

---

# **Voltus™-Fi Custom Power Integrity Solution User Guide**

**Product Version IC6.1.7  
July 2016**

---

© 2014-2016 Cadence Design Systems, Inc. All rights reserved.  
Printed in the United States of America.

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

**Trademarks:** Trademarks and service marks of Cadence Design Systems, Inc. (Cadence) 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 1-800-862-4522.

All other trademarks are the property of their respective holders.

**Restricted Print Permission:** This publication is protected by copyright and any unauthorized use of this publication may violate copyright, trademark, and other laws. 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. This statement grants you permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used solely for personal, informational, and noncommercial purposes;
2. The publication may not be modified in any way;
3. Any copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement; and
4. Cadence reserves the right to revoke this authorization at any time, and any such use shall be discontinued immediately upon written notice from Cadence.

**Disclaimer:** Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. The information contained herein is the proprietary and confidential information of Cadence or its licensors, and is supplied subject to, and may be used only by Cadence's customer in accordance with, a written agreement between Cadence and its customer. 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.

# Contents

---

<u>How This Manual Is Organized</u> .....	13
<u>Conventions Used in This Manual</u> .....	14
<u>Related Documents</u> .....	14
<u>Virtuoso Power System Documentation</u> .....	15
<u>Other Documentation</u> .....	15
<u>Additional Learning Resources</u> .....	15

## 1

<u>Product and Licensing Information</u> .....	17
<u>Voltus™-Fi Custom Power Integrity Solution Product Description</u> .....	18
<u>Voltus™-Fi Custom Power Integrity Solution-XL License</u> .....	19
<u>Checking Out Licenses for Voltus™-Fi Custom Power Integrity Solution-XL Product</u> .....	19
<u>Voltus-Fi-XL Compatibility with MMSIM and EXT Product Versions</u> .....	20
<u>Voltus-Fi-XL Compatibility with Voltus IC Power Integrity Solution</u> .....	20

## 2

<u>Getting Started</u> .....	23
<u>Product and Installation Information</u> .....	24
<u>Supported and Compatible Platform</u> .....	24
<u>Specifying the 64-Bit Version</u> .....	24
<u>Launching Voltus-Fi-XL</u> .....	26
<u>Running Voltus-Fi-XL from the GUI</u> .....	26
<u>Launching Voltus-Fi-XL from the Virtuoso® ADE L/ADE XL Window</u> .....	27
<u>Launching Voltus-Fi-XL from the Virtuoso® Layout Suite</u> .....	44
<u>Running Voltus-Fi-XL in the Batch Mode</u> .....	49
<u>Accessing Documentation and Help</u> .....	50
<u>Launching Cadence Help From the Command Prompt</u> .....	50
<u>Accessing Documentation and Help From the Voltus-Fi-XL GUI</u> .....	50
<u>Other Source of Information</u> .....	50

### 3

<u>Product Overview</u> .....	51
<u>Voltus-Fi-XL Product Overview</u> .....	52
<u>Key Product Features and Description</u> .....	52

### 4

<u>Data Preparation</u> .....	55
<u>Voltus-Fi-XL Flow Overview</u> .....	56
<u>Spectre® APS/XPS EMIR Flow Overview</u> .....	56
<u>Data Requirements for Voltus-Fi-XL</u> .....	59
<u>xDSPF Requirements</u> .....	60
<u>Simulation Requirements and Setup</u> .....	61
<u>Setup for Visualization of IR and EM Results</u> .....	76
<u>EM Rule File Requirements</u> .....	78
<u>Outputs of EMIR Analysis</u> .....	92

### 5

<u>xDSPF Generation using Quantus</u> .....	93
<u>Overview</u> .....	94
<u>Data Requirements for xDSPF Generation</u> .....	94
<u>Generating xDSPF</u> .....	94

### 6

<u>Static Current Analysis</u> .....	101
<u>Static Current Analysis</u> .....	102
<u>Static EMIR Flow in Spectre</u> .....	102
<u>Static Current Analysis Overview</u> .....	102
<u>Static Current Analysis Flow</u> .....	104
<u>Data Requirements for Static Current Calculation</u> .....	104
<u>Setting Up and Running Static Current Analysis</u> .....	104
<u>Static Current Analysis Report</u> .....	113

# 7

<u><a href="#">IR Drop Analysis Results</a></u> .....	115
<u><a href="#">Overview</a></u> .....	117
<u><a href="#">Signal Net IR Drop Analysis</a></u> .....	117
<u><a href="#">Power Gate Support</a></u> .....	118
<u><a href="#">Static EMIR Analysis</a></u> .....	119
<u><a href="#">Static Power Grid Solver</a></u> .....	120
<u><a href="#">Solid Shape Highlighting</a></u> .....	122
<u><a href="#">QRC Shape Database-Based Flow</a></u> .....	122
<u><a href="#">Displaying Finer Color Gradient for Selected Layers in IR and EM Plots</a></u> .....	123
<u><a href="#">Specifying Multiple Simulation Result Files for EMIR Analysis</a></u> .....	124
<u><a href="#">Viewing the IR Drop Analysis Results</a></u> .....	125
<u><a href="#">Displaying and Querying EMIR Results</a></u> .....	135
<u><a href="#">Viewing EMIR Violations in the Annotation Browser</a></u> .....	135
<u><a href="#">Querying EMIR Results</a></u> .....	139
<u><a href="#">Specifying the Display Options for IR/EM Plots</a></u> .....	139
<u><a href="#">Types of IR Drop Analysis Plots</a></u> .....	155
<u><a href="#">IR, IRAVG – IR Drop Plots</a></u> .....	156
<u><a href="#">RC, RCAVG, and RCRMS – Resistor Current Plots</a></u> .....	156
<u><a href="#">IV – Transistor Voltage Plot</a></u> .....	159
<u><a href="#">PI – Powergate Current Plot</a></u> .....	160
<u><a href="#">PV – Powergate Voltage Plot</a></u> .....	163
<u><a href="#">REffective – Effective Resistance Plot</a></u> .....	165
<u><a href="#">Plotting the Least Resistive Path</a></u> .....	166
<u><a href="#">The LRP Browser</a></u> .....	168
<u><a href="#">Shorting Layers for LRP Analysis</a></u> .....	173
<u><a href="#">Displaying LRP for the Tap on the Selected Marker</a></u> .....	174
<u><a href="#">Supporting LRP Analysis in Power-Gated Designs</a></u> .....	175
<u><a href="#">Calculating Effective Resistance between any Two Nodes on a Net</a></u> .....	178
<u><a href="#">Generating EMIR Analysis Reports in Voltus-Fi-XL</a></u> .....	182
<u><a href="#">Reports Generated using the EMIR Control File</a></u> .....	182
<u><a href="#">Default Reports Generated for EMIR Analysis</a></u> .....	183
<u><a href="#">Reports Generated for Multiple Bin Files for Different Nets from the Same Testbench</a></u> ..	189
<u><a href="#">Reports Generated for Multiple Bin Files from Different Testbenches in a Design</a></u> ..	189

## 8

<u>EM Analysis Results</u> .....	197
<u>Overview</u> .....	198
<u>Types of EM Analysis</u> .....	198
<u>Design Resistor EM Current Analysis</u> .....	200
<u>Viewing the EM Analysis Results</u> .....	200
<u>Displaying Current Directions for EM Plots</u> .....	209
<u>Reporting the Direction of Currents in the EM Result File</u> .....	210
<u>Generating EM Reports in Voltus-Fi-XL</u> .....	211
<u>Types of EM Analysis Plots</u> .....	211
<u>Tap Current Plots</u> .....	212
<u>Resistor Current Plots</u> .....	213
<u>RJ – Current Density – max</u> .....	215
<u>JAVG – Current Density – avg</u> .....	217
<u>JABSAVG – Current Density – avgabs</u> .....	219
<u>JACRMS – Current Density – rms</u> .....	221

## 9

<u>Self-Heating Effect Analysis</u> .....	225
<u>Overview</u> .....	226
<u>Data Requirements for SHE Analysis</u> .....	227
<u>Compatible Software Versions for SHE Analysis</u> .....	227
<u>Flow Overview</u> .....	228
<u>Performing SHE Analysis</u> .....	229
<u>Running PVS LVS</u> .....	229
<u>Running Quantus QRC Parasitic Extraction</u> .....	229
<u>Running Post-Layout Simulation using Spectre® APS</u> .....	230
<u>Viewing SHE Analysis Plots in Voltus-Fi-XL</u> .....	231
<u>Types of Self-Heating Analysis Plots</u> .....	232
<u>Thermal-DeltaT Regions</u> .....	233
<u>Thermal-DeltaT Tiles</u> .....	234
<u>RJ SH - Current Density - max (SH)</u> .....	235
<u>JAVG_SH - Current Density - avg (SH)</u> .....	235
<u>JACRMS_SH - Current Density - rms (SH)</u> .....	235

<u>Re-Evaluation of EM Limit Based on dTemp</u>	236
<u>EMIR Control File Options for Self-Heating Effect Analysis</u>	236
<u>Running SHE Analysis in the Batch Mode</u>	239
<u>Generating Reports for SHE Analysis</u>	239

## 10

<u>Structural Analysis</u>	243
<u>Overview</u>	244
<u>Checks Performed for Structural Analysis</u>	244
<u>Top Via Missing</u>	245
<u>Bottom Via Missing</u>	246
<u>Missing Via Connections</u>	246
<u>Via Coverage Ratio</u>	248
<u>Skewed Via Ratio</u>	249
<u>Running Structural Analysis</u>	250
<u>Examples of Structural Analysis Results</u>	255
<u>Show Unconnected from Top</u>	255
<u>Show Unconnected from Bottom</u>	256
<u>Bottom / Top Via Ratio</u>	257
<u>Missing Via Connections</u>	258

## 11

<u>What-If EMIR Analysis</u>	261
<u>Overview</u>	262
<u>ECO Commands Supported in Voltus-Fi-XL</u>	263
<u>Performing What-If Analysis in Voltus-Fi-XL</u>	264
<u>Pre-Filling the Run Simulation Form</u>	272
<u>Output</u>	273
<u>Options for Performing What-If Analysis</u>	273

## 12

<u>Power-Grid View Generation</u>	281
<u>Overview</u>	282
<u>Data Requirements and Flow for PGV Creation</u>	282

<u>Open Access (OA)-Based Library Generation</u>	286
<u>Generating Power-Grid Views</u>	286
<u>Outputs of PGV Generation</u>	289
<u>Verifying the PGVs</u>	290

## 13

<b><u>Batch Mode Execution</u></b>	291
<u>Overview</u>	293
<u>Batch Command for xDSPF Generation</u>	294
<u>extract_xdspf</u>	294
<u>Batch Commands for Static Current Analysis</u>	300
<u>ground_pins</u>	301
<u>power_pin</u>	302
<u>run_static_simulation</u>	303
<u>setup_simulation</u>	306
<u>Batch Commands for IR Reports</u>	308
<u>load_ir_results</u>	308
<u>print_ir_report</u>	308
<u>print_rlp_report</u>	312
<u>pin_2_pin_res</u>	313
<u>Batch Commands for EM Reports</u>	315
<u>load_em_results</u>	315
<u>print_em_report</u>	316
<u>Batch Commands for SHE Analysis</u>	320
<u>load_view</u>	320
<u>print_thermal_report</u>	321
<u>Batch Commands for PGV Generation</u>	323
<u>create_pgv</u>	323
<u>set_mmsim_pgv_nets</u>	326
<u>Batch Commands for Querying</u>	328
<u>search_res_id</u>	328
<u>res_id_info</u>	329
<u>search_node_id</u>	332
<u>node_id_info</u>	333

### 14

<b>Environment Variables</b> .....	335
<u>Setting Up the Voltus-Fi-XL Environment Variables</u> .....	336
<u>VFI_LIC_QUEUE</u> .....	336
<u>VOLTUSFI_DELTA_T</u> .....	336
<u>VOLTUSFI_DYNAMIC_AC_PEAK</u> .....	336
<u>VOLTUSFI_SPLIT_ACDC_RULES</u> .....	337

### 15

<b>Variables</b> .....	339
<u>Overview</u> .....	341
<u>Using the set_variable Command in the Voltus-Fi-XL Command File</u> .....	341
<u>set_variable</u> .....	341
<u>Using the emirutil Command in the EMIR Control File</u> .....	342
<u>Setting the Configuration Variable in the Batch Mode</u> .....	342
<u>Using the Variables in the Voltus-Fi-XL GUI</u> .....	343
<u>applyRThreshold</u> .....	344
<u>consolidatedReport</u> .....	344
<u>deltaT</u> .....	344
<u>disableLayoutScale</u> .....	344
<u>dynamicACPeak</u> .....	345
<u>geounit</u> .....	346
<u>geounit_wl</u> .....	346
<u>geounit_xy</u> .....	346
<u>optimisticEMRuleSelection</u> .....	346
<u>powerRailSupplyNets</u> .....	346
<u>powerRailRules</u> .....	347
<u>powerRailGroundNets</u> .....	347
<u>process_scale</u> .....	347
<u>reportUnmatchedRuleResistor</u> .....	348
<u>rmsMultiplierPin</u> .....	348
<u>rmsMultiplierSignal</u> .....	348
<u>shEMEffect</u> .....	348
<u>shHeatSink</u> .....	348

<u>shParamFile</u>	349
<u>shXMLFile</u>	349
<u>splitACDCRules</u>	349
<u>Tj</u>	349
<u>Tlife</u>	349
<u>useEncryptedWidth</u>	349
<u>useRecoveryFactor</u>	350
<u>viaCountFromResistance</u>	350

## 16

<u>SKILL Functions</u>	351
<u>Overview</u>	352
<u>Voltus-Fi-XL SKILL Functions</u>	352
<u>vsaCreateMarker</u>	354
<u>vsaOpenLayout</u>	355
<u>vsaLoadNets</u>	356
<u>vsaLoadSecondaryLayers</u>	357
<u>vsaSetEMIRConfig</u>	358
<u>vsaSetEMResultsFile</u>	359
<u>vsaSetEMLayerMapFileName</u>	360
<u>vsaSetEMTechFileName</u>	361
<u>vsaSetEMTypes</u>	362
<u>vsaSetDfiiLayerMapFileName</u>	363
<u>vsaSetQRCData</u>	364
<u>vsaSelectFailedNets</u>	365
<u>vsaSetEMOnlyICTFile</u>	366
<u>vsaSetEMPlot</u>	367
<u>vsaLoadEM</u>	369
<u>vsaSetIRRResultsFile</u>	370
<u>vsaSetIRThreshold</u>	371
<u>vsaSetIRPlot</u>	372
<u>vsaSetLRPShortedLayers</u>	374
<u>vsaLoadIR</u>	375

**17**

<b><u>File Formats</u></b> .....	377
<u>Static Current Analysis Files</u> .....	378
<u>Activity File</u> .....	379
<u>Clock File</u> .....	380
<u>Value Change Dump (VCD) File</u> .....	381
<u>Power File</u> .....	384
<u>Rail Analysis Files</u> .....	385
<u>ICT File</u> .....	386
<u>EM Only ICT File</u> .....	389
<u>EM Data File</u> .....	391
<u>Layer Map File</u> .....	393
<u>DFII Layer Map File</u> .....	394
<u>Self-Heating Effect Analysis File</u> .....	395
<u>shParamFile</u> .....	395
<u>Power-Grid View Creation Files</u> .....	397
<u>Format and Contents of Trigger File</u> .....	397
<u>LEF Layermap File</u> .....	400
<u>PGDB Layermap File</u> .....	401
<u>Common Command Language (CCL) File</u> .....	402

**18**

<b><u>Appendix A – Third-Party DSPF</u></b> .....	405
<u>Scenario – Missing Number of Vias</u> .....	405
<u>Resolving the Missing \$N Issue in Third-Party DSPF</u> .....	406

<b><u>Index</u></b> .....	407
---------------------------	-----

## **Voltus-Fi Custom Power Integrity Solution User Guide**

---

# About This Manual

---

This manual describes the Voltus™-Fi Custom Power Integrity Solution-XL (Voltus-Fi-XL) product that is used to perform transistor-level power integrity analysis, which includes multi-mode simulation (MMSIM) electromigration and voltage drop (EMIR) analysis for transistor designs.

The target designs for Voltus-Fi-XL include analog, analog/mixed-signal (AMS), and custom digital designs of large sizes that are created using Virtuoso.

## How This Manual Is Organized

This manual covers the following topics:

- [Product and Licensing Information](#)
- [Getting Started](#)
- [Product Overview](#)
- [Data Preparation](#)
- [xDSPF Generation using Quantus](#)
- [Static Current Analysis](#)
- [IR Drop Analysis Results](#)
- [EM Analysis Results](#)
- [Self-Heating Effect Analysis](#)
- [Structural Analysis](#)
- [What-If EMIR Analysis](#)
- [Power-Grid View Generation](#)
- [Batch Mode Execution](#)
- [Environment Variables](#)
- [Variables](#)

- [SKILL Functions](#)
- [File Formats](#)
- [Appendix A – Third-Party DSPE](#)

## Conventions Used in This Manual

This document uses the following syntactical conventions:

- Commands and parameters are given in *Courier* font:  
`command_name`
- Menu commands and the fields in dialog boxes in the graphical user interface are given in *Helvetica italic* font:  
*File – Save*  
An en dash (–) separates the menu name and the command name.
- Variables for which you are to substitute a value are given in *Courier italic* font:  
`filename, cellname, layer_name`
- File names are given in *Courier* font:  
`library.lib`
- Angle brackets enclose optional parameters:  
`<options>`
- Vertical bars (|) in commands indicate choices.
- Square brackets indicate that you must select one of the choices:  
`[a | b | c]`
- Use white space (tabs or spaces) to separate a command and its arguments.

## Related Documents

For more information about related products, see the following documents. You can access these and other Cadence documents using the Cadence Help online documentation system.

## **Virtuoso Power System Documentation**

- For information about how to perform power-grid and electromigration analysis using Power IR/EM, see the [\*Power IR/EM User Guide\*](#).
- For information about important Cadence Change Requests (CCRs) for Power IR/EM, including solutions for working around known problems, see [\*Power IR/EM Known Problems and Solutions\*](#).
- For information about new and enhanced features, see the [\*Virtuoso Power System What's New\*](#).

## **Other Documentation**

- [\*Virtuoso® Design Environment User Guide\*](#)
- [\*Virtuoso® Analog Design Environment L User Guide\*](#)
- [\*Virtuoso® Spectre® Circuit Simulator and Accelerated Parallel Simulator User Guide\*](#)
- [\*Virtuoso® Layout Suite XL User Guide\*](#)

## **Additional Learning Resources**

Cadence provides various [Rapid Adoption Kits](#) (RAKs) that demonstrate how to use Virtuoso applications in your design flows. These kits contain design databases and instructions on how to run the design flow.

The following RAK is available for Voltus-Fi-XL: Voltus-Fi EMIR Analysis Workshop

**Note:** The link provided in this section opens in a separate web browser window when clicked in Cadence Help.

# **Voltus-Fi Custom Power Integrity Solution User Guide**

## About This Manual

---

## **Product and Licensing Information**

---

- [Voltus™-Fi Custom Power Integrity Solution Product Description](#) on page 18
- [Voltus™-Fi Custom Power Integrity Solution-XL License](#) on page 19
- [Checking Out Licenses for Voltus™-Fi Custom Power Integrity Solution-XL Product](#) on page 19
  - [Voltus-Fi-XL Compatibility with MMSIM and EXT Product Versions](#) on page 20
  - [Voltus-Fi-XL Compatibility with Voltus IC Power Integrity Solution](#) on page 20

## **Voltus™-Fi Custom Power Integrity Solution Product Description**

This release of the Voltus™-Fi Custom Power Integrity Solution (Voltus-Fi) software includes the following product:

**Table 1-1 Voltus-Fi Product**

Product Name	Abbreviation	Product Number	Description
Voltus-Fi Custom Power Integrity Solution-XL	VTS-Fi-XL	VTS500	<p>Performs multi-mode simulation (MMSIM) electromigration and voltage drop (EMIR) analysis for power and signal nets using the simulation database generated by Spectre® APS/XPS. It is tightly integrated within Virtuoso.</p> <p>The Voltus-Fi Custom Power Integrity Solution-XL license is enabled for multi-CPU usage and each license can run up to 4 CPUs.</p>

**Note:** The base license of Voltus-Fi Custom Power Integrity Solution-XL can run up to 4 CPUs, which you can stack to access more CPUs. This means if you purchase two Voltus-Fi Custom Power Integrity Solution-XL licenses, you can run up to 8 CPUs.

## **Voltus™-Fi Custom Power Integrity Solution-XL License**

When you run a command to use a product, a license is checked out. Each product has a unique license string (also called a license key).

The following table lists the Voltus-Fi product name and the corresponding license string.

**Table 1-2 Product Name and Corresponding License String**

Product Name	License String
Voltus-Fi Custom Power Integrity Solution-XL (VTS-Fi-XL)	Voltus_Power_Integrity_Fi_XL



With the Voltus-Fi Custom Power Integrity Solution-XL license, you can also run the Virtuoso Power System (VPS) product, VPSL or Power IR/EM.

For more information about the above product licenses, see the “Product and Licensing Information” chapter in the following manual:

- ❑ [Power IR/EM User Guide](#)

## **Checking Out Licenses for Voltus™-Fi Custom Power Integrity Solution-XL Product**

The Voltus-Fi Custom Power Integrity Solution-XL product is integrated in the Virtuoso® Design Environment. The results of the analyses performed by Voltus-Fi Custom Power Integrity Solution-XL are displayed in the Virtuoso GUI, overlaid on the design layout.

Running Voltus-Fi Custom Power Integrity Solution-XL GUI requires check-out availability of the following license set:

- Cadence® Design Framework II (DFII) license (known as the “product 111 license”). Once a DFII license check is performed successfully, you will be able to access the Voltus-Fi Custom Power Integrity Solution-XL functionality.
- Virtuoso® Analog Design Environment XL (ADE XL)

**Note:** For information on licenses for the Virtuoso® Design Environment, see “Virtuoso Design Environment Licensing Setup” in the [Virtuoso Software Licensing and](#)

## **Voltus-Fi Custom Power Integrity Solution User Guide**

### Product and Licensing Information

---

#### Configuration User Guide.

The Voltus-Fi Custom Power Integrity Solution-XL product leverages existing Cadence tools for performing simulation but does not include licenses for these tools. These licenses must be purchased separately.

You require a license for one of the following simulators:

- Virtuoso® Accelerated Parallel Simulator (APS) with Virtuoso MMSIM Power option
- Spectre® Extensive Partitioned Simulator (XPS) with Virtuoso MMSIM Power option

**Note:** The Virtuoso MMSIM Power option is available from MMSIM13.1 release onwards. This option enables EMIR analysis with Spectre® APS and Spectre® XPS base products.

### **Voltus-Fi-XL Compatibility with MMSIM and EXT Product Versions**

The table below provides the mapping between the Voltus-Fi-XL product version and the corresponding EXT and MMSIM versions that are compatible with it. It is recommended that you use these version specifications to avoid erroneous results.

Voltus-Fi-XL Version	EXT Versions	MMSIM Versions
Voltus-Fi-XL 6.1.7	EXT 14.2.x	MMSIM 14.1.0 ISRx
Voltus-Fi-XL 6.1.7 ISR1	EXT 15.1.x	MMSIM 15.1.0 ISRx
Voltus-Fi-XL 6.1.7 ISR2		
Voltus-Fi-XL 6.1.7 ISR3	EXT 14.2.x	MMSIM 14.1.0 ISRx
Voltus-Fi-XL 6.1.7 ISR4	EXT 15.1.x EXT 15.2.x	MMSIM 15.1.0 ISRx
Voltus-Fi-XL 6.1.7 ISR5	EXT 15.1.x EXT 15.2.x	MMSIM 14.1.0 ISRx MMSIM 15.1.0 ISRx

### **Voltus-Fi-XL Compatibility with Voltus IC Power Integrity Solution**

The table below provides the mapping between the Voltus-Fi-XL product version and the corresponding Voltus IC Power Integrity Solution (Voltus) version that is compatible with it. It is recommended that you use this version specification to avoid erroneous results.

## **Voltus-Fi Custom Power Integrity Solution User Guide**

### Product and Licensing Information

---

<b>Voltus-Fi-XL Version</b>	<b>Voltus Version</b>
Voltus-Fi-XL 6.1.7 ISR5	Voltus IC Power Integrity Solution 16.1

**Voltus-Fi Custom Power Integrity Solution User Guide**  
Product and Licensing Information

---

## **Getting Started**

---

- [Product and Installation Information](#) on page 24
- [Supported and Compatible Platform](#) on page 24
- [Specifying the 64-Bit Version](#) on page 24
- [Launching Voltus-Fi-XL](#) on page 26
  - [Running Voltus-Fi-XL from the GUI](#) on page 26
    - [Starting the Virtuoso® Design Environment](#) on page 26
    - [Launching Voltus-Fi-XL from ADE L](#) on page 42
    - [Launching Voltus-Fi-XL from the Virtuoso® Layout Suite](#) on page 44
  - [Running Voltus-Fi-XL in the Batch Mode](#) on page 49
- [Accessing Documentation and Help](#) on page 50

## Product and Installation Information

The Voltus-Fi-XL product is launched within the Virtuoso Platform. To use this product, you must have the Virtuoso® IC 6.1.6 release version.

To check whether this product is available in your Virtuoso® Design Environment, type the following in the UNIX window, shell, or xterm:

```
vfibatch -v
```

This prints the product version available in your Virtuoso Design Environment.

For basic information and procedures required to install Cadence® products, see [Cadence Installation Guide](#).

For information about how to configure the Virtuoso® Design Environment, see “Setting up the Virtuoso Software”, in [Virtuoso Software Licensing and Configuration User Guide](#).

## Supported and Compatible Platform

Voltus-Fi-XL is supported only on the following platform:

- Linux (only 64-bit)

## Specifying the 64-Bit Version

You can run Voltus-Fi-XL in only 64-bit mode. To specify the 64-bit version, you need to set the CDS\_AUTO\_64BIT environment variable before starting the software. For more information see the section below.

### Using the CDS\_AUTO\_64BIT Environment Variable

To run 64-bit versions of all or some applications, complete the following steps before starting the software:

1. If you are using the `lnx86` operating system, verify that it supports 64-bit applications.
2. Set the CDS\_AUTO\_64BIT environment variable.

For example,

- To run all applications in 64-bit mode, run the following command:

# **Voltus-Fi Custom Power Integrity Solution User Guide**

## Getting Started

---

```
setenv CDS_AUTO_64BIT ALL
```

- ❑ To run specific applications in 64-bit mode, such as UltraSim and Voltus-Fi-XL, and all other applications in 32-bit mode, run the following command:

```
setenv CDS_AUTO_64BIT ultrasim:vfibatch
```

## **Launching Voltus-Fi-XL**

You can run Voltus-Fi-XL interactively from the GUI or you can execute it as a batch command.

Both methods are detailed in this chapter.

- [Running Voltus-Fi-XL from the GUI](#)
- [Running Voltus-Fi-XL in the Batch Mode](#)

### **Running Voltus-Fi-XL from the GUI**

You can run Voltus-Fi-XL from the Virtuoso® Design Environment in two ways. You can run it either from the Virtuoso® Analog Design Environment (ADE) L/ADE XL, or from the Virtuoso® Layout Suite. This information is covered in detail in the following sections.

- [Starting the Virtuoso® Design Environment](#)
- [Launching Voltus-Fi-XL from the Virtuoso® ADE L/ADE XL Window](#)
- [Launching Voltus-Fi-XL from the Virtuoso® Layout Suite](#)

#### **Starting the Virtuoso® Design Environment**

To launch Voltus-Fi-XL either from Virtuoso® ADE L/ADE XL or Virtuoso® Layout Suite, you first need to start the design environment.

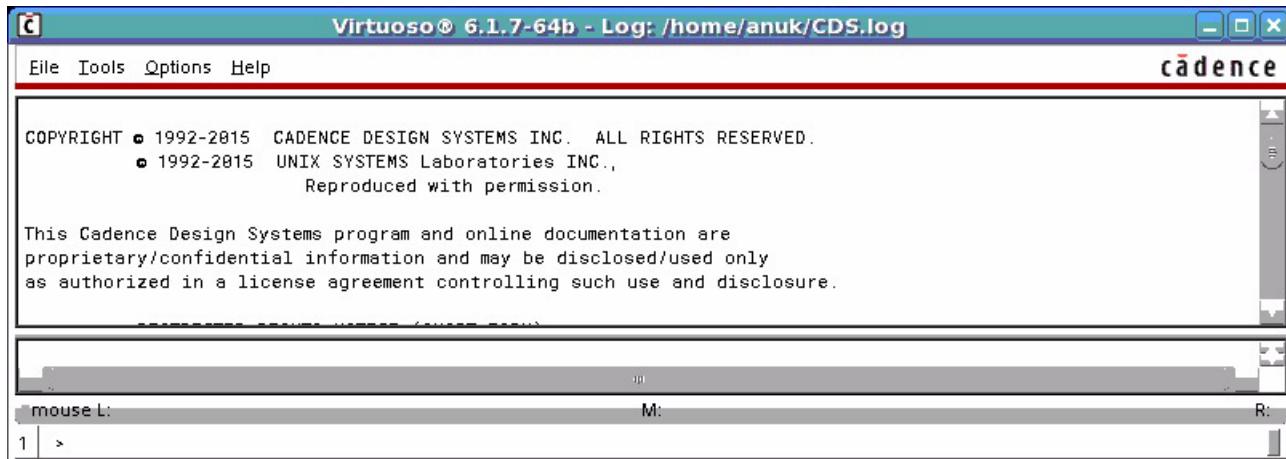
After setting up the Virtuoso® Design Environment, you can start Virtuoso by typing the following in the UNIX window, shell, or xterm:

`virtuoso &`

## Voltus-Fi Custom Power Integrity Solution User Guide

### Getting Started

The Command Interpreter Window (CIW) opens. You interact with the design environment from the CIW.



The title bar of the CIW contains the following information:

- The name of the workbench you are running (for example, `virtuoso`), and
- The path to the log file (`CDS.log`) that records the ongoing events of the design session

The contents of the log file are displayed in the output area.

### Launching Voltus-Fi-XL from the Virtuoso® ADE L/ADE XL Window

This method of launching Voltus-Fi-XL GUI involves opening the schematic view of the design in Virtuoso® Schematic Editor L, launching the ADE L/ADE XL window, setting up and running the simulation using the Spectre® APS/XPS simulators, creating the simulation database, and then launching Voltus-Fi-XL for layout analysis. The simulation database is used as input for running the EMIR analysis in Voltus-Fi-XL.

#### *Important*

The steps for launching Voltus-Fi-XL using ADE L are detailed below. The same steps can be used to launch Voltus-Fi-XL using ADE XL.

**Note:** To perform EMIR analysis in Voltus-Fi-XL using ADE L, ensure that you have set the following environment variable in your setup file:

```
setenv MMSIM_ADE_EMIR true
```

To run Voltus-Fi-XL from the ADE L window, perform the steps detailed in the following sections:

# Voltus-Fi Custom Power Integrity Solution User Guide

## Getting Started

---

- [Opening the Simulation Schematic View from the CIW](#)
- [Opening the ADE L Window](#)
- [Setting up a Simulation in the ADE L Window for Voltus-Fi-XL](#)
- [Setting up EMIR Analysis for Voltus-Fi-XL in the ADE L Window](#)
- [Running the Simulation in ADE L Window](#)
- [Launching Voltus-Fi-XL from ADE L](#)

### **Opening the Simulation Schematic View from the CIW**

The first step in this flow is to open the simulation schematic view of your design in the Virtuoso® Schematic Editor L. For this, use either the *File* menu or the *Tools* menu of the CIW.

#### ***Using the File Menu of CIW***

For information about how to open the Virtuoso® Schematic Editor L from the *File* menu in the CIW, see the “Working with Cellviews” chapter in [Virtuoso Design Environment User Guide](#).

#### ***Using the Tools Menu of CIW***

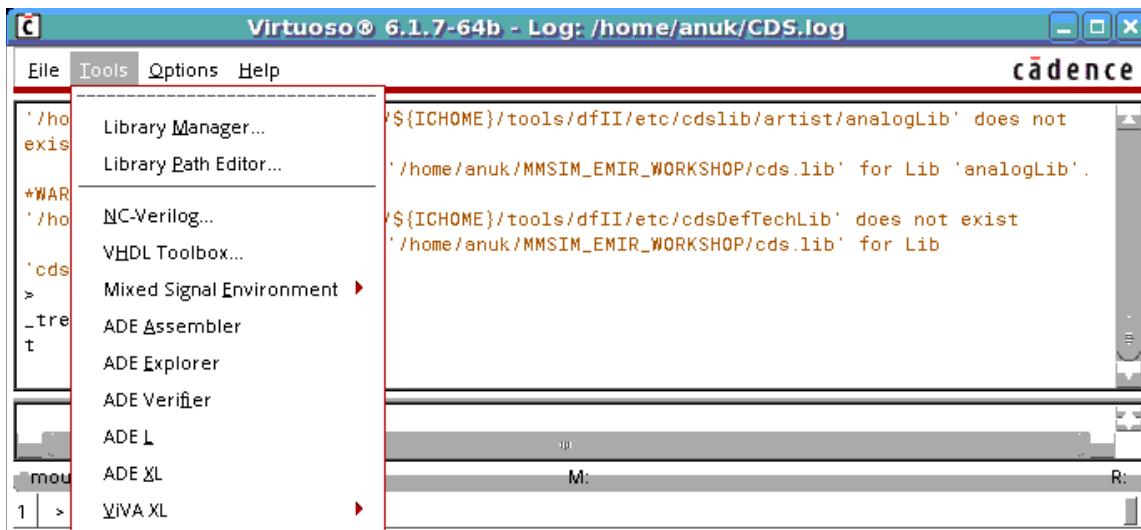
To open the Schematic Editor L from the *Tools* menu, perform the following steps:

- Choose *Tools – Library Manager*.

# Voltus-Fi Custom Power Integrity Solution User Guide

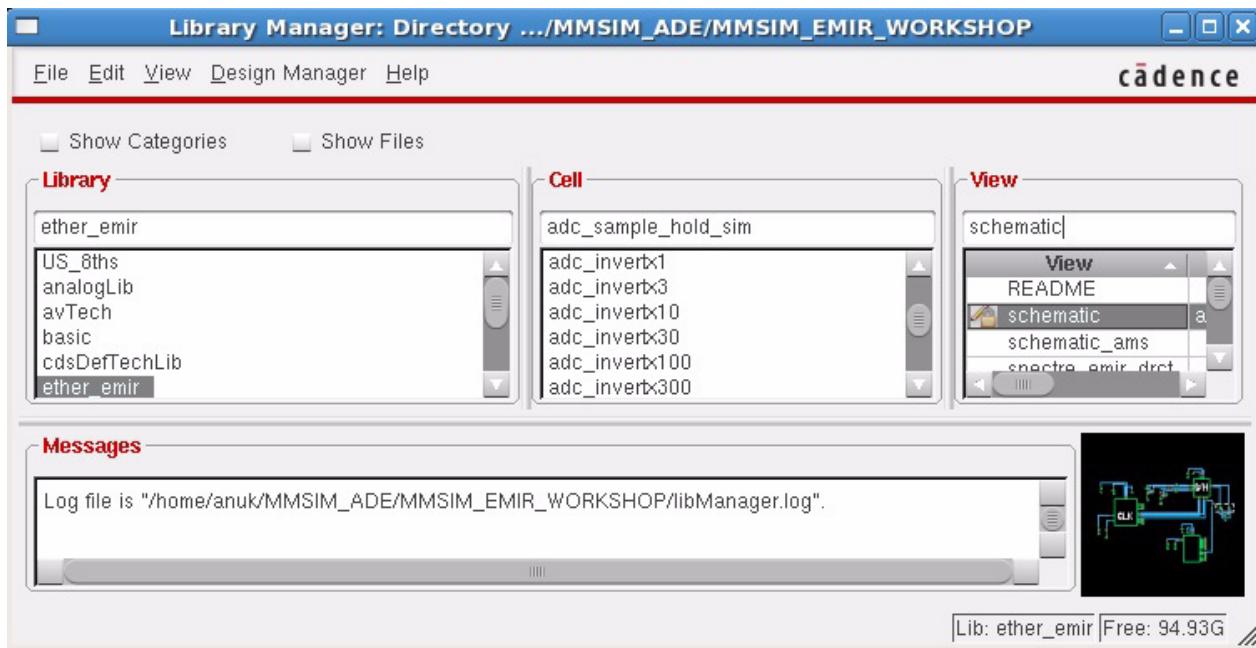
## Getting Started

**Figure 2-1 Opening the Library Manager Form**



The Library Manager form opens. This form is shown below.

**Figure 2-2 Library Manager Form**

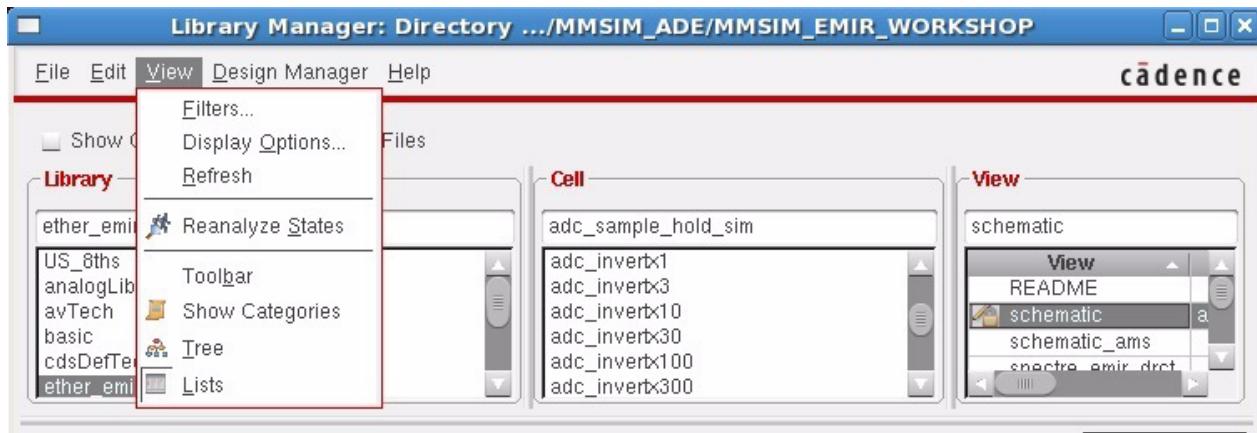


- In the Library Manager form, you can choose to display library information (library, cell, view, file, category) using either list boxes (in *View – Lists* mode) or a hierarchical tree structure (in *View – Tree* mode). This is shown in the figure below.

## Voltus-Fi Custom Power Integrity Solution User Guide

### Getting Started

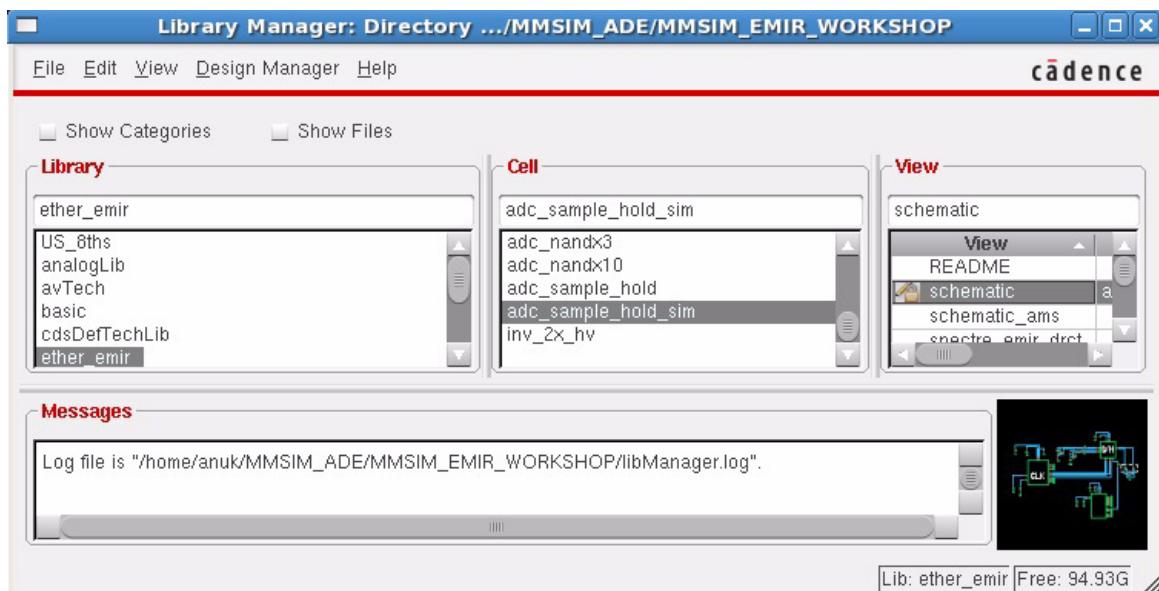
**Figure 2-3 Library Manager – View Tree and Lists Mode**



The information provided below is for *View – Lists* mode.

- To select a library and its corresponding cell and view on the Library Manager form, click the item names in the order: *Library*, *Cell*, and *View*. Select schematic in the *View* section. This is shown in the figure below.

**Figure 2-4 Library Manager – Specifying the Library, Cell, and Simulation Schematic View**



You can select a specific item that is not visible in the list box by typing the first part of the name in the active field at the top of the list box. As you type, the list scrolls to any matching names.

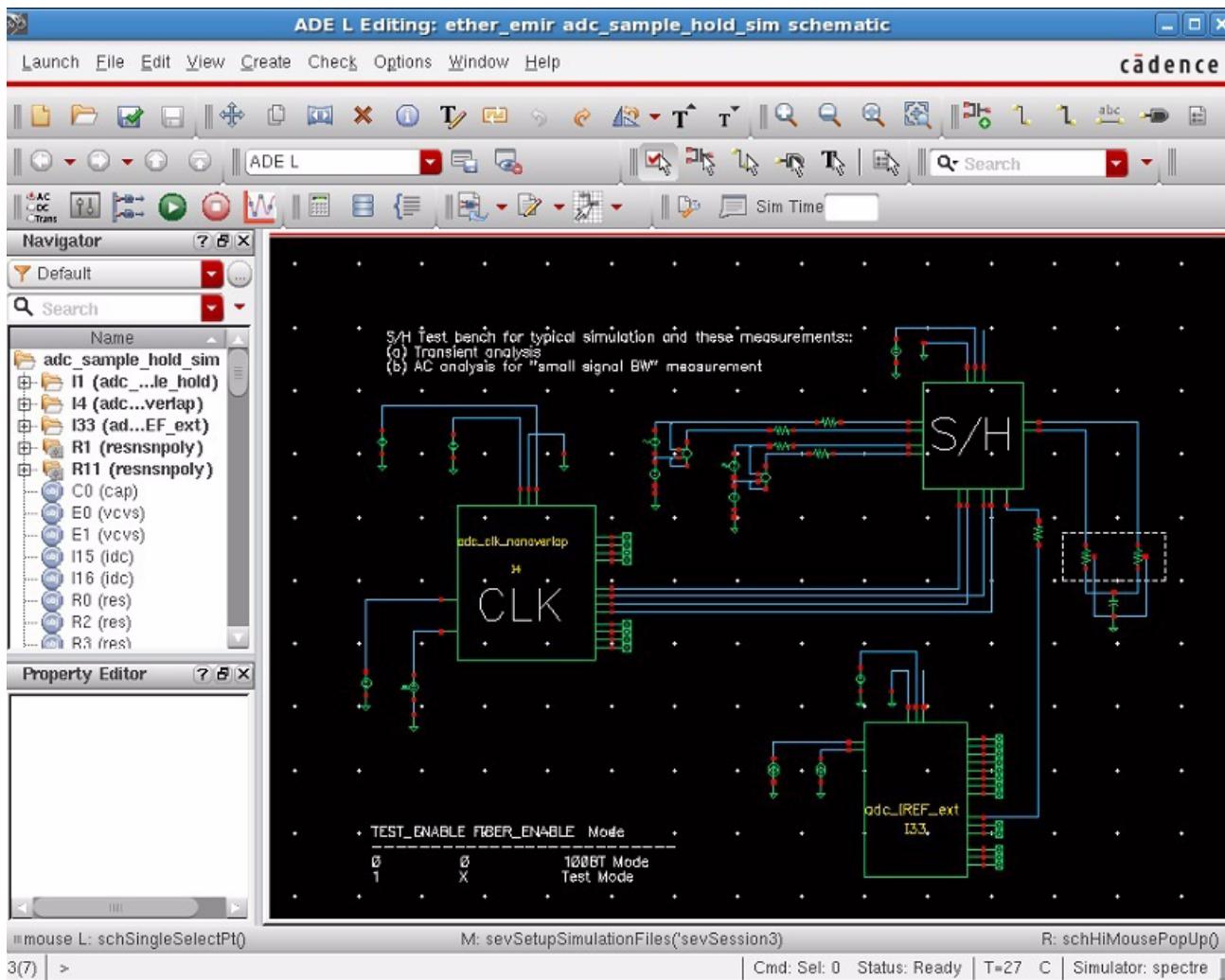
# Voltus-Fi Custom Power Integrity Solution User Guide

## Getting Started

**Note:** For more information on the Library Manager, see [Cadence Library Manager User Guide](#).

Once you have selected the simulation schematic view, double-click the view to open the Virtuoso Schematic Editor L. This is shown below.

**Figure 2-5 Opening the Virtuoso Schematic Editor L**



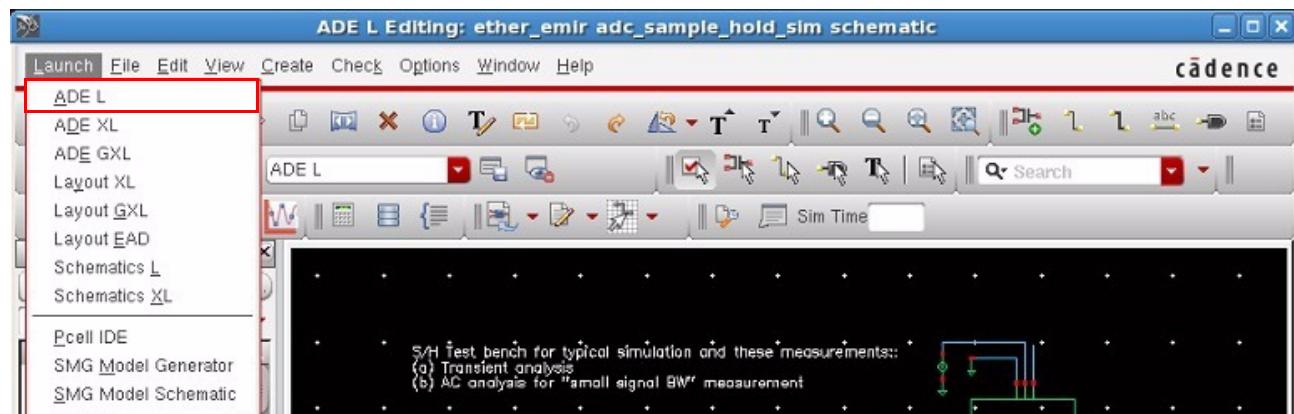
### Opening the ADE L Window

After opening the Virtuoso Schematic Editor L, you can open the ADE L window or the simulation window from the *Launch* menu. This is shown below.

# Voltus-Fi Custom Power Integrity Solution User Guide

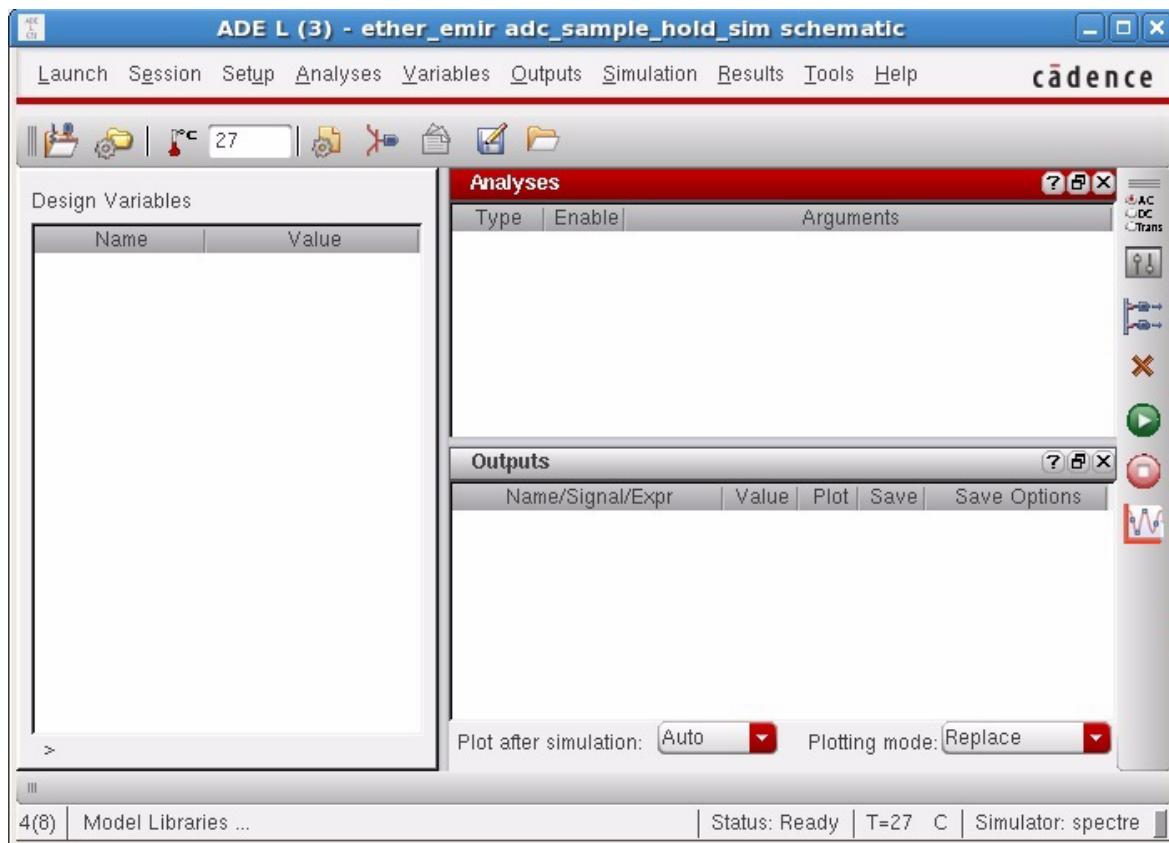
## Getting Started

**Figure 2-6 Opening the ADE L Window**



In the *Launch* menu of the Virtuoso Schematic Editor L, click *ADE L*. The ADE L window opens. This is shown below.

**Figure 2-7 The ADE L Window**



## Voltus-Fi Custom Power Integrity Solution User Guide

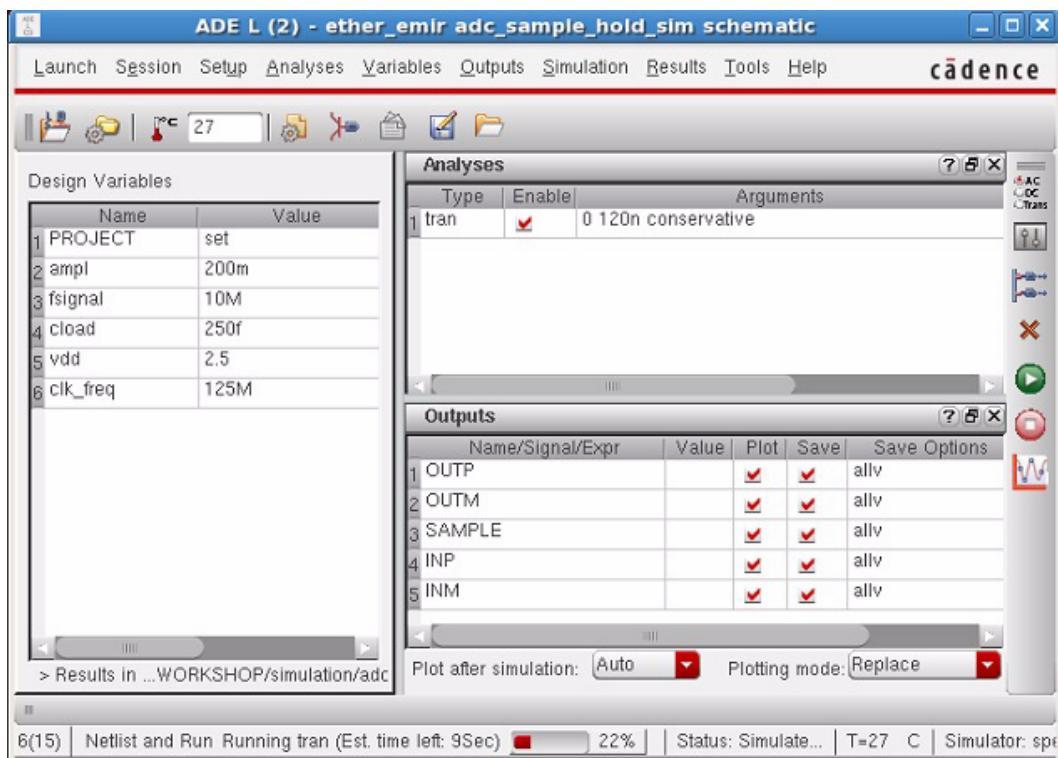
### Getting Started

Using the ADE L or the simulation window, you can run different simulations for a design. You can choose a simulator and the types of analyses you want to run for the design. You can also specify the design variables that you want to use for simulations. When you run a simulation, the results are saved in predefined formats and can be used by other tools such as the Virtuoso Visualization and Analysis tool for further analysis.

For detailed information about how to set up the ADE L environment for simulations, see the “Environment Setup” chapter in the *Virtuoso® Analog Design Environment L User Guide*.

After you have specified the settings in the ADE L window, the window will be populated as shown below.

**Figure 2-8 The ADE L Window with the Specified Settings**



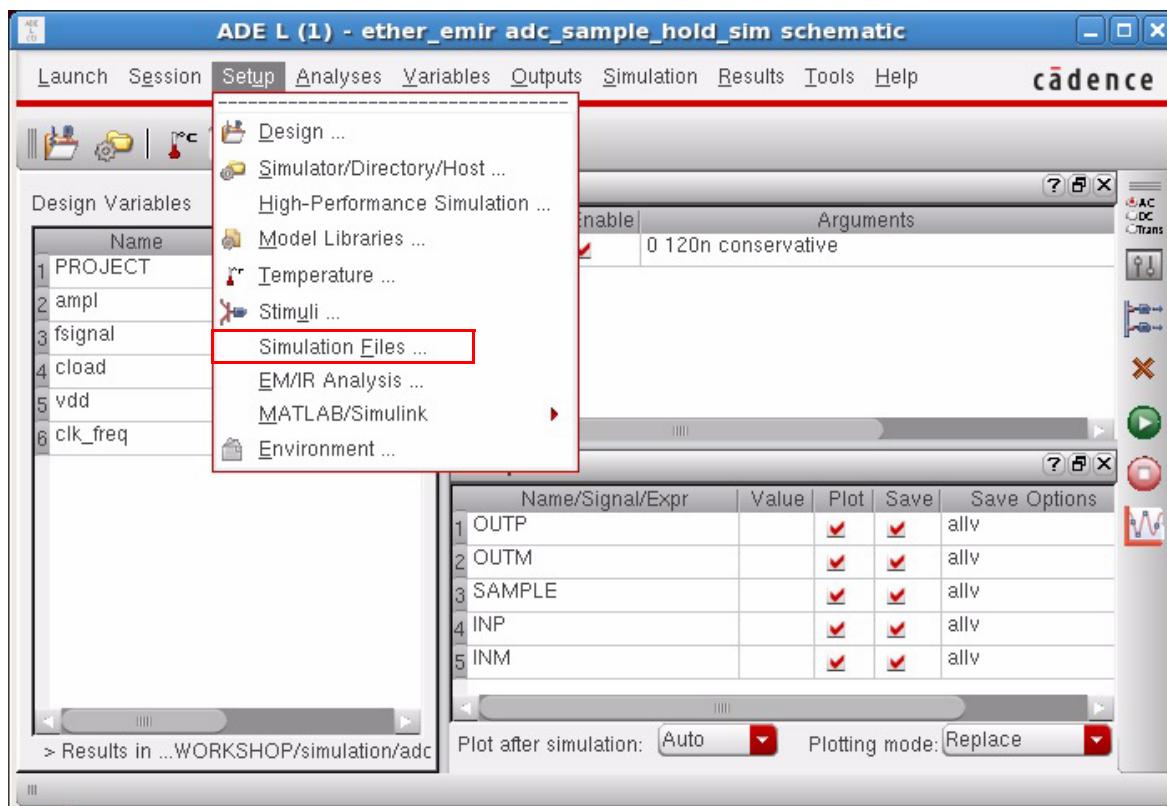
For performing EMIR analysis using Voltus-Fi-XL, you need to specify simulation settings and the EMIR analysis settings in the ADE L window. These are detailed in the subsequent sections.

## Setting up a Simulation in the ADE L Window for Voltus-Fi-XL

Before you run simulation, specify your simulation settings.

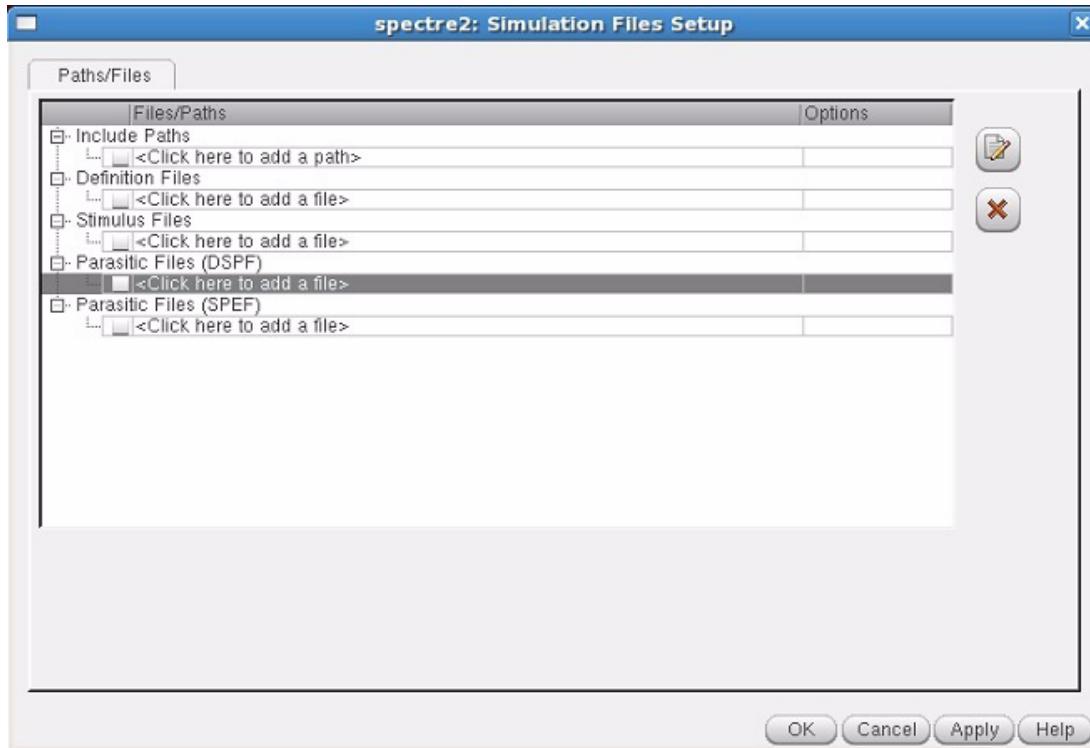
- In the Main menu of ADE L, click *Setup* and then *Simulation Files*. This is shown below.

**Figure 2-9 Setting up a Simulation for Voltus-Fi-XL in the ADE L Window**



- The Simulation Files Setup form opens. This form is shown below.

**Figure 2-10 Simulation Files Setup Form**

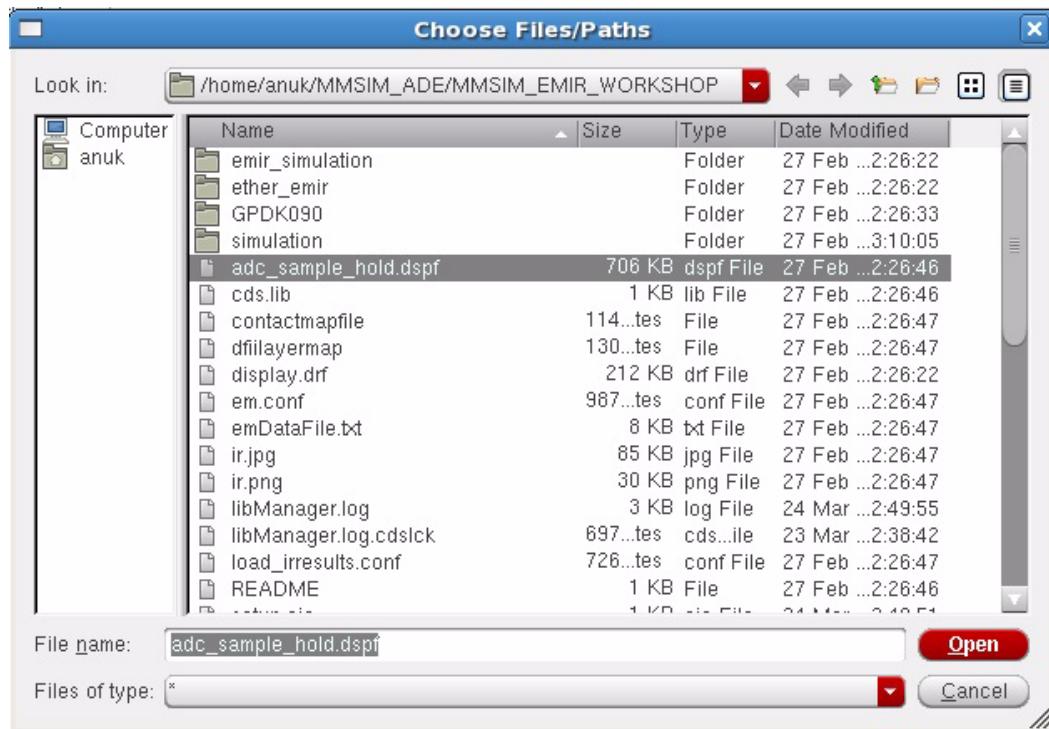


- In this form, specify the following files:
  - **The UTI commands file:** In the *Definition Files* tree, click where it says *<Click here to add a file>* and type the path and the filename for the UTI commands, or click the browse button to select and add the file using the Choose Files/Paths form. This file is required for generating power-grid views in Voltus-Fi-XL. For details of the syntax of the UTI commands to be included in the UTI command file, see [Data Requirements and Flow for PGV Creation](#) in the “Power-Grid View Generation” chapter.
  - **The DSPF file:** In the *Parasitic Files (DSPF)* tree, click where it says *<Click here to add a file>* and type the path and the filename you want to specify, or click the browse button to select and add the file using the Choose Files/Paths form. This form is shown below.

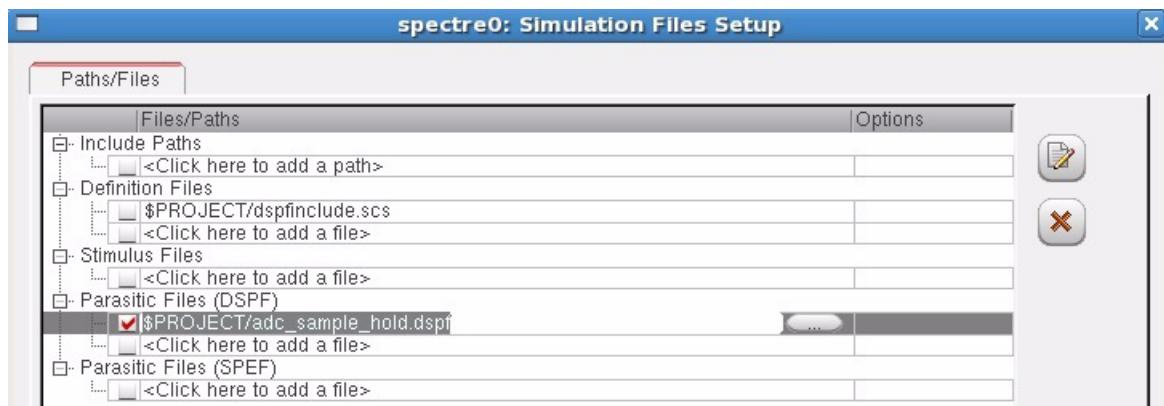
## Voltus-Fi Custom Power Integrity Solution User Guide

### Getting Started

**Figure 2-11 The Choose Files/Paths Form**



- In this form, select the Detailed Standard Parasitic Format (DSPF) file and click *Open*. The file is added in the Simulation files Setup form. You can also edit the file using the edit button provided in the form.



### Setting up EMIR Analysis for Voltus-Fi-XL in the ADE L Window

The next step is to specify the EMIR settings for Spectre® APS. These are required to perform EMIR analysis in Voltus-Fi-XL.

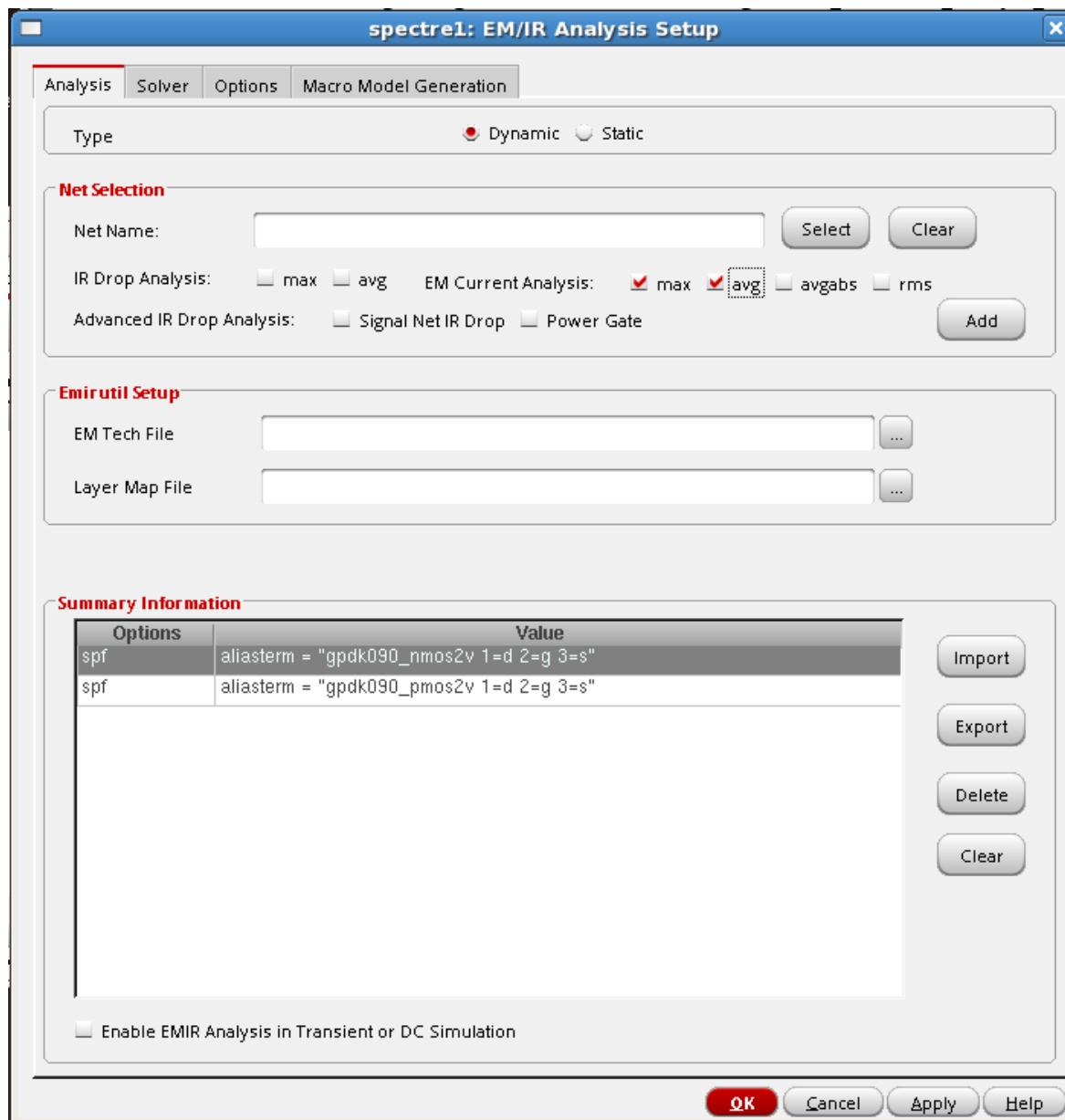
# Voltus-Fi Custom Power Integrity Solution User Guide

## Getting Started

**Note:** The EM/IR Analysis option is available in the Setup menu of the ADE L window only if you have set the `MMSIM_ADE_EMIR` environment variable to true:

- In the Main menu of ADE L, click *Setup* and then click *EM/IR Analysis*. The EM/IR Analysis Setup form opens. This form has three tabs, *Analysis*, *Solver*, and *Options*. This form is shown below.

**Figure 2-12 EM/IR Analysis Setup Form – Analysis Tab**

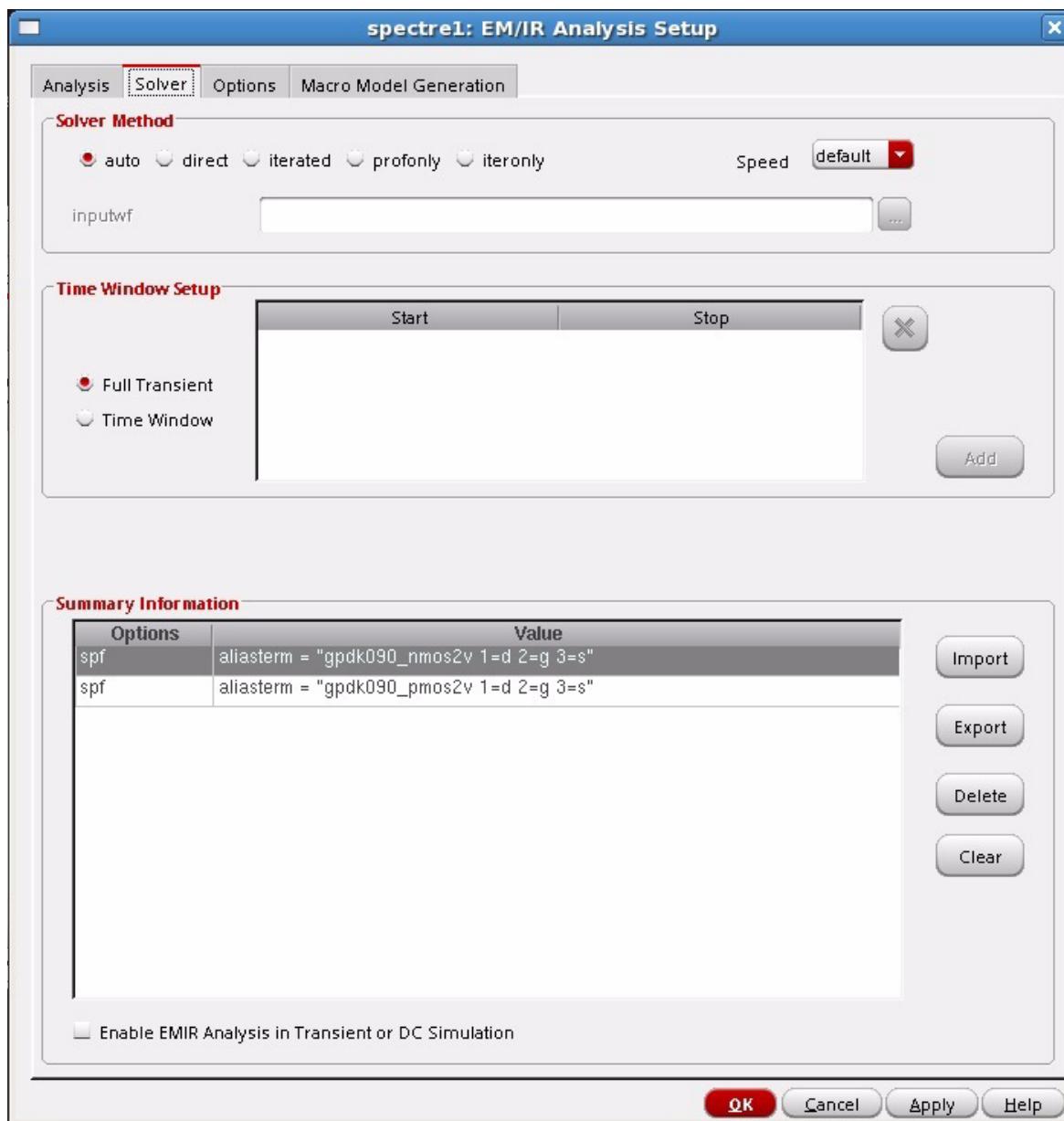


# Voltus-Fi Custom Power Integrity Solution User Guide

## Getting Started

In the *Analysis* tab of this form, ensure that you select the *EM Current Analysis* options, *max* and *avg*, in the *Net Selection* group box if you want to perform AC-Peak analysis later in Voltus-Fi-XL.

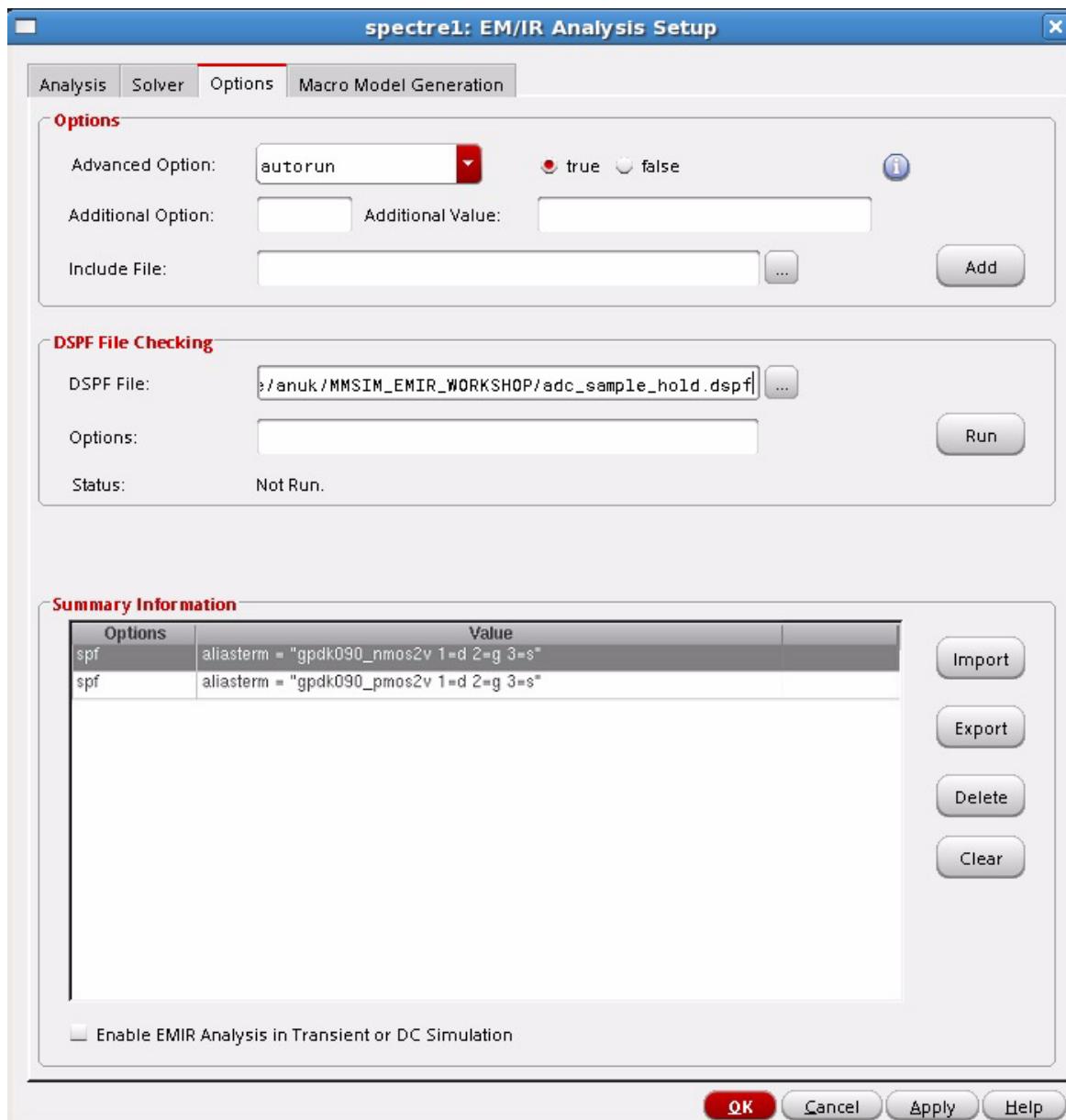
**Figure 2-13 EM/IR Analysis Setup Form – Solver Tab**



# Voltus-Fi Custom Power Integrity Solution User Guide

## Getting Started

**Figure 2-14 EM/IR Analysis Setup Form – Options Tab**



Verify the default settings that are displayed in the form. You can customize the settings in this form based on your requirements. After verifying the settings, click *OK*.

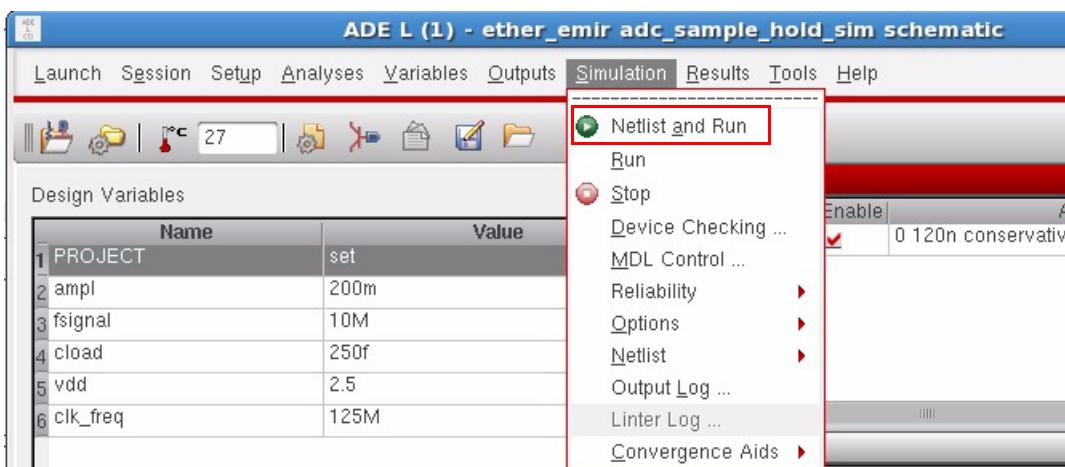
**Note:** For more information about specifying the EMIR settings in the above form, see the “Post-Layout Simulation” chapter in the [\*Virtuoso® Spectre® Circuit Simulator and Accelerated Parallel Simulator User Guide\*](#).

## Running the Simulation in ADE L Window

After completing the setup, you can now run a simulation to generate the simulation database.

- In the main menu of the ADE L window, click *Simulation* and then *Netlist and Run*. This is shown below. You should use this option when you run the simulation for the first time and also when you have edited your design. Using this option ensures that your design, the ADE setup, and the output netlist for simulation are synchronized.

**Figure 2-15 Running the Simulation from the ADE L Window**

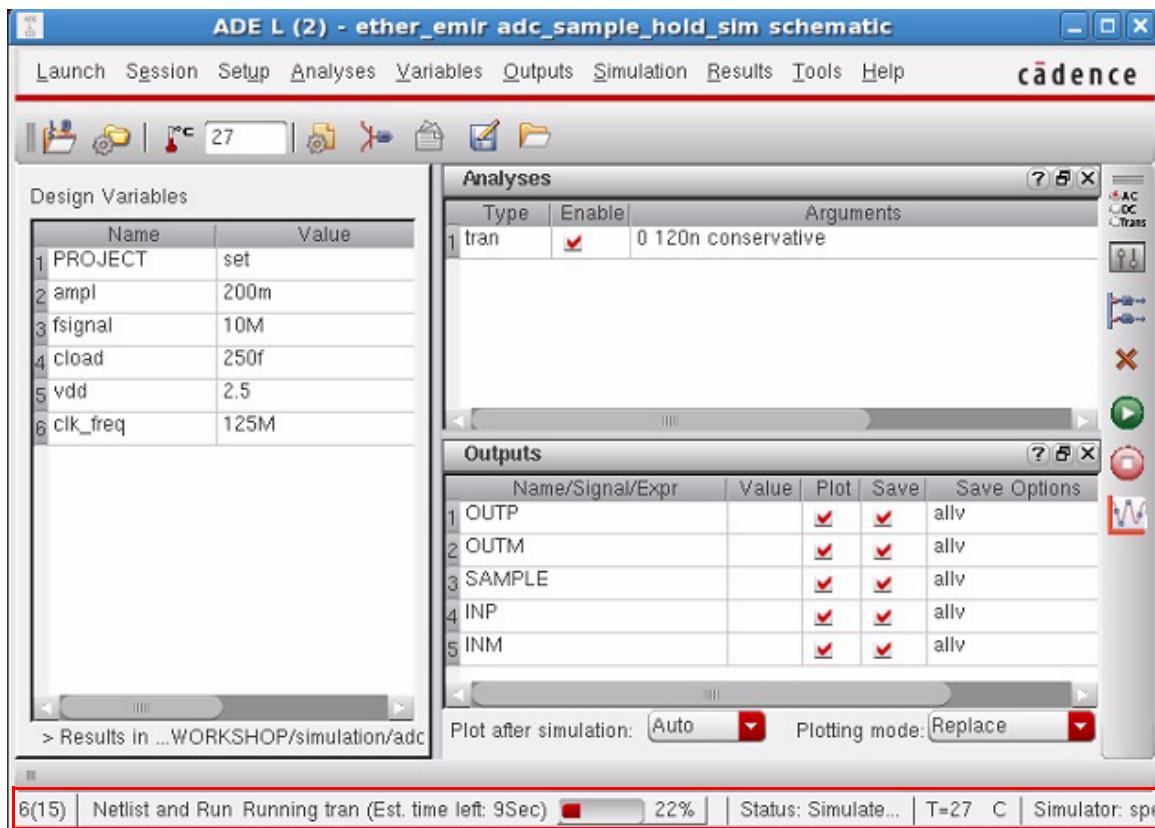


- A progress bar appears on the bottom right of the simulation window displaying the status of netlisting and simulation runs:
  - For netlisting, the progress bar displays the percentage completion status.
  - For simulation runs, the progress bar displays the analysis name, the estimated time to complete, and the percentage completion status. This is shown in the figure below.

# Voltus-Fi Custom Power Integrity Solution User Guide

## Getting Started

**Figure 2-16 Progress Bar Displaying the Status of the Simulation Run**

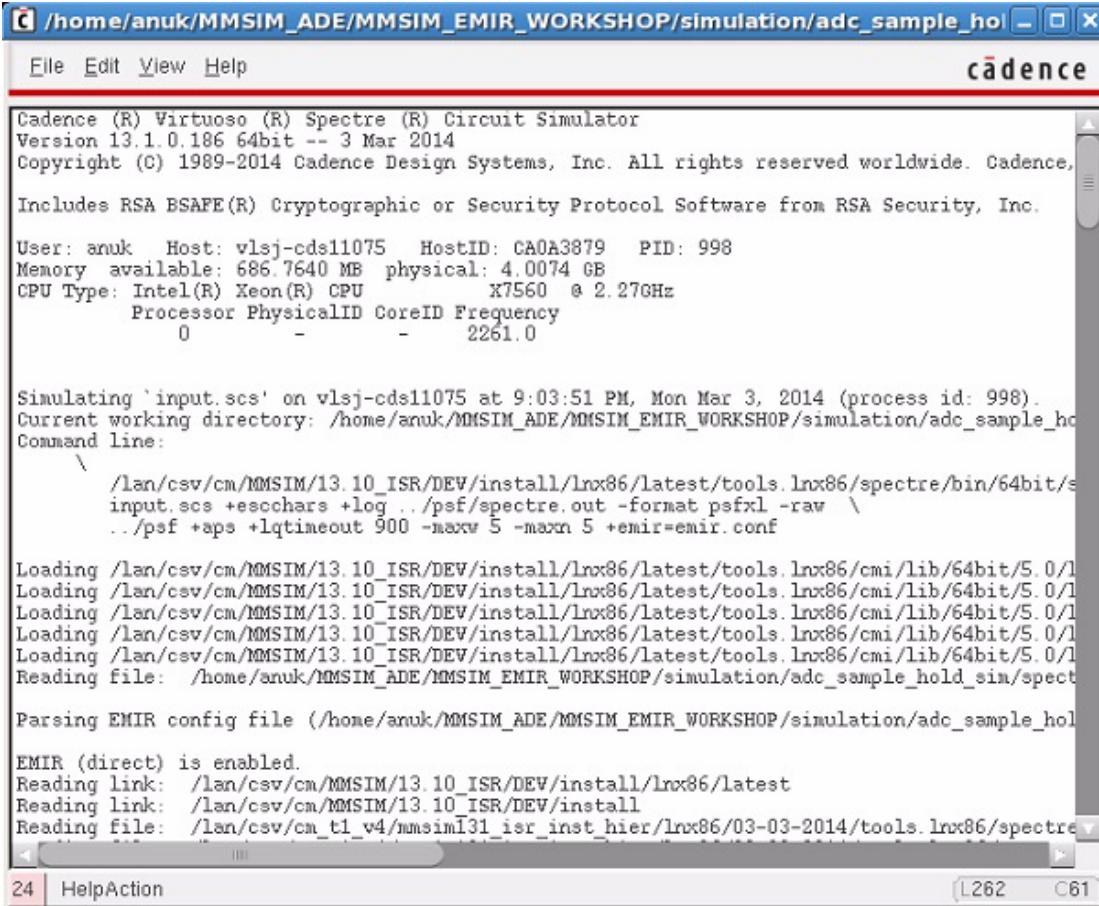


- You can view the details of the run in the UNIX window, shell, or xterm. This is shown below.

## Voltus-Fi Custom Power Integrity Solution User Guide

### Getting Started

**Figure 2-17 Displaying the Details of the Simulation Run**



The screenshot shows a terminal window titled 'Cadence' with the command line '/home/anuk/MMSIM\_ADE/MMSIM\_EMIR\_WORKSHOP/simulation/adc\_sample\_hold'. The window displays the following text:

```
Cadence (R) Virtuoso (R) Spectre (R) Circuit Simulator
Version 13.1.0.186 64bit -- 3 Mar 2014
Copyright (C) 1989-2014 Cadence Design Systems, Inc. All rights reserved worldwide. Cadence,
Includes RSA BSAFE(R) Cryptographic or Security Protocol Software from RSA Security, Inc.

User: anuk Host: vlsj-cds11075 HostID: CA0a3879 PID: 998
Memory available: 686.7640 MB physical: 4.0074 GB
CPU Type: Intel(R) Xeon(R) CPU K7560 @ 2.27GHz
Processor PhysicalID CoreID Frequency
          0      -      -     2261.0

Simulating 'input.scs' on vlsj-cds11075 at 9:03:51 PM, Mon Mar 3, 2014 (process id: 998).
Current working directory: /home/anuk/MMSIM_ADE/MMSIM_EMIR_WORKSHOP/simulation/adc_sample_hold
Command line:
  /lan/csv/cm/MMSIM/13.10_ISR/DEV/install/lnx86/latest/tools.lnx86/spectre/bin/64bit/s
  input.scs +escchars +log .. ./psf/spectre.out -format psfxml -raw \
  ./psf +aps +lqtimeout 900 -maxv 5 -maxm 5 +emir=emir.conf

Loading /lan/csv/cm/MMSIM/13.10_ISR/DEV/install/lnx86/latest/tools.lnx86/cmi/lib/64bit/5.0/1
Loading /lan/csv/cm/MMSIM/13.10_ISR/DEV/install/lnx86/latest/tools.lnx86/cmi/lib/64bit/5.0/1
Loading /lan/csv/cm/MMSIM/13.10_ISR/DEV/install/lnx86/latest/tools.lnx86/cmi/lib/64bit/5.0/1
Loading /lan/csv/cm/MMSIM/13.10_ISR/DEV/install/lnx86/latest/tools.lnx86/cmi/lib/64bit/5.0/1
Loading /lan/csv/cm/MMSIM/13.10_ISR/DEV/install/lnx86/latest/tools.lnx86/cmi/lib/64bit/5.0/1
Reading file: /home/anuk/MMSIM_ADE/MMSIM_EMIR_WORKSHOP/simulation/adc_sample_hold_sim/spectre

Parsing EMIR config file (/home/anuk/MMSIM_ADE/MMSIM_EMIR_WORKSHOP/simulation/adc_sample_hold_sim/spectre)
EMIR (direct) is enabled.
Reading link: /lan/csv/cm/MMSIM/13.10_ISR/DEV/install/lnx86/latest
Reading link: /lan/csv/cm/MMSIM/13.10_ISR/DEV/install
Reading file: /lan/csv/cm_t1_v4/nmsim131_isr_inst_hier/lnx86/03-03-2014/tools.lnx86/spectre
```

**Note:** For details of how to run the simulation in ADE L, see “Running a Simulation” chapter in *Virtuoso® Analog Design Environment L User Guide*.

### Launching Voltus-Fi-XL from ADE L

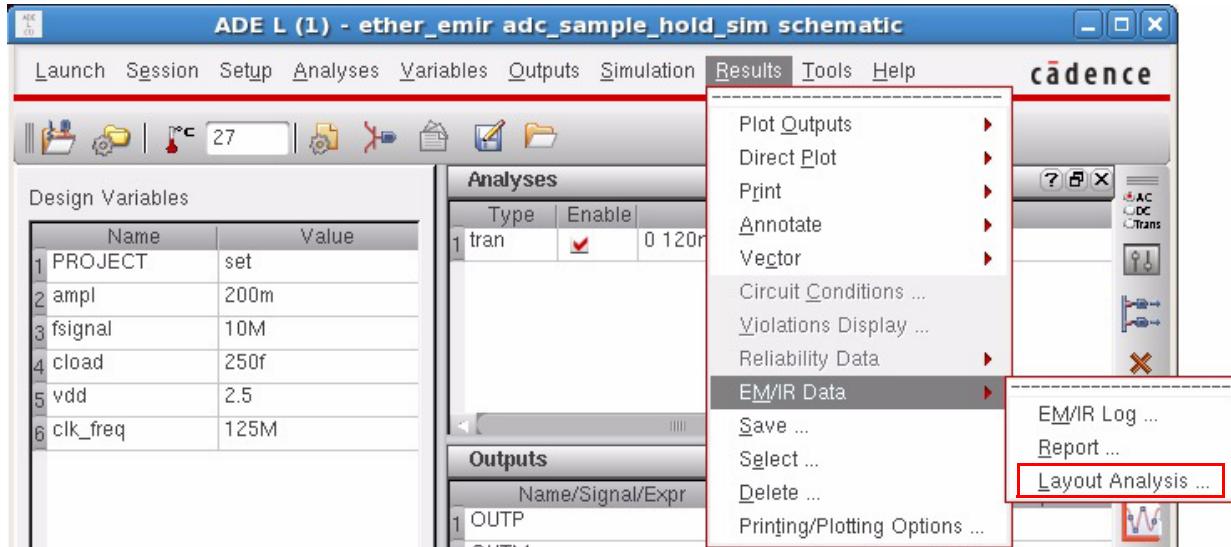
After running the simulation and generating the simulation database, you can now launch Voltus-Fi-XL and the Virtuoso® Layout Suite for viewing the results of the EMIR analysis.

- In the main menu of the ADE L window, click *Results*, *EM/IR Data*, and then *Layout Analysis*. This is shown below.

# Voltus-Fi Custom Power Integrity Solution User Guide

## Getting Started

**Figure 2-18 Launching Voltus-Fi-XL for Layout Analysis**



- The Select Layout View form opens. This is shown below.

**Figure 2-19 Select Layout View Form**



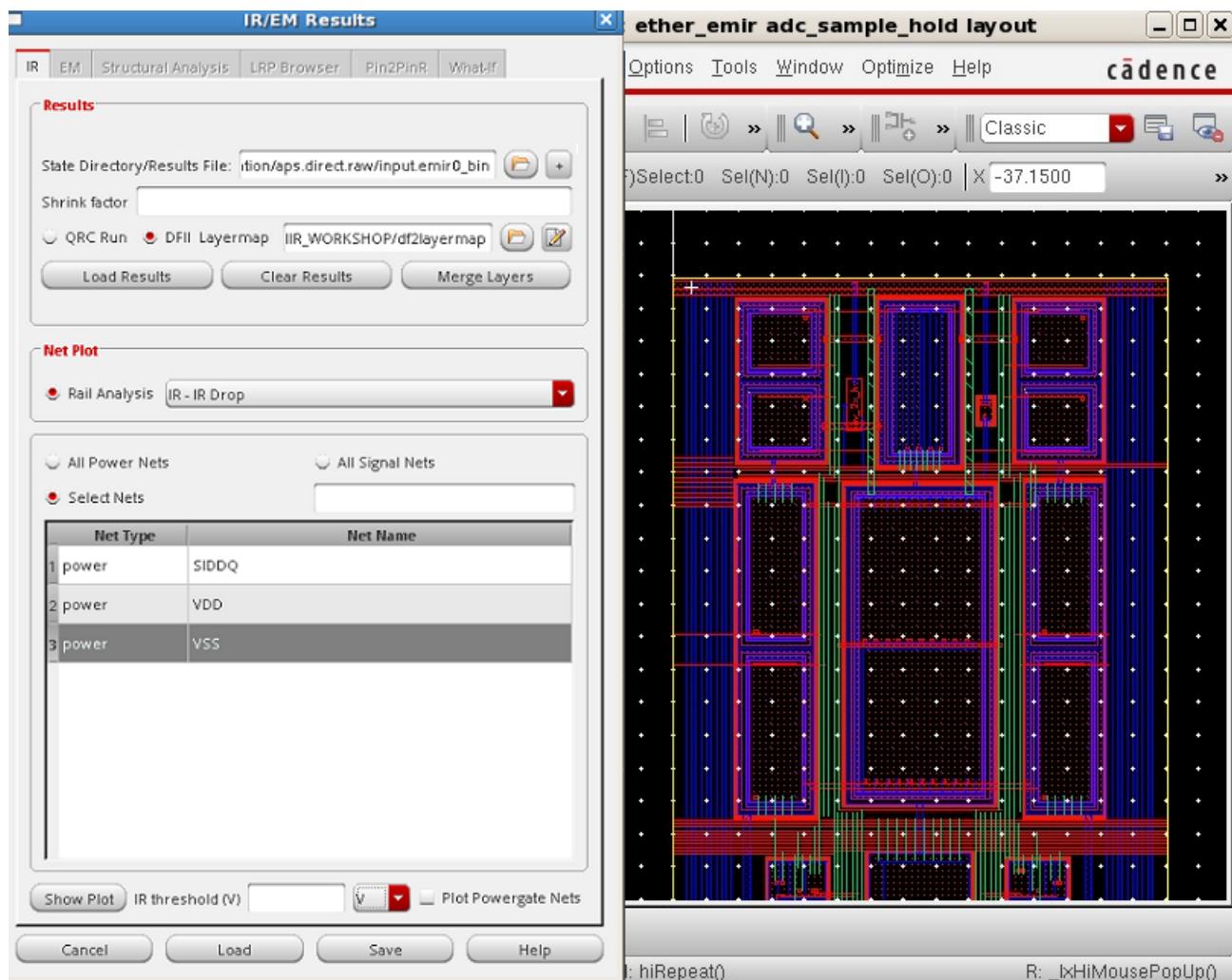
- In this form, select the cell name from the *Cell* cyclic field and click *OK*. The following windows open:
  - The Virtuoso® High Capacity Power IR/EM console
  - The Virtuoso® Layout Suite with the design displayed on the layout
  - The IR/EM Results form of Voltus-Fi-XL

**Note:** The IR/EM Results form is the main form of Voltus-Fi-XL using which you can load the EMIR analysis results, and view the plots for IR drop and EM violations on the layout to debug your design. This is shown below.

# Voltus-Fi Custom Power Integrity Solution User Guide

## Getting Started

**Figure 2-20 The IR/EM Results Form and the Virtuoso Layout Suite**



**Note:** For details of how to visualize EMIR results using the IR/EM Results form, see “[IR Drop Analysis Results](#)”.

## Launching Voltus-Fi-XL from the Virtuoso® Layout Suite

You can also launch Voltus-Fi-XL GUI from the main menu of the Virtuoso® Layout Suite. For this, the first step involves using the CIW to open the Library Manager form. This step is similar to the one detailed in the [Using the Tools Menu of CIW](#) section. The subsequent steps are detailed in the following sections:

- [Opening a Layout View from the Library Manager Form](#)
- [Running Voltus-Fi-XL from the Layout Suite](#)

# Voltus-Fi Custom Power Integrity Solution User Guide

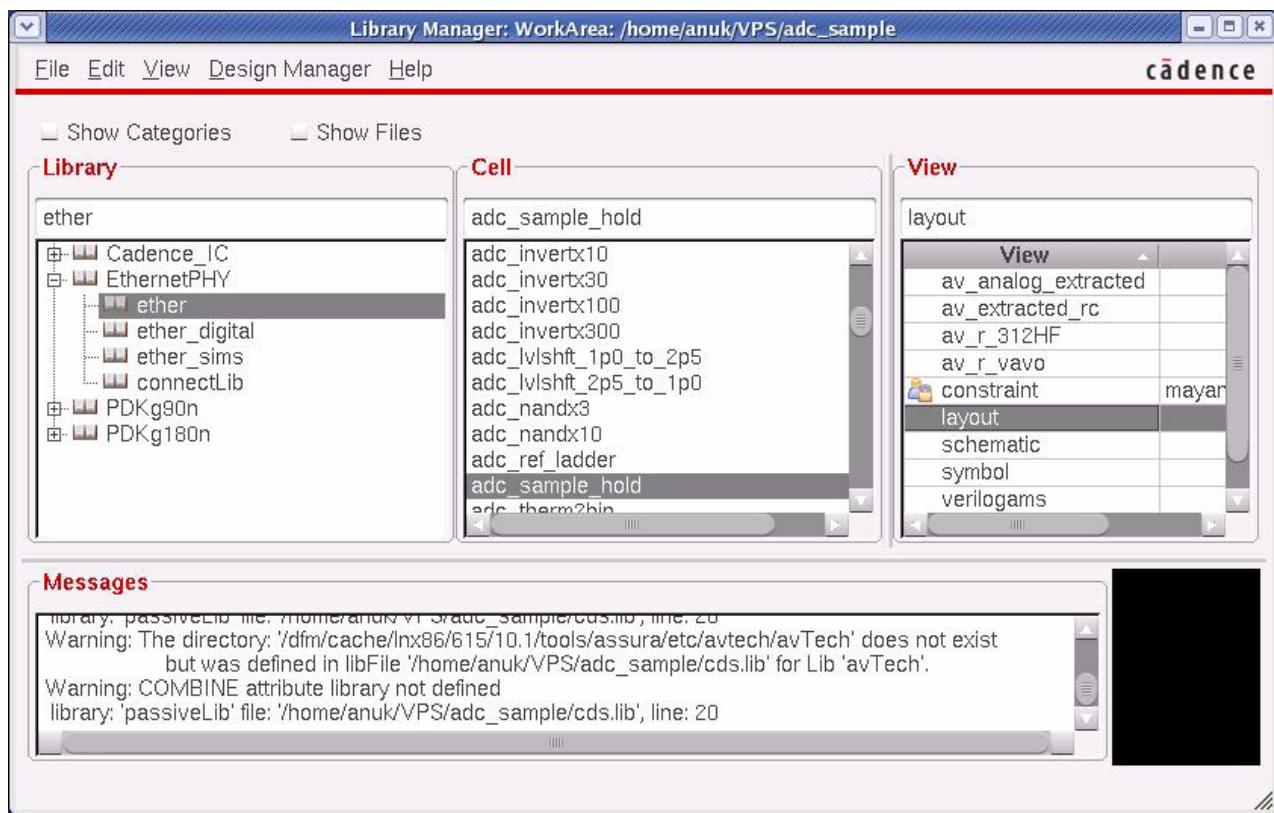
## Getting Started

### Opening a Layout View from the Library Manager Form

To open the layout view of the design, follow these steps:

- In the Library Manager form, select the *Library*, *Cell*, and the layout *View* of the design.

#### Opening the Layout View of the Design

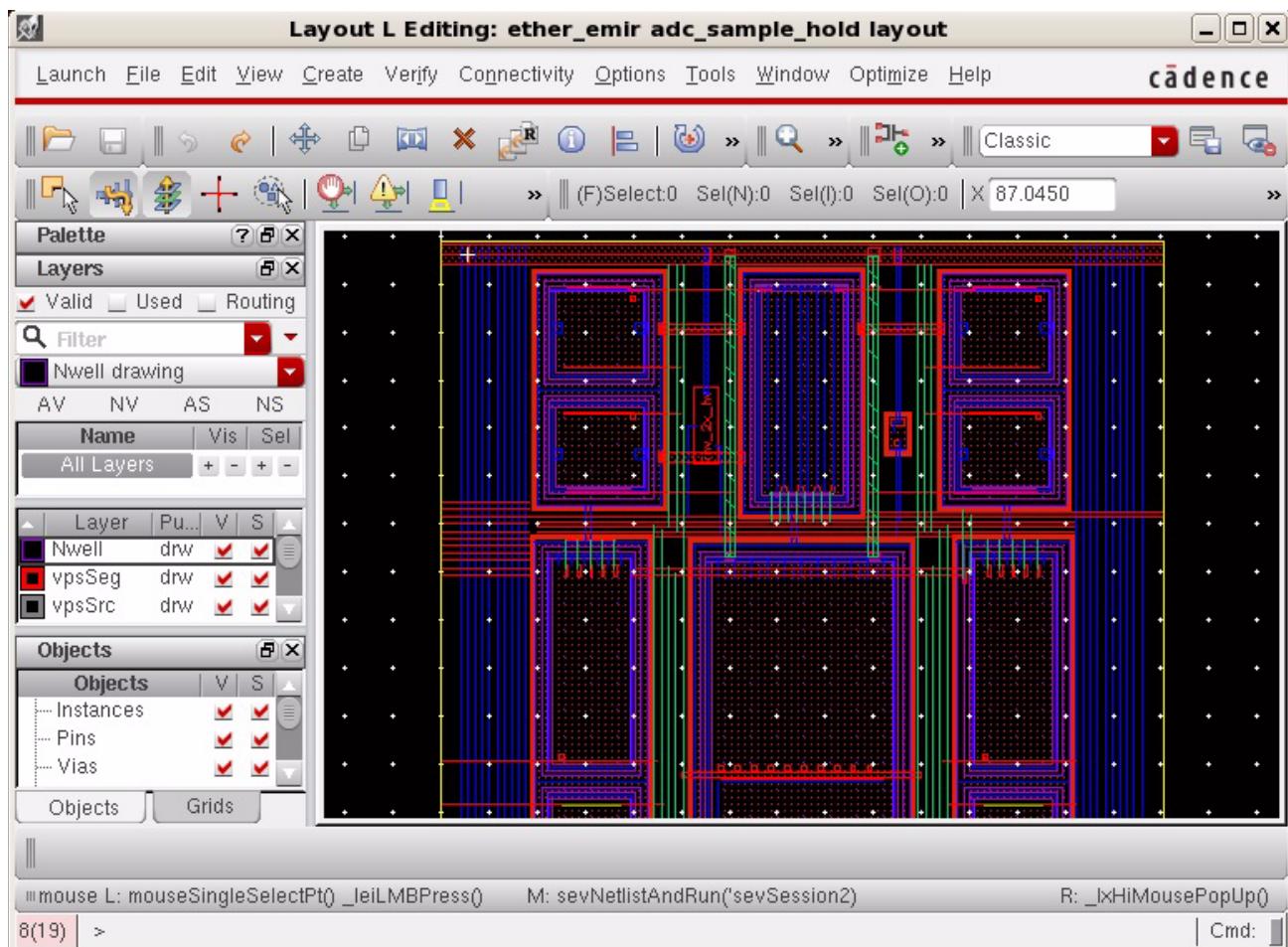


- Double-click the view to open the design in the Virtuoso® Layout Suite. This is shown below.

# Voltus-Fi Custom Power Integrity Solution User Guide

## Getting Started

**Figure 2-21 Opening the Virtuoso® Layout Suite**



### Running Voltus-Fi-XL from the Layout Suite

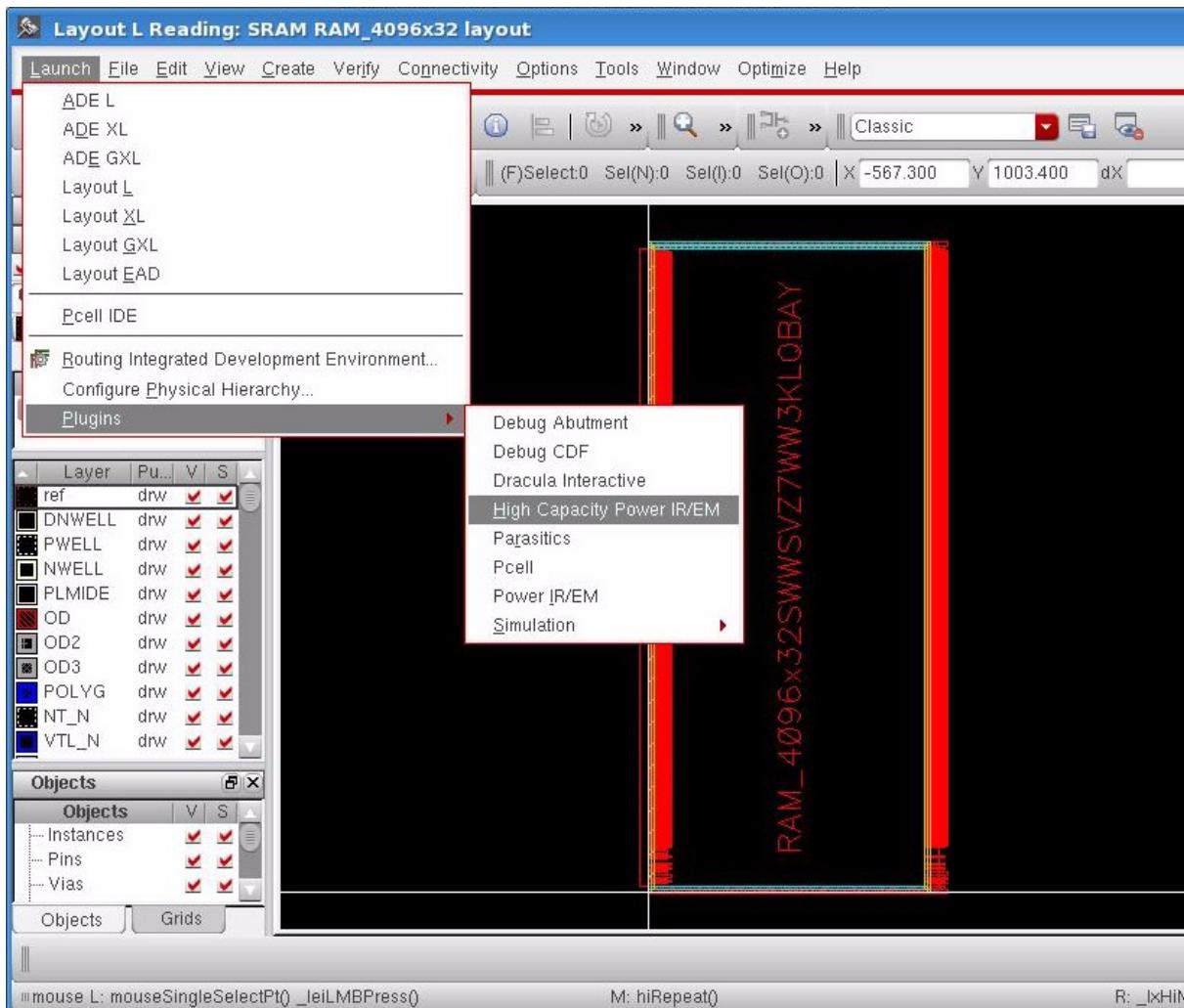
You can open the Voltus-Fi-XL GUI from the main menu of the Virtuoso Layout Suite by performing the following steps:

- Choose *Launch – Plugins – High Capacity Power IR/EM.*

# Voltus-Fi Custom Power Integrity Solution User Guide

## Getting Started

**Figure 2-22 Running Voltus-Fi-XL from the Layout Suite**

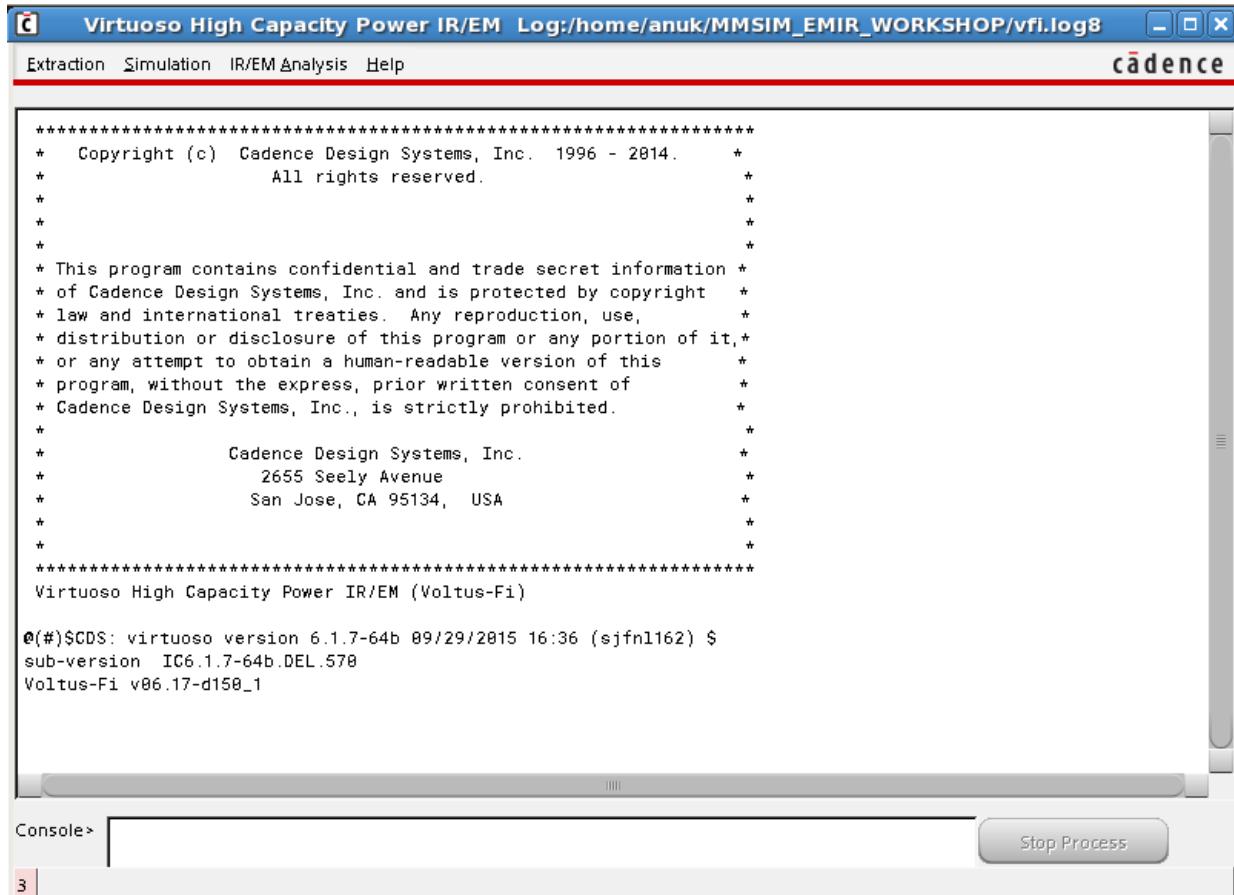


- The Virtuoso® High Capacity Power IR/EM console opens. This is shown below.

## Voltus-Fi Custom Power Integrity Solution User Guide

### Getting Started

**Figure 2-23 Opening the High Capacity Power IR/EM Console**



- The following main menus of Voltus-Fi-XL are available:
  - Extraction
  - Simulation
  - IR/EM Analysis
  - Help

**Note:** In addition to the menus, there is a *Stop Process* button in the console. This button is enabled when you run a process. Click this button to stop a process while it is running.

## **Running Voltus-Fi-XL in the Batch Mode**

You can run Voltus-Fi-XL as a batch command, running it from Virtuoso in no-graph mode. The batch mode is used when the simulation database is already generated and you want to print the IR/EM text reports without viewing the plots on the layout.

For details of how to run Voltus-Fi-XL in the batch mode, see the “[Batch Mode Execution](#)” chapter.

## Accessing Documentation and Help

You can access the Voltus-Fi-XL documentation and Cadence Help using the following methods:

- [Launching Cadence Help From the Command Prompt](#)
- [Accessing Documentation and Help From the Voltus-Fi-XL GUI](#)
- [Other Source of Information](#)

### Launching Cadence Help From the Command Prompt

1. Change to the following directory:

*installation\_dir/tools/bin*

2. Enter the following command:

*. /cdnshelp*

After launching Cadence® Help, press F1 or choose *Help – Contents* to display the help page for Cadence Help.

For more information see the [Cadence Help](#) manual.

### Accessing Documentation and Help From the Voltus-Fi-XL GUI

Click the *Help* button on the bottom right corner of a form. Clicking the *Help* button opens the *Voltus-Fi Custom Power Integrity Solution User Guide* entry for the form in the Cadence Help window.

### Other Source of Information

You can also get help on Cadence products by selecting Cadence Online Support on the Virtuoso *Help* menu. The Cadence Online Support menu opens Cadence Online Support website in your browser.

**Note:** You are required to have a Cadence Online Support account to access these materials.

## **Product Overview**

---

- [Voltus-Fi-XL Product Overview](#) on page 52
- [Key Product Features and Description](#) on page 52

# Voltus-Fi-XL Product Overview

The Voltus-Fi-XL product is used for transistor-level power-integrity analysis, which includes multi-mode simulation (MMSIM) electromigration and voltage drop (EMIR) analysis for transistor designs. The target designs include analog, analog/mixed-signal (AMS), and custom digital designs of large sizes that are created using Virtuoso.

This product is tightly integrated within Virtuoso. The results of the analyses are displayed on the Virtuoso layout. Voltus-Fi-XL also generates text reports of the analyses and lets you query the analyses results to view specific violations on the layout. This is used to identify and debug regions of high IR drop and EM violations in the design. After analyzing the EMIR results, you can create power-grid views (PGVs) of the design block, which can then be used in Voltus for mixed-signal analysis.

Voltus-Fi-XL uses the simulation database generated by simulators—Spectre® Accelerated Parallel Simulator (APS) and Spectre® eXtensive Partitioning Simulator (XPS)—as inputs to perform EMIR analysis.

A brief description of the capabilities of the two simulators is provided below.

- Spectre® APS is a new-generation SPICE simulator that provides high performance, high capacity circuit simulation with full Spectre accuracy. APS achieves maximum simulation performance by enabling multi-threading on multi-core and multi-CPU shared memory systems.
- Spectre® XPS is a new-generation transistor-level circuit simulator that emphasizes high simulation performance and large simulation capacity, and addresses the design and verification needs of full-chip low-power designs at advanced process nodes.

**Note:** For detailed information about Spectre® APS/XPS, see “Introducing the Virtuoso Spectre Circuit Simulator” in *Virtuoso® Spectre® Circuit Simulator and Accelerated Parallel Simulator User Guide*.

## Key Product Features and Description

The following table lists the key features of Voltus-Fi-XL.

**Table 3-1 Product Features – Voltus-Fi-XL**

Feature	Description
---------	-------------

# Voltus-Fi Custom Power Integrity Solution User Guide

## Product Overview

Extraction	<p>Supports the generation of the Detailed Standard Parasitic Format (xDSPF) file using Quantus QRC. This file provides all the parasitic information of the design for performing simulation using Spectre.</p> <p>For details, see the "<a href="#">xDSPF Generation using Quantus</a>" chapter.</p>
Simulation-based analysis	<p>Uses APS/XPS—direct or iterated method for EMIR analysis.</p> <p>For details of the two methods, see <a href="#">Spectre® APS/XPS EMIR Flow Overview</a> in the "Data Preparation" chapter.</p>
Static Current Analysis	<p>Calculates device currents, both peak and average, using Thunder. These current calculations are used during the Spectre simulation.</p> <p>For details, see the "<a href="#">Static Current Analysis</a>" chapter.</p>
IR drop analysis	<p>Supports signal net and power-rail IR drop analysis.</p> <p>For details, see the "<a href="#">IR Drop Analysis Results</a>" chapter.</p>
EM analysis	<p>Supports signal net and power-rail EM analysis</p> <p>Supports all types of EM analysis: peak, average, rms, AC-peak, and absavg</p> <p>Supports text-based ICT file for specifying EM rules</p> <p>Supports advanced EM rules with <code>qrcTechFile</code> for 20nm and below technologies.</p> <p>For backward compatibility, the <code>emDataFile.txt</code> file is supported for 28nm and above technologies.</p> <p>For details, see the "<a href="#">EM Analysis Results</a>" chapter.</p>
Results Display	<p>IR/EM plots annotation on Virtuoso layout</p> <p>Shape-based display for EMIR results</p> <p>IR/EM plots using the shape database generated by QRC</p>
PGV Generation	<p>Supports static and dynamic PGV generation for power pins.</p> <p>For details, see "<a href="#">Power-Grid View Generation</a>" chapter.</p>

## Voltus-Fi Custom Power Integrity Solution User Guide

---

Advanced debug features	<p>Supports structural analysis</p> <ul style="list-style-type: none"><li>■ Missing Via analysis</li><li>■ Via coverage ratio</li><li>■ Skewed via ratio</li></ul> <p>Supports Current Direction Reporting</p> <p>Supports Least-Resistive Path (RLRP) analysis</p> <p>Supports What-If Analysis</p> <p>Supports Self Heating Effect (SHE) analysis flow visualization.</p>
-------------------------	---

# Data Preparation

---

- [Voltus-Fi-XL Flow Overview](#) on page 56
  - [Spectre® APS/XPS EMIR Flow Overview](#) on page 56
- [Data Requirements for Voltus-Fi-XL](#) on page 59
  - [xDSPF Requirements](#) on page 60
  - [Simulation Requirements and Setup](#) on page 61
  - [EMIR Control File Options Supported in Voltus-Fi-XL](#) on page 62
- [Setup for Visualization of IR and EM Results](#) on page 76
  - [Loading the DFII Layer Map File by Default](#) on page 76
  - [Setting a Layer Index for Displaying Violations on the Layout](#) on page 77
- [EM Rule File Requirements](#) on page 78
  - [EM Rules in qrcTechFile](#) on page 78
  - [Rules for Defining EM Parameters](#) on page 80
  - [EM Rule Selection Priority](#) on page 86
  - [EM Data File](#) on page 91
- [Outputs of EMIR Analysis](#) on page 92

## **Voltus-Fi-XL Flow Overview**

The Voltus-Fi-XL-Spectre® Accelerated Parallel Simulator (APS)/Extensive Partitioned Simulator (XPS) flow is a two-step flow.

**Step1:** Simulation is run using Spectre® APS/XPS and the simulation database is generated. For details of the Spectre® APS/XPS flow, see [Spectre® APS/XPS EMIR Flow Overview](#).

**Step2:** The simulation output generated in the above step is provided as input to Voltus-Fi-XL, which performs the following electromigration and voltage drop (EMIR) analyses:

- Power IR/EM analysis
- Signal IR/EM analysis
- Structural analysis

Voltus-Fi-XL generates text reports of IR drop and EM violations and displays them as plots on the Virtuoso® layout. It also provides batch mode support for IR/EM analysis and report generation.

**Note:** Voltus-Fi-XL supports both static and dynamic EMIR analysis.

## **Spectre® APS/XPS EMIR Flow Overview**

With MMSIM12.1, Spectre® APS and Spectre® XPS, Cadence delivers a new EMIR solution. This dynamic power EMIR and signal EM capability uses a new technology, and is designed to provide higher capacity and better performance compared to existing EMIR solutions. Within this flow, Spectre® APS can be used for high accuracy EMIR analyses; while Spectre® XPS can be deployed for high performance and high capacity EMIR simulation.

In an EMIR flow, a circuit is evaluated together with the parasitic resistor and capacitor network, which models the IR drop or the EM effect. There are two general approaches of solving such a problem.

- 1. Direct EMIR analysis:** When high accuracy is needed, a brute-force simulation of the entire system – circuit plus parasitic resistances and capacitances – can be performed to accurately calculate EMIR of any net. This approach is called the “one-step” method, where the EMIR simulation performance and capacity is subject to the limitation of the circuit simulator being used.
- 2. Iterated EMIR analysis:** To conduct EMIR simulation on circuits with much larger power and signal nets, within a shorter time, there is an alternative approach that involves decoupling the nonlinear circuit simulation from the linear RC net analysis. You can iterate

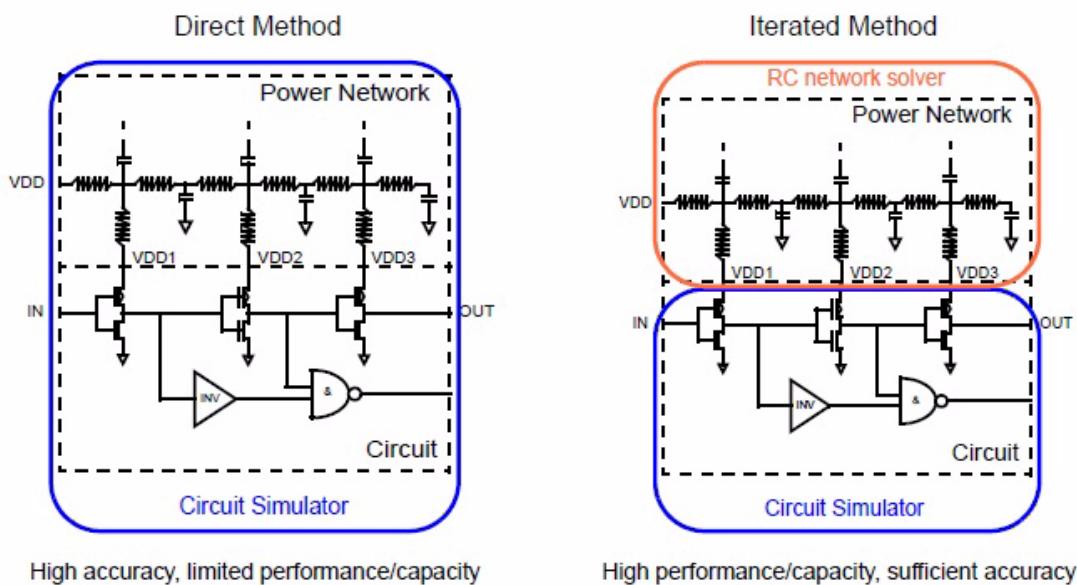
# Voltus-Fi Custom Power Integrity Solution User Guide

## Data Preparation

the linear RC net analysis by modifying the layout, however, the nonlinear circuit simulation is done only once. This approach is called the “two-step” method. The decoupling of the linear RC nets from the nonlinear circuit is not mathematically equivalent to the original design and certain inaccuracy is introduced but it provides the benefit of simulation performance and capacity.

The above two approaches are depicted in the figure below.

**Figure 4-1 Direct and Iterated EMIR Analysis Methods**



For high accuracy EMIR analysis of small design blocks, or designs with smaller numbers of RC nets, the direct EMIR analysis method is recommended, where Spectre® APS and Spectre® XPS SPICE mode are fully supported. Spectre® XPS does not support direct EMIR analysis.

To gain higher performance and higher capacity on medium to large designs, the iterated EMIR analysis method is recommended. Spectre® XPS is used for these large-scale simulations as it is able to take full advantage of the FastSPICE technology. Both, Spectre® APS and Spectre® XPS SPICE mode support iterated EMIR analysis.

The Spectre® APS/XPS EMIR flow requires a complete testbench that contains Detailed Standard Parasitic Format (DSPF) files – with parasitic and instance sections describing the circuit to be analyzed – and stimuli, device models, and so on. The presence of a `+emir` statement on the Spectre command line enables the EMIR analysis during circuit simulation. The details of EMIR analysis, including the type of analyses to be performed, nets to be analyzed, and the output to be generated, are specified using an EMIR configuration file.

## **Voltus-Fi Custom Power Integrity Solution User Guide**

### Data Preparation

---

Using either direct EMIR analysis or iterated EMIR analysis, the circuit simulation is performed first with various voltages and currents being calculated. Next, a standalone tool, `emirutil`, is called to post-process the simulation results and generate IR and EM reports. By default, this postprocessing step is invoked automatically.

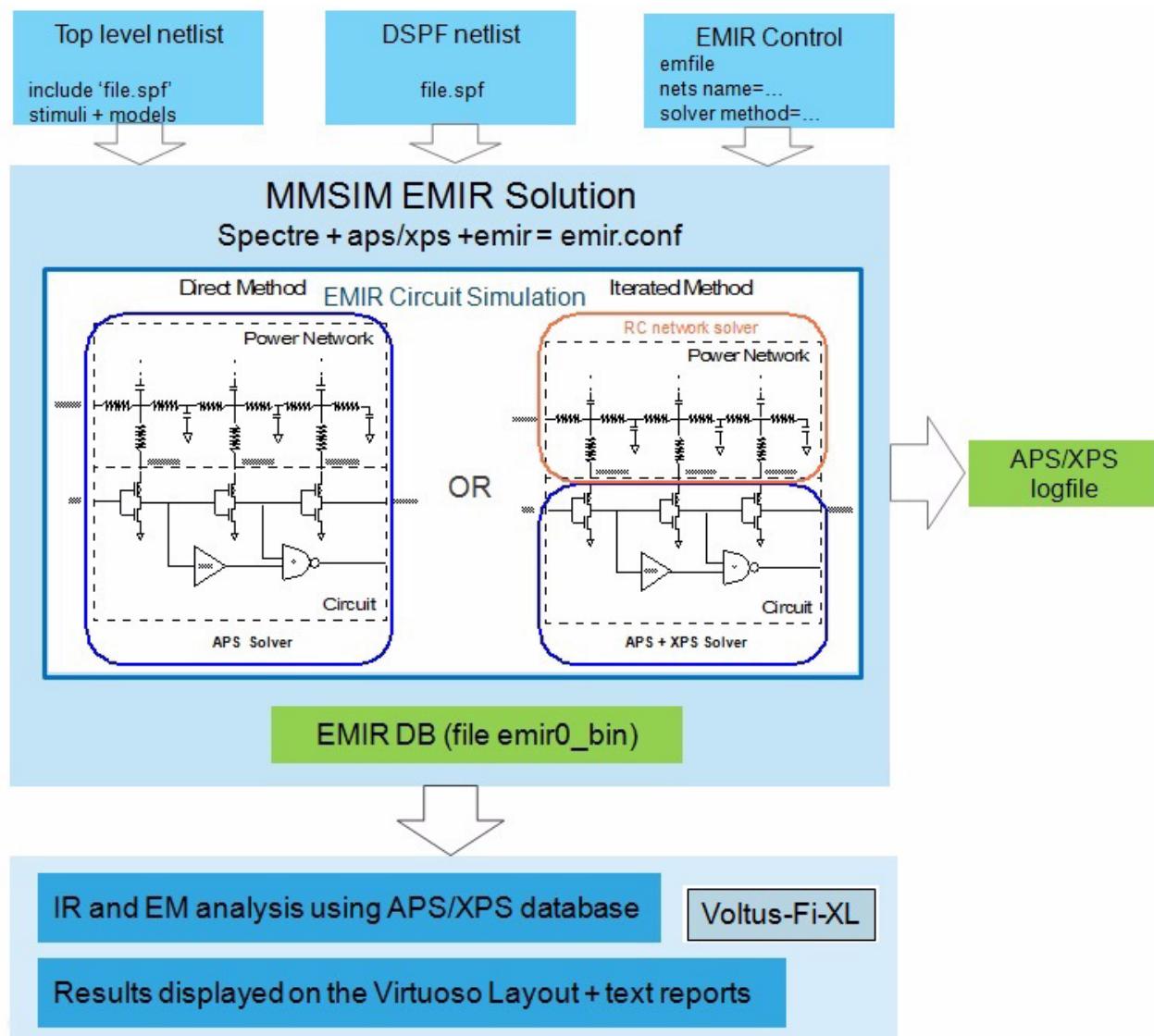
The output of the EMIR analysis is an EMIR text report, which lists the EM and/or IR information of the specified nets. In addition, an OA database is generated to enable the visualization of the EMIR results on the Virtuoso layout using Voltus-Fi Custom Power Integrity Solution tool.

This flow is shown in the below figure.

## Voltus-Fi Custom Power Integrity Solution User Guide

### Data Preparation

**Figure 4-2 Voltus-Fi-XL – Spectre® APS/XPS Data Flow**



For details of Spectre® APS/XPS EMIR flow and how to activate the EMIR analysis, see the EMIR Flow Setup section in the “Post-Layout Simulation” chapter of the [\*Virtuoso® Spectre® Circuit Simulator and Accelerated Parallel Simulator User Guide\*](#).

## Data Requirements for Voltus-Fi-XL

The following inputs are required for EMIR analysis in Voltus-Fi-XL:

- A simulation output directory that contains the EMIR database

- The `qrcTechFile` with EM Models for EM rules, text-based ICT file, or the `emDataFile.txt` file with EM rules
- Layer map file – this file is required to map the layer names in the simulation result file and the technology file
  - Note:** This file is required only if the layer names in the two files are different.
- APS/XPS-to-DFII layer map file for solid shape display and structural analysis
- Net names (optional)
- Information about the type of analysis to be performed (peak, avg, rms, absAvg)

The above data requirements are categorized under the following heads and explained in detail in subsequent sections:

- [xDSPF Requirements](#)
- [Simulation Requirements and Setup](#)
- [Setup for Visualization of IR and EM Results](#)
- [EM Rule File Requirements](#)

## **xDSPF Requirements**

The third-party netlist file or xDSPF is used to provide all the parasitic information of the design for performing simulation using the Spectre® APS/XPS simulators. The xDSPF must include the following information:

- All power ground nets extracted
- Layer information with each resistor
- x, y co-ordinates of the parasitic resistors, and
- Decoupled RC of all signal nets

DSPF files are created using the parasitic extraction tools and their content and format is heavily dependent on the extraction tool and the extraction runset. Often simulation problems occur due to problems in the DSPF file. To ensure that the DSPF file is compatible with the simulation requirements use the Standard Parasitic Format (SPF) Checker utility before using the file in the MMSIM EMIR flow.

The SPF Checker utility analyzes the DSPF file, reports problems that may cause simulation problems, and creates an EMIR config file with recommended mapping statements.

For details of how to use the SPF Checker, see the SPF Checker section in the “Post-Layout Simulation” chapter of *Virtuoso® Spectre® Circuit Simulator and Accelerated Parallel Simulator User Guide*.

**Note:** Ensure that there is no pin on the substrate shorting in the xDSPF. Set the extractor accordingly.

**Note:** If you are using third-party DSPF file, you may encounter some problems while performing the EMIR analysis. For possible issues and how to resolve them, see [Appendix A – Third-Party DSPF](#).

**Note:** Voltus-Fi-XL supports the generation of the xDSPF file using Quantus (QRC) both from the GUI and in the batch mode. For details of generating the xDSPF file from within Voltus-Fi-XL GUI, see “[xDSPF Generation using Quantus](#)” chapter. For details of the batch mode command used for generating the xDSPF file, see [Batch Command for xDSPF Generation](#) in the “Batch Mode Execution” chapter.

## Simulation Requirements and Setup

For viewing IR drop and EM analysis results in Voltus-Fi-XL, the simulation database generated using Spectre® APS/XPS simulators is loaded into Voltus-Fi-XL. This is shown in [Figure 4-2](#) on page 59 .

For EMIR analysis, ensure the following while performing simulation.

- Create a complete simulation testbench with the DSPF files containing the post-layout data of the design, including fingered devices and net parasitics. Use the `dspf_include` statement to read the DSPF content, as shown below.

```
dspf_include "sram.spf" (Spectre syntax)  
.dspf_include "sram.spf" (SPICE syntax)
```

**Note:** Though `include/.include` statements are also supported, it is recommended to use `dspf_include/.dspf_include` statements, which automatically handle duplicated subcircuit definitions and port order differences.



Use the SPF Checker utility before using the file in the MMSIM EMIR flow.

- Define the EMIR configuration using the EMIR control file (`emir.config`), and include the control file with the `+emir` statement on the Spectre command line, as shown below.

```
% spectre +aps +emir=emir.conf input.scs (APS solver)  
% spectre +xps +emir=emir.conf input.scs (XPS solver)
```

**Note:** The supported EMIR control file options are provided in [Supported EMIR Control](#)

File Options on page 62.

- Before plotting EMIR results, ensure that the simulation result file is available in the output directory of the simulation results.

The naming convention for the simulation results file is; \*.emir#\_bin. For example, the result file could be named, xps.emir0\_bin. This file is required for displaying IR and EM plots and generating text reports in Voltus-Fi-XL.

### **EMIR Control File Options Supported in Voltus-Fi-XL**

The following table summarizes the supported EMIR control file options:

**Table 4-1 Supported EMIR Control File Options**

Keyword	Option Set	Explanation	Default Value
net	name=[instance1. net1instance1.ne t2....]	Defines the nets for which the analysis is performed. instance defines the instance of the subcircuit containing the net. net defines the net name inside the subcircuit instance as it is defined in the DSPF *   NET definition. If the DSPF file is included on the top level, the instance name is not required. Wildcarding is supported for net names but not instance names.	none
	analysis=[imax iavg iavgpos iavgneg iavgabs irms vmax vavg]	peakEM, avgEM, avgEM <b>for</b> i>0, avgEM <b>for</b> i<0, avgabsEM, rmsEM, peakIR, avgIR  <b>Note:</b> To perform AC-Peak analysis in Voltus-Fi-XL, ensure that the imax and iavg options are specified in the control file.	iavg, vmax

# Voltus-Fi Custom Power Integrity Solution User Guide

## Data Preparation

---

Keyword	Option Set	Explanation	Default Value
	<code>pwrgate=[powerne t1 powernet2...]</code>	Enables power gate handling. You need to specify the power supply net driving the power gate and the internal power supply net driven by the pwrgate. If one power supply net drives multiple power gates, then you need to specify one statement with all internal power supply nets.	none
	<code>analysis=[sigvma x sigvavg] reftype=[avg max] [findsrc=yes]</code>	<p><code>sigvmax</code> - max signal net IR drop.  <code>sigvavg</code> - avg signal net IR drop.</p> <p><code>avg</code> - reference voltage for IR drop is the average voltage of all subnodes at every time point.</p> <p><code>max</code> - reference voltage for IR drop is the maximum voltage of all subnodes at every time point.</p> <p><code>findsrc=yes</code> - Use voltage from subnode which is connected to vsource. The feature automatically traces through the design or source resistors.</p>	none
	<code>analysis=[ipwc] pwc_threshold=1e -6</code>	Enables pulse-wise EM current calculation for violation visualization. The <code>pwc_threshold</code> option defines the current threshold for finding the start of the pulse. Any value below <code>pwc_threshold</code> is considered as zero. The <code>emirutil_dump_pwc_info=1</code> option generates an ascii report in the <code>&lt;name&gt;.pwc0_raw</code> file for debugging purpose only.	none
	<code>vref</code>	Defines the optional voltage reference value for <code>vmax</code> and <code>vavg</code> analyses.	

# Voltus-Fi Custom Power Integrity Solution User Guide

## Data Preparation

---

Keyword	Option Set	Explanation	Default Value
	<p><b>For versions before MMSIM15.1 ISR2, the syntax is as follows:</b></p> <pre>design_res_models=[name=&lt;cell_name&gt; w=&lt;width parameter name&gt; l=&lt;length parameter name&gt; layer=&lt;layer name&gt; unit=&lt;multiplier to get length/width * geounit in um&gt;]</pre> <p><b>For MMSIM15.1 ISR2 and later versions, the syntax is as follows:</b></p> <pre>design_res_models=[name=&lt;cell_name&gt; w=&lt;width parameter name&gt; l=&lt;length parameter name&gt; layer=&lt;layer name&gt; unit=&lt;multiplier to get length/width in m&gt;]</pre>	<p>Specifies the design resistor models for which EMIR analysis will be performed.</p> <p><b>Note:</b> The design resistor models are specified for cells and not for the instances of the cell.</p> <p>For example, for the design resistor provided in the xDSPF file as:</p> <pre>XR1 XR1#PLUS XR1#MINUS r_user wr=1e-06 lr=5e-06</pre> <p>In this example, width (<code>wr</code>) and length (<code>lr</code>) are in meters.</p> <p>The design resistor models will be specified in the following manner:</p> <p><b>Before MMSIM15.1 ISR2,</b></p> <pre>design_res_models=[name=r_use r w=wr l=lr layer=mt1 unit=1e6]</pre> <p><b>MMSIM15.1 ISR2 and later:</b></p> <pre>design_res_models=[name=r_use r w=wr l=lr layer=mt1 unit=1]</pre> <p>where,</p> <p><code>r_user</code> is the design resistor model name</p> <p><code>lr</code> is the length parameter name specified in the xDSPF</p> <p><code>wr</code> is the width parameter name specified in the xDSPF</p> <p><code>mt1</code> is the layer for the EM rules. The tool will apply <code>mt1</code> layer rules from the ICT file for this resistor</p>	none

# Voltus-Fi Custom Power Integrity Solution User Guide

## Data Preparation

---

Keyword	Option Set	Explanation	Default Value
		<p><b>Before MMSIM15.1 ISR2</b></p> <p><math>1e6</math> is the unit that is calculated as design resistor <math>W/L</math> in netlist * unit/geounit. For example:</p> <p>For <math>lr=5e-06</math> and <math>unit=1e6</math>, the length value of design resistor in um is as follows:</p> $l=5e-06*1e06=5\mu m$ <p><b>MMISM15.1ISR2 and later</b></p> <p><math>1</math> is the unit that is calculated as design resistor <math>W/L</math> in netlist * unit. For example:</p> <p>For <math>lr=5e-06</math> and <math>unit=1</math>, the length value of design resistor in meters is as follows:</p> $l=5e-06*1=5e-06m$	
solver	method=[auto   direct   iterated   profonly   iteronly]	<p>auto - direct for APS, iterated for XPS (default)</p> <p>direct - forced direct method, not available for XPS</p> <p>iterated - forced iterated approach, available for APS and XPS</p> <p>profonly - iterated flow - circuit profile generation</p> <p>iteronly - iterated flow - RC network iteration</p>	auto
	speed=[1 2 3 4 5 6 7 8]	<p>Defines the speed/accuracy trade-off for PN and SN RC network solver.</p> <p>1 - highest accuracy, lowest speed</p> <p>8 - lowest accuracy, highest speed</p>	5

## Voltus-Fi Custom Power Integrity Solution User Guide

### Data Preparation

---

Keyword	Option Set	Explanation	Default Value
	inputwf=file_name	Specifies the waveform file that contains the results from circuit simulation when <code>method=iteronly</code> is used.	none
static	ifile="filename"	Enables static EMIR analysis. The file defines the subckt instance port currents.  <b>Note:</b> Static EMIR analysis cannot be combined with the dynamic EMIR analysis.	

# Voltus-Fi Custom Power Integrity Solution User Guide

## Data Preparation

---

Keyword	Option Set	Explanation	Default Value
<b>emirutil</b>	techfile="emDataFile"	<p>Specifies the EM rule file in the emdatafile, qrctechfile, or ictfile format. Case-insensitive layer names are supported allowing easier match between the DSPF and emData file.</p> <p>The EM rule file can be predefined in the configuration file using the EMTECHFILE environment variable or in the .cdsinit file as follows:</p> <pre>envSetVal( "spectre.envOpts" "emTechFile" string &lt;path to the file&gt;</pre> <p>The rule file defined in the configuration file has higher priority followed by the rule file specified in .cdsinit.</p> <p>Alternatively, you can use the emdatafile="emDataFile" option to specify the techfile in the emdata format,</p> <p>qrctechfile="qrctechfile" to specify the QRC tech files, and ictfile="ictfile" to specify the file in ICT format.</p> <p>The EM rule file in emdatafile format can be predefined using the EMDATAFILE environment variable, or emRuleFile in cdsenv. The EM rule file defined in cdsenv has higher priority.</p> <p>The layermap file provides the mapping between the xDSPF and the layout database. It is applicable only for qrctechfile and ictfile.</p>	none

# Voltus-Fi Custom Power Integrity Solution User Guide

## Data Preparation

---

Keyword	Option Set	Explanation	Default Value
	EMOnlyICTFile="emOnlyICTFile"	<p>Specifies the process and EM models to be used for EM analysis.</p> <p><b>Note:</b> This file can only be provided with the <code>qrcTechFile</code> flow.</p>	none
	postTclCmdFile=[conf.tcl]	<p>Specifies the tcl file that contains user-specified commands for querying specific resistor and node information and creating customized EM and IR drop analysis reports.</p> <p>For details about the commands that can be written in the file (<code>conf.tcl</code>), see <a href="#">Batch Commands for Querying</a>.</p>	none
	layermapfile="mapfile"	Specifies the layer map file that provides the mapping between the layer names in the simulation database to the layer names in the technology file.	none
	autorun=[true   false]	<p>true - run <code>emirutil</code> automatically to generate a text report.</p> <p>false - manually run <code>emirutil</code>.</p>	true
	report=[text   html]	<p>Defines the type of report created by <code>emirutil</code>.</p> <p>text - produces only text report.</p> <p>html - creates the report in html format.</p> <p><b>Note:</b> Multiple entries are supported.</p>	text
	notation=[s   e]	<p>Notation for the text and html reports.</p> <p>e - engineering scale number (for example, 5.02m)</p> <p>s - scientific notation (for example, 5.02e-3)</p>	e

# Voltus-Fi Custom Power Integrity Solution User Guide

## Data Preparation

---

Keyword	Option Set	Explanation	Default Value
	sort_by_net=yes   no	<p>Report IR and EM results per net, or all nets.</p> <p>yes - report EMIR results per net</p> <p>no - combine EMIR results or all nets into one report.</p>	
	filter_ir_thresh old= <i>value</i>	<p>Defines the IR voltage drop threshold for the node being reported.</p> <p>filter_ir_threshold=0.01 reports all nodes with IR drop above 0.01V.</p>	
	filter_em_thresh old= <i>value</i>	<p>Defines the EM analysis threshold value for the resistors being reported. The value for filter_em_threshold is the J/Jlimit value.</p> <p>filter_em_threshold=1 reports only those resistors that have a J/Jlimit value above 1.</p> <p><b>Note:</b> Specifying this option is equivalent to filtering all passed resistors.</p>	
	geounit= <i>value</i>	<p>Defines the scaling of all DSPF geometry parameters (W, L, X, Y). It applies only to the *   NET section, and not to the instance section.</p>	1um
	geounit_XY	<p>Defines the scaling of DSPF geometry parameters X and Y. It applies only to the *   NET section, and not to the instance section.</p> <p><b>Note:</b> If the geounit_XY and geounit options are specified together, the geounit_XY option takes precedence over the geounit option.</p>	1um

# Voltus-Fi Custom Power Integrity Solution User Guide

## Data Preparation

---

Keyword	Option Set	Explanation	Default Value
	geounit_WL	<p>Defines the scaling of DSPF geometry parameters W and L. It applies only to the *   NET section, and not to the instance section.</p> <p><b>Note:</b> If the geounit_WL and geounit options are specified together, the geounit_WL option takes precedence over the geounit option.</p>	1um
	via_count=[ "useR " "useArea" ]	<p>Defines the method used for calculating the via count.</p> <p>useR - calculation based on via resistance (emdatafile_via_resistance/dspf_via_resistance).</p> <p>useArea - calculation based on via area (dspf_via_area/emdatafile_via_area).</p>	useR
	idirn=[true false]	Prints the current direction in the EM report ( <i>Current Direction</i> column). For metal resistors, the current direction is printed. That is, <i>W</i> means that the current is flowing from east to west. For via resistors, the direction from/to layer is printed.	false
	recovery_factor	<p>Defines the recovery factor for iavg calculation with recovery factor. If specified, then</p> $iavg=\max( iavgpos ,  iavgneg ) - recovery\_factor*\min( iavgpos ,  iavgneg ).$ <p>If not specified, then</p> $iavg=iavgpos+iavgneg$	

# Voltus-Fi Custom Power Integrity Solution User Guide

## Data Preparation

---

Keyword	Option Set	Explanation	Default Value
	Tlife=hours   years	<p>Specifies the lifetime for which the EM analysis will be performed. You can specify the value in either years or hours. For example:</p> <pre>emirutil Tlife=20,000 hrs</pre>	years
	print_em_report	<p>Defines the EM analysis report file to be generated in the Voltus-Fi-XL flow.</p> <p>filename: create report with specified file name</p> <p>net: create report for specified nets - all power nets (all_power), all signal nets (all_signal), all nets (all_nets), or net (netname).</p> <p>threshold - create report for resistors or nodes that have a threshold value of EM ratio above the specified value</p> <p>type - create report for specified analysis type; javg, jmax, jabsavg, jacpeak, jacrms, rc, rcavg, rcrms, tc, tcavg, or tcrms.</p> <p>rule - specifies the custom EM rule for which the report should be generated</p>	
	extendedreport=true	<p>Specifies that the following information should be included in the EM analysis report file:</p> <ul style="list-style-type: none"> <li>■ Td: time duration in micro second or total ‘On Time’ period</li> <li>■ dutyR: duty ratio</li> <li>■ Ilimit: current limit</li> </ul>	false

# Voltus-Fi Custom Power Integrity Solution User Guide

## Data Preparation

---

Keyword	Option Set	Explanation	Default Value
	reportUnmatchedRuleResistor=true   false	When set to true, specifies that while generating the EM analysis reports, the resistors that do not match any EM rule for the specified analysis in the ICT file should be reported in the "#.rpt_unmatch" file.	true
	reportAllResistor=true   false	When set to true, reports all resistors of the specified nets in the "#.rpt_all" file.  <b>Note:</b> If both, reportUnmatchedRuleResistor and reportAllResistor, are set to true, the reportAllResistor setting gets priority.	false
	consolidatedReport=true	Specifies that consolidated EM and IR reports for multiple bin files will be generated in Voltus-Fi-XL.	false
	worstResult=false	Specifies that the worst result will be reported in the consolidated EM and IR reports. This option is enabled by default when the consolidatedReport parameter is set to true.  For example,  consolidatedReport=true worstResult=true  <b>Note:</b> To view all the results in the consolidated report, set this option to false.	true

# Voltus-Fi Custom Power Integrity Solution User Guide

## Data Preparation

---

Keyword	Option Set	Explanation	Default Value
	layer_filter=[layer1 layer2...]	<p>Specifies the layers that are to be filtered out from the EMIR analysis reports.</p> <p>For example,</p> <pre>emirutil layer_filter=[cont odcont]</pre> <p><b>Note:</b> If you have specified this option and also set the <code>reportAllResistor</code> option to <code>true</code>, the resistor information for the specified layers will be filtered out from the reports.</p>	none
	disableLayoutScale=true	<p>When set to <code>true</code>, disables the layout scale factor defined in the ICT file for Voltus-Fi-XL EM analysis. The <code>layout_scale</code> factor is the ratio of the scaled dimensions divided by the drawn dimensions.</p> <p>The ICT file is defined based on the scaled dimensions. Quantus applies the <code>layout_scale</code> factor to the input design during extraction so that the design dimensions and the ICT file dimensions are consistent.</p>	false
time	window=[start1 stop1 start2 stop2 ...]	Time window to which the EMIR analysis is applied. Multiple non-overlapping windows are supported.	[0 tend]
spf	aliasTerm="alias library_terminal name"	Alias used when SPF instance terminal names do not match the cell terminal name in the libraries.	none
spgs	net=[net1 net2 ...]	Enables static power grid solver, and defines power nets to be analyzed.	none
	pwrgate=[powerne t1 powernet2 ...]	Defines power gates to be analyzed in static power grid solver.	none

# Voltus-Fi Custom Power Integrity Solution User Guide

## Data Preparation

---

Keyword	Option Set	Explanation	Default Value
	rshort=value	Resistors with R<rshort are shorted during static power grid analysis.	0
	tap2sub net=<net_name> include=[<tap_name>] exclude=[<tap_name>]	Converts tap to subnode.	none
	tap2pin net=<net_name> include=[<tap_name>] exclude=[<tap_name>]	Converts tap to pin.	none
	sub2tap net=<net_name> include=[<node_name>] exclude=[<node_name>]	Converts subnode to tap.	none
	sub2pin net=<net_name> include=[<node_name>] exclude=[<node_name>]	Converts subnode to pin.	none
	pin2sub net=<net_name> include=[<pin_name>] exclude=[<pin_name>]	Converts pin to subnode.	none

# Voltus-Fi Custom Power Integrity Solution User Guide

## Data Preparation

---

Keyword	Option Set	Explanation	Default Value
	pin2tap net=<net_name> include=[<pin_name>] exclude=[<pin_name>]	Converts pin to tap.	none

 **Important**

In earlier releases, Voltus-Fi-XL 6.1.6 ISR 8 and before, the geounit\* option was specified as follows:

geounit\* = 1e6 – when values in DSPF were in meter  
geounit\* = 1 – when values in DSPF were in um

From 6.1.6 ISR 9 release onwards, the behavior of the geounit\* option is modified as follows to synchronize it with emirutil:

geounit\* = 1 – when values in DSPF are in meter  
geounit\* = 1e-6 – when values in DSPF are in um

**Note:** Every net statement only supports one name and one analysis definition. If different analyses are required, use multiple net statements.

The following example performs the EMIR analysis on the x1 instance. EM and IR analyses are performed on net VDD and VSS to measure vmax and iavg. EM analysis is performed on all other nets to calculate the irms value. The current density limits for the EM report are read from file ./em\_dir/qrcTechFile.txt. In this release, only one instance is allowed for EMIR analysis per Spectre run.

### Example EMIR control file (emir.config)

```
net name=[X0.VDD X0.VSS] analysis=[vmax vavg] net name=[X0.*] analysis=[iavg imax]
emirutil techfile = qrcTechfile layermapfile = contactmapfile

emirutil deltaT=10.0
emirutil dbu=500
emirutil dynamicACPeak=false
emirutil applyRThreshold=0.05
emirutil rmsMultiplierPin=1.0
emirutil rmsMultiplierSignal=1.0
emirutil useEncryptedWidth=true
net design_res_models = [name=resTwoTerm l=lr w=wr layer=mt2 unit=1e6]
```

```
solver method=[iterated]
```

For details on how to specify variables in the EMIR control file, see the “[Variables](#)” chapter.

## Setup for Visualization of IR and EM Results

Voltus-Fi-XL displays plots for the specified IR drop and EM analysis results on the Virtuoso layout. As a requirement for plotting IR drop and EM results on the layout, the APS/XPS-to-DFII layer map file must be specified. This file provides the mapping between the layer names in the extractor-generated xDSPF file to the DFII layer names.

Multiple extraction layers can be mapped to the same DFII layer. This layer map file is also used for performing structural analysis.

**Note:** For details of the file format and to view a sample file, see [DFII Layer Map File](#) in the “File Formats” chapter.



If you do not specify the correct mapping in the DFII layermap file or if some layers are not included in this file then resistors or nodes of those layers will not be highlighted on the layout because there will be no shapes attached to them.

## Loading the DFII Layer Map File by Default

You can specify the path to the DFII Layer Map file in the GUI for both IR drop and EM analysis results in the *DFII Layermap file* field provided in both the IR and EM tabs of the IR/EM Results form. You can also use a variable to load the layer map file by default. For this, specify the DFII layer map file in either the `.cdsinit` or the `.cdsenv` file by using the `vfiDfiiLayerMapFile` environment variable. The syntax and example of specifying the path to the DFII layer map file is provided below.

- In the `.cdsinit` file, specify the following:

```
envSetVal("voltus_fi.results" "vfiDfiiLayerMapFile" 'string'  
"MMSIM_EMIR_workshop/df2layermap")
```

Where,

`voltus_fi.results` is the name of the tool

`vfiDfiiLayerMapFile` is the name of the variable

`MMSIM_EMIR_workshop` is the path to the layer map file

`df2layermap` is the name of the layer map file

- In the .cdsenv file, specify the following:

```
voltus_fi.results vfiDfiiLayerMapFile string  
"MMSIM_EMIR_workshop/df2layermap"
```

### **Setting a Layer Index for Displaying Violations on the Layout**

Voltus-Fi-XL creates layers for displaying the EMIR violations when you plot them on the Virtuoso Layout. By default, the software uses more than a hundred layers for displaying violations. It creates these layers by using a layer index value or layer number that is equal to or greater than 256.

However, if you are already using the same layer number as that used by Voltus-Fi-XL, a layer number conflict will occur and a warning will be issued. To avoid this warning, specify a different value for the layer number. You can specify the layer index value in either the .cdsinit or the .cdsenv file by using the `vfiLayerIndex` environment variable. The syntax and example of specifying the variable is provided below.

- In the .cdsinit file, specify the following:

```
envSetVal("voltus_fi.results" "vfiLayerIndex" 'int value')
```

Where,

`voltus_fi.results` is the name of the software

`vfiLayerIndex` is the name of the variable

`int` is integer

`value` is the layer index value

For example,

```
envSetVal("voltus_fi.results" "vfiLayerIndex" 'int 270')
```

- In the .cdsenv file, specify the following:

```
voltus_fi.results vfiLayerIndex int value
```

For example,

```
voltus_fi.results vfiLayerIndex int 270
```

## EM Rule File Requirements

Before running EMIR analysis in Voltus-Fi-XL, you need to specify the EM reliability rules.

The EM reliability rules can be specified in any of these files: the QRC technology file (`qrcTechFile`), the ICT file, or the EM data file (`emDataFile`).

The ICT file contains the process information for the design and information about the EM rules. This is a text-based file that can be edited using any text editor.

You can either specify the ICT file directly as an input for EM analysis, or you can use the information in the ICT file to update the `qrcTechFile`. You can view a sample [ICT File](#) in the “File Formats” chapter.

For more information about the ICT file, see “Creating the ICT File” chapter in *QRC Techgen Reference Manual*.

The EM reliability rules that can be specified in the `qrcTechFile` and the `emDataFile` are detailed below.

### EM Rules in `qrcTechFile`

This section provides details of the current density (`jmax`) keywords that can be specified in the `qrcTechFile` and the rules for defining EM parameters. It covers the following topics:

- [Parameters for Limit-Based Analysis](#)
- [Current Density \(JMAX\) Keywords](#)
- [Rules for Defining EM Parameters](#)
- [Rules for Specifying Via Area](#)
- [Rules for Specifying EM Analysis Type for Power and Ground Nets](#)
- [EM Rule Selection Priority](#)

**Note:** Currently, Voltus-Fi-XL only supports parameters for limit-based EM analysis.

#### ***Parameters for Limit-Based Analysis***

The following parameters can be specified in the `qrcTechFile`:

**Table 4-2 Limit-Based Analysis Parameters**

Parameter Name	Description
em_vcwidth <value>	Optional parameter for via that defines the minimum size via that is used by the software to identify single square via.
em_W_n	Optional parameter that specifies the wire width in microns below which a line is considered narrow. All lines are considered wide unless you specify otherwise using this parameter  <b>Note:</b> It helps to select the rule between em_jmax_*_w and em_jmax_*_n. If w < em_W_n , then em_jmax_*_n is selected, otherwise em_jmax_*_w selected.

**Note:** For more information about limit-based EM model syntax, see the EM\_Models section in “Creating the ICT File” chapter of *QRC Techgen Reference Manual*.

### **Current Density (JMAX) Keywords**

The table below lists the current density keywords that are supported in Voltus-Fi-XL.

**Table 4-3 Current Density Keywords**

Keyword	Description
em_jmax_dc_avg*	Optional keyword that specifies the DC AVG analysis in Voltus-Fi-XL
em_jmax_ac_avg*	Optional keyword that specifies the AC AVG analysis in Voltus-Fi-XL
em_jmax_dc_peak*	Optional keyword that specifies DC PEAK analysis in Voltus-Fi-XL
em_jmax_ac_peak*	Optional keyword that specifies AC_PEAK analysis in Voltus-Fi-XL
em_jmax_dc_rms*	Optional keyword that specifies DC RMS analysis in Voltus-Fi-XL

# Voltus-Fi Custom Power Integrity Solution User Guide

## Data Preparation

Keyword	Description
em_jmax_ac_rms*	Optional keyword that specifies AC RMS analysis in Voltus-Fi-XL
em_jmax_dc_absavg*	Optional keyword that specifies DC average absolute analysis in Voltus-Fi-XL
em_jmax_ac_absavg*	Optional keyword that specifies AC average absolute analysis in Voltus-Fi-XL

Where star '\*' implies that '\_w' and '\_n' rules are also supported.

For more information about the above keywords, refer to "Creating the ICT File" chapter of *QRC Techgen Reference Manual*.

### Rules for Defining EM Parameters

An example of the EM rule for defining the em\_jmax\_dc\_avg parameter is provided below:

```
[em_jmax_dc_avg <value> | <value_1> <area_1/width_1> [...] | EQU <fn(E)>]  
[jmax_factor <temp1> <scale1> [<temp2> <scale2> ...]]  
[jmax_lifetime <lifetime1> <scale1> [<lifetime2> <scale2>.....]]  
[current_direction up | down | both]  
[conditions]  
[single]  
[power_rail/power_grid]  
[priority==<priority no.>]  
[device=="<device model names>"]  
[sub_conductor=="<subconductor names>"]  
[color=="<list of color no.>"]  
[mask==<mask no.>]
```

**Note:** Currently, the via\_range construct is not supported in Voltus-Fi-XL. If it is specified in the EM rule, it will be ignored by the software.

```
[via_range <value>]
```

All other EM parameters are defined with the same structure but apply to different characteristics of the em\_model (peak current versus average for instance). The following rules apply to all the em\_model parameters:

### ■ Equations in EM Model

An EOL or one of the qualifiers (`jmax_factor`, `current_direction`, or `single`) marks the end of an equation specifying the current density limit. If the equation resolves into two separate equations, then this will result in an error.

For EQU, the following operators (case-insensitive) are supported:

- SQRT
- LOG
- EXP
- \* / + -
- ()
- e
- ^

**Note:** Square brackets ‘[ ]’ are not supported. A comma ‘,’ implies multiple equations, and results in an error. The period ‘.’ is ambiguous and should not be used to indicate multiplication (use \* instead).

Spaces are permitted to make an equation more readable.

Variables within equations (such as `deltaT`, `w` and `l`) will be populated by the analysis tool at run-time. Use the `em_variables` parameter in the process definition to globally declare variables that are used within the various `em_models`.

The equation definition ends when reaching the end of line (EOL) or another keyword (`jmax_factor`, `current_direction`, or `single`).

## ■ Order Dependency

The actual current density limit (value, piece-wise linear or PWL pairs, or equation) should follow immediately after the current density keyword (`em_jmax_dc_avg`). Any of the qualifiers (`jmax_factor`, `current_direction`, or `single`) will come after that. The order of the qualifiers is not important.

## ■ Units of Values

The unit should be specified in accordance with the setting defined by the `em_conductor_unit`, `em_via_unit`, and `em_via_area_unit` parameters defined in the process definition. Units are specified as:

- For a single value or an equation:
  - A/cm<sup>2</sup> or mA for metal layers (width based)

# Voltus-Fi Custom Power Integrity Solution User Guide

## Data Preparation

---

- A/via or mA for via layers (per via)
- For a PWL table:
  - A/cm<sup>2</sup> or mA, with width specified in microns for metal layers
  - A/cm<sup>2</sup> or mA, with area specified in square microns for via layers

For more information, refer to the Process section of the “Creating the ICT File” chapter in *QRC Techgen Reference Manual*.

## ■ JMAX Factor

```
jmax_factor <temp1> <scale1> [<temp2> <scale2> ...]
```

`jmax_factor` is the optional scaling factor to use at different temperatures compared to the reference temperature (defined by `em_tref` in the process definition). The temperature for `jmax_factor` should be specified in degrees Celsius.

Scaling factor is a positive integer: >1 to scale up, <1 to scale down, or 1 for no scale effect.

**Note:** When setting scale factors for multiple temperatures, they should be specified in an ascending sequence to enable interpolation.

## ■ Format of the Conditions

```
Condition = [L/W/Lu/Wu/Lb/Wb/Lv2v/Td/r/a/N < | <= | > | >= | == | != <value in microns/micro seconds>] | [Itolerance < | <= | > | >= | == | != <value in percentage>]
```

```
Conditions = [condition] *
```

The `L` parameter specifies the length-based jmax values in microns.

The `w` parameter specifies the width-based jmax values in microns.

`L` and `w` are applicable for both metals and vias.

`Lu`, `Lb`, `Wu`, and `Wb` are applicable for vias only.

`Lv2v` and `Itolerance` are applicable only for the `power_rail` rule.

Where,

`L` = length

`W` = Width

`Lu` = Upper metal length

`Lb` = Bottom metal length

`Wu` = Upper metal width

# Voltus-Fi Custom Power Integrity Solution User Guide

## Data Preparation

---

$W_b$  = Lower metal width

$L_{v2v}$  = Distance between terminal vias

$I_{tolerance}$  = Difference in the current values of terminal vias. For example, if a difference of up to 5 percent is acceptable, then specify ' $I_{tolerance} <= 0.05$ '.

$T_d$  = Time duration in micro second or total ‘On Time’ period

$r$  = Duty ratio

$a$  = Area of the vias

$N$  = Number of vias in the via array

Voltus-Fi-XL supports metal length/width rules above and below vias, where, the dimensions to be checked for the rules are different for the metal above and the metal below. In this case,  $L_u/W_u$  refers to dimensions of the metal above the via and  $L_b/W_b$  refers to dimensions of the metal below the via.

For more information, refer to “Creating the ICT File” chapter of *QRC Techgen Reference Manual*.

### ■ Single

The keyword, `single` can be specified to differentiate between a single square cut via and square viaarray. The following example shows how to use the `single` keyword:

```
em_jmax_dc_avg PWL 0.022 0.01 jmax_factor 105 1.1 110 1.0 115  
0.9 120 0.8 125 0.7 130 0.6 140 0.5 150 0.4 current_direction up  
Lb > 4 Wb >= 0.05 current_direction up single
```

### ■ JMAX Lifetime

The keyword, `jmax_lifetime` provides the ability to set the scaling factor that applies to the current density limits for different lifetimes. The syntax is as follows:

```
jmax_lifetime <lifetime1> <scale1> [<lifetime2> <scale2>.....]
```

**Note:** The software will take the unit of lifetime from the `em_lifetime_units` parameter specified in the process section of the ICT file.

### ■ Current Direction

`current_direction [up | down | both]` can be specified for via current direction. The syntax of `current_direction` is:

- ❑ `up` - means the direction of the current is from bottom to top and specifies to use the length and width of the metal below  $L_b/W_b$  for the VIA Jmax factor
- ❑ `down` - means the direction of the current is from top to bottom and specifies to use the length and width of the metal above  $L_u/W_u$  for VIA Jmax factor
- ❑ `both` - This rule is only applicable when the direction of the current is uncertain. This rule is unlikely to be applied since the software calculates the current direction

## ■ Power Rail/Power Grid

The keyword `power_rail` or `power_grid` can be specified to enable the particular EM rule for power-rail analysis. When power-rail analysis is enabled, an EM rule with the `power_rail` or `power_grid` keyword will have a higher priority than any other rule.

```
em_jmax_dc_avg PWL 0.022 0.01 jmax_factor 105 1.1 110 1.0 115  
0.9 120 0.8 125 0.7 130 0.6 140 0.5 150 0.4  
Lb > 4 Wb >= 0.05 current_direction up  
power_rail Lv2v <= 5 Itolerance <= 0.05
```

## ■ ApplyR

The keyword `applyR` can be used to modify the threshold value of ratio “*r*” in AC-Peak analysis. The default value of this parameter is 0.05. Modifying this parameter is useful in scenarios where the value of “*r*” is very small. A small “*r*” value results in an unreasonable increase in the `Ipeak_ac` limit. You can avoid such scenarios by resetting the value of “*r*” in the ICT file. For example:

```
em_jmax_ac_peak EQU 4.0 r < 0.01 applyR 0.01
```

## ■ Priority

Specifies the priority of a rule. It is an integer number. Currently the only value that is supported is 1. The syntax is as follows:

```
priority==<integer no.>
```

**Note:** The rule with `priority==1` is only enabled when the variable '`powerRailRules=n10_special`' is set and when the `power_grid` and `power_rail` keywords are also specified.

## ■ Device

This keyword is used to specify different EM rules for device resistors, which are subsets of other layers. The syntax is as follows:

```
device=="device model names"
```

For example, the following keyword specifies that the particular EM rule is only applicable to device with the model names, “`devRA devRB`”.

```
device=="devRA devRB"
```

## ■ Subconductor

This keyword is used to specify different EM rules for subconductors, which are subsets of poly or other layers. The syntax is as follows:

```
sub_conductor=="subconductor names"
```

For example, the following keyword specifies that the particular EM rule is only applicable to subconductor layers, `pploy` and `gpoly`.

```
sub_conductor=="pploy gpoly"
```

■ **bridge\_via**

This keyword is used to specify that the particular EM rule will apply only to bridge vias.

■ **Color**

This keyword is used to specify that the particular EM rule is only applicable to the resistor with the specified color number.

For example,

```
color=="2 4 5" specifies that the particular EM rule is only applicable to the resistor with property, $M=2 , $M=4, or $M=6.
```

■ **Masks**

This keyword is used to specify that the particular EM rule is only applicable to the resistor with the specified mask number.

For example,

```
mask==2 specifies that the particular EM rule is only applicable to the resistor with property, $M=2.
```

### ***Rules for Specifying Via Area***

You can specify PWL in either one of the following ways:

■ Provide PWL for a specific area:

```
em_jmax_* PWL Value_1 area_1 ... Value_N areaN
```

For example,

```
em_jmax_* PWL 1.3 2.0 1.7 3.0
```

■ Provide PWL for a specified area range:

```
em_jmax_* EQU <equation with 'a'> conditions
```

For example,

```
em_jmax_* EQU (1.3 * a)/2.0 a>=0.0 a<2.5
```

```
em_jmax_* EQU (1.7 * a)/3.0 a>=2.5 N>=3
```

```
em_jmax_* EQU (1.9 * a)/3.0 a>=2.5 N<3
```

**Rules for Specifying EM Analysis Type for Power and Ground Nets**

Set the variable, `splitACDCRules`, to `true` for specifying different rules for EM analysis of power and signal nets. You can set this variable in the following ways:

- `setenv VOLTUSFI_SPLIT_ACDC_RULES true`
- `set_variable splitACDCRules true`

When this variable is specified using the `set_variable` command, it takes priority over the `setenv` variable.

- Set the variable, `splitACDCRules` to `true` in the Variables form. You can open this form from the EM tab of the IR/EM Results form.

The table below shows the rules for power and signal nets both when the variable is set and not set.

**Table 4-4 Rules for Power and Signal Nets**

<code>splitACDCRules false</code>	<code>splitACDCRules true</code>	
<b>Power and Signal Nets</b>	<b>Power Nets</b>	<b>Signal Nets</b>
<code>em_jmax_dc_peak</code>	<code>em_jmax_dc_peak</code>	<code>em_jmax_ac_peak</code>
<code>em_jmax_dc_avg</code>	<code>em_jmax_dc_avg</code>	<code>em_jmax_ac_avg</code>
<code>em_jmax_ac_rms</code>	<code>em_jmax_dc_rms</code>	<code>em_jmax_ac_rms</code>
<code>em_jmax_dc_absavg</code>	<code>em_jmax_dc_absavg</code>	<code>em_jmax_ac_absavg</code>

**EM Rule Selection Priority**

This section details the order of priority in which the EM rules are applied by Voltus-Fi-XL based on the specified keywords. The order of priority of keywords in descending order is provided below:

1. device --> **highest priority**
2. sub\_conductor
3. bridge\_via
4. priority

5. power\_grid/power\_rail
6. color
7. mask
8. current\_direction
9. Conditions
10. Area matching in case of PWL
11. Base rule
12. Optimistic/pessimistic rules --> **lowest priority**

These are detailed below.

■ **device**

When a device is specified in the EM rule file, the software first looks for the `device` keyword and applies the EM rules to the specified device.

For example, for a device with model name, `devRA`, if the following rules are specified in the ICT file:

1. `em_jmax* ... device=="devRA"`
2. `em_jmax* ... device=="devRB"`
3. `em_jmax* ...`

Then the software follows the order of priority provided below:

- Match rule 1 because it has a matching `device` keyword
- Match rule 3 if rule 1 does not match
- Skip rule 2 because it is for device, `devRB`

■ **sub\_conductor**

When a sub conductor is specified in the EM rule file, the software first looks for the `sub_conductor` keyword and applies the EM rules to the specified sub conductor.

For example, for a resistor on layer, `ppoly`, if the following rules are specified in the ICT file:

1. `em_jmax* ... sub_conductor=="ppoly"`
2. `em_jmax* ... sub_conductor=="gpoly"`
3. `em_jmax* ...`

Then the software follows the order of priority provided below:

# Voltus-Fi Custom Power Integrity Solution User Guide

## Data Preparation

---

- Match rule 1 because it has a matching `sub_conductor` keyword
- Match rule 3 if rule 1 does not match
- Skip rule 2 because it is for sub conductor, `gpoly`

### ■ `bridge_via`

When a `bridge_via` is specified in the EM rule file, the software first looks for the `bridge_via` keyword and applies the EM rules having `bridge_via` keyword for the bridge via. In case, the via is not bridge via, rules having `bridge_via` keyword will be ignored.

For example, for a bridge via, if the following rules are specified in the ICT file:

1. `em_jmax*` ... `bridge_via`
2. `em_jmax*` ...

Then the software follows the order of priority provided below:

- Match rule 1 because it is a bridge via
- Match rule 2 if rule 1 does not match

In case, the via is not bridge via, then the software will match rule 2.

### ■ `priority`

When a priority is specified in the EM rule file, in the following manner:

1. `em_jmax*` ... `priority==1`
2. `em_jmax*` ...

Then the software follows the order of priority provided below:

- Match rule 1 when `powerRailRules=n10_special` is set
- Match rule 2 if rule 1 does not match

### ■ `power_rail/power_grid`

When the `power_rail/power_grid` keyword is specified in the EM rule file, in the following manner:

1. `em_jmax*` ... `power_grid`
2. `em_jmax*` ...

Then the software follows the order of priority provided below:

- Match rule 1 when the power grid analysis is enabled
- Match rule 2 if rule 1 does not match

## Voltus-Fi Custom Power Integrity Solution User Guide

### Data Preparation

---

#### ■ color

When the `color` keyword is specified in the EM rule file, in the following manner:

1. `em_jmax* ... color=="1 3 5"`
2. `em_jmax* ...`

For example, if a resistor has color number 5, specified using `$M=5`, then the software follows the order of priority provided below:

- Match rule 1 for the specified color number
- Match rule 2 if rule 1 does not match

#### ■ mask

When the `mask` keyword is specified in the EM rule file, in the following manner:

1. `em_jmax* ... mask==2`
2. `em_jmax* ...`

For example, if a resistor has mask number 2, specified using `$M=2`, then the software follows the order of priority provided below:

- Match rule 1 for the specified mask number
- Match rule 2 if rule 1 does not match

#### ■ current\_direction

When the `current_direction` keyword is specified in the EM rule file, in the following manner:

1. `em_jmax* ... current_direction up`
2. `em_jmax* ... current_direction down`
3. `em_jmax* ...`

Then, for a resistor with `current_direction up`, the software follows the order of priority provided below:

- Match rule 1 because it has a matching `current_direction` keyword
- Match rule 3 if rule 1 does not match
- Skip rule 2 because the current direction is not matching

#### ■ Conditions

When conditions are specified in the EM rule file, in the following manner:

1. `em_jmax* ... L <=5`
2. `em_jmax ... L > 5`
3. `em_jmax* ...`

# Voltus-Fi Custom Power Integrity Solution User Guide

## Data Preparation

---

Then the software follows the order of priority provided below:

- Match rules 1 and 2 because conditions are specified
- Match rule 3 if rules 1 and 2 are not matching

### ■ Area Matching from the PWL

When the PWL for specific via areas are specified in the EM rule file, in the following manner:

1. em\_jmax\* ... PWL AREA\_1 VAL\_1 AREA\_2 VAL\_2 L <=5
2. em\_jmax ... PWL AREA\_3 VAL\_3 AREA\_4 VAL\_4 L <=5

Then the software follows the order of priority provided below:

- Match the exact areas provided in the rule file
- The software does not interpolate for unmatched areas (areas other than AREA\_1, AREA\_2, AREA\_3, and AREA\_4 in above example)

### ■ Base Rule Selection

When the PWL for specific via area is specified in the EM rule file, **without any conditions** in the following manner:

```
em_jmax* ... PWL AREA_1 VAL_1 AREA_2 VAL_2
```

Then the software follows the order of priority provided below:

- Match the exact areas from the base rule
- The software does not interpolate for unmatched areas (areas other than AREA\_1 and AREA\_2 in above example)

### ■ Optimistic/Pessimistic Rule Selection

When multiple rules are selected after all conditions match, the software selects the optimistic/pessimistic value based on whether the variable, `optimisticEMRuleSelection`, is set to true or false.

By default, the software selects the pessimistic value.

For example, when the following rule set is provided:

1. em\_jmax\* EQU 1.5 ...
2. em\_jmax\* EQU 3.5 ...
3. em\_jmax\* EQU 2.5 ...

- Rule 1 is selected by default because it is more pessimistic as compared to rules 2 and 3

## Voltus-Fi Custom Power Integrity Solution User Guide

### Data Preparation

---

- When the variable, `optimisticEMRuleSelection`, is set to `true`, then rule 2 is selected because it is more optimistic as compared to the other rules
- Rule 2 is skipped because it is neither the most pessimistic nor the most optimistic

#### **Example**

Consider the two scenarios provided below for EM rule selection.

The given rule set is as follows:

1. `em_jmax*... current_direction up L>=5`
2. `em_jmax*... current_direction up L<5`
3. `em_jmax*... current_direction down L>=5`
4. `em_jmax*... L<5`

**Case 1:** EM rule selection for a resistor with current direction “up” and with `L==3` will be as follows:

- The tool will select rules 1 and 2 because they match the current direction “up”
- Of the two selected rules, the tool will discard rule 1 because of the non-matching condition, `L>=5` and it will select rule 2 because of the condition, `L==3`, which matches the condition `L<5`

The final rule selection in Case 1 will be rule 2.

**Case 2:** EM rule selection for a resistor with current direction “down” and `L==2` will be as follows:

- The tool will first select rule 3 as it matches the current direction “down”
- The tool will then discard the selected rule 3 because of the non-matching condition, `L>=5`
- The tool will then select rule 4 from the remaining rules, as rule 4 does not have any current direction condition but it matches the condition, `L<5`

The final rule selection in Case 2 will be rule 4.

#### **EM Data File**

An EM data file specifies the technology information, such as current density limits, and provides a mapping between the layers for highlighting. For a sample EM Data file, see [EM Data File](#).



EM data file support is provided only for backward compatibility. It supports all the rules that were supported by APS EMIR. Any new rules are supported in the qrcTechFile and the ICT file.

For detailed information about the EM data file command syntax, see the “EM Data File” section in the “Netlist-Based EM/IR Flow” chapter of the *Virtuoso® UltraSim Simulator User Guide*.

## Outputs of EMIR Analysis

As output, Voltus-Fi-XL produces the following:

- Power-rail and signal IR drop and EM analysis result plots displayed on the Virtuoso layout
- Power-rail and signal IR drop and EM analysis result reports
- Structural analysis results displayed on the Virtuoso layout
- Power-grid views

## **xDSPF Generation using Quantus**

---

- [Overview](#) on page 94
- [Data Requirements for xDSPF Generation](#) on page 94
- [Generating xDSPF](#) on page 94

## Overview

The third-party netlist file or the xDSPF file is used to provide all the parasitic information of the design for performing simulation using the Spectre® APS/XPS simulators.

The xDSPF file is created using an extractor. This file is then used as input for performing simulation. The simulation database that is generated is used to perform EMIR analysis in Voltus-Fi-XL.

However, if you are using Quantus QRC for extraction, you can also generate the xDSPF file from within Voltus-Fi-XL.

For this, you can use either the GUI or the batch mode. In the batch mode, run the `extract_xdspf` command. For details of this command, see [Batch Command for xDSPF Generation](#) in the “Batch Mode Execution” chapter.

The sections below provide details for creating the xDSPF file from the Voltus-Fi-XL GUI.

## Data Requirements for xDSPF Generation

Before generating the xDSPF file, ensure the following:

- The Quantus hierarchy is available as the software calls the Quantus binary from this path.
- The PVS or Calibre® run is performed successfully to generate the PVS or Calibre database, which is the input database for generating the xDSPF file using Quantus.

## Generating xDSPF

To create xDSPF file using Quantus in Voltus-Fi-XL, click *Extraction – Extract DSPF*. The Extract xDSPF form opens. This form is shown below.

## Voltus-Fi Custom Power Integrity Solution User Guide

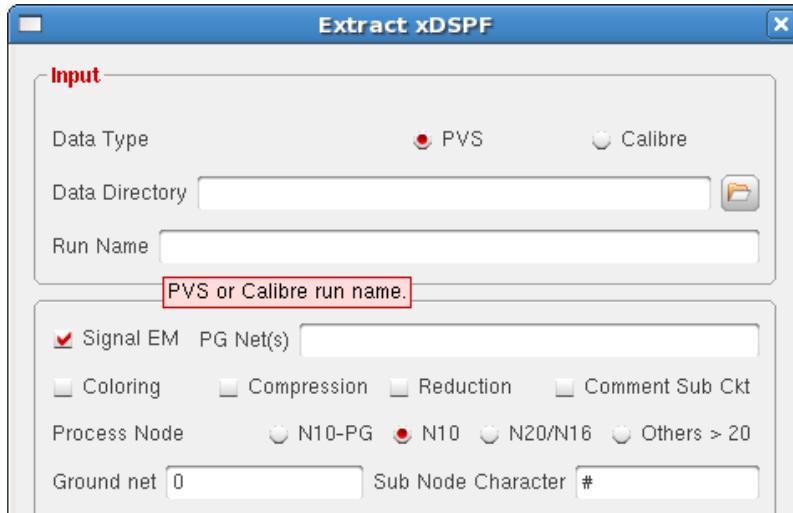
### xDSPF Generation using Quantus

Figure 5-1 The Extract xDSPF Form



In this form, you can provide the information detailed below. The software provides support in the form of tips about the information to be filled out in the various fields. This tool tip appears when you hover over a field in the form. This is shown below.

**Figure 5-2 Tool Tips for Extract xDSPF Form**



In the Extract xDSPF form, provide the following information:

- For *Data Type*, select either *PVS* or *Calibre®*. This specifies whether the input data will come from *PVS* or *Calibre*. By default, *PVS* is selected.
- Specify the *Data Directory*, which provides the path to the directory in which the input data is stored.
- Specify the *Run Name*, which is the name of the PVS or Calibre run.
- Specify the *Process Node*. The following options are available:
  - *N10-PG*: for N10 process nodes with PG rules
  - *N10*: for N10 process nodes
  - *N20/N16*: for process nodes from N16 to N20
  - *Others > 20*: for process nodes greater than N20
- Select *Signal EM* to specify that signal nets should be included in the xDSPF file. When *Signal EM* is selected, the following command is added to the Quantus command file or the common command language (CCL) file, in order to perform coupled RC extraction (*rc\_decoupled*) for all nets.

**Note:** In *rc\_decoupled* extraction, an RC network for the specified net(s) with both parasitic resistance and capacitance decoupled to ground is extracted.

```
extract \
    -selection "all" \
    -type "rc_decoupled"
```

When *Signal EM* is not selected, you need to provide the names of the power or ground nets on which `rc_decoupled` extraction will be performed. For all other nets, `c_only_coupled` extraction is performed.

**Note:** In `c_only_coupled` extraction, only parasitic capacitance coupled between the specified net or nets is extracted.

For example, when the following command is specified, `rc_decoupled` extraction is performed on all nets:

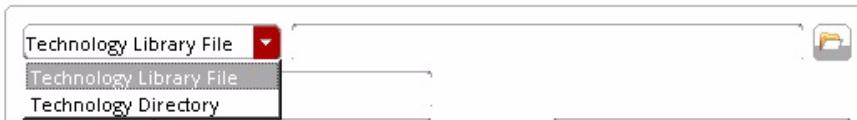
```
extract \
    -selection "all" \
    -type "c_only_decoupled"
```

When the following command is specified, `rc_decoupled` extraction is performed on nets specified in the `vfiQrc_nets` file. For all other nets, `c_only_coupled` extraction is performed.

```
extract \
    -selection "nets_file ./vfiQrc_nets"
    -type "rc_decoupled"
```

- Specify *PG Net(s)* that should be included in the xDSPF file. This option is specified only when the *Signal EM* option is unselected. If the *Signal EM* option is unselected and *PG Nets* are not provided, a pop-up window opens asking for the list of PG Nets.
- Select *Coloring* to enable color-aware EM analysis.
- Select *Compression* to enable xDSPF to be generated in a compressed format.
- Select *Reduction* to enable the `reduce_i_cards` option in the CCL file that is used for generating the xDSPF file. If a net is capacitance only, it is qualified for reduced I card or instance card treatment. This means the I cards, and the corresponding dummy resistors are not printed in the results.
- Select *Comment Sub Ckt* to comment out the top sub circuit statements from the xDSPF file.
- Select *Sub Conductor Model* to specify the parasitic resistor models for subconductor layers that should be included during extraction. This is used to support different EM rules for poly layers during EM analysis.
- Specify the *Ground net* on which the extraction is to be performed. By default, 0 is specified.
- Specify the *Sub Node Character* that will be used as subnode identification. By default, # is specified.

- Specify the *Fracture Via Count* that will be used to divide pseudo vias into segments during extraction.
- Specify the *Via Array Spacing*, which is the distance, floating point micron value, that will be used as the maximum distance criteria for grouping vias within the same array. This means that if the distance between vias is less than the specified maximum distance, they are grouped within the same array. The default value of via array spacing is 0.35 times the via size for N10, N16, and N20 processes. For other processes, the default is *auto*, in which case the software sets the via spacing automatically.  
Click this option to specify a value other than the default value of 0.35.
- From the drop-down list, either specify the *Technology Library File* that provides the path of the technology file to be used for reading the technology data or the name of the *Technology Directory* that contains the technology file.



- Specify the *Technology Name*, which is the technology file name to be used for extraction. This field is optional if the technology directory has been specified.
- Specify the *Temperature* at which the extraction will be performed. By default, a temperature of 85 degree Celsius is used.
- Specify the number of CPUs to be used during the Quantus extraction run in the *No of CPUs* field. By default, the field shows 8 as the number of CPUs.
- Specify the *SubCkt Pin Order File* to provide the order of pins to be used during extraction. The pin order is specified in the xDSPF file using the .SUBCKT and .ENDS statements. You can use this option to provide a file with a different pin order that will override the order specified using the default .SUBCKT statement in the xDSPF file.

Below are examples of pin orders in the xDSPF file before and after specifying the pin order file.

#### Without the sub-circuit pin order file

```
.SUBCKT top In_001 VDD Out_001 VSS In_002 Out_002 In_003 Out_004 ...
.ENDS
```

#### With the sub-circuit pin order file

```
.SUBCKT top VDD VSS In_001 Out_001 In_002 Out_002 In_003 Out_004 ...
.ENDS
```

- Specify the *Customized Extract Commands File* to specify the file with user-customized extract commands. When this file is specified, the Voltus-Fi-generated CCL will not have any extract commands and the commands from the customized extract commands file will be added in the Voltus-Fi-generated CCL.

A sample customized extract command file is as follows:

```
extract \
    -selection "all" \
    -type "rc_coupled"
```

**Note:** You can create customextract file by using either single or multiple extract settings. Ensure that the commands are in the Quantus QRC CCL file syntax as this file is added as is in the Voltus-Fi-generated CCL.

- Specify the *Parasitic Blocking device cells* file to specify the file containing parasitic blocking device cell names, if any. Once a cell is declared a blocking cell, all the parasitic RCs inside the cell are not extracted. If the cell content overlaps top cell routing, then, the top cell routing's RCs in the overlapping regions are also not extracted.
- Specify the *Directory*, which is the output directory in which the xDSPF file will be saved. If this file is not specified, a pop-up window opens asking for the name of the directory.
- Specify the name of the *xDSPF* file that will be generated. If the file name is not specified, the software generates a file with the name, <runName>.dspf, by default.
- Click *Generate CCL* to create the CCL or the Quantus command file that will be used to run Quantus. By default, vfiQuantus.ccl is created and the file opens in the Voltus-Fi-XL console. You can view the file and modify it, if required.
- Click *Generate and Run* to create the CCL file and run Quantus to create the xDSPF file in a single step. This is useful when you do not want to view the CCL file before running Quantus.
- Click *Run* to select the CCL file to use for running Quantus. This is useful if you have modified the auto-generated CCL file in the previous step. The log is saved both in the "vfiQuantus.log" and in the Voltus-Fi-XL console.

**Note:** For information about the format of the Voltus-Fi-generated CCL file and its example, see [Common Command Language \(CCL\) File](#) in the “File Formats” chapter.

**Voltus-Fi Custom Power Integrity Solution User Guide**  
xDSPF Generation using Quantus

---

---

## **Static Current Analysis**

---

- [Static Current Analysis](#) on page 102
  - [Static EMIR Flow in Spectre](#) on page 102
- [Static Current Analysis Overview](#) on page 102
- [Static Current Analysis Flow](#) on page 104
  - [Data Requirements for Static Current Calculation](#) on page 104
  - [Setting Up and Running Static Current Analysis](#) on page 104
    - [Setting Up and Running Peak Static Current Analysis](#) on page 104
    - [Setting Up and Running the Average Static Current Analysis](#) on page 109
- [Static Current Analysis Report](#) on page 113

## Static Current Analysis

Static current analysis is performed in Voltus-Fi-XL using Thunder. It is performed before running a simulation in Spectre. The static current analysis output file, `static_current.txt`, is used as an input for performing a Spectre simulation. Following are the key aspects of this feature:

- Support for both, peak and average modes for static analysis
- Support for both, GUI and TCL commands

### Static EMIR Flow in Spectre

A static EMIR analysis allows the evaluation of IR drop and EM currents based on user-provided subcircuit instance current consumption w/o running a transient or DC simulation. The user-provided currents are distributed to the tap devices based on the W/L ratio of the devices in the design. The IR drop and EM current analysis is performed based on the current at each tap device.

The static EMIR analysis is enabled with the “`static ifile`” statement in the EMIR config file.

```
net name=[I1.VDD I1.VSS] analysis=[vavg iavg]  
static ifile="static_currents.txt"
```

Since the analysis is static, the IR drop `vmax` and `vavg`, and the EM current `imax`, `irms`, and `iavg` values will be the same, and just one can be selected in the statement.

For more information, see the “Static EMIR Analysis” section in the Postlayout Simulation chapter of the [Virtuoso® Spectre® Circuit Simulator and Accelerated Parallel Simulator User Guide](#).

## Static Current Analysis Overview

Static current analysis can either be static peak current analysis (`ipeak`) or static average current analysis (`iavg`). Static peak current analysis is based on the peak saturation transistor currents. Because of simplicity and ability to find problems quickly in power grids, static current analysis is the most commonly performed analysis.

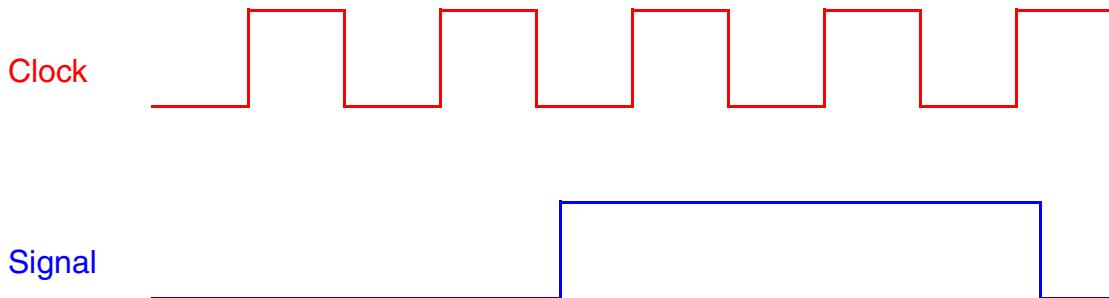
Static average current analysis or activity-based analysis resolves the distribution of currents on your power grid better than the static peak current analysis. The activity-based approach

assumes that you have a mechanism such as a Verilog simulator to compute and report the relative activity of the nets in the design. These relative activities can be used in conjunction with net capacitances to estimate the average current load of each gate in the design. This form of analysis clearly provides more realistic current data than the Ipeak approach.

### **Definition of Activity, Duty Cycle, and Transition Density**

**Activity** means the probability of all the signal nets in design switching from 0->1 or 1->0 in one clock cycle.

For instance, if the activity of a net or instance is 0.1, then the power engine assumes that the net or instance will switch from 0->1 or 1->0 once every ten clock cycles.



For the above diagram,

$$\text{Activity} = (\text{Number of } (0 \rightarrow 1 \text{ or } 1 \rightarrow 0) \text{ transitions} / \text{Number of clock cycles}) \\ = 2/5 = 0.4$$

**Duty Cycle** means the probability that a signal net has the value of 1.

For example, if the signal of a net is 1 for 2ns in the total simulation time of 10ns, then the duty cycle of net is 0.2. The duty cycle of the signal in the previous diagram is 0.5 (2.5/5). However, if a signal is Z or X for some time and 0 for the rest of the time then the duty cycle of the signal is 0.

**Transition Density** means the number of times the signal toggles from 0->1 or 1->0 in 1 second.

For the previous diagram, if you assume that one clock cycle is 4ns, then:

$$\text{Transition Density} = 1e+08 \quad (2/20\text{ns})$$

## Static Current Analysis Flow

In this section, the following topics will be covered:

- [Data Requirements for Static Current Calculation](#)
- [Setting Up and Running Static Current Analysis](#)

### Data Requirements for Static Current Calculation

The following data is required to perform static current calculations:

- A Quantus (QRC)-generated DSPF file
- SPICE model libraries and SPICE corners
- An Activity, clock, or value change dump (VCD) file—These inputs are required for the average static current analysis

### Setting Up and Running Static Current Analysis

The following topics are covered in this section:

- [Setting Up and Running Peak Static Current Analysis](#)
- [Setting Up and Running the Average Static Current Analysis](#)

#### Setting Up and Running Peak Static Current Analysis

Peak static current ( $I_{peak}$ ) analysis gives the peak current (saturated current) of every transistor connected to the power nets.

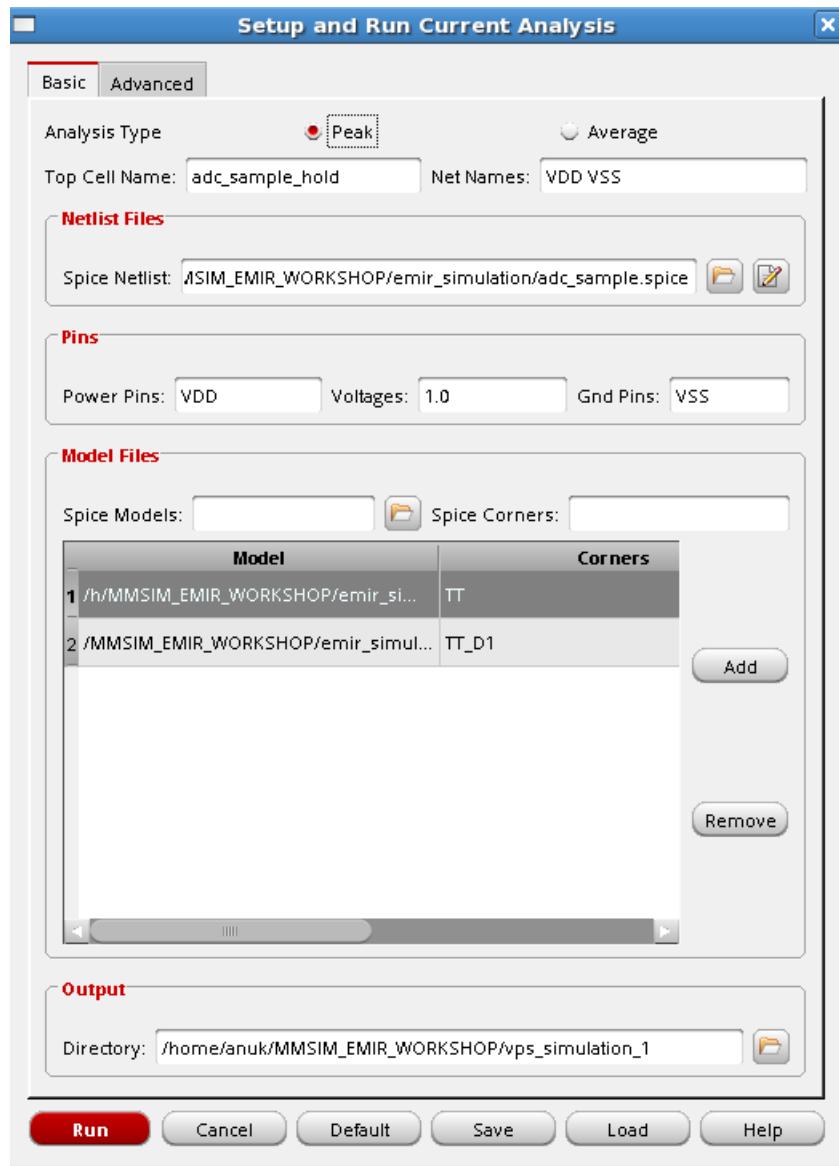
To set up and run the static current analysis mode for peak current analysis, perform the following steps:

- In the Virtuoso® console, choose *Simulation* and then *Setup and Run Simulation*. The Setup and Run Current Analysis form opens. The form has two tabs, *Basic* and *Advanced*. The *Basic* tab is shown below.

# Voltus-Fi Custom Power Integrity Solution User Guide

## Static Current Analysis

**Figure 6-1 Setting Up and Running the Peak Static Current Analysis – Basic Tab**



On the *Basic* tab, provide the following information:

- Select *Peak* as the *Analysis Type*.
  - The *Top Cell Name* is populated by default from the layout.
  - Specify the *Net Names* for which you want to calculate the peak current.
  - In the *Netlist Files* group box, specify the *Spice Netlist* file, which is the DSPF file.
- Note:** The Spice netlist file can also be edited using the edit button provided next to this

# Voltus-Fi Custom Power Integrity Solution User Guide

## Static Current Analysis

---

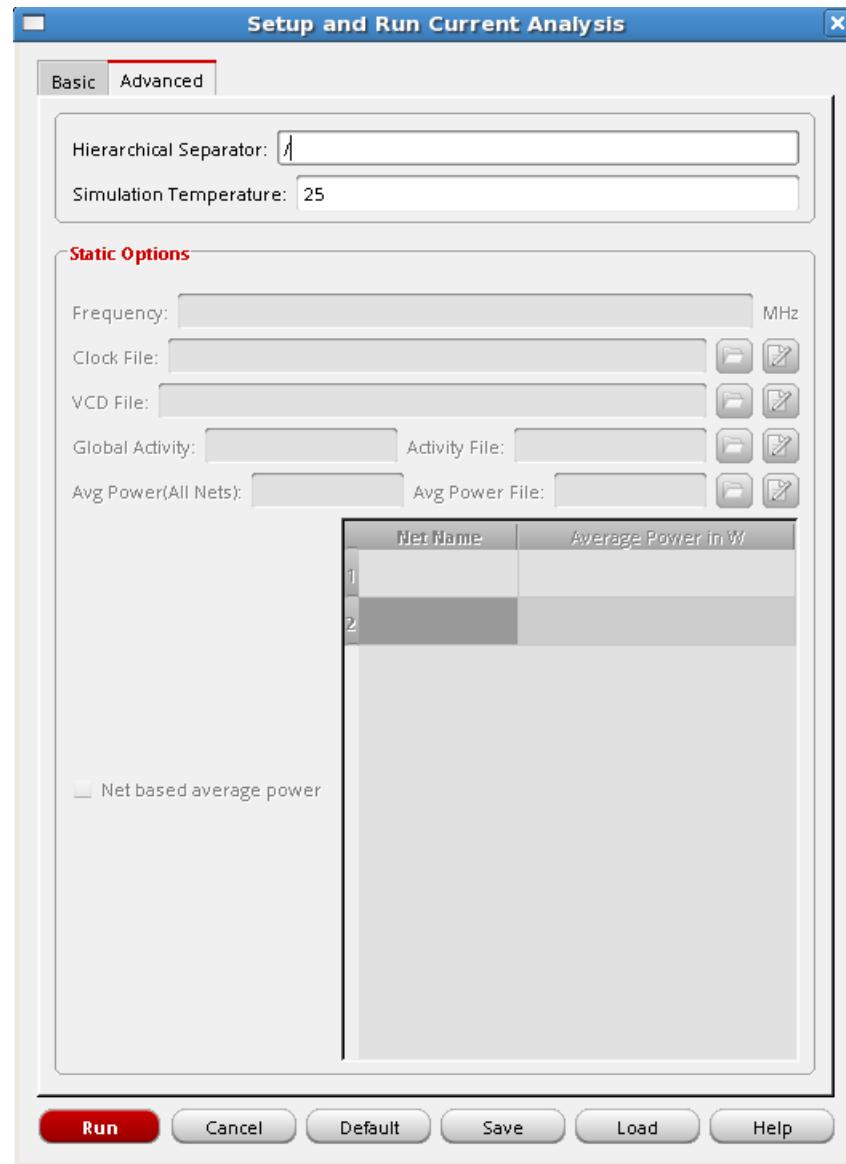
field.

- In the *Pins* group box, specify the *Power Pins* and *Gnd Pins* (ground pins) for which current analysis is to be performed. Specify the corresponding *Voltages* for the power pins.
- In the *Model Files* group box, choose the *Spice Models* and the corresponding *Spice Corners* used by the SPICE netlist. For this, perform the following steps:
  - Specify the path of the SPICE model files.
  - Specify the corresponding *Spice Corners* to be used from this model file.
  - Click *Add* to add the SPICE model – SPICE corner pair to the table.
  - Continue till all the models and their corresponding corners are specified.
  - Click *Remove* to remove a SPICE model – SPICE corner pair from the table.
- In the *Output* group box, specify the *Directory*, which is the output directory in which all the current data will be saved. By default, for every static current analysis run, the tool will save the results in the directory with the naming convention, “vps\_simulation\_#”, where the number is increased incrementally for each run. For example, vps\_simulation\_1, vps\_simulation\_2, and so on.
- Select the *Advanced* tab. This form is shown below.

## Voltus-Fi Custom Power Integrity Solution User Guide

### Static Current Analysis

**Figure 6-2 Setting Up and Running the Peak Static Current Analysis – Advanced Tab**



On the *Advanced* tab, provide the following information:

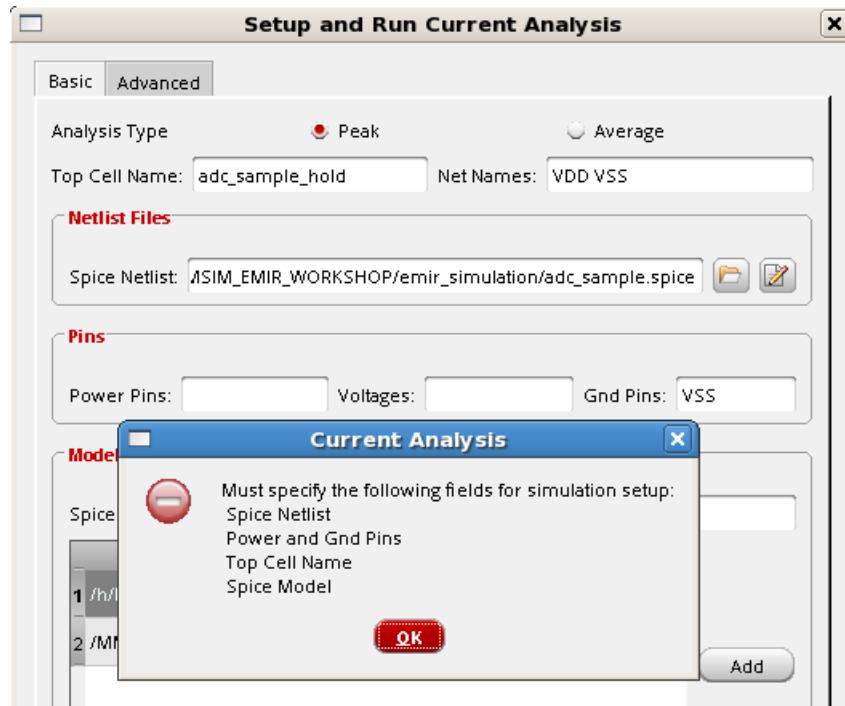
- Specify the *Hierarchical Separator*. This is required only if the netlist used is hierarchical.
- Specify the *Simulation Temperature*, in degree Celsius, at which to perform the simulation.

**Note:** The *Static Options* group box is disabled for the peak current analysis because these options are specified only for the average static current analysis.

# Voltus-Fi Custom Power Integrity Solution User Guide

## Static Current Analysis

- Click *Run*. If any required input is missing in the form, a pop window opens to list all the required inputs. This is shown below.



### TCL Commands:

An example of the set of TCL commands for the peak static current calculation is as follows:

```
setup_simulation\  
  -spice_netlist design.spice\  
  -topcell design_name\  
  -output_directory ./vps_simulation_1  
spice_model\  
  -spice_lib ./models/model1.1\  
  -spice_corner {TT TT_dio}  
spice_model\  
  -spice_lib ./models/model2.1.special\  
  -spice_corner {TT_sr}  
power_pin\  
  -pin_name VDD\  
  -voltage 0.9  
ground_pins VSS  
run_static_simulation\
```

## Voltus-Fi Custom Power Integrity Solution User Guide

### Static Current Analysis

---

```
-method Ipeak\  
-nets {VDD VSS}
```

For details of the above commands, see [Batch Commands for Static Current Analysis](#) in the “Batch Mode Execution” chapter.

### Setting Up and Running the Average Static Current Analysis

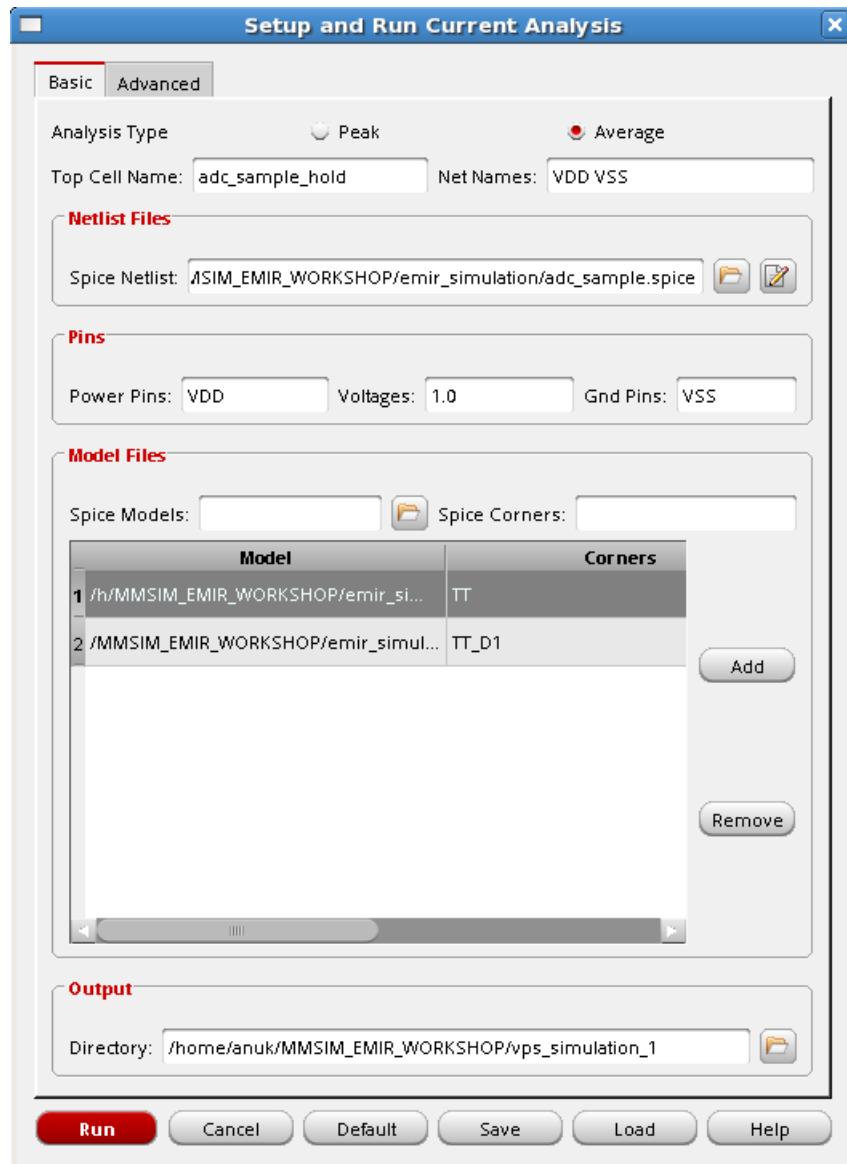
Average static current analysis is a method of analyzing a circuit to obtain the average current flowing through each tap point without using time-based simulation.

For the average (lavg) static current calculation, the information in the *Basic* tab of the Setup and Run Current Analysis form remains the same as that filled in for the peak static current analysis, except for specifying the *Analysis Type*. This is shown below.

# Voltus-Fi Custom Power Integrity Solution User Guide

## Static Current Analysis

**Figure 6-3 Setting Up and Running the Average Current Analysis – Basic Tab**

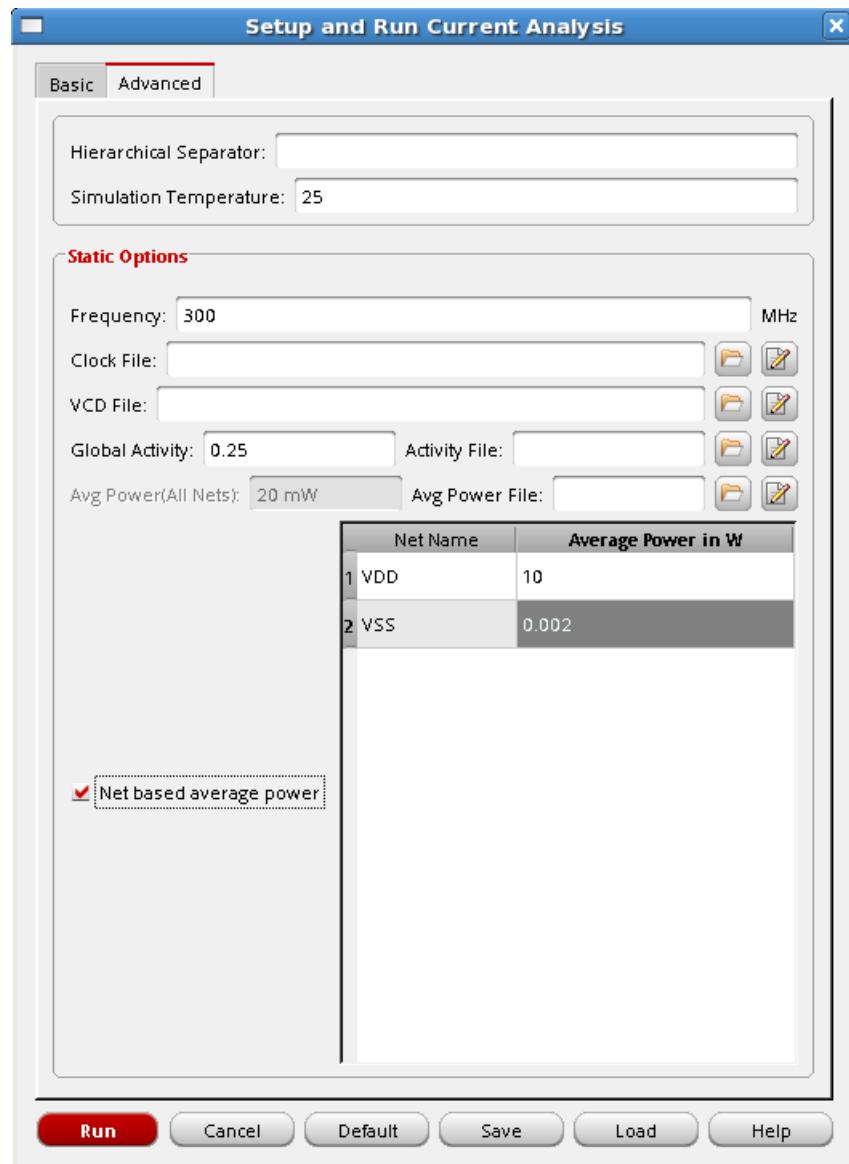


- Choose *Average* as the *Analysis Type*.
- Specify the *Spice Netlist* file, which is the DSPF file.
- Select the *Advanced* tab. This form is shown below.

# Voltus-Fi Custom Power Integrity Solution User Guide

## Static Current Analysis

**Figure 6-4 Setting Up and Running the Average Current Analysis – Advanced Tab**



In the *Static Options* group box, provide the following information:

- Specify the dominant *Frequency* of the design.
- Specify the *Clock File*.
- Specify the *VCD File*. This file contains information about the number of transitions for each net.
- Specify a value for *Global Activity* in the design. This value represents the average number of times all the unset nodes switch in a clock cycle. Alternately, you can specify

the path of the *Activity File*, which contains the data for the activity factor on any node of the circuit.

- Specify the *Avg Power (All Nets)* of the design. This is the average power value for all the nets. The default unit is W (Watts). If the value is in any other unit, you must specify the unit. For example, 20mW. You can also specify the path of the *Avg Power File*, which contains the average power for specific sub circuits.
- Select *Net based average power* to specify different power values for different nets. When you check this option, a list box is enabled. The list box has two columns, “Net Name” and “Average Power in W”. The “Net Name” column is already populated with the names of nets specified on the *Basic* tab. For each net, specify the power value in the “Average Power in W” column.

**Note:** The *Avg Power (All Nets)* and *Net based average power* options are mutually exclusive.

- Click *Run*.

**Note:** You can also create or edit the clock file, VCD file, activity file, and the average power file by using the edit buttons provided against each of these options in the form.

For more information about the formats of above files, see [Static Current Analysis Files](#) in the “File Formats” chapter.

### ***TCL Commands:***

The example TCL command for the average static current analysis is as follows:

```
setup_simulation\
    -spice_netlist adc_sample_hold.spice\
    -topcell adc_sample_hold\
    -output_directory ./vps_simulation_1\
spice_model models/spectre/gpdk090.scs\
    -spice_corner {NN}
power_pin\
    -pin_name VDD\
    -voltage 2.5
ground_pins VSS
run_static_simulation\
    -method Iavg\
    -nets {VDD VSS}\
    -freq 333\
    -global_activity 0.25\
```

-average\_power 20mW

For details of the above commands, see [Batch Commands for Static Current Analysis](#) in the “Batch Mode Execution” chapter.

## Static Current Analysis Report

The output of the static current analysis in Voltus-Fi-XL using Thunder is a text report that is saved in the output directory. The report is used by Spectre to run a simulation and create the simulation result file, which is then used for EMIR analysis and visualization in Voltus-Fi-XL. The default text report generated for both peak and average current analysis is `static_current.txt`.

### Format:

Node Name                    Static Current

### Format Description:

Node Name specifies the name of the node for which the static current is being reported.

static current specifies the value of the static current on the specified node.

### Sample Report

X0.AbcI2/avI33/avD30_32	9.83e-05
X0.AbcI2/avI30/avD30_35	9.83e-05
X0.AbcI86/avD27_135	1.96e-05
X0.AbcI86/avD27_134	1.96e-05
X0.AbcI86/avD27_133	1.96e-05
X0.AbcI86/avD27_122	1.96e-05
X0.AbcI86/avD27_121	1.96e-05
X0.AbcI86/avD27_120	1.96e-05

# **Voltus-Fi Custom Power Integrity Solution User Guide**

## Static Current Analysis

---

---

# IR Drop Analysis Results

---

- [Overview](#) on page 117
- [Signal Net IR Drop Analysis](#) on page 117
- [Power Gate Support](#) on page 118
- [Static EMIR Analysis](#) on page 119
- [Static Power Grid Solver](#) on page 120
- [Solid Shape Highlighting](#) on page 122
- [Displaying Finer Color Gradient for Selected Layers in IR and EM Plots](#) on page 123
- [Specifying Multiple Simulation Result Files for EMIR Analysis](#) on page 124
- [Viewing the IR Drop Analysis Results](#) on page 125
- [Displaying and Querying EMIR Results](#) on page 135
  - [Viewing EMIR Violations in the Annotation Browser](#) on page 135
  - [Querying EMIR Results](#) on page 139
  - [Specifying the Display Options for IR/EM Plots](#) on page 139
- [Types of IR Drop Analysis Plots](#) on page 155
  - [IR, IRAVG – IR Drop Plots](#) on page 156
  - [RC, RCAVG, and RCRMS – Resistor Current Plots](#) on page 156
  - [IV – Transistor Voltage Plot](#) on page 159
  - [PI – Powergate Current Plot](#) on page 160
  - [PV – Powergate Voltage Plot](#) on page 163
- [Plotting the Least Resistive Path](#) on page 166
  - [The LRP Browser](#) on page 168

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

---

- ❑ [Shorting Layers for LRP Analysis](#) on page 173
- ❑ [Displaying LRP for the Tap on the Selected Marker](#) on page 174
- ❑ [Supporting LRP Analysis in Power-Gated Designs](#) on page 175
- [Calculating Effective Resistance between any Two Nodes on a Net](#) on page 178
- [Generating EMIR Analysis Reports in Voltus-Fi-XL](#) on page 182
  - ❑ [Reports Generated using the EMIR Control File](#) on page 182
  - ❑ [Default Reports Generated for EMIR Analysis](#) on page 183
  - ❑ [Reports Generated for Multiple Bin Files for Different Nets from the Same Testbench](#) on page 189
  - ❑ [Reports Generated for Multiple Bin Files from Different Testbenches in a Design](#) on page 189

## Overview

Voltus-Fi-XL uses the simulation database generated by Spectre® APS/XPS simulators and displays the results of the IR drop analysis on the Virtuoso layout.

It also generates text reports of the analyses and lets you query the analyses results to view specific violations on the layout. This is used to debug the high IR drop regions in the design.

Batch mode support is provided for loading IR drop analysis results and generating text reports. For details of the batch commands used to load and print IR drop result reports, see [Batch Commands for IR Reports](#) in the “Batch Mode Execution” chapter.

## Signal Net IR Drop Analysis

By default, IR drop analysis can only be applied to power nets, which are nets driven by DC or constant voltage sources. The advanced feature, signal net IR drop analysis, lets you perform IR drop analysis on signal nets, which are nets that are not connected to any DC or constant voltage source.

In the signal net IR drop analysis, the maximum or average IR drop is reported using the `analysis=[sigvmax sigvavg]` statement in the EMIR control file (`emir.config`). The IR drop is reported as the difference between the reference voltage and the voltage of the sub node (`vref-vnode`). The reference voltage for calculating the IR drop value at each sub node and time point is defined as the average voltage of all sub nodes, or the maximum voltage of all sub nodes (`reftype=[avg max]`).

The following is an example of the EMIR control file (`emir.config`):

```
net name=[X1.*] analysis=[sigvmax] reftype=[max]
solver method=direct
```

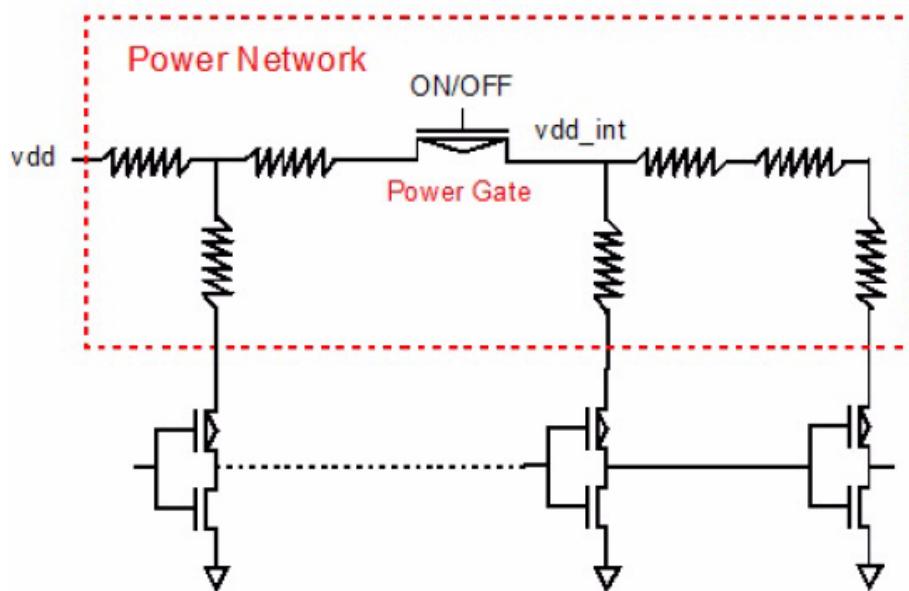
In the above example, the signal net IR drop analysis is performed for all nets `X1.*`. The maximum IR drop for the sub nodes of each net is reported. For the IR drop analysis at each net and time point, the sub node with the highest voltage is picked as the reference voltage. When using `reftype=[max]` for a net with three sub nodes: `net1_1` (1V), `net1_2` (2V), and `net1_3` (3V), the detected reference voltage will be 3V, and the IR drop calculated will be 2V for `net1_1`, 1V for `net1_2`, and 0V for `net1_3`. When using `reftype=[avg]` the reference voltage will be 2V, and the IR drop calculated will be 1V for `net1_1`, 0V for `net1_2`, and -1V for `net1_3`.

For details of the supported EMIR control file options that can be specified, see [EMIR Control File Options Supported in Voltus-Fi-XL](#) in the “Data Preparation” chapter.

## Power Gate Support

Power networks may contain power gates which enable or disable the power supply in the circuit. These power gates split the power supply RC network into two parts. The RC network driving the power gate, and the RC network being driven by the power gate. This is shown in the figure below. The MMSIM EMIR analysis can handle such power gates.

**Figure 7-1 Power Gates**



To invoke the power gating handling, add the power gate setting in the EMIR control file (`emir.config`), as shown below.

```
net pwrgate=[vdd vdd_int] analysis=[vavg]
```

Both, the power supply net (`vdd`) driving the power gates, and the internal power supply net (`vdd_int`) driven by the power gates, need to be specified in the net statement in any order. Next, the required analyses need to be defined, as shown above. The IR drop report includes both the power supply net in the usual report file name, for example, `input.rpt_ir` and the internal power supply net is reported in a file, such as `input.rpt_pwg` or `input.emirtap.rpt_pwg` for direct and iterated methods respectively. As for the EM analyses, they are performed as usual, if specified.

In addition, an important parameter `Ton` is also be reported in the `.rpt_pwg` file. It reports the time taken to power up the terminal of the internal power supply net to 95% of the power supply level (VDD).

**Note:** The parameter `Ton` can be infinity if the internal power supply level does not reach 95% of VDD.

The subsequent sections cover the various options available in Voltus-Fi-XL for viewing the IR drop analysis results on the Virtuoso layout.

## Static EMIR Analysis

A static EMIR analysis evaluates IR drop and EM currents based on user-provided subcircuit instance current consumptions without running a transient or DC simulation. The user-provided currents are distributed to the tap devices based on the W/L ratio of the devices in the design. The IR drop and EM current analysis is performed based on the value of current at each tap device.

The static EMIR analysis is enabled with the “`static ifile`” statement in the EMIR configuration file.

```
net name=[I1.VDD I1.VSS] analysis=[vavg iavg]
static ifile="static_currents.txt"
```

Since the analysis is static, the IR drop `vmax` and `vavg`, and the EM current `imax`, `irms`, and `iavg` values will be the same, and just one of these can be selected in the statement.

The subckt instance currents are defined (in A) in the “`static_currents.txt`” file with the subckt instance name, the subckt port name, and the value of current flowing in the port.

```
I1 VDD 0.001
I1.I2 VDD 0.005
I1.I2/I3 VDD 0.00001
I1.I2/I3/I4 VDD 0.000005
I1 VSS ?0.005
```

The accuracy of the static EMIR analysis depends greatly on the detailed current information in the `static_ifile`. It is highly recommended that you provide the current consumption for each subckt instance.

The results of the static EMIR analysis are written into the same IR drop and EM current text reports, and into the same binary EM database as the dynamic EMIR analysis. Static EMIR analysis cannot be combined with the dynamic analysis. You can perform either a static or a dynamic EMIR analysis at one time. When running the static EMIR analysis all other Spectre analyses like DC, TRAN and so on will be ignored.

The static EMIR analysis can also be applied to designs with power gates. In that case the `pwrgate` statement needs to be added to the EMIR configuration file as follows:

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

---

```
net name=[I1.VDD] analysis=[vavg iavg]
static ifile="static_currents.txt"
net pwrgate=[I1.VDD I1.VDD_INT] analysis=[vavg iavg]
```

and the power gate current is required to be added to the “static\_currents.txt” file as follows:

```
I1 VDD 0.001
I1 VDD_INT 0.0005
```

**Note:** To support users setting up the static EMIR flow, the static EMIR current file is automatically generated when running a dynamic EMIR analysis (file extension: static).

## Static Power Grid Solver

The static power grid solver (SPGS) feature of Spectre can be used in Voltus-Fi-XL to calculate all pin-to-tap resistances based on the description of a DSPF file and the options set in the EMIR configuration file. The resistances calculated by SPGS are electrically equivalent resistances, and not the summation of resistors. The calculation assumes that all pins are connected together to form a global pin. After calculation, an ordered list of resistances between the global pin and all taps is generated.

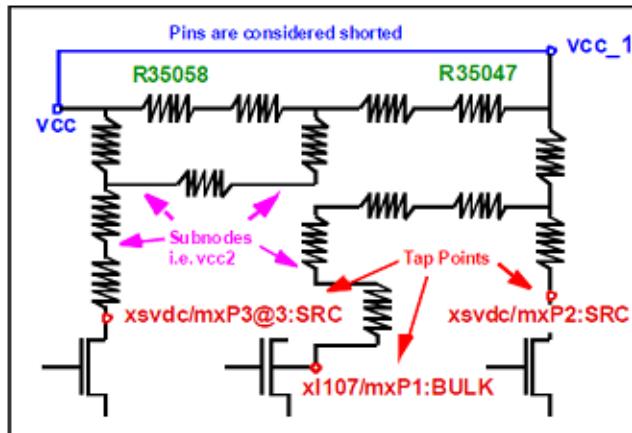
This feature can also be used to calculate the pin to pin resistance between any two nodes of the same net. For details, see [Calculating Effective Resistance between any Two Nodes on a Net](#).

The diagram below shows the definition of pin, sub nodes, and tap nodes in the DSPF file.

**Figure 7-2 Pin, Sub Node, and Tap Node Information in the DSPF File**

# Voltus-Fi Custom Power Integrity Solution User Guide

## IR Drop Analysis Results



```
*|NET vcc 0PF
*|P (vcc_1 B 0 -1.441 419.479)          pll.spf
*|P (vcc B 0 2158.41 419.491)
*|| (xl107/mxP1:BULK xl107/mxP1 BULK B 0 2128.46 392.112)
*|| (xsvdc/mxP3@3:SRC xsvdc/mxP3@3 SRC B 0 1837.78 431.24)
*|| (xsvdc/mxP2:SRC xsvdc/mxP2 SRC B 0 1837.78 429.16)
*|S (vcc:2 40.43 193.665)
...
R35047 vcc_1 vcc_7 0.001 $I=0.004 $w=10 $lvl=195
...
```

The static EMIR analysis is enabled with the “spgs net=...” statement in the EMIR configuration file.

```
spgs net=[i1.vcc i1.vss]
spgs rshort=1e?]12
spgs pwrgate = [i1.vcc i1.outp]
```

The net statement defines the power nets to be analyzed. Resistors in the power net can be optionally shorted with the rshort options. If the power net contains a power gate, then the pwrgate option needs to be defined in the configuration file.

The SPGS flow requires the DSPF file to be included in the Spectre input file with the dspf\_include statement.

```
dspf_include "pll.spf"
```

The power nets need to be connected to voltage sources, and device models need to be defined for the devices in the instance section in the DSPF file. For running the SPGS feature, Spectre is run as in the regular EMIR flow.

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

---

```
spectre +aps input.scs +emir=spgs.conf
```

The SPGS report provides a list of the pin to tap node resistors ordered highest to lowest. This is shown below.

Count	R	W/L	(x:y)	(x:y ) in GDSII	name
#1	1920.2579	5.00	(1440:366)	(1440:366)	xI107/mxP1:BULK
#2	1577.1482	10.00	(1440:476)	(1440:476)	xsvdc/mxP3@3
#3	1353.6123	4.00	(1446:509)	(1446:509)	xsvdc/mxP2:SRC

It is recommended to run the SPGS feature separately and not together with any other EMIR analysis. SPGS also supports the conversion between pin, tap node, and sub node. The following statements can be specified in the EMIR configuration file:

```
convert tap node MPM3@44:s of net I1.vdd to a pin, and sub node VDD:5 to a tap node.  
spgs tap2pin net=[i1.vdd] include=[MPM3@44:s]  
spgs sub2tap net=[i1.vdd] include=[VDD:5]
```

All conversion options are described in the [EMIR Control File Options Supported in Voltus-Fi-XL](#) table in the “Data Preparation” chapter.

## Solid Shape Highlighting

The rail analysis plots that are displayed on the Virtuoso Layout can be viewed as either stick diagrams or solid shape highlights. In the solid shape display, the plots highlight the full shape of the resistors, while in the stick diagram display, the resistors are connected with thin lines over the layout shapes.

The solid shape display is more useful for designers because it allows them to see the shapes that are failing the analysis. There are two flows for solid shape display. You can either specify the DFII layermap file or the QRC run name for viewing solid shape display for plots. When you specify the QRC run name, the plots are displayed based on the shape database generated by QRC.

For details of how to specify the two options in the GUI, see [Viewing the IR Drop Analysis Results](#).

## QRC Shape Database-Based Flow

To display plots using the shape database generated by QRC, you must ensure the following:

- The QRC command file or the CCL file should include the following command:

```
extraction_setup -analysis em
```

For details, see the `extraction_setup` command in the “QRC Command Files” chapter of the *Quantus QRC Extraction Users Manual*.

- QRC should be set in the path and its version should be compatible with the Voltus-Fi-XL version being used.

## Displaying Finer Color Gradient for Selected Layers in IR and EM Plots

In the DFII layermap flow, for solid shape display of IR/EM violations on the Virtuoso layout, Voltus-Fi-XL locates all the nodes lying on a shape and then colors the shape with the worst violation value among all resistors connected to these nodes.

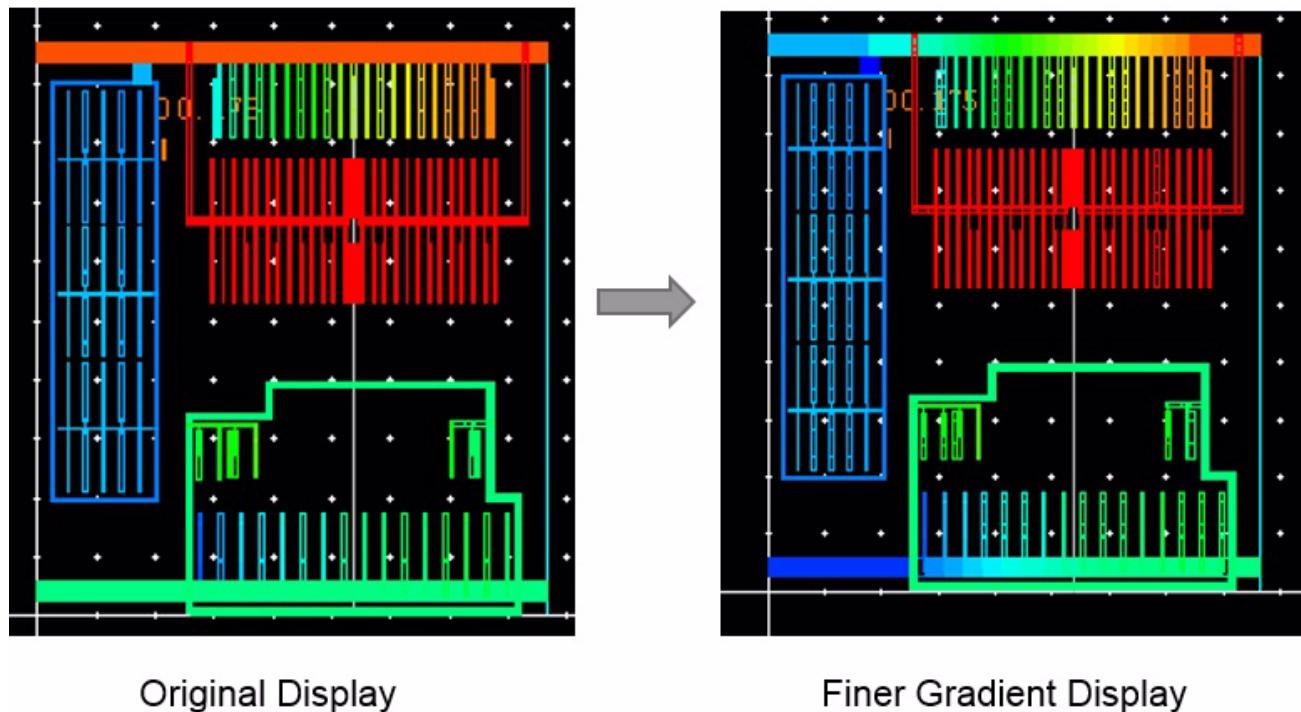
However, sometimes resistors do not align well with the shapes used to display the violations. This may be because the shapes are large and have multiple nodes and resistors. For a finer display of the color gradient for violations in different segments of such layers, you can select layers, both in the IR and EM tabs of the IR/EM Results form. For the selected layers, the software will split the shape polygons into subpolygons and then it will color the subpolygons with the worst violation value for all resistors connected to the nodes in the split shape.

For this, specify the *DFII Layermap* file and then click the *Finer Gradient* button provided on the IR or EM tabs of the IR/EM Results form. In the *Finer Gradient Layer Selection* pop-up, select the PGDB layers for which you want to view finer gradient. For details of the steps involved, see [Viewing the IR Drop Analysis Results](#).

**Note:** Perform this step before loading the IR or EM Results. After loading the results, when you click *Show Plot*, you can see the finer gradient for the selected layers.

A comparison of finer gradient display versus the original display is shown in below figure.

**Figure 7-3 Viewing the Finer Gradient Display**



## Specifying Multiple Simulation Result Files for EMIR Analysis

While performing EMIR analysis in Voltus-Fi-XL, multiple bin files can be specified, both in the GUI mode as well as in the batch mode.

In the GUI, a + button is provided for the *State Directory/Results File* field on both the IR and EM tabs of the IR/EM Results form. Use this to add multiple bin files for EMIR analysis. For details, see [Viewing the IR Drop Analysis Results](#).

In the batch mode, different options of the `vfibatch` command are used to specify multiple bin files in different scenarios. The following use models are supported:

- **Specifying multiple bin files that contain simulation results for different nets from the same testbench.** This is useful in large designs, where simulation is run separately for different nets of the same testbench to improve the runtime. In such cases, you can create `emir_bin` for different nets in different bin files. In the batch mode, the `-db` option of the `vfibatch` command is used to generate reports in this scenario.

Consider an example where separate bin files are created for nets, VDD and VSS. The emir.conf files for the two nets will be as follows:

```
emir.conf.vdd net name=[X0.VDD] analysis=[iavg imax sigvmax vmax] emirutil  
techfile = qrcTechfile layermapfile = contactmapfile solver method=[iterated]  
emir.conf.vss net name=[X0.VSS] analysis=[iavg imax sigvmax vmax] emirutil  
techfile = qrcTechfile layermapfile = contactmapfile solver method=[iterated]
```

For details on how the EMIR analysis reports are generated for the above scenario in the batch mode, see [Reports Generated for Multiple Bin Files for Different Nets from the Same Testbench](#).

- **Specifying multiple bin files that contain simulation results from different testbenches for the same design.** You can specify multiple bin files that are located either in the same directory or in different directories. In the batch mode, the -db\_path and -db\_multi\_path options of the vfibatch command are used to generate reports in these scenarios, respectively. For details on how the EMIR analysis reports are generated for these scenarios in the batch mode, see [Reports Generated for Multiple Bin Files from Different Testbenches in a Design](#).

## Viewing the IR Drop Analysis Results

To plot EMIR results in Voltus-Fi-XL, ensure the following:

- The emir mode is enabled while performing simulation.
- The simulation result file or bin file is available in the output directory of the simulation results. The naming convention of the simulation result file is, \*.emir#\_bin. For example, your result file could be named, xps.emir0\_bin.

Follow these steps to view the IR drop analysis results.

- In the High Capacity Power IR/EM console, choose *IR/EM Analysis – Rail Analysis Results*. The IR/EM Results form opens. This form is shown below.

The IR/EM Results form has the following tabs: *IR*, *EM*, *Structural Analysis*, *LRP Browser*, *Pin2PinR*, and *What-If*.

Click the *IR* tab to plot the IR drop analysis results.

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

**Figure 7-4 IR/EM Results Form – IR Tab**



On this tab, perform the following steps:

- In the *Results* group box, specify the *State Directory / Results File* that stores the results of the simulation. The naming convention of the result file is \*.emir#\_bin.
- Click the + button to specify multiple bin files or simulation result files. For details, see [Specifying Multiple Simulation Result Files for EMIR Analysis](#).
- Specify the *Shrink factor*, if any, by which the xDSPF was shrunk. This is used for flows where the xDSPF is generated with a shrunk technology but the layout remains on the

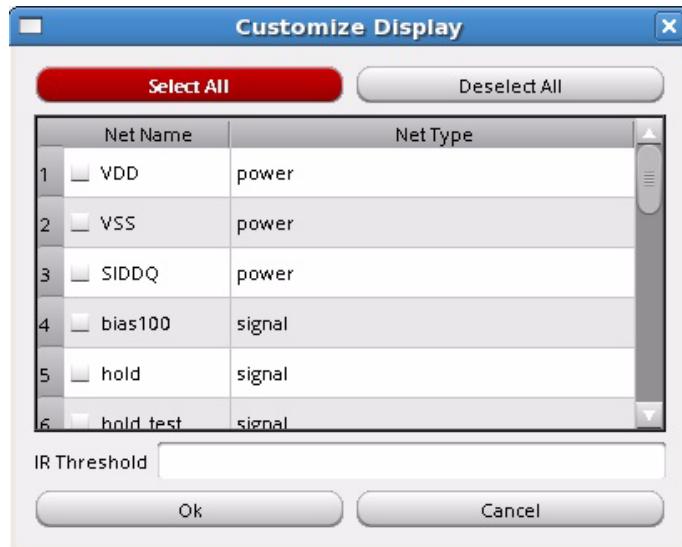
## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

original technology. For example, if the original xDSPF was shrunk by 80%, the shrink factor should be specified as 0.8. The shrink factor value is less than 1.

- Click *Customize Display* to select the nets for which you want to view the display. The Customize Display pop-up window opens. This helps save time as results are loaded only for the selected nets and not for all nets. You can also specify the *IR Threshold* value in this window. This is shown below.

**Figure 7-5 Customize Display Form**



The *Customize Display* button is enabled only before the results are loaded. It is disabled once the results are loaded.

- Specify either the *QRC Run* or the *DFII Layermap*. When either of these options is specified, the plots displayed on the layout show solid shape highlighting. For more information, see [Solid Shape Highlighting](#).

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

- Select *QRC Run*. The QRC Run Location form opens. In this form, specify the QRC run details, which include the path to the QRC *Run Directory* and the QRC *Run Name*.



The display now generated will be based on the shape database generated by QRC. For more information about the setup requirements for the QRC Run flow, see [Solid Shape Highlighting](#).

- Select the DFII layermap file, which is the APS/XPS-to-DFII layer map file. You can create a new layer map file or edit an existing file, using the edit button provided next to the field. This layer map file is the same file that is used in structural analysis. For more information about the file format and description, see [DFII Layer Map File](#) in the “File Formats” chapter.

You can also specify this file by using the `vfiDfiiLayerMapFile` environment variable either in the `.cdsenv` or the `.cdsinit` file. For more information, see [Loading the DFII Layer Map File by Default](#) in the “Data Preparation” chapter.

The following considerations apply to the DFII layermap flow:

- When the DFII layermap file is specified, the visibility of the DFII layers on the layout is in sync with the layers selected in the *Layers* group box or the layer selection window of the Display form.
- When the correct mapping in the DFII layermap file is either not specified or if some layers are missing in this file then the resistors or nodes of those layers will not be highlighted on the layout because there will be no shapes attached to them.

## Voltus-Fi Custom Power Integrity Solution User Guide

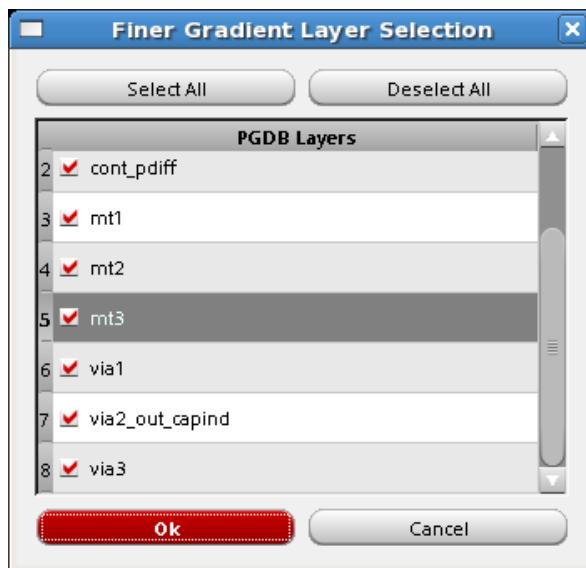
### IR Drop Analysis Results

- When this file is not specified, the pop-up-window shown below opens. If you click Yes, the plots are displayed using stick diagrams.



- For the DFII layermap flow, click *Finer Gradient* to view the finer gradient for the selected layers. When you click this button, the *Finer Gradient Layer Selection* pop-up window opens. This is shown in below figure.

**Figure 7-6 Selecting Layers for Viewing Finer Gradient**



In this window, select the PGDB layers and click *OK*. For details, see [Displaying Finer Color Gradient for Selected Layers in IR and EM Plots](#).

- Click *Load Results* to load the results. You can click *Clear Results* to clear the results.
- Once the results are loaded, the *Rail Analysis* cyclic field in the *Plot Type* group box becomes enabled. From the drop-down list, select the analysis type for which you want to display the results. The following options are available:
  - **IR – IR Drop:** analyzes and reports peak voltage drop
  - **IRAVG – IR Avg Drop:** analyzes and reports average voltage drop
  - **RC – Resistor Current:** analyzes and reports peak resistor currents

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

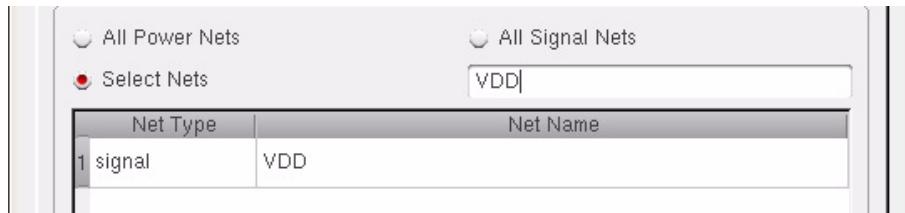
- ❑ **IV – Transistor Voltage:** analyzes and reports transistor-based supply voltage data
- ❑ **PI – Powergate current:** analyzes and displays the currents through the resistors when power gates are connected
- ❑ **PV – Powergate voltage:** analyzes and displays the IR drop across power-switch instances. This plot can be used to analyze and debug regions of high IR drop inside the power-gated block of the design
- ❑ **RCAVG – Average Resistor Current:** analyzes and reports average resistor currents
- ❑ **RCRMS – RMS Resistor Current:** analyzes and reports RMS resistor currents
- ❑ **REffective – Effective Resistance:** analyzes and displays the pin to node resistances

**Note:** For details of the above plots, see “[Types of IR Drop Analysis Plots](#)”.

- Click *All Power Nets* to view the plots for all power nets. An example of an IR plot for all signal nets is shown in [Figure 7-3](#) on page 133.
- Click *All Signal Nets* to view the plots for all signal nets.
- Click *Select Nets* to specify the nets for which you want to view IR plots. You can select one or more nets from the list of nets provided in the list box. The list box lists *Net Type* and the corresponding *Net Name*.

You can search for specific nets for which you want to plot results by typing the net name in the text field provided in the form. The list box is updated to display information about the specified net.

**Note:** The net name is case-sensitive. This is shown in the image below.



An example of an IR plot for selected nets is shown in [Figure 7-3](#) on page 134.

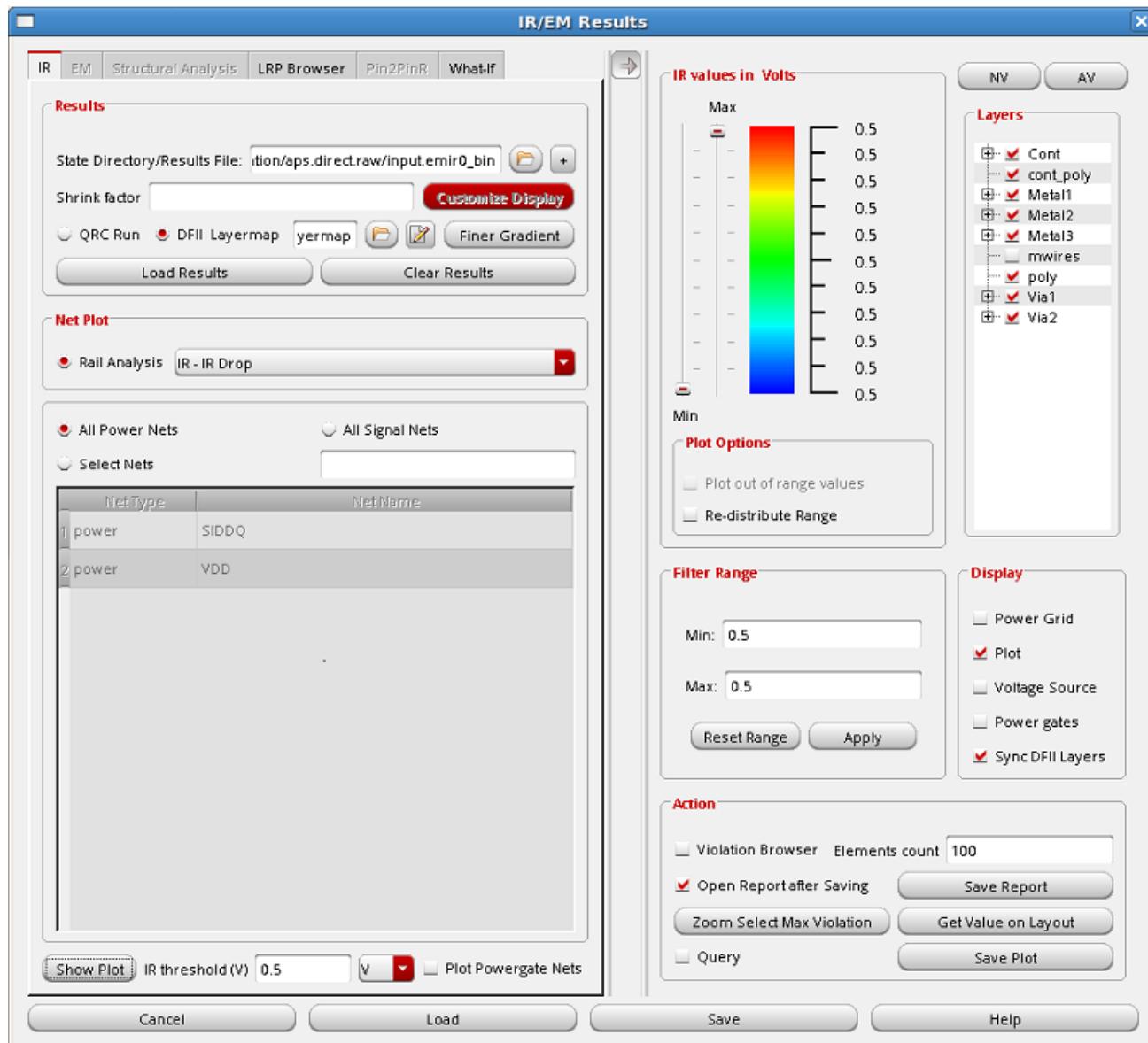
- Click *Show Plot* to view the plot on the Virtuoso layout. When you click *Show Plot*, the IR/EM Results form expands to show the available plot display options. You can use these options to customize the displayed plots. These options are explained in detail in the [Specifying the Display Options for IR/EM Plots](#) section.

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

- Specify *IR threshold* for viewing IR drop violations above the specified threshold value. Select V or mV to specify the threshold voltage in volt and millivolt, respectively. When this value is specified, the *Min - Max* slider range is updated to reflect the specified minimum violation value and the plot is updated on the layout. For example, in the image below, the IR threshold value is specified as 0.25. So, the *Min* value of the slider range is updated to 0.25.

**Figure 7-7 Specifying the Threshold Value for Viewing IR Drop Violations**



- The *Plot Powergate Nets* option is used for plotting the IR drop analysis results of power-gated designs. In this case, when you click the *Select Nets* option and select a

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

---

net from the list box, all the switched nets for the selected always-ON net also get selected. The software plots results for all the switched nets together. This option is turned off by default.

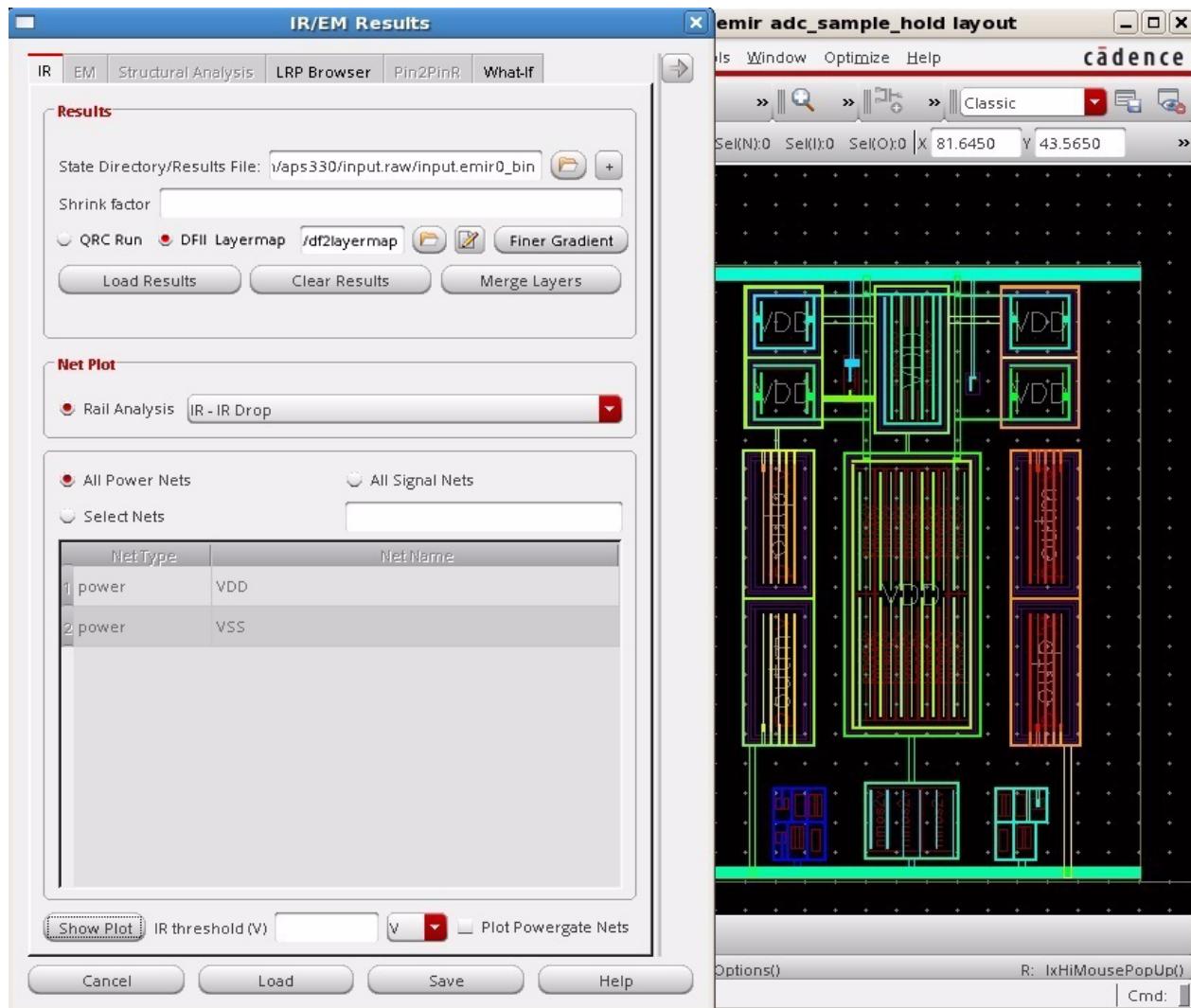
- Click *Save* to save the specified settings in a configuration file.
- Click *Load* to automatically load settings saved in a previous run in the IR/EM Results form. For this, you can save the settings in a `.vpsconfig` file. When you click *Load*, the software searches for the `.vpsconfig` file. If the file is available, the settings specified in this file are restored in the form. A sample `.vpsconfig` file is shown in [Example 7-1](#) on page 134.

You can also load the EMIR control file or `emir.config` file using this button. When this file is specified, the technology file and layermap file information provided in this file is used for EMIR analysis. For more information about the EMIR control file, see [Simulation Requirements and Setup](#) in the “Data Preparation” chapter.

# Voltus-Fi Custom Power Integrity Solution User Guide

## IR Drop Analysis Results

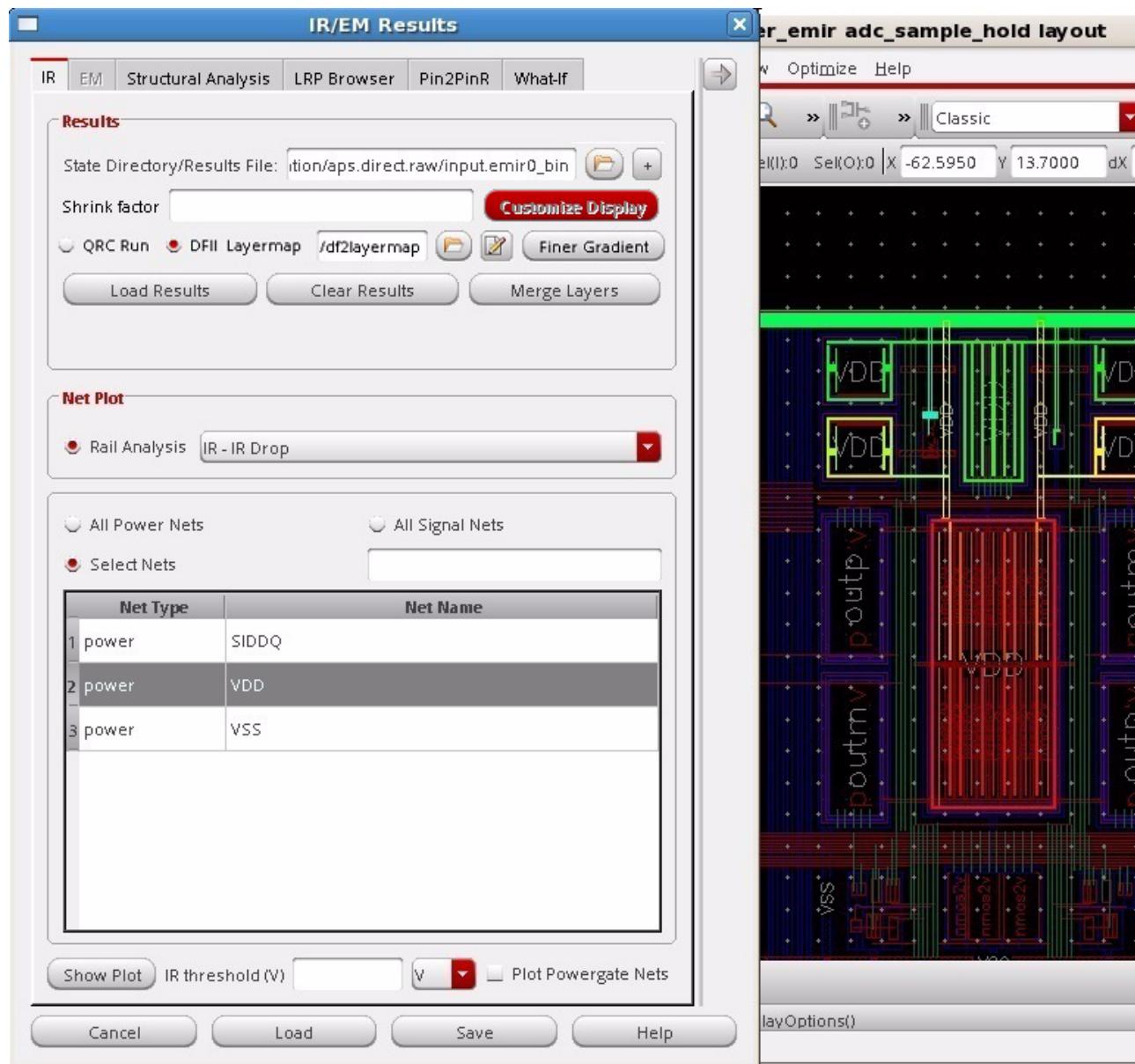
**Figure 7-8 Viewing the IR Plot for All Power Nets**



# Voltus-Fi Custom Power Integrity Solution User Guide

## IR Drop Analysis Results

**Figure 7-9 Viewing IR Plot for Selected Nets**



### Example 7-1 Sample Configuration File

A sample .vpsconfig file is shown below.

```
setvar vps_results_IR  
{/icd/epsrd_t1nb_002/babita/xps/demo/design/raw.078/xps.emirtap.emir0_bin}  
setvar vps_results_EM  
{/icd/epsrd_t1nb_002/babita/xps/demo/design/raw.078/xps.emirtap.emir0_bin}
```

```
setvar vps_results_IR_Flow {mmsim}
setvar vps_results_IR_shrink_factor {}
setvar vps_results_EM_shrink_factor {}
setvar vps_results_EM_tech_file
{/icd/epsrd_t1nb_002/babita/xps/demo/qrcTechFile}
setvar vps_results_EM_layermap_file
{/icd/epsrd_t1nb_002/babita/xps/demo/contactmapfile}
setvar vps_results_EM_peak_dc true
setvar vps_results_EM_avg_dc true
setvar vps_results_EM_abs_avg_dc false
setvar vps_results_EM_peak_ac false
setvar vps_results_EM_rms_ac false
```

## Displaying and Querying EMIR Results

Voltus-Fi-XL lets you specify the plot display settings for the IR drop and EM plots that are displayed on the Virtuoso layout.

The plot display options include a *Min - Max* slider that lets you customize the range of violations you want to view. A continuous RGB gradient is used to highlight the worst violation regions in the design. The vast range of color options provided by the RGB gradient makes it easier to view a range of violations on the layout. The available plot display options of the IR/EM Results form are shown in [Figure 7-12](#) on page 140.

The details of specifying the display options in this form are provided in [Specifying the Display Options for IR/EM Plots](#) section.

In addition to specifying settings for plots displayed on the layout, there are options to query the layout for viewing specific violations. The results of the query are highlighted in the EM/IR tab of the Annotation Browser.

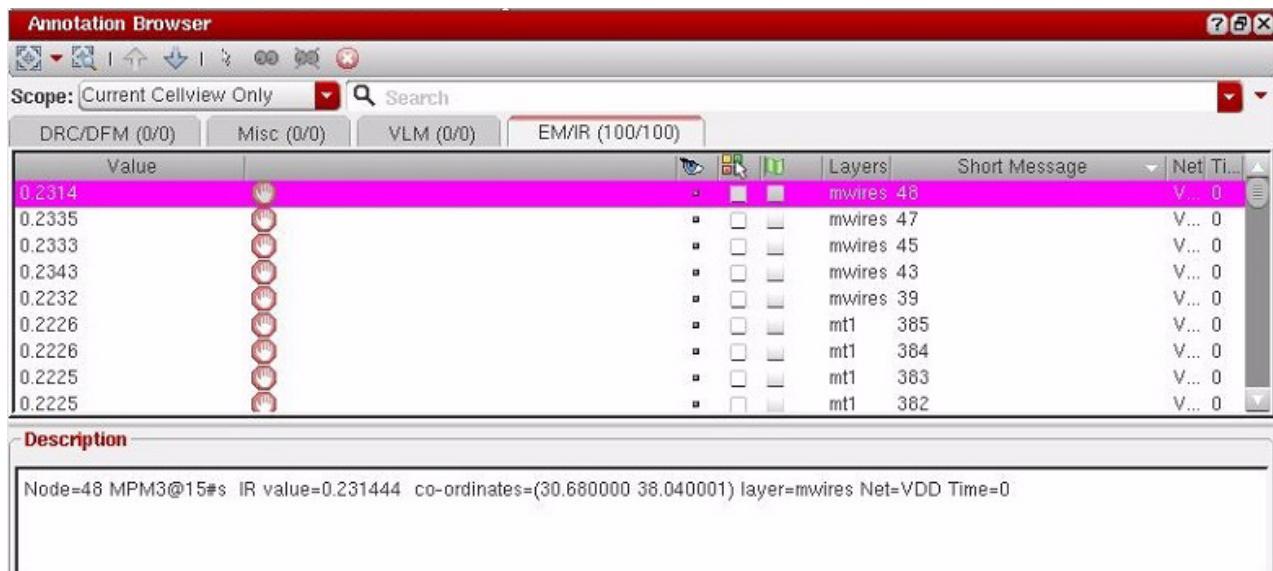
## Viewing EMIR Violations in the Annotation Browser

The Annotation Browser lets you view and manage violation markers generated for the design. When you click *Violation Browser* in the IR/EM Results form, the Annotation Browser window appears in the right corner of the Virtuoso layout window and the EM/IR tab is open by default. This is shown below.

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

**Figure 7-10 EM/IR Tab of the Annotation Browser**



The EM/IR tab is used for viewing the violations of IR drop and EM analyses and it is enabled only when Voltus-Fi-XL is launched. The number of violations displayed in the EM/IR tab is based on the number specified in the *Elements count* field in the IR/EM Results form.

#### Information Displayed in the EM/IR Tab of the Annotation Browser

The following information is available in columns in the EM/IR tab of the Annotation Browser. The columns that are displayed vary for different analyses types. The complete list of available columns and the information provided in them is as follows:

- Value: provides the violation value
- Short Message: provides the name of the node or the resistor
- Layers: provides the name of the layer on which the violation is located
- Density: provides the current density value across the resistor
- Current: provides the current value across the resistor
- Current Limit: provides the current rules as per the specified EM rules
- Length: provides the resistor length value
- Width: provides the resistor width value
- Resistance: provides the resistance value of the resistor

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

---

- Number of Vias: provides the number of vias in the via array. This is valid only for via layers
- Needed No. of Vias: provides the number of vias needed to pass the EM violation. This is valid only for via layers.
- Via Area: provides the area of the via array
- Needed Width: provides the minimum width required for metal layer resistors to pass the EM violation
- Net: provides the name of the net
- Power Gate Transistor: provides the name of the power-gate transistor instance. This is valid only for power-gated resistors

By default, the columns displaying the resistor/node name, layers, and violation values are visible for all types of plots.

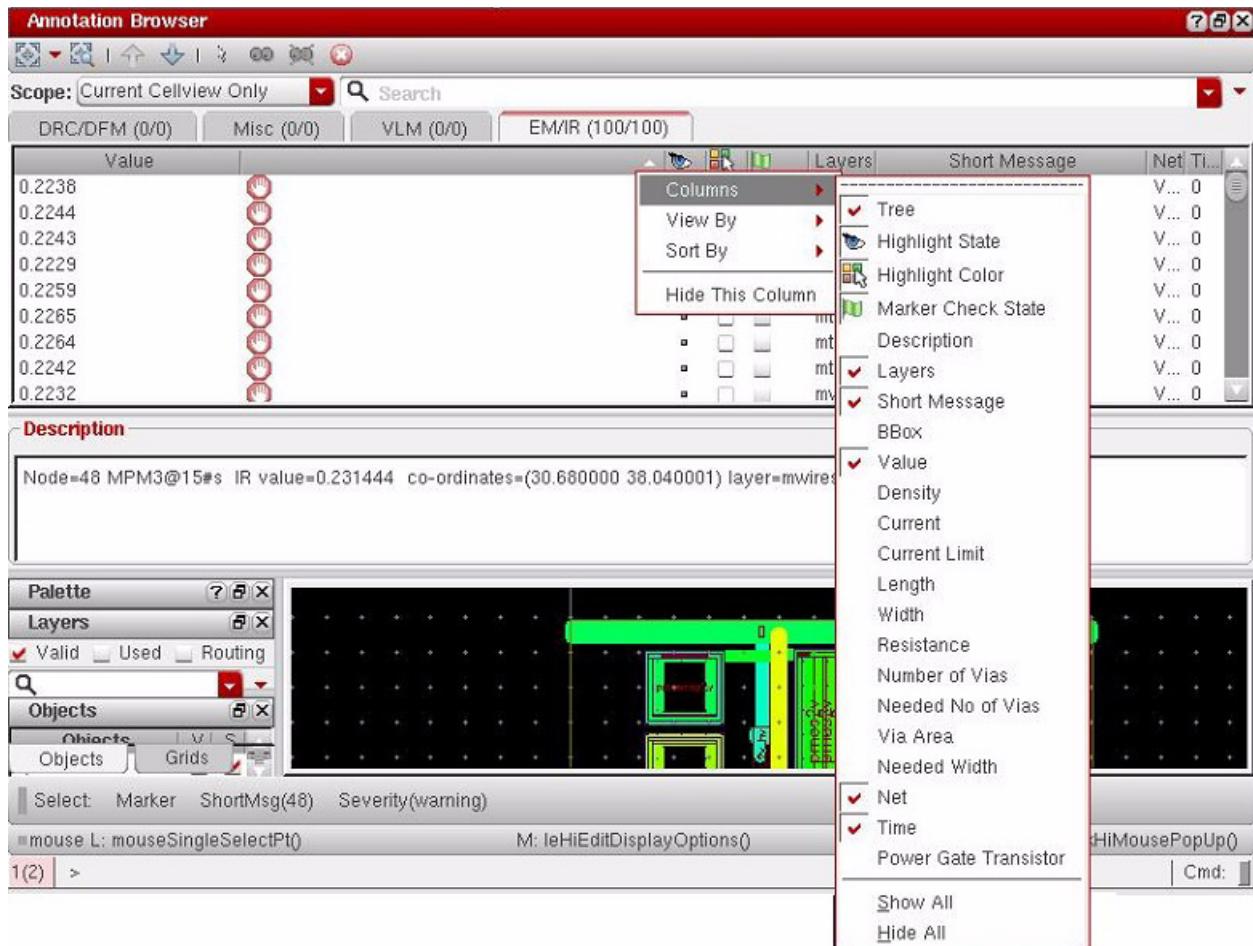
**Note:** The information about columns available for different analyses types for IR plots is provided in the [Types of IR Drop Analysis Plots](#) section in this chapter and for EM plots in the [Types of EM Analysis Plots](#) section of the “EM Analysis Results” chapter.

You can add or remove columns to customize the information you want to view in the Annotation Browser. For this, right-click the column heading, select *Columns* from the drop-down list, and then select the columns you want to view in the tab. This is shown below.

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

**Figure 7-11 Selecting Columns in the EM/IR Tab**



#### Information Displayed in the Description Section of the Annotation Browser

Detailed information about the selected node is provided in the *Description* section of the Annotation Browser. For example, for an RJ analysis type, the following information is provided:

```
Net = VSS Resistor=r1875 RJ value=3.155 layer=mt1 co-ordinates=(12.017000-  
23.025000,11.225000-23.020000) resistance=0.247689 length=0.7875um width=0.24um  
needed Width=0.757201um
```

For an IR drop analysis type, the following information is provided:

```
Node=459 VSS#453 IR value=0.00495278 co-ordinates=(58.334999 27.959999) layer=mt1  
Net=VSS Time=7.64774e-08
```

You can view the units for corresponding values in the description section.

**Note:** The entries in the Annotation Browser honor the user-specified plot display settings in the display options specified in the IR/EM Results form. Therefore, the results of the query comply with the user-specified plot settings.

For more information about how to use the Annotation Browser and manage the different views, see the “Layout XL Assistants” chapter in *Virtuoso® Layout Suite XL User Guide*.

## Querying EMIR Results

The following layout query features are supported in Voltus-Fi-XL.

- **Viewing worst violations:** Select an area on the layout and zoom into the worst violation in the selected area by using the *Zoom Select Max Region* button provided in the Display form, which is the expanded section of the IR/EM Results form. The worst violation in the selected area is highlighted in the Annotation Browser. For details, see [Viewing Worst Violations](#).
- **Viewing all violations in selected area:** Select an area on the layout and retrieve the values of all violations in the selected area by using the *Get Value on Layout* button provided in the Display form. All violations in the selected area are highlighted in the Annotation Browser. For details, see [Viewing all Violations in Selected Area](#).
- **Querying Specific Points and Areas on the Layout for Resistor Information:** Click on any point or select an area on the layout to display the resistor information for that point or for all points that lie within that area in the console. This query feature is independent of the Annotation Browser. For this, a *Query* check box is provided in the Display form. For details, see [Querying Specific Points and Areas on the Layout for Resistor Information](#).

## Specifying the Display Options for IR/EM Plots

After you load the IR drop or the EM analysis results on the IR and EM tabs of the IR/EM Results form, select the nets for which you want to view the plots, and click *Show Plot*. The IR/EM Results form expands to provide options for customizing the plot display on the layout. This expanded form is shown below. You can click the arrow provided on the top-left corner of the form to hide or unhide the Display form.

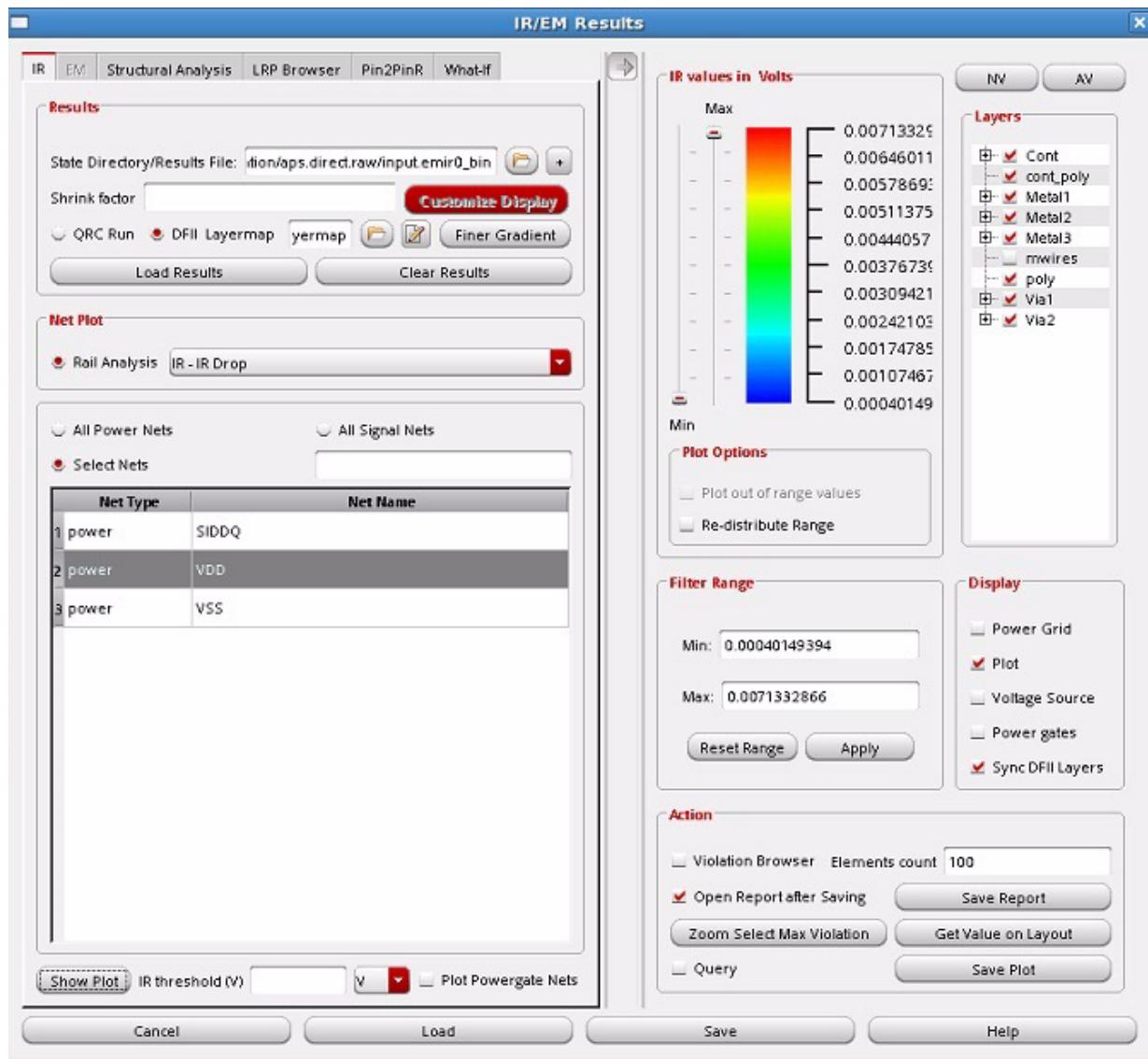
The display options provided in the IR/EM Results form are described in detail below.

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

---

**Figure 7-12 Specifying the Display Options for Plots**



### Selecting Layers for Viewing Violations

The following options are provided in the *Layers* group box of the Display form for selecting layers:

- Click *AV* or *NV* to select or deselect all layers. Use the check boxes provided against layer names to select or deselect specific layers. By default, all layers are selected.

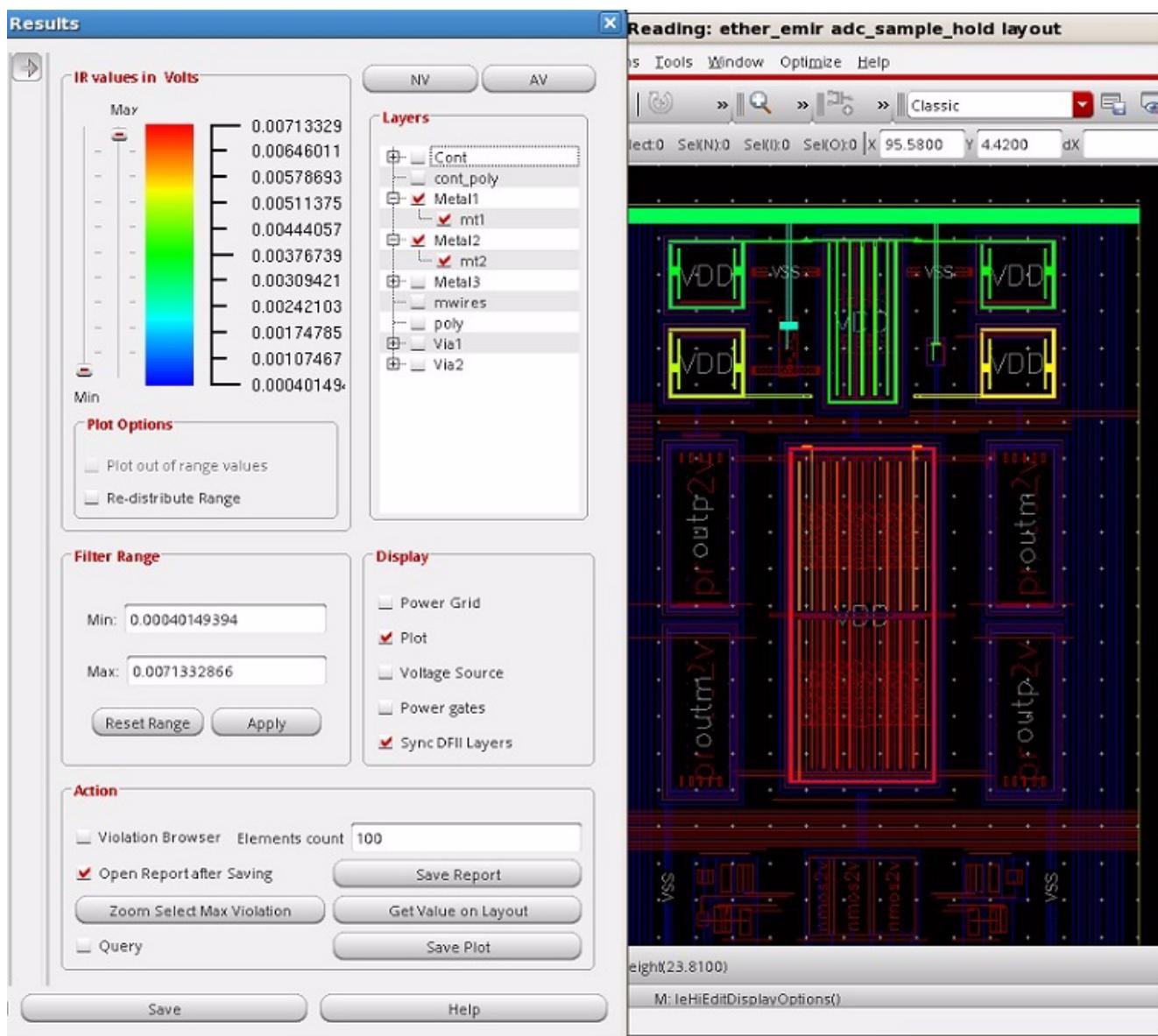
# Voltus-Fi Custom Power Integrity Solution User Guide

## IR Drop Analysis Results

When you update your layer selection, the plot on the layout is updated automatically. The image below shows the layout display of the total violation range for the selected layers for an IR drop analysis type plot.

**Note:** For all layers defined in the DFII layermap file, the DFII layer names and the corresponding extraction layer names are listed in a tree view. For example, when you click Metal1, you will see the extraction layer name, mt1. This is shown in below image.

**Figure 7-13 Displaying the Total Violation Range for Specified Layers on the Layout – IR Analysis Type**



## Customizing the Range for Viewing Violations

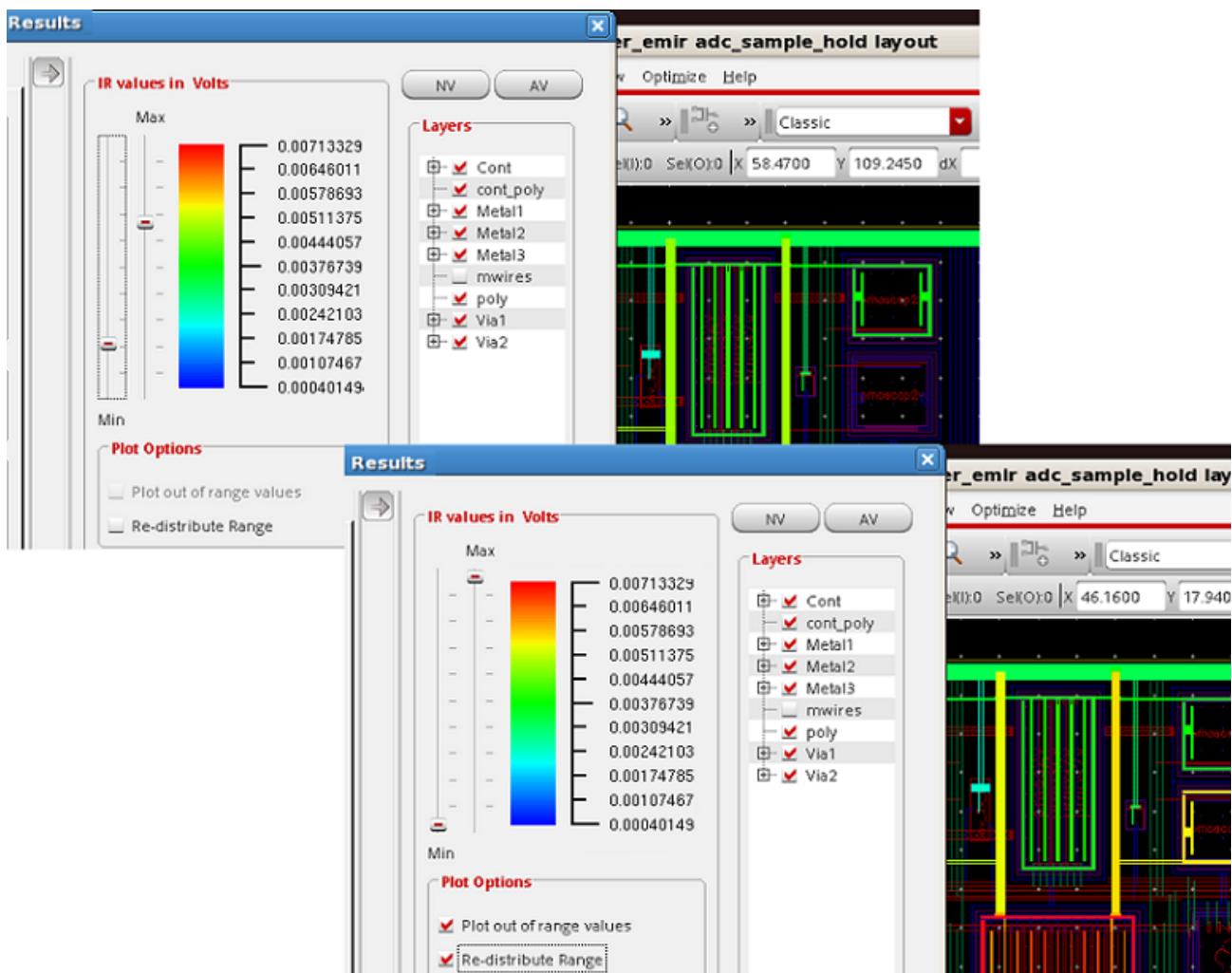
The following options are provided in the Display form for customizing the range for viewing violations.

- *Min* and *Max* sliders are provided for viewing violations that fall within the specified slider range. Move the *Min* - *Max* sliders up and down. The display highlights are updated according to the new slider positions. The *Min* - *Max* slider range functionality works differently in the following scenarios:
  - When the *Re-distribute Range* option in the Plot Options group box is unchecked, moving the *Min* - *Max* sliders only updates the plot display, which means, the violation range outside the color spectrum of this range will be toggled. For example, in [Figure 7-12](#) on page 143, the *Min* and *Max* slider range is changed from showing violations between “green” and “light blue” to “red” and “light blue”.
  - When the *Re-distribute Range* option is checked, moving the *Min* - *Max* sliders re-distributes the range on the ruler and the plot is updated to reflect this new range. This will also update the *Min* and *Max* values in the Filter Range group box.
- The RGB color spectrum is provided next to the slider for viewing violations in different colors.
- The ruler range of violation values, for all or selected layers, is provided next to the color spectrum.

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

**Figure 7-14 Customizing the Range for Viewing IR Drop Violations for Specified Layers**



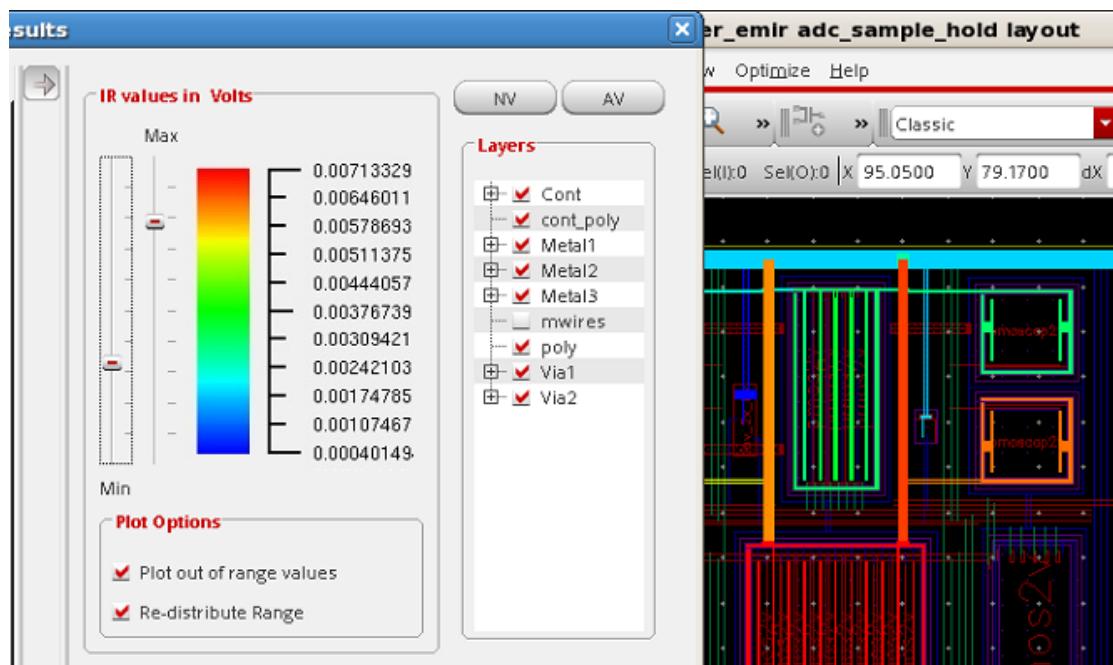
- In the Plot Options group box, select *Plot out of range* to view the violation values outside the specified *Min - Max* slider range. For example, in the image below, the selected range is between “green” and “light blue” but since the *Plot out of range values* is selected, the violations above and below the slider marks are also displayed in “deep red” and “deep blue”, respectively.

**Note:** The *Plot out of range values* option is only enabled when the *Re-distribute Range* option is selected.

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

**Figure 7-15 Viewing IR Drop Violations for the Out of Range Values**

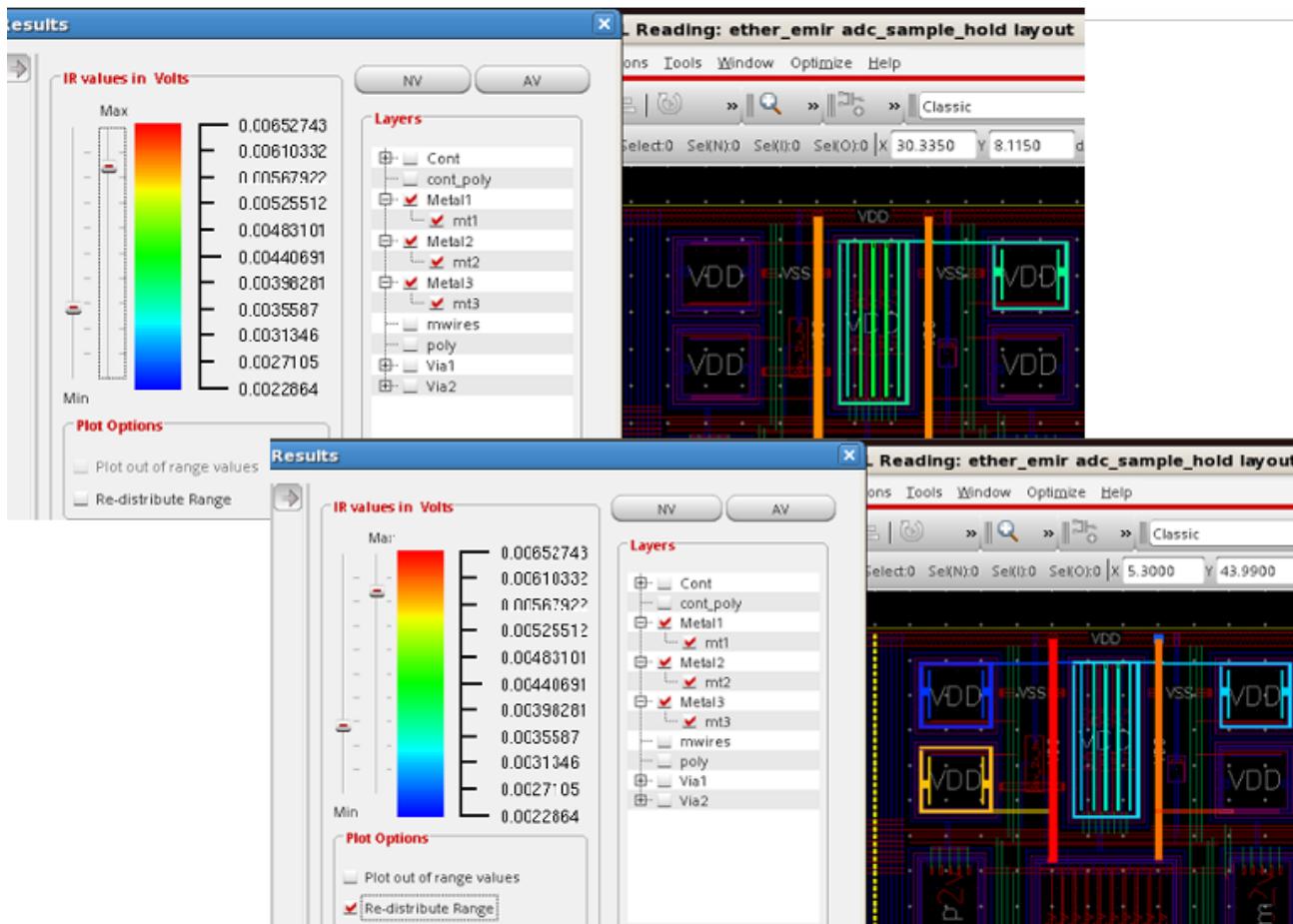


- Select *Re-distribute Range* to view the complete spectrum, from “deep red” to “deep blue”, for the specified layer selection. For example, in the image below, when this option is not selected, the slider range only shows the color range for the violations in the selected range, from “orange” to “light blue”. However, when this option is selected, the slider re-distributes the range to show the complete color spectrum, from “deep red” to “deep blue”, for the selected range.

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

**Figure 7-16 Re-Distributing the Color Spectrum for the Selected Range**



- In the Filter Range group box, specify the *Min* and *Max* values, which are the minimum and maximum values for the filter range. When a plot type is chosen on the IR or EM tab, the filter values are filled in automatically in these fields. To change the default values, you can either type the new Min and Max values and click *Apply* or enter. The plot is updated automatically.
- Click *Reset Range* to restore the default values of filters in the *Min* and *Max* fields.

### Customizing the Display Options for Plots

The following options are used to customize the plot displays:

- Click *Power Grid* to view the power grids. This option is disabled for EM plots.
- Click *Plot* to view the plots.

- Click *Voltage Source* to view the voltage source locations. This option is disabled for EM plots.
- Click *Power gates* to view the power gates on the layout. This option is relevant only for power-gated designs.
- Click *Sync DFII Layers* to toggle the display of the DFII layers for the layers selected in the Layers group box . By default, this option is checked, which means the layers are synchronized.

### **Viewing and Managing Violations on the Layout**

The following options are provided in the Action group box of the Display form for viewing and managing violations on the Virtuoso layout:

- Click *Violation Browser* to view and manage the violation markers generated for the current design. The Annotation Browser window opens on the right side of the Virtuoso Layout and displays the top violations. This is shown in [Figure 7-12](#) on page 147.

**Note:** For more information, see [Viewing EMIR Violations in the Annotation Browser](#).

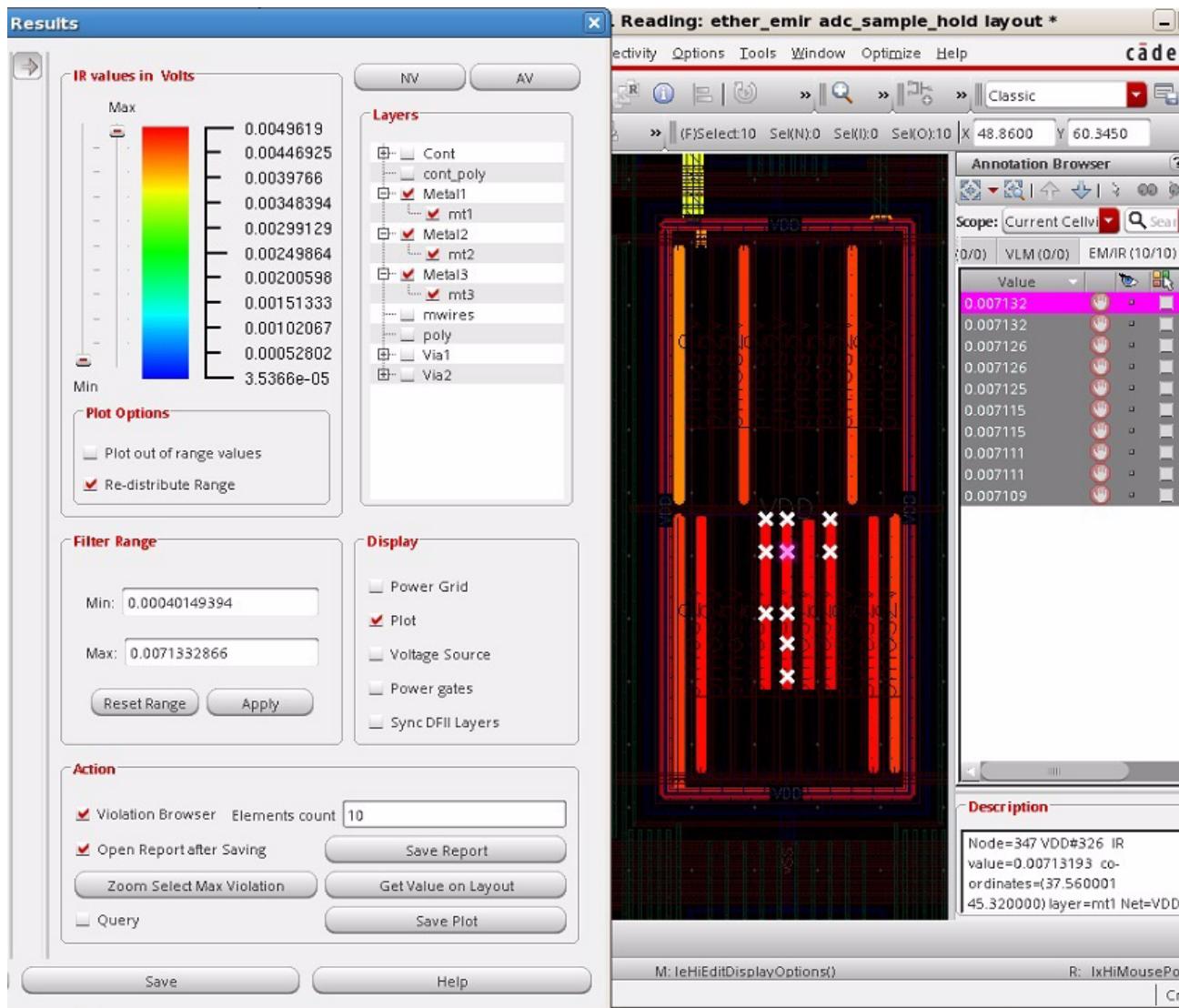
- Specify the number of violations you want to view in the *Elements count* field. For example, you can choose to view only the top ten violations.

The image below shows the top ten violations listed in the EM/IR tab of the Annotation Browser and the violation markers in the plot on the Virtuoso layout.

# Voltus-Fi Custom Power Integrity Solution User Guide

## IR Drop Analysis Results

**Figure 7-17 Viewing the Top Ten Violations on the Layout**



If the number of elements specified is 1000 and above, then the list of violations in the Annotation Browser is suppressed by default. To view the violations, select *Unsuppress Expansion* from the context menu of the Annotation Browser.

**Note:** For more information about how to use the Annotation Browser and manage the different views, see the “Layout XL Assistants” chapter in *Virtuoso® Layout Suite XL User Guide*.

# Voltus-Fi Custom Power Integrity Solution User Guide

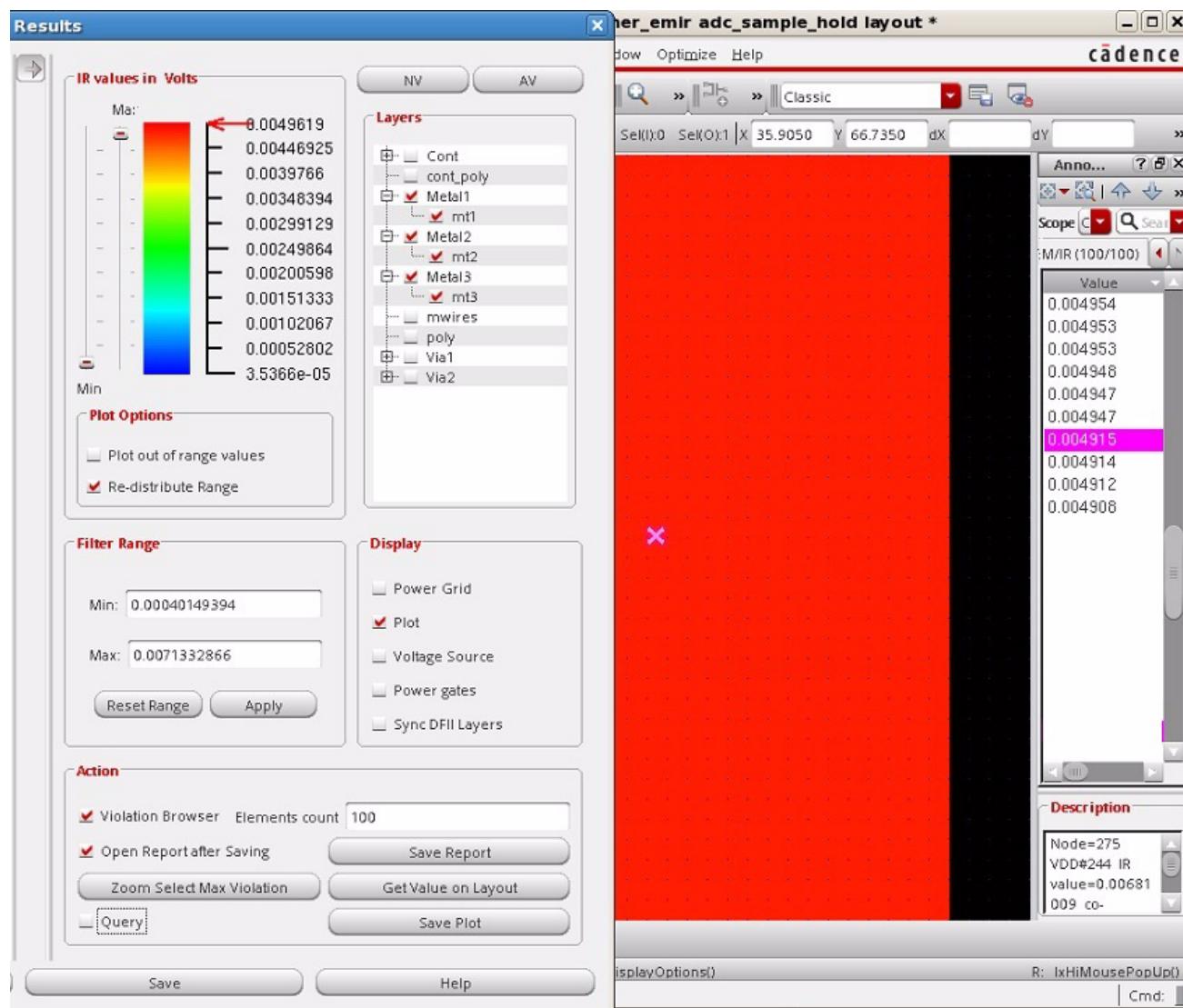
## IR Drop Analysis Results

### Viewing Worst Violations

Click *Zoom Select Max Violation* to zoom into and highlight the maximum violation in the selected area on the layout. For this, first click *Zoom Select Max Violation*, and then select an area on the layout to locate and zoom into the maximum violation in that area. The Annotation Browser opens and the node or presistor with the worst violation in the selected area is highlighted in the browser.

In addition, the maximum violation in the selected area is marked on the ruler in the IR/EM Results form with a red arrow. This is shown in the image below.

**Figure 7-18 Zooming into the Maximum Violation in the Selected Area**



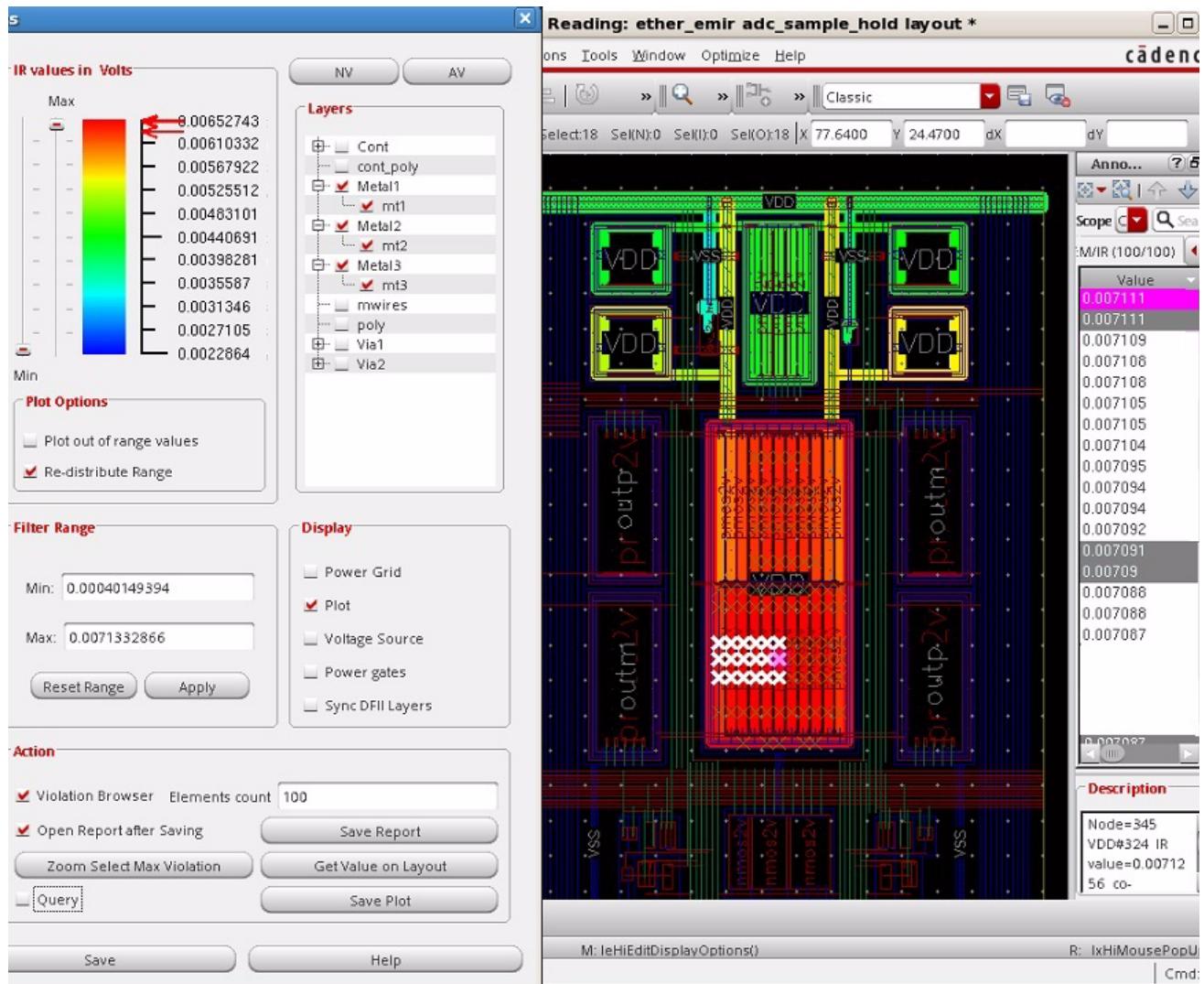
# Voltus-Fi Custom Power Integrity Solution User Guide

## IR Drop Analysis Results

### Viewing all Violations in Selected Area

Click *Get Value on Layout* to retrieve the values of violations in the selected area from the layout view. For this, first click *Get Value on Layout* and then select an area on the layout. The violations in the selected area are marked on the ruler with red arrows and are highlighted in the Annotation Browser. If there are multiple violations in the selected area, all violations will be marked in the Annotation Browser. This is shown in the image below.

**Figure 7-19 Retrieving Violation Values from the Layout**

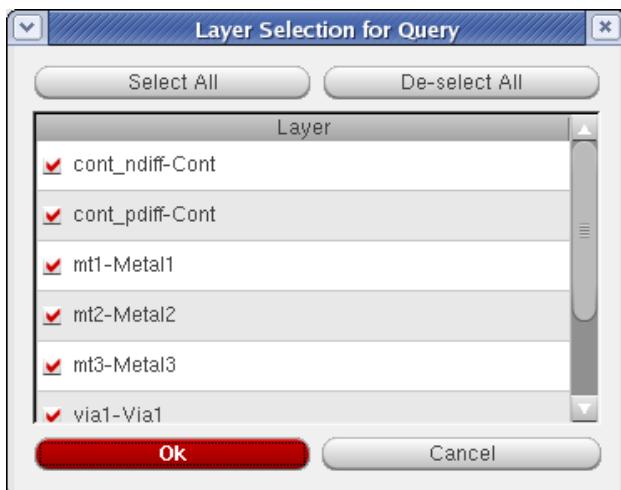


### Querying Specific Points and Areas on the Layout for Resistor Information

Select *Query* to display the resistor information for a point or points within a specified area on the layout. For this, first select *Query* and then use the left mouse button to click on any point or to select an area on the layout.

The Layer Selection for Query form opens. In this form, select the layers for which you want to perform the query and click *OK*. By default, all layers that are selected in the *Layers* group box of the Display form are selected in this form. This form is shown below.

**Figure 7-20 Layer Selection for Query Form**



**Note:** The above form opens when multiple layers are selected in the *Layers* group box or the layer selection window.

The result of the query or the resistor information for the selected point or for points in the selected area is displayed in the console. This is shown in below figures.

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

**Figure 7-21 Displaying Query Results for a Selected Point in the Console**

The screenshot shows the Virtuoso High Capacity Power IR/EM Log window titled "Virtuoso High Capacity Power IR/EM Log:/home/anuk/MMSIM\_EMIR\_WORKSHOP/vfi.log30". The menu bar includes Extraction, Simulation, IR/EM Analysis, Tools, and Help. The Cadence logo is in the top right. The main area displays query results for three points: ri837, rh725, and a third point whose details are partially visible. The results are presented in a tabular format with columns for name, layer, netName, node1, and node2. The third point's data is as follows:

	name	layer	netName	node1	node2
	ri837	mt1	IRmaxNode1	IRavgNode2	IRavgNode2
			0.216787	0.216823	0.216823
			Ipeak	Iavg	layerType
			0.00551244	0.00551244	Metal
		x	y		
		31.475	81.955		

INFO : Query Results for x = 23.935000 y= 80.980000

	name	layer	netName	node1	node2
	ri837	mt1	IRmaxNode1	IRavgNode2	IRavgNode2
			0.223327	0.221428	0.223327
			Ipeak	Iavg	layerType
			0.00251003	0.00251003	Metal
		x	y		
		22.795	83.142		

	name	layer	netName	node1	node2
	rh725	mt2	IRmaxNode1	IRavgNode2	IRavgNode2
			0.218385	0.219557	0.219557
			Ipeak	Iavg	layerType
			0.00524421	0.00524421	Metal
		x	y		
		24.787	82.082		

Console> [Stop Process] 3 | Rail Analysis Results

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

**Figure 7-22 Displaying Query Results for a Selected Area in the Console**

The screenshot shows the Virtuoso High Capacity Power IR/EM Log window with the title bar "Virtuoso High Capacity Power IR/EM Log:/home/anuk/MMSIM\_EMIR\_WORKSHOP/vfi.log30". The menu bar includes Extraction, Simulation, IR/EM Analysis, Tools, and Help. The Cadence logo is in the top right. The main window displays command-line output and two tables of query results.

```
INFO : Loading results for IR plots display success.
Time Msg : Time taken 1.00 seconds. User cpu 0.00 seconds. System cpu 0.00 seconds.
Time Msg (IR plot) : Time taken 0.00 seconds. User cpu 0.00 seconds. System cpu 0.00 seconds.
INFO: layout X-Y are 9820:1480 by 65395:101110
INFO : Query Results for x = 29.850000 y= 81.420000
ERROR : No layer selected
INFO: layout X-Y are 9820:1480 by 65395:101110
INFO : Query Results for x1=26.125000 y1=79.885000 x2=36.860000 y2=84.925000
name      layer          netName        node1           node2
ri1008    mtl            IRmaxNode2   IRavgNode1   VSS#383
          IRmaxNode1   0.214857     0.213931     IRavgNode2
          0.214857     0.213931     0.214857     0.213931
          Ipeak         Iavg          Irms          layerType
          0.00229209   0.00229209   0.00229209   Metal
          x              y
          31.540        82.584
name      layer          netName        node1           node2
ri989     mtl            IRmaxNode2   IRavgNode1   VSS#365
          IRmaxNode1   0.211135     0.214857     IRavgNode2
          0.211135     0.214857     0.211135     0.214857
          Ipeak         Iavg          Irms          layerType
          0.00322035   0.00322035   0.00322035   Metal
          x              y
          31.540        80.199
```

Console> [ ] Stop Process

3 Rail Analysis Results

You can select the nets and the layers for which you want to query results at particular points or areas in the IR/EM Results form. The results for all selected nets and layers will be displayed in the console.

#### **Important**

When the *Query* check box is selected, the left mouse button can be used to click on a series of points or select multiple areas on the layout for retrieving resistor information. At this time, you cannot perform any other operation with the left mouse button. To use this button for any other layout operation, unselect the *Query* option.

### Saving the IR Drop Analysis Result Plots and Reports

The following options are provided for saving the plots and reports of IR drop analysis results:

- Click *Save Plot* to save a screen capture of the plot. When you click this button, the Export Image form opens. This form lets you take a screen capture of the plot displayed in the Virtuoso layout and save it to a standard image format. This form is shown below.

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

**Figure 7-23 Export Image Form**



You can save images in any of these: BMP, JPG, PNG, PPM, XBM, and XPM. To specify the format, use the appropriate suffix with the filename. Note, however, that PNG is the recommended format; not all features are supported by the other formats.

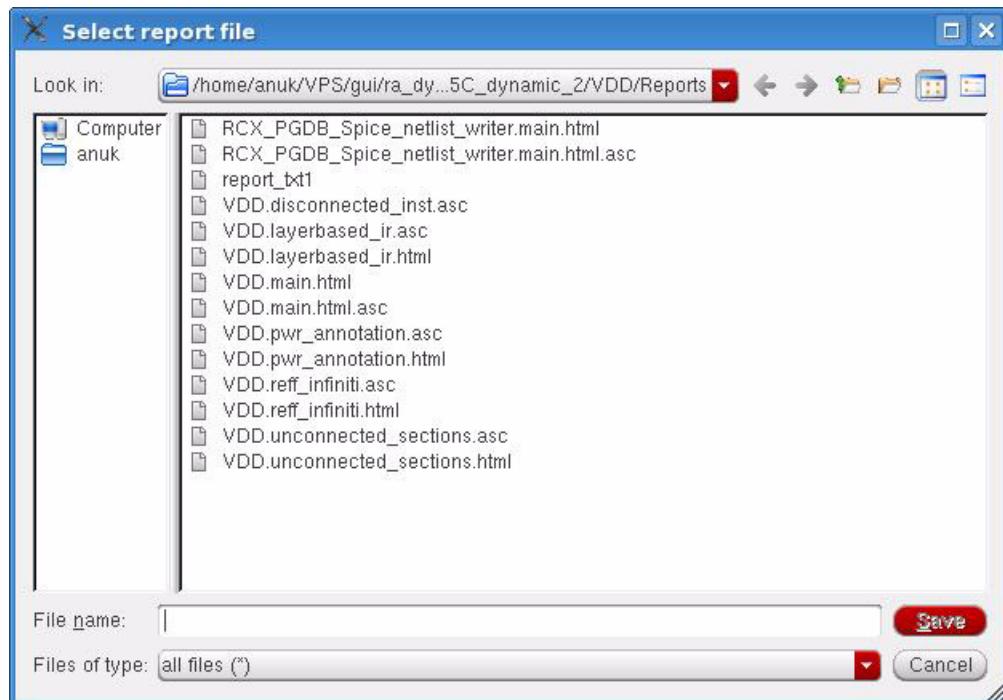
**Note:** For more information about the settings that can be specified in this form, see “Screen Capture Functions” in the *Cadence® User Interface SKILL Reference* guide.

- Click *Save Report* to save the report for the analysis type as a text file. When the *Save Report* button is clicked, the Select report file dialog box opens. You can choose an

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

existing file to save the report, or you can save the report in a new file. The dialog box is shown below.

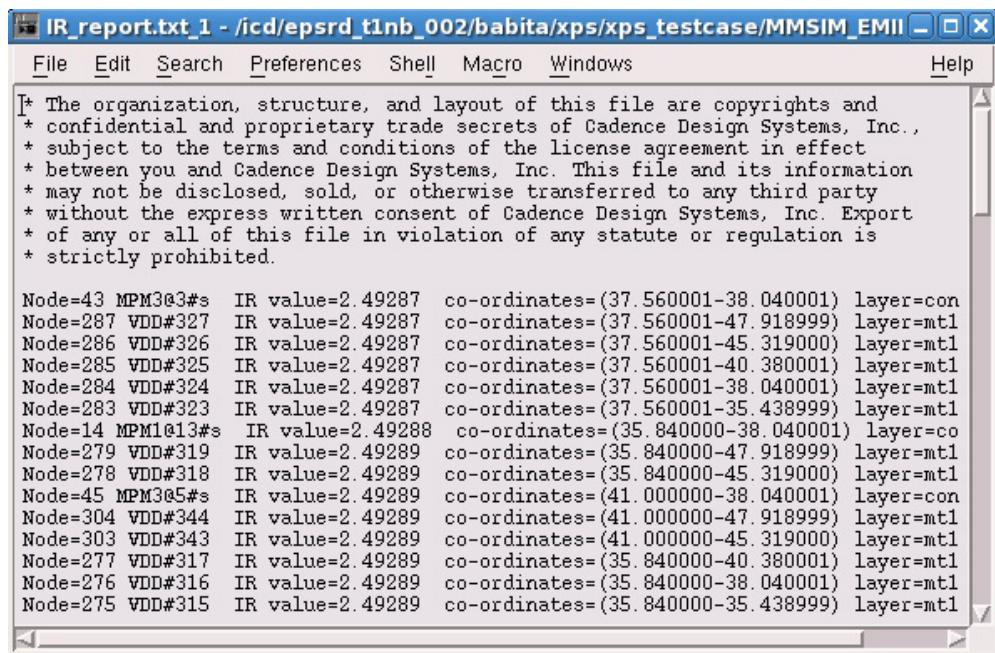


**Note:** To save and print the report for all elements, leave the *Elements count* field empty and then select *Save Report*.

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

- Click *Open Report after Saving* to view the report in the text editor. A sample report for IR drop analysis that opens when you select this option and click *Save Report* is shown below.



The screenshot shows a Windows-style text editor window titled "IR\_report.txt\_1 - /icd/epsrd\_tlnb\_002/babita/xps/xps\_testcase/MMSIM\_EMII". The menu bar includes File, Edit, Search, Preferences, Shell, Macro, Windows, and Help. The main text area contains a copyright notice and a list of nodes with their IR values and coordinates:

```
* The organization, structure, and layout of this file are copyrights and
* confidential and proprietary trade secrets of Cadence Design Systems, Inc.,
* subject to the terms and conditions of the license agreement in effect
* between you and Cadence Design Systems, Inc. This file and its information
* may not be disclosed, sold, or otherwise transferred to any third party
* without the express written consent of Cadence Design Systems, Inc. Export
* of any or all of this file in violation of any statute or regulation is
* strictly prohibited.

Node=43 MPM3@3#s IR value=2.49287 co-ordinates=(37.560001-38.040001) layer=con
Node=287 VDD#327 IR value=2.49287 co-ordinates=(37.560001-47.918999) layer=mt1
Node=286 VDD#326 IR value=2.49287 co-ordinates=(37.560001-45.319000) layer=mt1
Node=285 VDD#325 IR value=2.49287 co-ordinates=(37.560001-40.380001) layer=mt1
Node=284 VDD#324 IR value=2.49287 co-ordinates=(37.560001-38.040001) layer=mt1
Node=283 VDD#323 IR value=2.49287 co-ordinates=(37.560001-35.438999) layer=mt1
Node=14 MPM1@13#s IR value=2.49288 co-ordinates=(35.840000-38.040001) layer=co
Node=279 VDD#319 IR value=2.49289 co-ordinates=(35.840000-47.918999) layer=mt1
Node=278 VDD#318 IR value=2.49289 co-ordinates=(35.840000-45.319000) layer=mt1
Node=45 MPM3@5#s IR value=2.49289 co-ordinates=(41.000000-38.040001) layer=con
Node=304 VDD#344 IR value=2.49289 co-ordinates=(41.000000-47.918999) layer=mt1
Node=303 VDD#343 IR value=2.49289 co-ordinates=(41.000000-45.319000) layer=mt1
Node=277 VDD#317 IR value=2.49289 co-ordinates=(35.840000-40.380001) layer=mt1
Node=276 VDD#316 IR value=2.49289 co-ordinates=(35.840000-38.040001) layer=mt1
Node=275 VDD#315 IR value=2.49289 co-ordinates=(35.840000-35.438999) layer=mt1
```

- Click *Save* to save the form settings in a configuration file. This includes saving all the settings specified for loading the IR/EM results and the display options for the plots. Click *Load* to load these settings in a subsequent session.

## Types of IR Drop Analysis Plots

All rail analysis plots are available in the drop-down list in the *Rail Analysis* field in the IR/EM Results form.

Following types of IR drop analysis results plots are available:

- [IR, IRAVG – IR Drop Plots](#)
- [RC, RCAVG, and RCRMS – Resistor Current Plots](#)
- [IV – Transistor Voltage Plot](#)
- [PI – Powergate Current Plot](#)
- [PV – Powergate Voltage Plot](#)
- [REffective – Effective Resistance Plot](#)

## IR, IRAWG – IR Drop Plots

Analyzes and reports voltage drop. The IR drop plot is displayed on the Virtuoso layout. The IR drop data is saved inside the state directory or the simulation result file of the net. The worst case voltage drop can be displayed using the *Violation Browser* check box. The segments of the design with high IR drop can then be debugged.

There are two types of IR plots available in the drop-down list – **IR - IR Drop** and **IRAVG - IR Avg Drop**. The IR Drop plot displays the peak voltage drop while the IRAWG plot displays the average voltage drop.

### Information Displayed in the Annotation Browser for IR Drop Analysis

The EM/IR tab of the Annotation Browser displays the following columns for this analysis type:

- Value
- Layers
- Short Message
- Net
- Time

The IR plot is shown in the [Displaying and Querying EMIR Results](#) section.

## RC, RCAVG, and RCRMS – Resistor Current Plots

Analyzes and reports resistor currents. After observing IR drop plots, the next step is to understand how the current flows in the design to create the generated IR drop plot. It is important to examine resistor current (RC) plots because they exhibit behaviors that are not obvious from examining only IR drop plots. The RC plots show current-flow trends that you either might not have expected or current from several power pins reconverging in the middle of the chip to create high currents in some wires.

There are three types of RC plots available in the drop-down list – **RC - Peak Resistor Current**, **RCAVG - Average Resistor Current**, and **RCRMS - RMS Resistor Current**.

### Information Displayed in the Annotation Browser for RC Plot Types

The EM/IR tab of the Annotation Browser displays the following columns for RC - Peak, RCAVG, and RCRMS plot types:

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

---

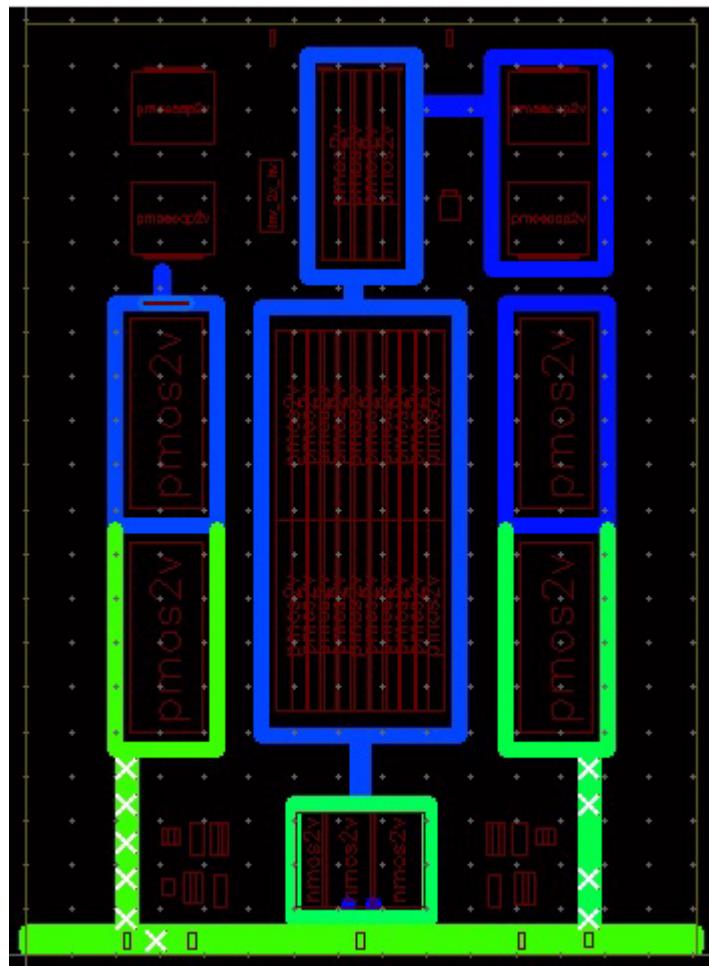
- Value
- Layers
- Short Message
- Width
- Length
- Resistance
- Via Area
- Number of Vias

The plots shown below display peak and average resistor currents. The violation range is customized by using the *Min Max* sliders in the IR/EM Results form.

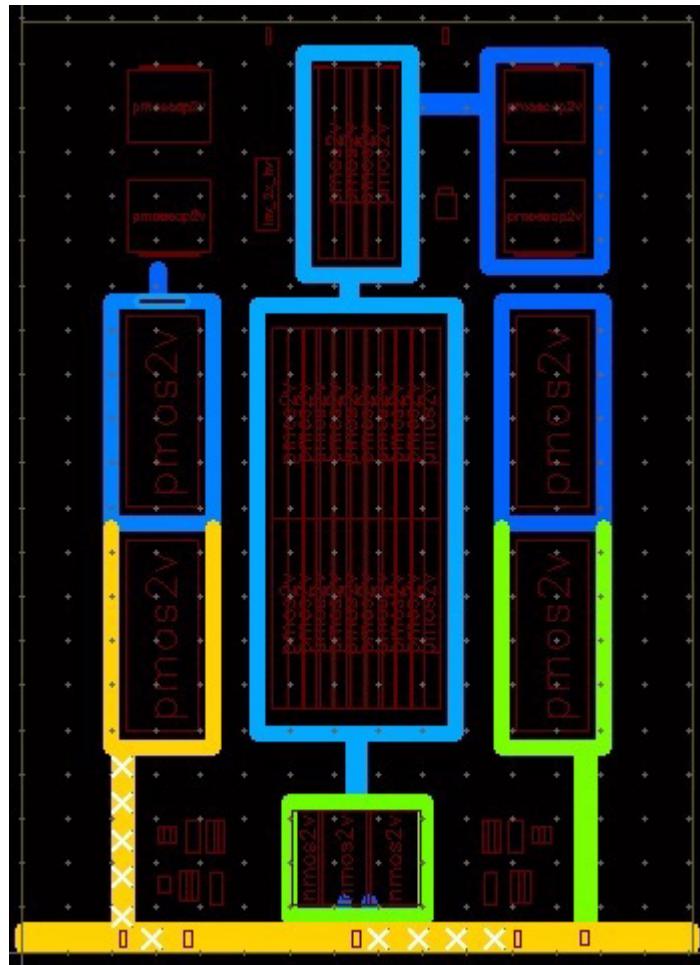
# Voltus-Fi Custom Power Integrity Solution User Guide

## IR Drop Analysis Results

**Figure 7-24 RC – Peak Resistor Current Plot**



**Figure 7-25 Average Resistor Current Plot**



## IV – Transistor Voltage Plot

Analyzes and reports the transistor-based supply voltage data. The transistor voltage is determined using power and ground IR drop waveforms inside the switching window of the transistor.

### Information Displayed in the Annotation Browser for Transistor Voltage Plot Type

The EM/IR tab of the Annotation Browser displays the following columns for transistor voltage analysis:

- Value
- Layers

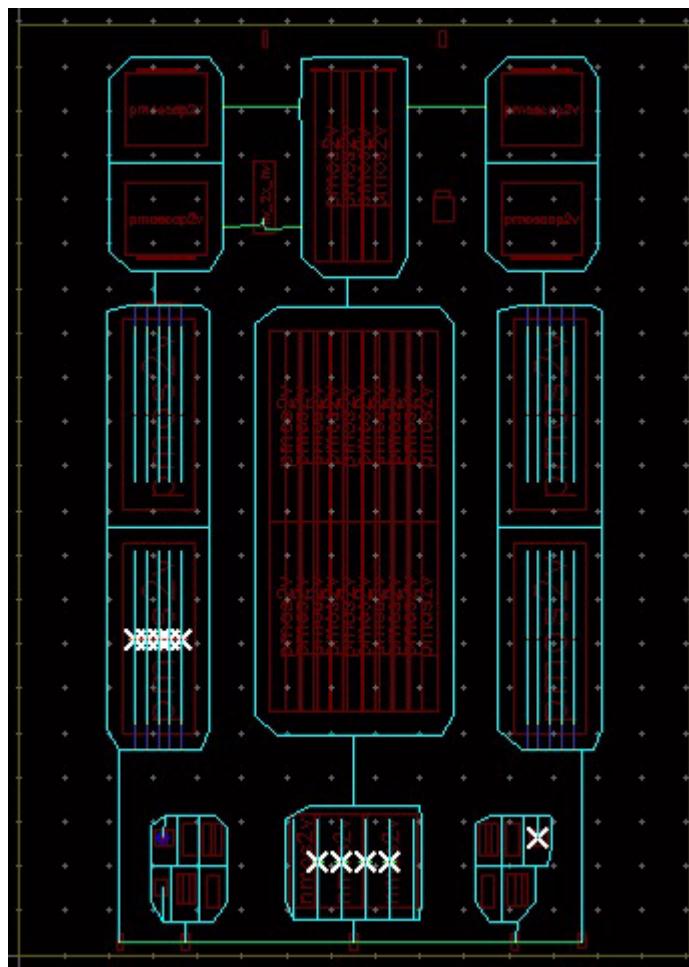
# Voltus-Fi Custom Power Integrity Solution User Guide

## IR Drop Analysis Results

- Short Message
- Net
- Time

This plot is shown below.

**Figure 7-26 Transistor Voltage Plot**



## PI – Powergate Current Plot

This plot is only applicable to power-gated design and analysis.

This plot displays the currents across resistors when power gates are connected. This plot is used to analyze whether the current through power gates exceeds the saturation current or

not, and if it does, to debug such instances in the design. The saturation current and the on-resistance of the power switches are characterized and stored inside the power-grid library of the power-gate transistor.

### **Information Displayed in the Annotation Browser for Powergate Plot Types**

The EM/IR tab of the Annotation Browser displays the following columns for both, powergate current and powergate voltage plot types:

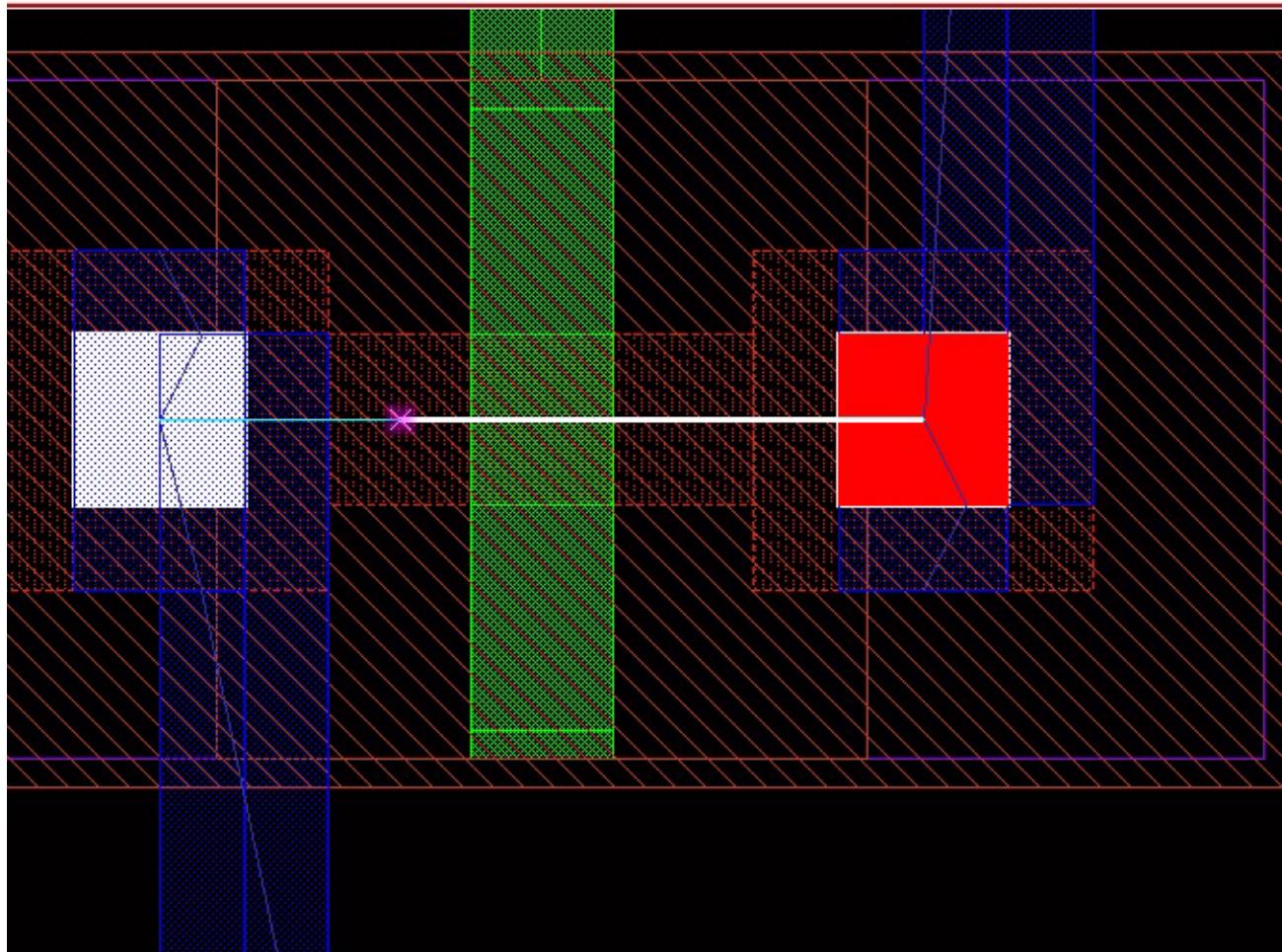
- Value
- Layers
- Short Message
- Resistance
- Power Gate Transistor

This PI plot is shown below. In this plot, the resistor with the maximum current violation is zoomed into using the *Zoom Select Max Violation* option in the IR/EM Results form.

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

**Figure 7-27 Powergate Current Plot**



A sample PI report is shown below:

# Voltus-Fi Custom Power Integrity Solution User Guide

## IR Drop Analysis Results

```
pi RESULTS
BINARY FILE = /vols/ssvrd_t1_010/nanduc/VPS/CCRs/ruchin_pg_case/vps_pg_design/header_vpsml/output.pg/xps.emir0_bin
RESULTS FILE CREATED = 2013-Dec-03 23:17:20 (2013-Dec-04 07:17:20 GMT)
USER SUPPLIED VALUES:
    RESULTS TYPE      = TRANSIENT
    TRANSIENT START   = 0
    TRANSIENT STOP    = 3e-08
    SIM TEMPERATURE   = 25 C
|
----- "VDDG" NET: Vref = 1.200000V -----
max

Resistor  Idsat  pi Value      Power Gate Transistor  X1       Y1       X2       Y2       Resistance
rk167     NA     4.906e-06   XM16      3.179000  4.335000  3.210000  4.335000  3.75
rk166     NA     4.90562e-06  XM15      3.179000  2.125000  3.210000  2.125000  3.75
rk162     NA     1.01019e-06  XM10      2.114000  1.260000  2.145000  1.260000  3.75
rj159     NA     1.00065e-06   XM8       1.759000  3.855000  1.790000  3.855000  3.75
rk164     NA     9.62062e-07  XM13      2.544000  0.395000  2.575000  0.395000  3.75
rj157     NA     9.5882e-07   XM6       1.049000  3.855000  1.080000  3.855000  3.75
rj158     NA     9.58674e-07  XM7       1.759000  2.555000  1.790000  2.555000  3.75
rk165     NA     9.58373e-07  XM14      3.179000  1.260000  3.210000  1.260000  3.75
rk163     NA     4.82571e-07  XM11      2.469000  2.125000  2.500000  2.125000  3.75
rj156     NA     4.7966e-07   XM5       1.049000  1.260000  1.080000  1.260000  3.75
```

## PV – Powergate Voltage Plot

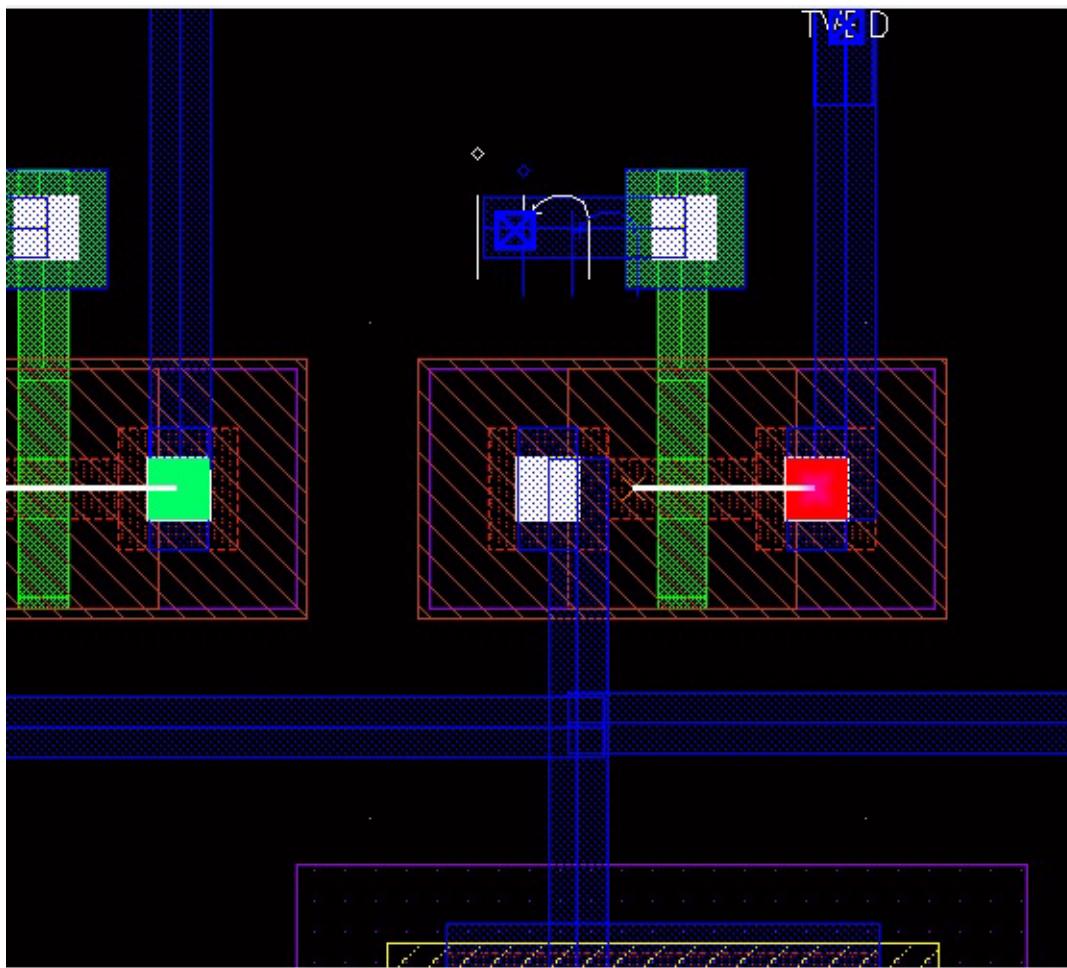
This plot displays the IR drop across power-switch instances. This plot can be used to analyze and debug the regions of IR drop inside the power-gated block. For example, if the IR drop, across power gate is already high, the IR drop inside the power-gated block will be much higher. In this case, the power-gate placement will have to be refined or more power gates will have to be added to resolve the IR drop problem.

This plot is shown below.

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

**Figure 7-28 Powergate Voltage Plot**



A sample PV report is shown below:

# Voltus-Fi Custom Power Integrity Solution User Guide

## IR Drop Analysis Results

---

pv RESULTS

```
BINARY FILE | = /vols/ssvrd_t1_010/nanduc/VPS/CCRs/ruchin_pg_case/vps_pg_design/output.pg/xps.emir0_bin
RESULTS FILE CREATED = 2013-Dec-03 23:18:03 (2013-Dec-04 07:18:03 GMT)
USER SUPPLIED VALUES:
    RESULTS TYPE      = TRANSIENT
    TRANSIENT START    = 0
    TRANSIENT STOP     = 3e-08
    SIM TEMPERATURE   = 25 C

----- "VDDG" NET: Vref = 1.200000V -----
max

Resistor  Idsat  pv Value  Power Gate Transistor X1          Y1          X2          Y2          Resistance
rj159     NA     0.51355 XM8       1.759000  3.855000  1.790000  3.855000  3.75
rj157     NA     0.513548 XM6       1.049000  3.855000  1.080000  3.855000  3.75
rk167     NA     0.513542 XM16      3.179000  4.335000  3.210000  4.335000  3.75
rj158     NA     0.513535 XM7       1.759000  2.555000  1.790000  2.555000  3.75
rk164     NA     0.51353  XM13      2.544000  0.395000  2.575000  0.395000  3.75
rj156     NA     0.513523 XM5       1.049000  1.260000  1.080000  1.260000  3.75
rk162     NA     0.513521 XM10      2.114000  1.260000  2.145000  1.260000  3.75
rk163     NA     0.51352  XM11      2.469000  2.125000  2.500000  2.125000  3.75
rk165     NA     0.513462 XM14      3.179000  1.260000  3.210000  1.260000  3.75
rk166     NA     0.513273 XM15      3.179000  2.125000  3.210000  2.125000  3.75
```

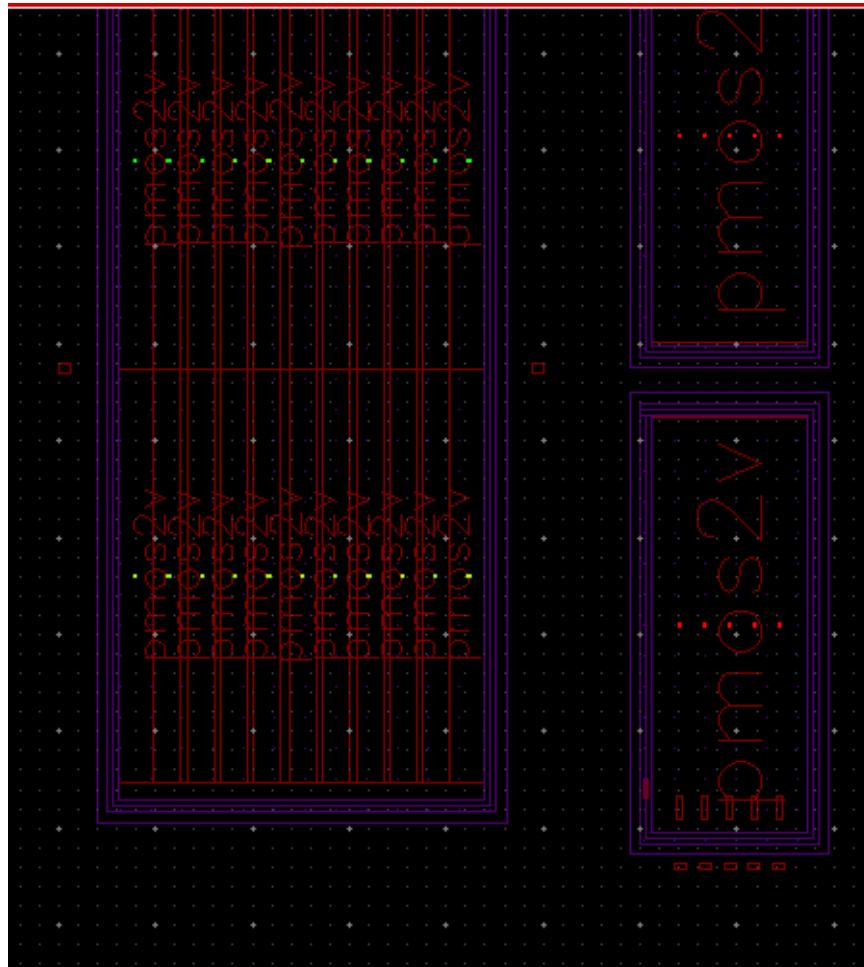
## REffective – Effective Resistance Plot

Uses the data generated by the static power grid solver (SPGS) to display the pin-to-node resistances on the Virtuoso layout. For more information about how resistances are calculated and reported by the SPGS, see [Static Power Grid Solver](#).

In this plot type, you can also plot the least-resistive path for a net. For details, see [Plotting the Least Resistive Path](#).

The REffective plot for all nets is shown below.

**Figure 7-29 REffective Plot for All Nets**



## Plotting the Least Resistive Path

The least-resistive path (LRP) plot lets you identify weakly connected instances in the design during early stages of power planning. The resistance for an instance pin is calculated as the total resistance along the least resistance path. If an instance has multiple power pins connected to the power grid, the LRP plot uses the pin with the worst (highest) resistance value to plot the instance-based data. This plot highlights the current path for the selected instance to the voltage source. A long LRP usually results in high resistance and potentially high voltage drop.

In addition to identifying and displaying the worst IR drop violations, the LRP feature in Voltus-Fi-XL lets you plot LRP on demand for any node or tap. You can view the LRP for any high IR drop node by selecting it on the layout. There are two options provided in the GUI, *Get layout*

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

---

*Tap and Get Marker Tap*, to let you select a node on the GUI and view its LRP and to select an object in the Annotation Browser and view the LRP for the node on the selected marker. For details, see [The LRP Browser](#) and [Displaying LRP for the Tap on the Selected Marker](#), respectively.

The use model for LRP analysis is detailed below.

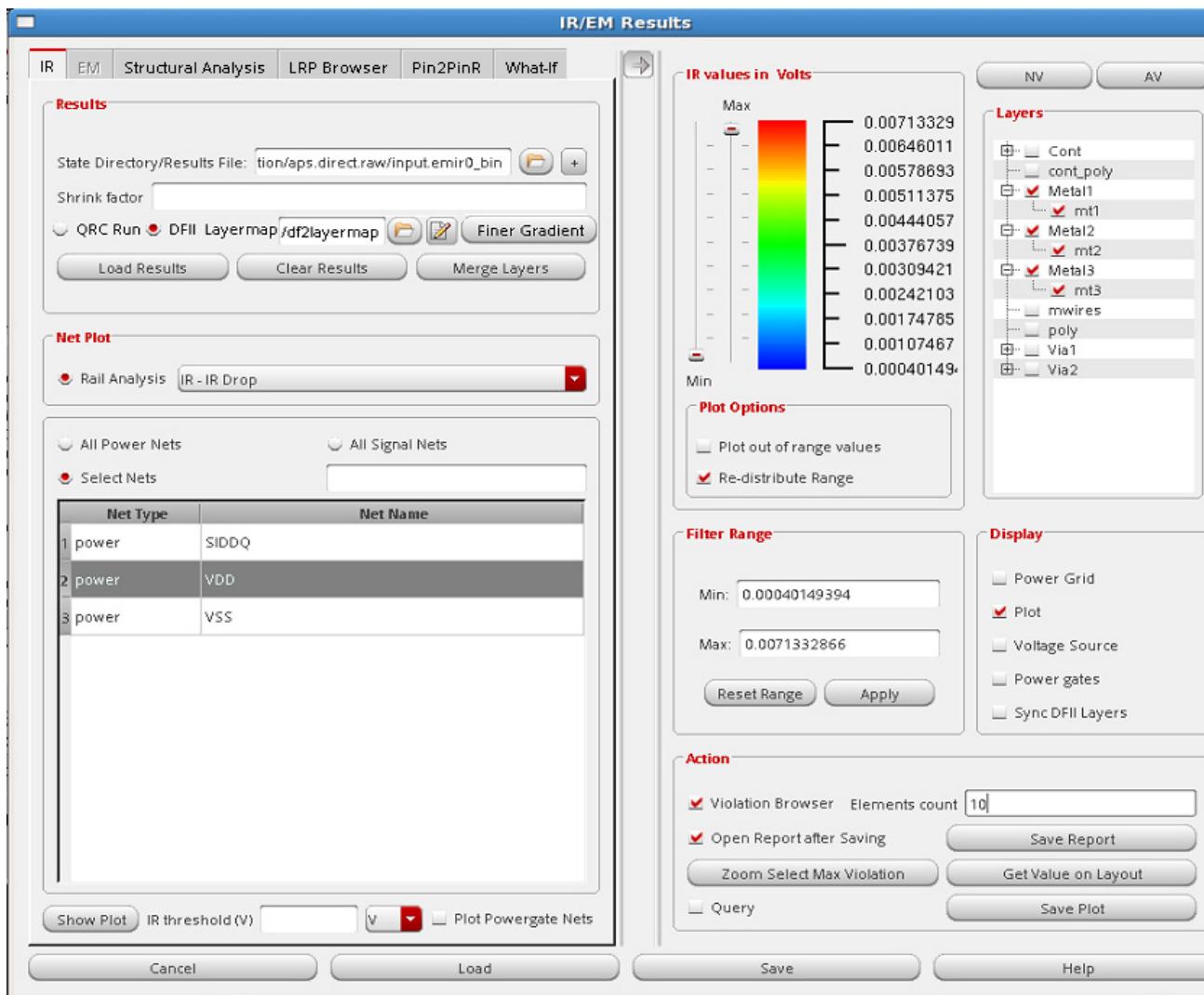
To view the LRP plot for a net, perform the following steps after loading the IR drop results:

- Click *Select Nets* to select the net for which you want to view the LRP plot.  
**Note:** The LRP plot feature works for one net at a time. It can only be used in the “Select Nets” mode and not in the “All Nets” mode.
- Click *Show Plot*. The LRP Browser tab in the IR/EM Results form gets enabled. This is shown below.

# Voltus-Fi Custom Power Integrity Solution User Guide

## IR Drop Analysis Results

**Figure 7-30 IR/EM Results Form – Plotting the Least-Resistive Path**



## The LRP Browser

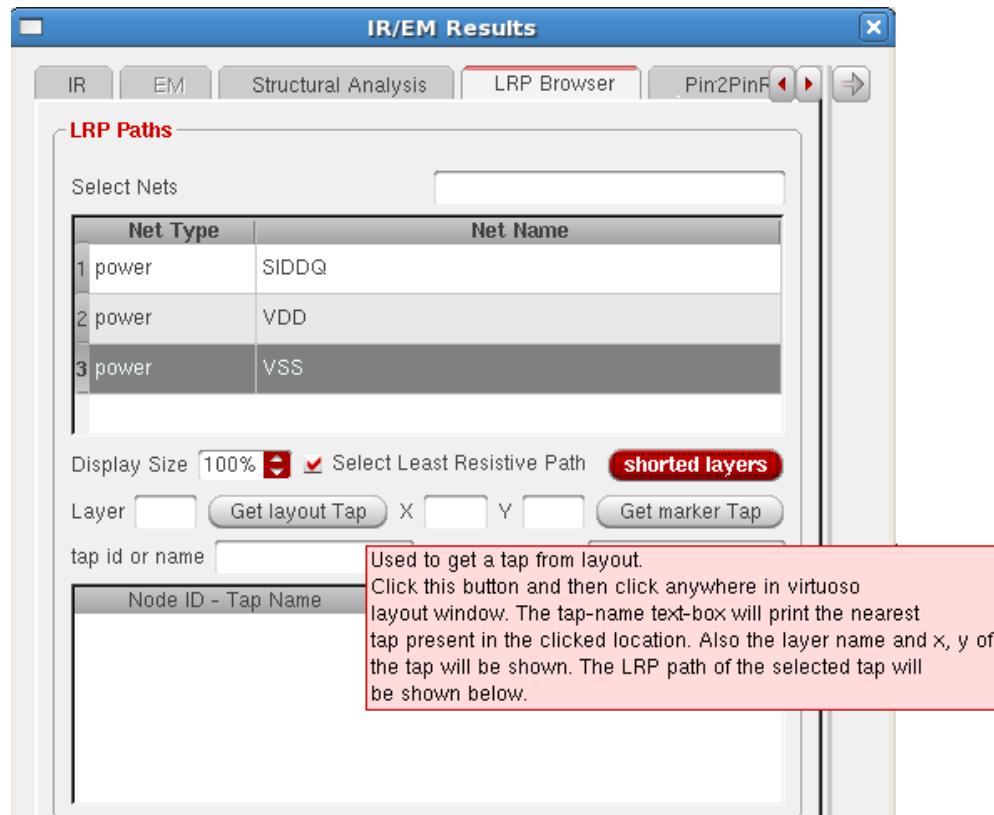
The LRP Browser tab lets you specify nets, layers, and nodes for which you want to view the LRP plots. The various options provided for this are detailed below.

Voltus-Fi-XL provides support in the form of tips about the information to be filled out in the various fields in the LRP Browser form. This tool tip appears when you hover over a field in the form. This is shown below.

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

**Figure 7-31 Tool Tips for LRP Browser Tab**

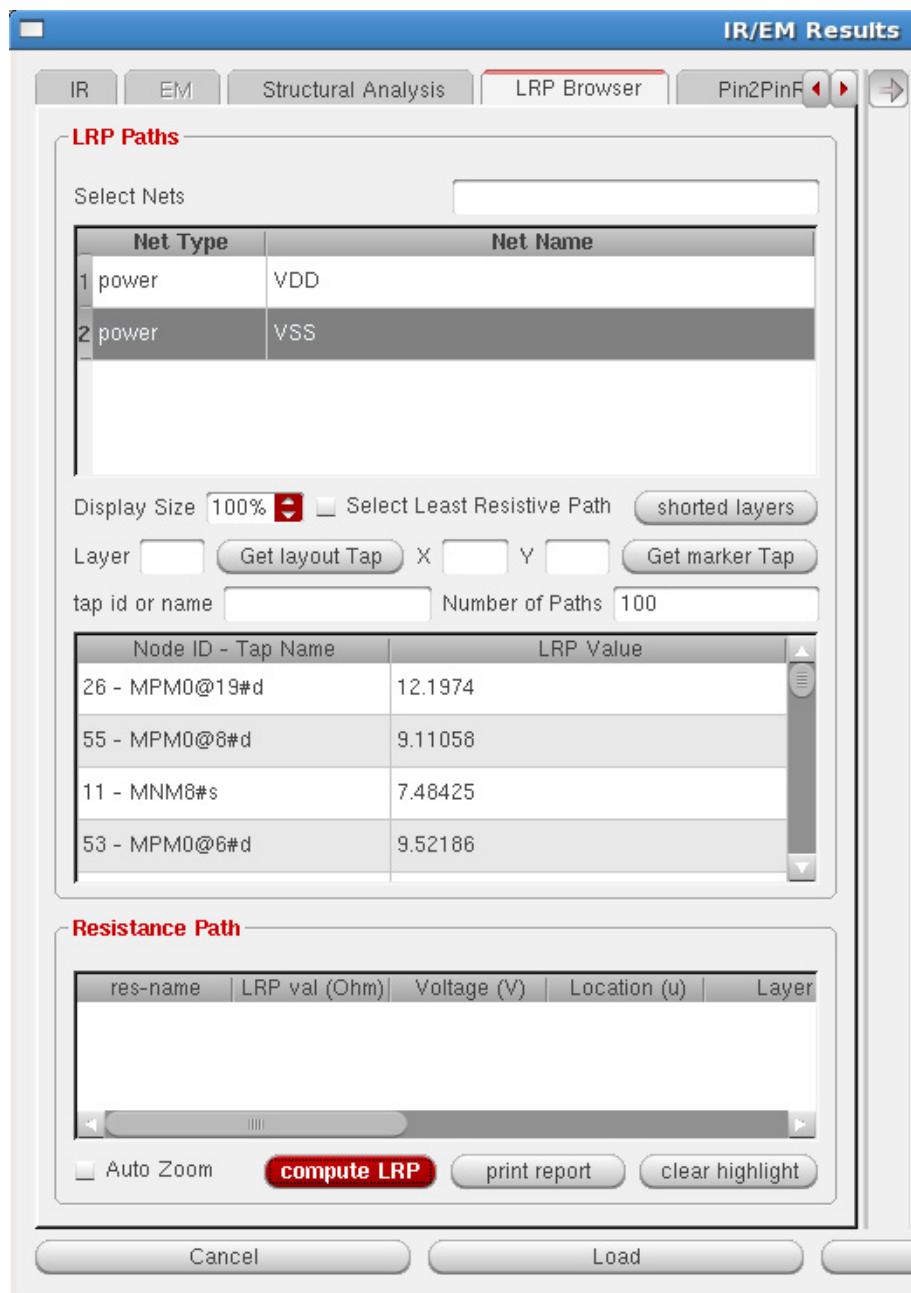


To start, click the LRP Browser tab of the IR/EM Results form. The form shown below opens.

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

Figure 7-32 LRP Browser Tab of the IR/EM Results Form



There are two group boxes in this tab, the LRP Paths and Resistance Path.

- In the LRP Paths group box, specify the net for which you want to perform the LRP analysis in the *Select Nets* field. The net selected in the IR tab is selected by default.

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

---

- Click *Compute LRP* to compute LRP values for all taps of the selected net. The information is populated in the Node ID - Tap Name and LRP Value columns of the table in the LRP Paths group box.
- Specify the *Display Size* in percentage.
- Click *Select Least Resistive Path* to view the least-resistive path on the layout.
- Click *shorted layers* to short specific layers for the LRP analysis. For details, see [Shorting Layers for LRP Analysis](#).
- Click *Get layout Tap* to select a tap on the layout. Click this button and then click anywhere in the Virtuoso layout window. The tap-name text box will print information about the X- and Y-coordinates, and the *Layer* name for the tap present nearest to the location of the click.
- Click *Get Marker Tap* to view the LRP values for a tap, if present, on a selected marker. For this, enable the Annotation Browser. For details, see [Displaying LRP for the Tap on the Selected Marker](#).
- Type the name of the tap-node in the *tap id or name* to retrieve the LRP value for the specific tap-node. Tap names can be regular expressions, such as \*<sup>MPM</sup> shown in the figure above. The LRP table shows all taps matching the regular expression. This saves time taken to scroll through the list of nodes to find information for a specific node.

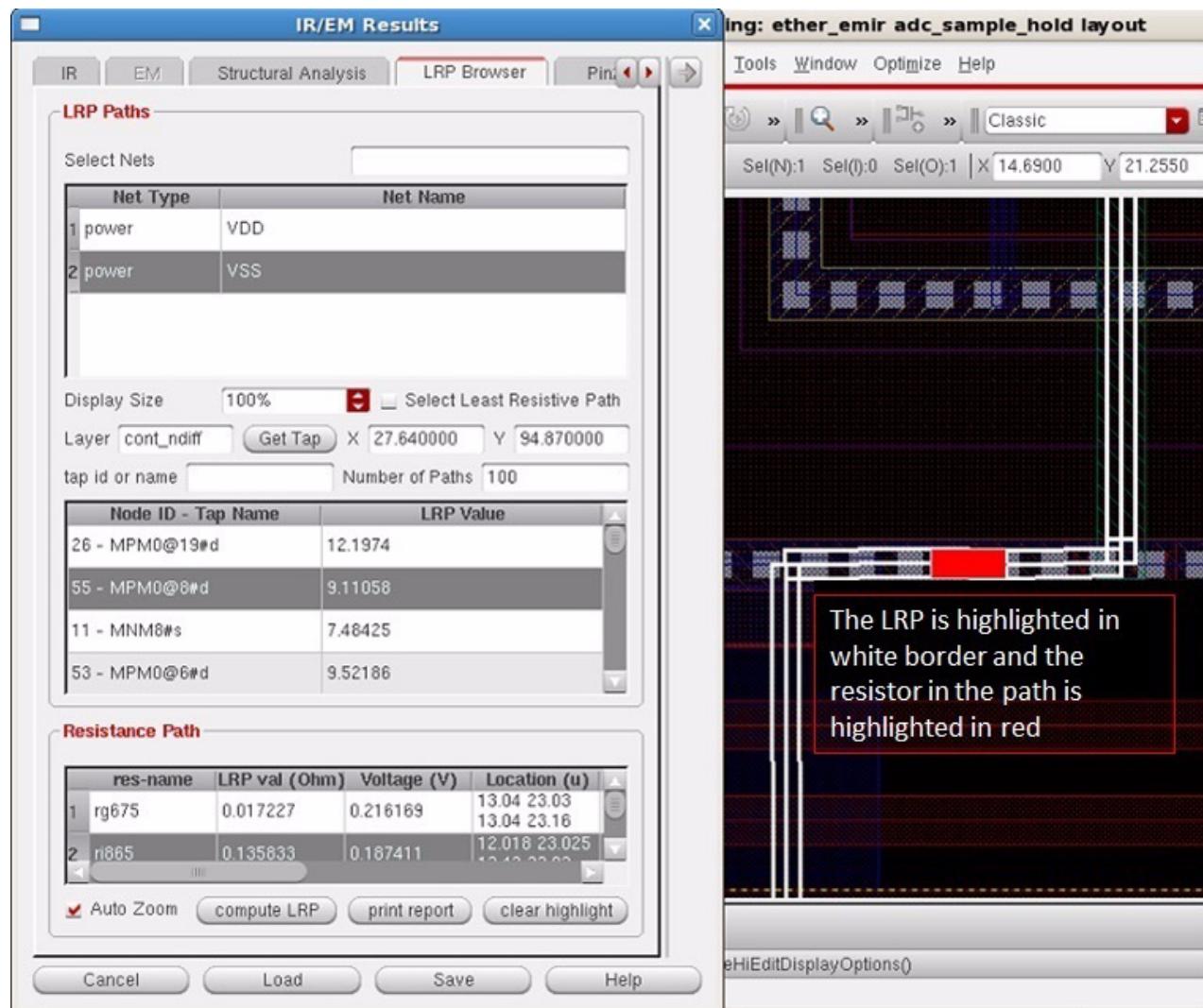
**Note:** You can type the name of any node in this field, including a node that is not included in the list of violations. In this case, the table in the *Resistance Path* group box shows the information for the node of the selected net and the plot displays the LRP path.

- Specify the *Number of Paths* or the number of violations that you want to view. By default, top 100 violations are listed.
- The table shows the LRP for the selected net to all instances or taps defined for the net in the *Node ID - Tap Name* column and their corresponding LRP values in the *LRP Value* column. For each instance, the table shows the cumulative resistance on the path.
- Select an instance in the list to view its LRP on the layout. When an instance is clicked, the LRP path is highlighted in red in the Virtuoso layout. This is shown in below image.
- In the *Resistance Path* group box of the LRP Browser tab, you can view the list of resistors for a path. Select *Auto Zoom* to automatically zoom to the resistor being selected in the path on the layout. The LRP path is highlighted with a white border and the resistor from the path is highlighted in red. This is shown below.

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

**Figure 7-33 Least-Resistive Path for a Resistor Highlighted on the Virtuoso Layout**



**Note:** If the tables in the LRP browser tab do not show any entries, ensure the following:

- ❑ The selected net should have at least one pin location or voltage-source location
- ❑ The selected net should have taps connected to it
- Click *Print Report* to print the LRP values of the specific resistors in a text file. The Select report file pop-up window opens. Specify the name of the report file and click *Save*. The report opens in the console. A sample report is shown in [Default Reports Generated for EMIR Analysis](#).
- Click *Clear highlight* to clear the least-resistive path highlighted on the layout.

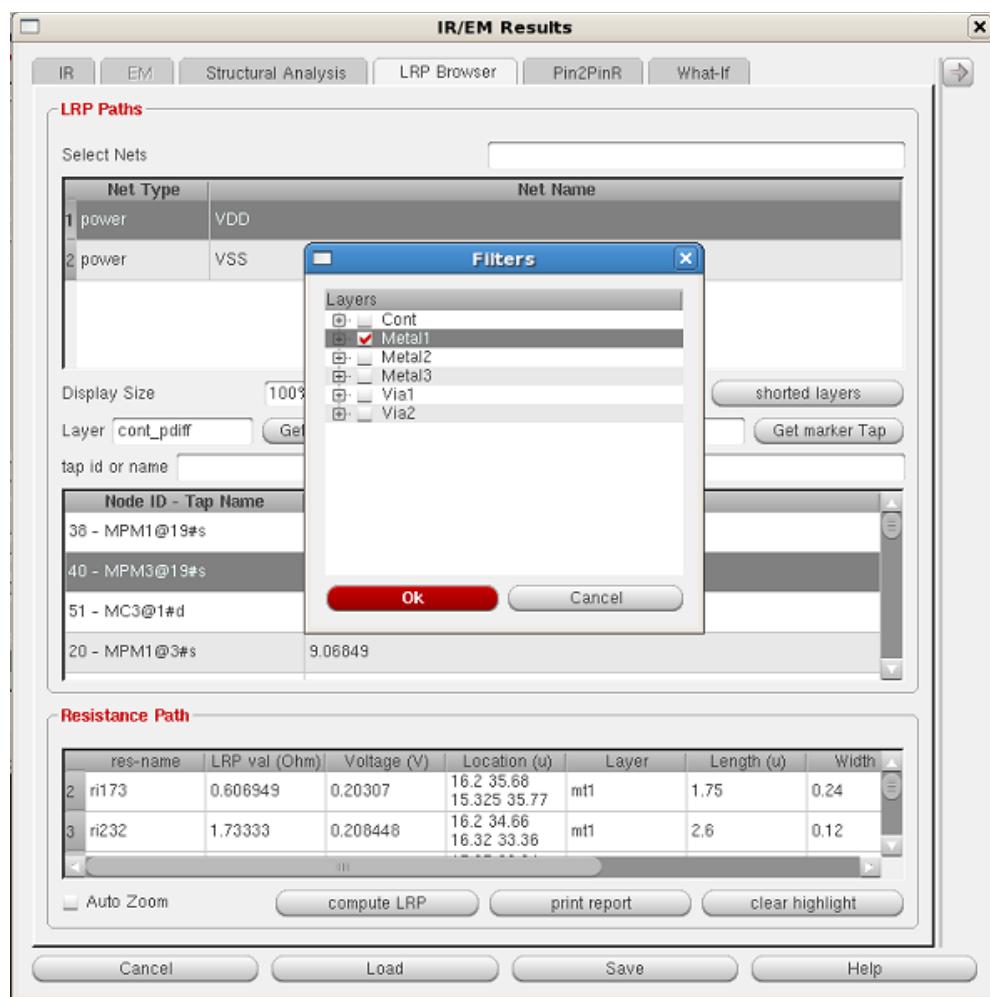
## Shorting Layers for LRP Analysis

While computing LRP for a tap-node or instance, you can short specific layers. When you do this, all resistance on the specified layer is shorted, that means, the LRP display shows a value of 0 for all resistors on the shorted layer. This is shown in below images.

When you click *shorted layers*, the *Filters* pop-up window opens. In this window, select the layer you want to short. You can short multiple layers.

For example, in the image below, layer Metal1 is shorted.

**Figure 7-34 Specifying the layers to be shorted for the LRP Analysis**

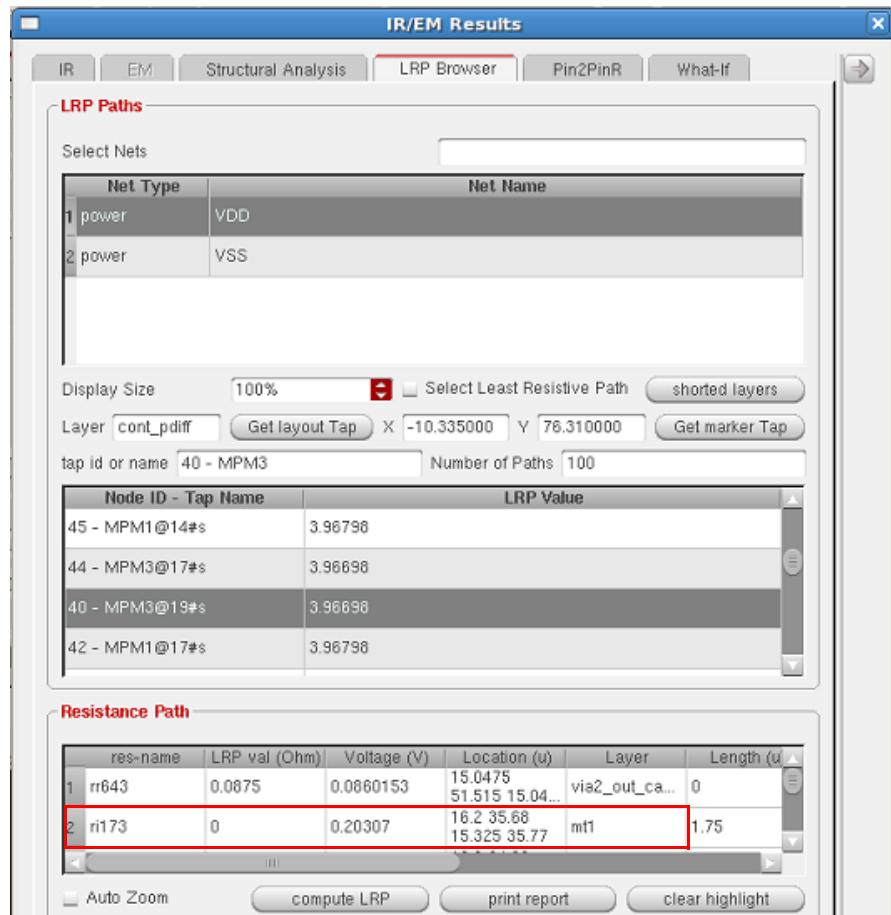


When a layer is shorted, the Layer column in the Resistance Path group box shows a value of 0 in the LRP val (Ohm) column for all resistors on that layer. For example, in the image below, the LRP value for resistor, ri173 on layer, m1 is 0.

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

**Figure 7-35 LRP Value for Resistors on the Shorted Layer**



### Displaying LRP for the Tap on the Selected Marker

You can display the LRP for a tap on the selected marker in the Annotation Browser. When you select a violation marker in the Browser, it is highlighted on the layout. If the marker is associated with a tap name, you can view the LRP for the tap. This is particularly useful when either REffective or the transistor voltage plot is being shown and you want to see the LRP for the selected marker tap.

The steps are detailed below.

- In the IR tab, select *IV – Transistor Voltage* or *REffective – Effective Resistance* as the plot type and click *Show Plot*.
- In the Display form, select *Violation Browser*. The Annotation Browser window opens in the layout. Select a violation and the violation marker is displayed on the layout.

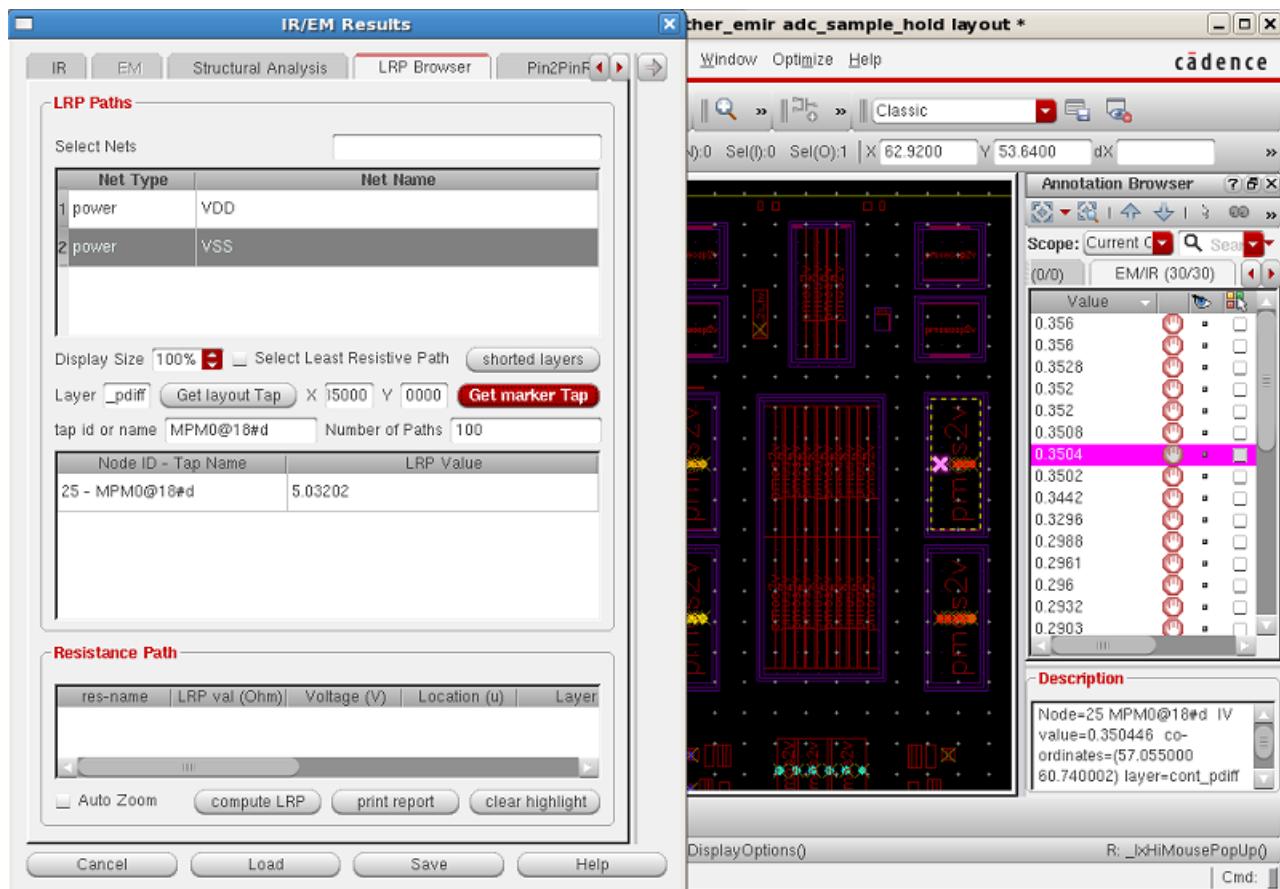
## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

- In the LRP Browser tab, click *Get Marker Tap*. The information for the tap – the x- and y- co-ordinates and the layer name – on the marker is populated in the LRP browser tab.
- Click the tap name in the LRP table and then click *Compute LRP* to compute the LRP value for the tap and to view the least-resistive path for the tap on the layout.

An example is shown below.

**Figure 7-36 Displaying LRP for the Tap on the Selected Marker**



## Supporting LRP Analysis in Power-Gated Designs

In Voltus-Fi-XL, you can view the LRP plot for power-gated designs. An always-ON net LRP plot shows the LRP for all tap-nodes in the always-ON net and for all nodes connected to the switched net.

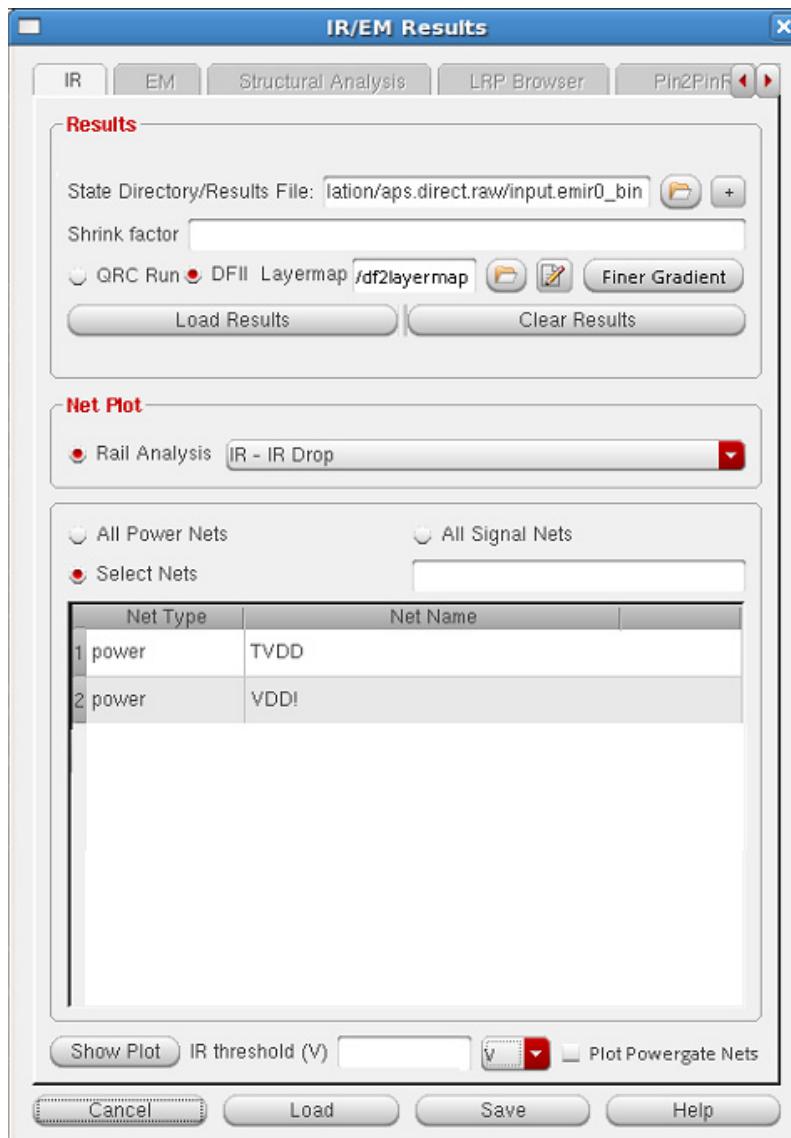
## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

The steps performed to select the LRP analysis type in the IR/EM Results form are the same as those described in the previous section. In addition, select the always-ON net. For example, the net TVDD, shown in below image is an always-ON net.

**Note:** The net, VDD!, shown in below image, is a switched net.

**Figure 7-37 Selecting the Always-On Net for Viewing the LRP**

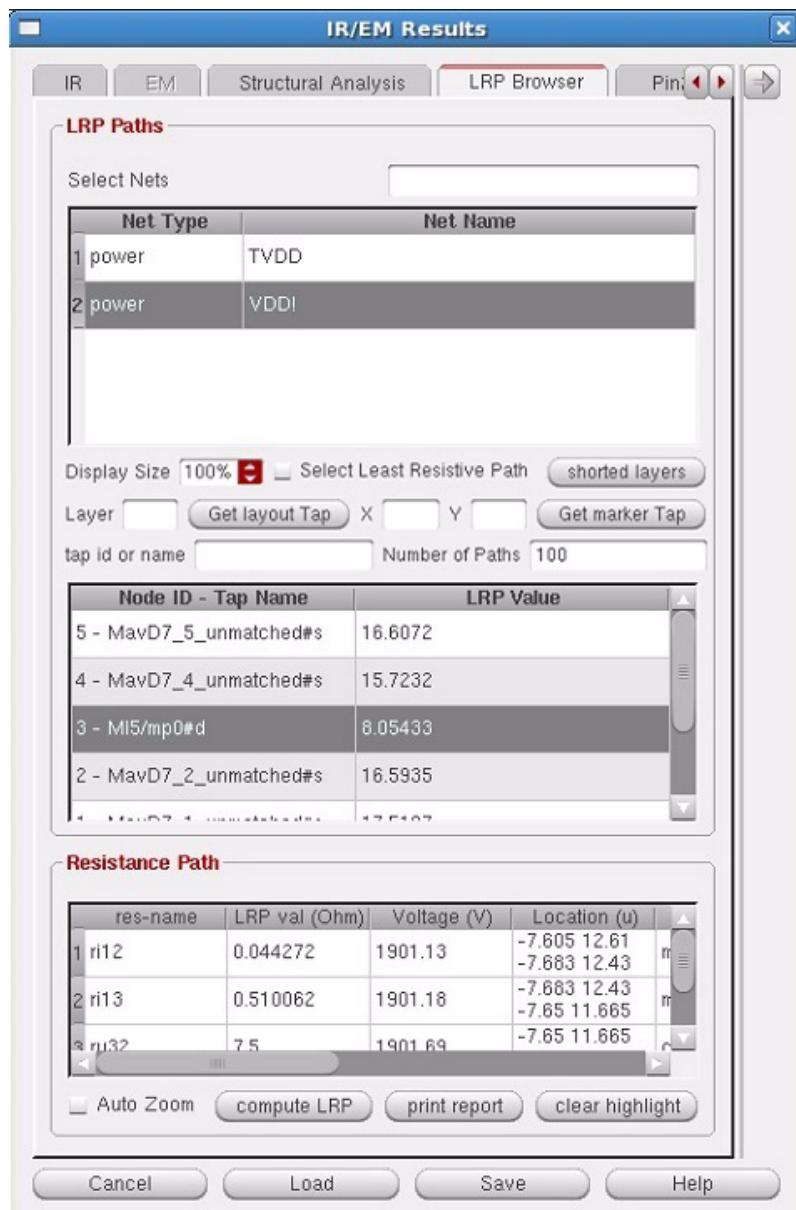


Select the LRP Browser Tab. This is shown below.

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

Figure 7-38 LRP Browser Tab for Power-Gated Designs



In the above form, the *LRP Paths* group box displays the LRP values for the tap nodes connected to the always-ON net. It also shows the LRP values for the nodes connected to the switched net.

The table in the *Resistance Path* group box shows the resistance for the node selected in the LRP Paths table above.

**Note:** The resistance value of the resistor, "r<sub>ON</sub>", is assumed to be zero ohms.

## LRP Reports

You can create RLRP reports in the batch mode by using the `print_rlp_report`. You can also generate and view the report by using the *print report* option in the LRP Browser tab in the GUI. You can specify the name of the report file and click *Save*. The report opens in the console. A sample report is shown in [Default Reports Generated for EMIR Analysis](#).

## Calculating Effective Resistance between any Two Nodes on a Net

In Voltus-Fi-XL, you can calculate the effective resistance between any two nodes (pins, tap nodes, or subnodes), either on the same net or on different layers of the same net. This feature is supported both in the GUI and in the batch mode.

This feature uses the DSPF file, used for Spectre simulation, and the SPGS feature of Spectre. It works only for IR drop analysis. For more information about the SPGS flow, see [Static Power Grid Solver](#) section.

In the batch mode, this feature is enabled using the `pin_2_pin_res` command. For details, see “[Batch Mode Execution](#)”.

In the GUI mode, this feature is enabled through the *Pin2PinR* tab provided in the IR/EM Results form. By default, this tab is disabled. To enable this tab, perform the following steps:

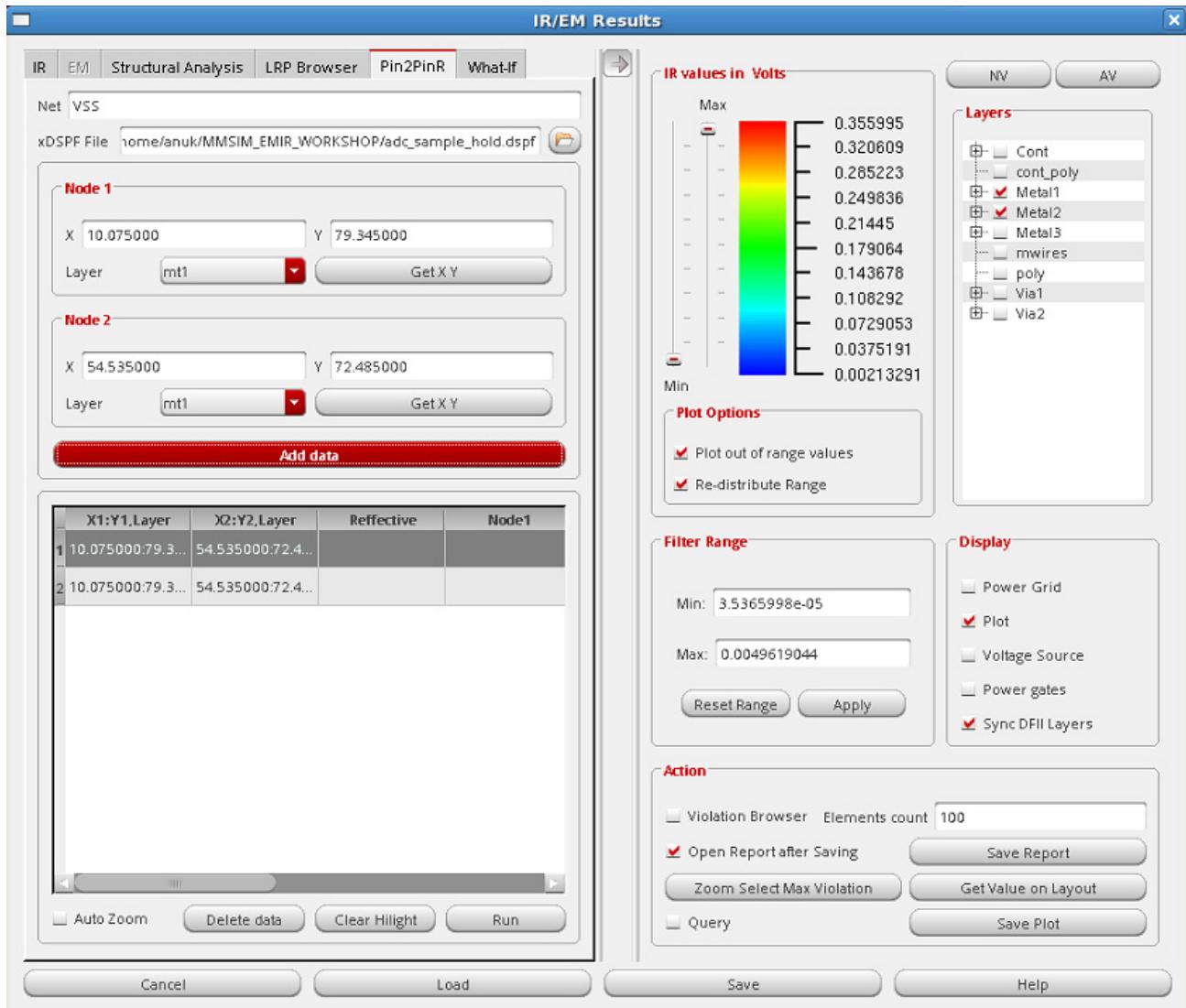
- Select the IR tab.
- Load the IR drop analysis results.
- Select a net and click Show Plot.

The *Pin2PinR* tab is enabled. This is shown below.

# Voltus-Fi Custom Power Integrity Solution User Guide

## IR Drop Analysis Results

**Figure 7-39 IR/EM Results form – Pin2PinR Tab**



To calculate effective resistance between two nodes on a net, provide the following information on this tab:

- On the *Pin2PinR* tab, the *Net* field is populated with the name of the net selected in the *IR* tab. You can change your selection of nets in the *Layers* group box.
- Specify the *xDSPF File*, which was used for Spectre simulation.
- In the *Node 1* group box, provide the following information for the first node for which resistance is to be calculated:

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

- ❑ Select the *Layer* on which you want to calculate the node's resistance from the drop-down list. By default, this field is populated with the name of the layer selected in the Display form. If multiple layers are selected in the Display form, all the layer names will appear in the drop-down list.
- ❑ Specify the *X*- and *Y*-co-ordinates for the node. You can type the co-ordinates in the *X* and *Y* fields. Alternatively, you can get the co-ordinates from the layout. For this, first click *Get X Y* and then select a point on the specified layer in the layout. The co-ordinates of this point are populated in the *X* and *Y* fields. If there is no node on the specified point, the tool will locate the node nearest to the specified point and populate its location in the *X* and *Y* fields.
- In the *Node 2* group box, provide the same information as above for the second node.  
**Note:** You can choose a node either on the same layer or on a different layer.
- Click *Add Data* to add the location of the two nodes in the table provided in the form. You can add multiple pairs of co-ordinates in the table.
- Click *Run* to calculate the effective resistance between the specified nodes. The table is updated with the effective resistance between the node pairs. This is shown in the figure below.  
**Note:** After the run, the effective resistance values are also dumped in the log file, *vfi.log*.

**Figure 7-40 Effective Resistance Information Displayed in the Table**

	X1:Y1,Layer	X2:Y2,Layer	Reflective	Node1	Node2	Node1 X:Y	Node2 X:Y
1	34.470000:76.045000,mt1-Metal1	24.235000:82.075000,mt2-Metal2	1.44033	VSS#362	VSS#288	36740:76190	26650:82180
2	43.730000:77.305000,mt1-Metal1	25.920000:81.935000,mt2-Metal2	1.4703	VSS#398	VSS#287	42388:76190	27350:82270

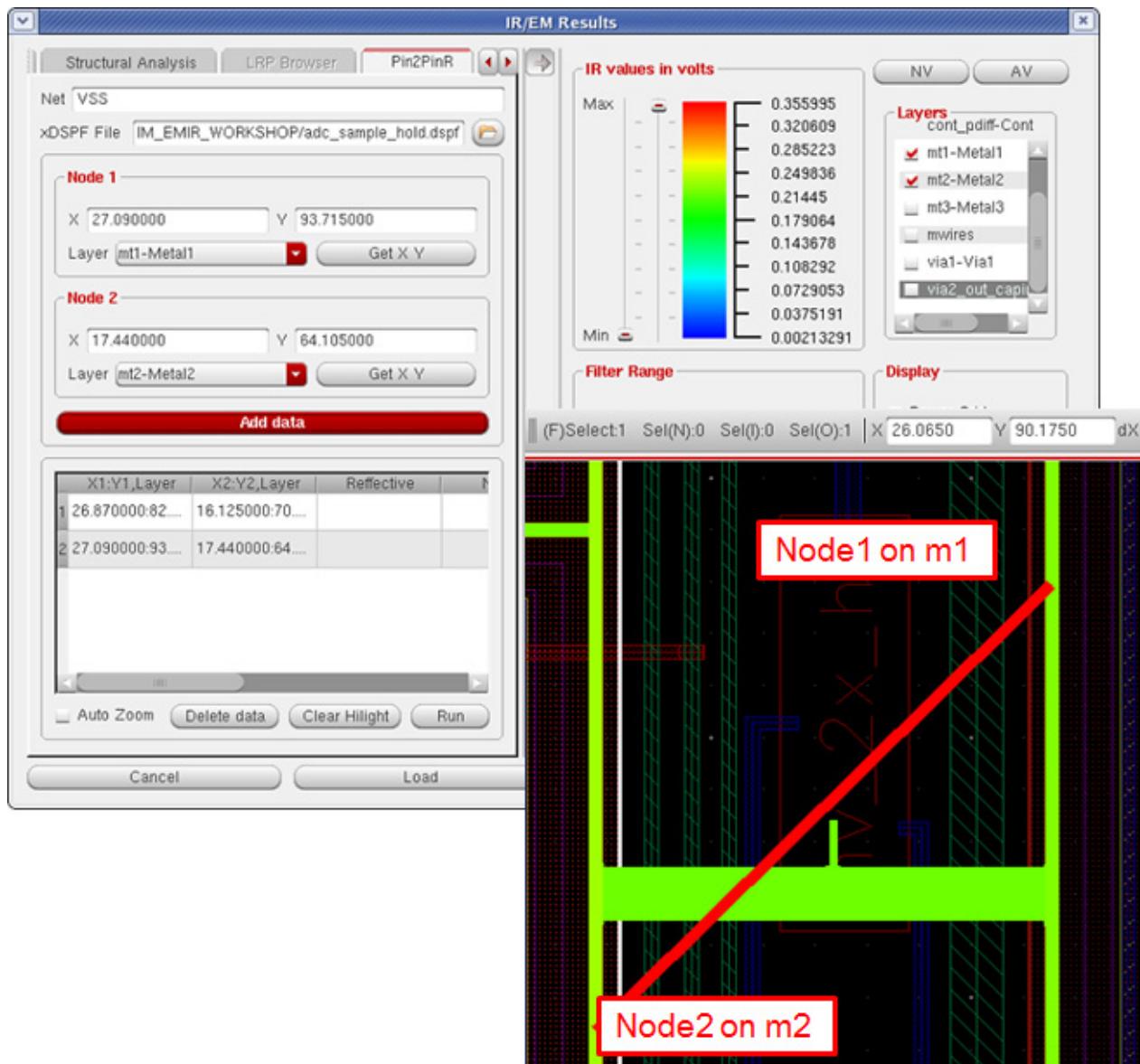
- To highlight the effective resistance path on the layout after the run, select the row for which you want to calculate effective resistance in the table.  
This is shown in below image. In the image, the effective resistance path between Node 1 specified on layer, m1 and Node 2 specified on layer, m2 is highlighted in red.

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

---

**Figure 7-41 Effective Resistance Path Highlighted on the Layout**



- Select *Auto Zoom* to automatically zoom into the highlighted path on the layout when a row is selected in the table.
- Click *Delete data* to delete the data added in any row in the table.
- Click *Clear Hilight* to clear the highlighted path on the layout.

## Generating EMIR Analysis Reports in Voltus-Fi-XL

The following topics are covered in this section:

- [Reports Generated using the EMIR Control File](#)
- [Default Reports Generated for EMIR Analysis](#)
- [Reports Generated for Multiple Bin Files from Different Testbenches in a Design](#)

### Reports Generated using the EMIR Control File

You can generate both textual and html reports for EMIR results in Voltus-Fi-XL from the simulation results by specifying the EMIR control file (`emir.conf`), the EMIR database file, and the output directory. In the EMIR control file, the `emirutil` command's `report` option is set to `text` or `html` depending upon the required output.

For example, `emirutil report=text` will generate a text report.

The syntax of the command to generate text or html reports for EMIR results in Voltus-Fi-XL is as follows:

```
vfibatch -control <confFileName> -db <binFileName> -outdir <outDirName>
```

For example,

```
vfibatch -control emir.conf -db xps.emir0_bin -outdir out1
```

OR

```
emirreport -control emir.conf -db xps.emir0_bin
```

**Note:** Specifying the output directory is optional.

The syntax of the command to generate reports for EMIR results using multiple bin files is as follows:

```
vfibatch -control emir.conf.report -db "binFileName1 binFileName2"
```



For details of the supported EMIR control file options that can be specified, see [EMIR Control File Options Supported in Voltus-Fi-XL](#) in the “Data Preparation” chapter. For details on how to specify variables using the `emirutil` command in the EMIR control file, see [Using the emirutil Command in the EMIR Control File](#) in the “Variables” chapter.

## Default Reports Generated for EMIR Analysis

By default, Voltus-Fi-XL generates the following reports for EMIR analysis:

- IR drop analysis reports with the extension, #.rpt\_ir
- EM analysis reports with the extension, #.rpt\_em
- Pin current reports with the extension, #.rpt\_pin
- Summary report (summary.rpt), which includes reports for all analyses (IR drop and EM) performed in the current run
- RLRP analysis report (RLRP.rpt), which includes the LRP values of the instances or tap-nodes of the specified net

**Note:** All default reports print the software version number in the header. You can view this in the sample reports shown in the subsequent sections.

### IR Report Format

In the IR drop analysis report file, information is provided in the following format for each net. The type of analysis is mentioned for each net.

```
----- "VDD" NET: Vref = 2.500000V -----
--  
max  
Resistor    ir value    Layer    X1     Y1     X2     Y2     Res    Length   Width    Ipeak
```

### Sample IR Drop Analysis Report

A part of sample IR drop analysis report is shown below.

---

```
VOLTAGE DROP RESULTS
VERSION = v06.17-e021_1
BINARY FILE      =
/home/anuk/MMSIM_EMIR_WORKSHOP/emir_simulation/aps.direct.raw/input.emir0_bin
RESULTS FILE CREATED = 2015-Dec-16 23:15:18 (2015-Dec-17 07:15:18 GMT)
USER SUPPLIED VALUES:
  RESULTS TYPE      = TRANSIENT
  TRANSIENT START    = 0
  TRANSIENT STOP     = 1.2e-07
  SIM TEMPERATURE   = 27 C
  AVERAGE           = nil
```

# Voltus-Fi Custom Power Integrity Solution User Guide

## IR Drop Analysis Results

---

```
PEAK          = t
SIGVMAX       = t
SIGVAVG       = nil
----- "VDD" NET: Vref = 2.5V -----
max
VOLTAGE-DROP   NETNAME      TIME        LAYER      X           Y
(V)            (s)          (um)        (um)
7.133m         MPM3@3#s    68.391n    cont        37.560     38.040
7.133m         MPM1@12#s   68.391n    mwires     37.560     38.040
7.132m         VDD#327    68.391n    metall1   37.560     47.920
7.132m         VDD#326    68.391n    metall1   37.560     45.320
7.126m         VDD#325    68.391n    metall1   37.560     40.380
```

## EM Report Format

In the EM analysis report file, information is provided in the following format for each net. The type of analysis is mentioned for each net.

```
----- NET "VDD" -----
avg
Pass/Fail %  Resistor   layer   Current   Width   PathLength I limit
X1    Y1    X2    Y2    J/JMAX   Res    ViaArea  No of needed width/#via  J limit
```

## Sample EM Analysis Report

The image below shows a part of the EM analysis report.

# Voltus-Fi Custom Power Integrity Solution User Guide

## IR Drop Analysis Results

### ELECTROMIGRATION ANALYSIS RESULTS

```
VERSION = v06.17-e021_1
BINARY FILE          = /home/anuk/MMSIM_EMIR_WORKSHOP/emir_simulation/aps.direct.raw/input.emir0_bin
RESULTS FILE CREATED = 2015-Dec-16 23:15:18 (2015-Dec-17 07:15:18 GMT)
USER SUPPLIED VALUES:
  RESULTS TYPE      =
  TRANSIENT START    = 0
  TRANSIENT STOP     = 1.2e-07
  SIM TEMPERATURE   = 27 C
  Tj                 = 110
  deltaT             = 5
  AVERAGE            = t
  AVERAGE ABS        = t
  AVERAGE POS        = nil
  ACPEAK              = nil
  CUSTOM              = nil
  RMS                 = t
  PEAK                = t

----- NET "VDD" -----
avg

Pass/Fail % Resistor layer Current      Width PathLength I limit X1      Y1      X2      Y2
          (mA)       (um)   (um)           (mA)
fail+249.1 rs396   vial  1.25894    NA    0       0.3606 30.09 71.54  30.095 71.54
fail+213.2 rs400   vial  1.1294   NA    0       0.3606 45.02 71.54  45.02  71.54
fail+133.7 ri176   mt1   0.617015  0.24  583.338  0.264  29.51 71.36  29.17  71.36
fail+97.6  ri173   mt1   0.521607  0.24  583.338  0.264  32.4   1.36  30.65  71.54
fail+52.2  ri172   mt1   0.401795  0.24  583.338  0.264  33.01 71.36  32.4   71.36
fail+49.4  rs397   vial  0.336716  NA    0       0.2254 33.63 99.64  33.63  99.64
```

The table below lists the analysis type and the corresponding section heading in the EM report file.

EM analysis type	Section heading in the EM Report File
em_jmax_dc_avg	avg
em_jmax_dc_peak	max
em_jmax_ac_rms	rms
em_jmax_ac_peak	max
em_jmax_abs_avg	Absavg

### ***Generating a Report for All Resistors***

You can generate a report to view the resistor information for all resistors in the design. To generate this report, use the `emirutil` command to set `reportAllResistor` to true in the emir configuration file.

```
emirutil reportAllResistor=true
```

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

---

For more information, see [EMIR Control File Options Supported in Voltus-Fi-XL](#) in the “Data Preparation” chapter.

A report, with the extension, “#.rpt\_all” is generated. The format of the report is the same as that of the “.#rpt\_em” report.

#### Pin Current Report Format

In the pin current report file, the `Imax`, `Iavg` and `Irms` currents for pins for each net are reported.

```
-----NET "VDD"-----
Name      Imax      Iavg      Irms
```

#### ***Sample Pin Current Report***

A part of a sample pin current report is shown below.

```
PIN CURRENT RESULTS

VERSION          = v15.10-dev
BINARY FILE      = /xps.raw/xps.emirtap.emir0_bin
RESULTS FILE CREATED = 2016-Jan-27 01:42:30 (2016-Jan-27 09:42:30)
USER SUPPLIED VALUES:
    RESULTS TYPE     = tran
    TRANSIENT START   = 0
    TRANSIENT STOP    = 1e-08

----- NET "VDD" -----

Name      Imax      Iavg      Irms
        (A)        (A)        (A)
VDD       22.0183 m    1.38584 m   2.75948 m

----- NET "VSS" -----

Name      Imax      Iavg      Irms
        (A)        (A)        (A)
VSS      -23.4348 m   -1.90594 m   3.04235 m
```

#### Summary Report Format

In the summary report, there are two tables of information for each net.

The first table provides, in its header, the name of the net, the simulation temperature, and the measurement window or the start and stop time for the analyses. For each net, the table

# Voltus-Fi Custom Power Integrity Solution User Guide

## IR Drop Analysis Results

---

lists the layer name and the corresponding maximum value of each analysis type for that layer. Information for different types of analyses is displayed in different columns.

The format of the first table is as follows:

---

Net name	GND
Sim temperature	125
Measurement window	0 - 5.9e-08

---

Layer	iavg
-------	------

---

The second table lists the layer names. For each layer and analysis type, there are two columns of information. The first column lists the number of resistors on which the analysis was run (`analysistype1_run_on`) and the second column lists the number of resistors that were ignored in the analysis (`analysistype1_num_skip`). The two columns are repeated for different types of analyses.

The format of the second table is as follows:

---

Layer	analysistype1_run_on	analysistype1_num_skip	analysistype2_run_on	analysistype2_num_skip
-------	----------------------	------------------------	----------------------	------------------------

---

## Sample Summary Report

A part of the summary report is shown below:

---

Net name	GND
Sim temperature	125
Measurement window	0 - 5.9e-08

---

Layer	vmax	irms	iavg	imax
metal1	0.0137511	0.0541541	1.92388	0.339854
metal2	0.0098085	0.541685	11.6608	0.478398
metal3	0.00587761	0.227175	1.9191	0.445334
metal4	0.00451982	0.109318	0.778421	0.11097
n_odcont	0.022525	0	0.425099	0
n_poly	0.00261004	0	0	0
n_polycont	0.00261902	0	2.87543e-08	0
p_odcont	0.00614363	0	0.146129	0
p_poly	0.00261004	0	0	0

---

# Voltus-Fi Custom Power Integrity Solution User Guide

## IR Drop Analysis Results

---

Layer	vmax_run_on	vmax_num_skip	irms_run_on	irms_num_skip
metal1	15537	0	15537	0
metal2	7253	0	7253	0
metal3	2253	0	2253	0
metal4	616	0	616	0
n_odcont	6526	0	0	6526
n_poly	222	0	0	222
n_polycont	80	0	0	80
p_odcont	3291	0	0	3291
p_poly	34	0	0	34

Net name	VDD
Sim temperature	125
Measurement window	0 - 5.9e-08

Layer	vmax	irms	iavg	imax
metal1	0.0288203	0.0742177	1.0235	0.621502
metal2	0.0141003	0.46877	10.0913	1.45709
metal3	0.00556588	0.191977	1.40789	0.184331
metal4	0.00391154	0.113201	0.562096	0.106243
n_odcont	0.00902663	0	0.0742855	0
n_poly	0.00260217	0	0	0

## RLRP Report Format

In the RLRP analysis report file, information is provided in the following format for each net.

RLRP paths from source-pin to following taps for net : netName

Tap name: : total resistance:

res-name: coord coord resistance: layer: Length: Width:

## Sample RLRP Report

RLRP paths from source-pin to following taps for net : TVDD

Tap name : MavD7\_4\_unmatched#d : total resistance : 15.5925

res-name r161 : coord (-3.520000 7.300000 ) coord (-3.550000 6.832500 ) resistance : 0.592453 layer : mt1 Length : 0.935 Width : 0.12

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

---

```
res-name ru76 : coord (-3.550000 6.832500 ) coord (-3.550000 6.832500 ) resistance : 15 layer : cont_pdiff Length : 0.12 Width : 0.12

Tap name : MavD7_1_unmatched#d : total resistance : 18.0641
res-name ri60 : coord (-3.520000 7.300000 ) coord (-3.522500 7.595000 ) resistance : 0.401827 layer : mt1 Length : 0.5895 Width : 0.12

res-name ri59 : coord (-3.522500 7.595000 ) coord (-3.535000 7.634000 ) resistance : 0.061827 layer : mt1 Length : 0.0775 Width : 0.12

res-name ri58 : coord (-3.535000 7.634000 ) coord (-4.160000 7.634000 ) resistance : 0.649321 layer : mt1 Length : 1.2495 Width : 0.155

res-name ri57 : coord (-4.160000 7.634000 ) coord (-4.190000 7.642500 ) resistance : 0.035386 layer : mt1 Length : 0.06 Width : 0.155

res-name ri54 : coord (-4.190000 7.642500 ) coord (-4.840000 6.832500 ) resistance : 1.91573 layer : mt1 Length : 2.94 Width : 0.12
```

## Reports Generated for Multiple Bin Files for Different Nets from the Same Testbench

This section describes the use model for analyzing IR/EM results when multiple emir bin files are available for different nets of the same testbench. This is useful in large designs, where simulation is run separately for different nets to improve the runtime.

In the batch mode, the following command is used to generate separate IR/EM reports for multiple bin files:

```
vfibatch -control emir.conf.report -db "xps.vdd/xps.raw/xps.emirtap.emir0_bin  
xps.vss/xps.raw/xps.emirtap.emir0_bin" -text abc
```

**Note:** The bin filenames are separated by a space as shown above. The `-text` option is used to specify the name of the result file that will include the results for both nets. In the above example, the following report files will be generated for IR and EM analysis, respectively:

```
abc.rpt_ir  
abc.rpt_em
```

## Reports Generated for Multiple Bin Files from Different Testbenches in a Design

This section describes the use model for analyzing IR/EM results when multiple emir bin files are available for the same design, but different testbenches. The following reports can be generated for such designs in Voltus-Fi-XL.

- **Simulation deck-based reports:** In this scenario, separate EM and IR reports are generated for each simulation deck or the bin file.
- **Consolidated report for all simulation decks:** In this scenario, consolidated EM and IR reports are generated for all simulation decks or emir bin files.

### **Creating the Simulation Deck-Based Reports**

This use model requires that the multiple `#.emir_bin` files should be available in the same directory. For example, `test.emir0_bin`, `test.emir1_bin`, `test.emir2_bin`, and so on.

In the batch mode, the following command is used to generate separate IR/EM reports for multiple bin files:

```
vfibatch -control conf file name -db_path  
path_of_directory_containing_emir_bin_files -outdir outDirName
```

For example,

```
vfibatch -control emir.conf -db_path test/test.raw -outdir out
```

This command will generate separate IR/EM reports for each emir bin file.

For example, if there are two bin files, `test.emir0_bin` and `test.emir1_bin`, then the following reports will be generated:

- `out/test.rpt_em` and `out/test.rpt_ir` will be generated corresponding to the `test.emir0_bin` file
- `out/test.rpt_em1` and `out/test.rpt_ir1` will be generated corresponding to `test.emir1_bin` file

**Note:** The format of these reports is the same as that of the default IR/EM reports generated in case of a single `emir_bin` file run.

### **Consolidated Reports for All Simulation Decks**

A single consolidated EM report and IR report will be generated comprising results from all simulation decks (`#.emir_bin` files). To enable the generation of consolidated reports, set the following:

- In the GUI, open the Variables form available in the EM tab of the IR/EM Results form, and set the EM variable, `consolidatedReport`, to true. By default, this variable is set to false.

OR

- In the batch mode, set the following in the `emir.conf` file:

```
emirutil consolidatedReport=true
```

**Note:** The `vfibatch` command having the option, "`-db_path/-db_multi_path`" with `emir` variable "consolidatedReport" set to "true" in the `emir` configuration file will generate the consolidated reports for both, EM and IR analyses.

By default, when the `consolidatedReport` option is set to true, the `worstResult` option is also set to true, which means only the worst violation will be reported.

```
emirutil consolidatedReport=true  
emirutil worstResult=true
```

To view all the results in the consolidated report, set the `worstResult` option to false.

For details of the above options, see [EMIR Control File Options Supported in Voltus-Fi-XL](#) in the “Data Preparation” chapter.

There are two possible scenarios in the creation of consolidated reports:

- Scenario 1, when all `emir` bin files are in the same directory
- Scenario 2, when the `emir` bin files are in different directories

These scenarios are detailed below.

### ***Scenario 1: The emir bin Files are in the Same Directory***

In this scenario, there are multiple `emir` bin files in the same directory. For example, `test.emir0_bin`, `test.emir1_bin`, `test.emir2_bin`, and so on.

To generate a consolidated report in this scenario, use the following command in the batch mode:

```
vfibatch -control conf_file_name -db_path  
path_of_directory_containing_emir_bin_files -outdir outDirName
```

**Note:** The above command uses the `-db_path` option to specify the path of the directory containing all the bin files.

For example,

```
vfibatch -control emir.conf -db_path test/test.raw -outdir out
```

***Scenario 2: The emir bin Files are in Different Directories***

To generate a consolidated report in this scenario, use the following command in the batch mode:

```
vfibatch -control conf file name -db_multi_path  
name_of_file_containing_paths_to_emir_bin_files -outdir  
outDirName
```

**Note:** The above command uses the `-db_multi_path` option to specify the name of the file that contains the paths to the emir bin files containing the simulation results of different testbenches of the same design but in different directories.

For example,

```
vfibatch -control emir.conf -db_multi_path emir_bin_files -outdir  
out
```

Where, the file, “*emir\_bin\_files*” contains the following information:

```
test/test.raw/test.emir0_bin  
test1/test1.raw/test1.emir0_bin  
test2/test2/raw/test2/emir0_bin
```

A part of a sample consolidated report is shown below.

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

**Figure 7-42 Consolidated EM Report Sample**

```
pass-30.0 ra2 via2 8.41499e-07 NA 0.55 2.2 4.78 9.897 4.78 9.834 1.55227e-07 1500 33000 1 1
0.0666667 test.emir0_bin

pass-100.0 ra2 via2 5.41499e-07 NA 0.55 2.2 4.78 9.897 4.78 9.834 1.55227e-07 1500 33000 1 1
0.0666667 test.emir1_bin

pass-100.0 ra2 via2 3.41499e-07 NA 0.55 2.2 4.78 9.897 4.78 9.834 1.55227e-07 1500 33000 1 1
0.0666667 test.emir2_bin

----- NET "VSS" -----
avg

Pass/Fail % Resistor layer Current Width PathLength I limit X1 Y1 X2 Y2 J/JMAX Res ViaArea No of
needed width/#via J limit emir_bin

(mA) (um) (um) (mA) (nm^2) vias (um/#) (A/um)

fail+40.0 ra4 via2 3.41 NA 0.55 2.2 4.78 9.897 4.78 9.834 1.55227e-07 1500 33000 1 1
0.0666667 test.emir1_bin

fail+30.0 ra4 via2 3.00 NA 0.55 2.2 4.78 9.897 4.78 9.834 1.55227e-07 1500 33000 1 1
0.0666667 test.emir0_bin

fail+20.0 ra4 via2 2.70 NA 0.55 2.2 4.78 9.897 4.78 9.834 1.55227e-07 1500 33000 1 1
0.0666667 test.emir2_bin

pass-30.0 ra5 via2 8.41499e-07 NA 0.55 2.2 4.78 9.897 4.78 9.834 1.55227e-07 1500 33000 1 1
0.0666667 test.emir0_bin

pass-100.0 ra5 via2 5.41499e-07 NA 0.55 2.2 4.78 9.897 4.78 9.834 1.55227e-07 1500 33000 1 1
0.0666667 test.emir1_bin

pass-100.0 ra5 via2 3.41499e-07 NA 0.55 2.2 4.78 9.897 4.78 9.834 1.55227e-07 1500 33000 1 1
0.0666667 test.emir2_bin
```

## Voltus-Fi Custom Power Integrity Solution User Guide

### IR Drop Analysis Results

Figure 7-43 Consolidated IR Report Sample

```
VOLTAGE DROP RESULTS

VERSION = v15.10-dev.
RESULTS FILE CREATED = 2015-Oct-28 01:45:33 (2015-Oct-28 08:45:33 GMT)

----- "VG" NET: Vref = 0.000000V -----
max

Resistor    ir value    Layer   X1     Y1     X2     Y2     Res    Length  Width  Ipeak  emir_bin
              (u)        (u)      (mA)
ra3         0.00402345  m1      4.78   9.834   4.78   9.734  2000   0.1    0.02   0.000924894 test.emir2_bin
ra3         0.00302345  m1      4.78   9.834   4.78   9.734  2000   0.1    0.02   0.000924894 test.emir1_bin
ra3         0.00202345  m1      4.78   9.834   4.78   9.734  2000   0.1    0.02   0.000924894 test.emir0_bin

ra2         0.00223525  via2    4.78   9.897   4.78   9.834  1500   NA     NA     0.000924894 test.emir0_bin
ra2         0.00203525  via2    4.78   9.897   4.78   9.834  1500   NA     NA     0.000924894 test.emir1_bin
ra2         0.00193525  via2    4.78   9.897   4.78   9.834  1500   NA     NA     0.000924894 test.emir2_bin

----- "VDD" NET: Vref = 0.900000V -----
max

Resistor    ir value    Layer   X1     Y1     X2     Y2     Res    Length  Width  Ipeak  emir_bin
              (u)        (u)      (mA)
rx13        0.0965851   m1      4.78   9.834   4.78   9.734  2000   0.1    0.02   0.0214634 test.emir2_bin
rx13        0.0865851   m1      4.78   9.834   4.78   9.734  2000   0.1    0.02   0.0214634 test.emir0_bin
rx13        0.0765851   m1      4.78   9.834   4.78   9.734  2000   0.1    0.02   0.0214634 test.emir1_bin
```

In the above report, the IR results for the resistors "ra3" and "ra2" for net VG, and resistor, "rx13" for net, VDD from all emir\_bin files are reported. By default, for consolidated reports, the emirutil variable, worstResult is set to true. Therefore, the report only includes the worst result for each resistor, which is the first row of information for each resistor shown in the above report.

## **Voltus-Fi Custom Power Integrity Solution User Guide**

### IR Drop Analysis Results

---

However, for the above report, to view all results, the `worstResult` variable is set to `false`. In this case, the IR results for the resistors are sorted so that the worst case is reported first and best case is reported last.

**Voltus-Fi Custom Power Integrity Solution User Guide**  
IR Drop Analysis Results

---

## **EM Analysis Results**

---

- [Overview](#) on page 198
- [Types of EM Analysis](#) on page 198
- [Design Resistor EM Current Analysis](#) on page 200
- [Viewing the EM Analysis Results](#) on page 200
- [Displaying Current Directions for EM Plots](#) on page 209
- [Displaying Current Directions for EM Plots](#) on page 209
- [Reporting the Direction of Currents in the EM Result File](#) on page 210
- [Generating EM Reports in Voltus-Fi-XL](#) on page 211
- [Types of EM Analysis Plots](#) on page 211
  - [Tap Current Plots](#) on page 212
  - [Resistor Current Plots](#) on page 213
  - [RJ – Current Density – max](#) on page 215
  - [JAVG – Current Density – avg](#) on page 217
  - [JABSAVG – Current Density – avgabs](#) on page 219
  - [JACRMS – Current Density – rms](#) on page 221

## Overview

Voltus-Fi-XL uses the simulation database generated by Spectre® APS/XPS simulators and displays the results of the electromigration (EM) analysis on the Virtuoso layout.

It also generates text and html reports of the analyses and lets you query the analyses results to view specific violations on the layout. This is used to debug the high EM violation regions in the design.

Batch mode support is provided for loading EM analysis results and generating reports. For details of the batch commands used to load and print EM analysis result reports, see [Batch Commands for EM Reports](#) in the “Batch Mode Execution” chapter.

## Types of EM Analysis

This section details the analysis types that are available in Voltus-Fi-XL.

- **Average (avg)**—Calculates the current density violations based on the average value of the DC current for metal lines, vias, and contacts.

$$I_{avg} = \left[ \left( \int_0^{\tau} I(t) dt \right) / \tau \right]$$

where “T” is time and “ $I(\tau)$ ” is value of current.

- **RMS**—Calculates the current density violations based on the root mean square (RMS) of the DC current for metal lines, vias, and contacts.

$$I_{rms} = \left[ \left( \int_0^{\tau} I(t)^2 dt \right) / \tau \right]^{1/2}$$

where “T” is time and “ $I(\tau)$ ” is value of current.

- **Peak (max)**—Calculates the current density violations based on the maximum DC current for metal lines, vias, and contacts.

$I_{peak} = I_{peak\_dc} = \text{maximum value of } I$

## Voltus-Fi Custom Power Integrity Solution User Guide

### EM Analysis Results

---

- **AC-Peak**—Calculates the current density violations based on the peak AC current for metal lines, vias, and contacts. It is applied to periodic AC or pulsed DC signals.

**Note:** To perform AC-Peak analysis in Voltus-Fi-XL using the ADE L flow, ensure that while setting up simulation, you have enabled the *EM Current Analysis* options *max* and *avg* in the *Analysis* tab of the EM/IR Analysis Setup form. For more information, see Setting up EMIR Analysis for Voltus-Fi-XL in the ADE L Window in the “Getting Started” chapter.

$$I_{peak\_ac} = I_{peak}$$

**Note:** The above values are from simulation.

$$I_{peak\_ac(limit)} = I_{peak\_dc(limit)} / (r)^{1/2}$$

where the  $I_{peak\_dc(limit)}$  is specified in the ICT file

and duty ratio “r” is:

$$r = I_{abs-avg} / I_{peak}, \text{ by default ("false")}$$

**Note:** The  $I_{abs-avg}$  and  $I_{peak}$  values are from simulation.

And

$r = T_d / T_{total}$ , when the `dynamicACPeak` variable is set as either “true” or “multiplePeak”.

$T_d$  = the time duration in micro second or the total “On Time” period, where  $\text{abs}(I) > \text{max}(\text{abs}(I)) / 2.0$  during transient analysis

$T_d$  = the time duration of maximum peak, when `dynamicACPeak` is set to “true”

$T_d$  = sum of time durations of different peaks,  $T_{d1} + T_{d2} + T_{dN} \dots$  (when `dynamicACPeak` is set to “multiplePeak”)

$T_{total}$  = total transient time

**Note:** The software replaces the value of “r” with the value of `applyRThreshold` if the value of “r” is < than that of `applyRThreshold`. This is because a small “r” value results in an unreasonable increase in the  $I_{peak\_ac}$  limit. To avoid this scenario, use the `applyR` keyword to reset the value of “r” in the ICT file or change the value of the `applyRThreshold` variable, either in the Variables form or in the batch mode.

- **Avg-Abs (avgabs)**—Calculates the current density violations based on the average of the absolute current for metal lines, vias, and contacts.

$$I_{abs-avg} = \left[ \left( \int_0^{\tau} |I(t)| dt \right) / \tau \right]$$

where “ $\tau$ ” is time and “ $I(t)$ ” is value of current.

## Design Resistor EM Current Analysis

By default, the EM current analysis is only performed on parasitic resistors defined in the RC section of the xDSPIF file. If required, EM analysis for design resistors in the xDSPIF instance section can be enabled. This analysis can be applied to primitive resistor elements, or resistors defined with subckt definitions.

```
net design_res_models=[resistor] analysis=[iavg] (primitive resistor)
net design_res_models=[name=rhim] analysis=[iavg] (resistor subckt rhim)
```

For the EM current analysis the parameters \$l, \$w, and \$lvl are expected to be defined for primitive resistor calls, and l and w for subckt resistor calls in the xDSPIF instance section. If parameter names are differently defined in the xDSPIF file, the mapping can be performed in the config file statement.

```
net design_res_models=[name=rhim_m l=lr w=wr] analysis=[iavg]
```

Layer information, and units for l/w can be defined in the EMIR config file statement.

```
net design_res_models=[name=rhim_m layer=mt1 unit=1] analysis=[iavg]
```

The design resistor related EM results are reported in the `design.rpt_em` file.

## Viewing the EM Analysis Results

Before plotting EMIR results, ensure that the `emir` mode was enabled while performing the simulation. Also, check that the simulation result file is available in the output directory of the simulation results. The naming convention of the simulation result file is, `*.emir#_bin`. For example, your result file could be named, `xps.emir0_bin`.

Follow these steps to view the EM analysis results.

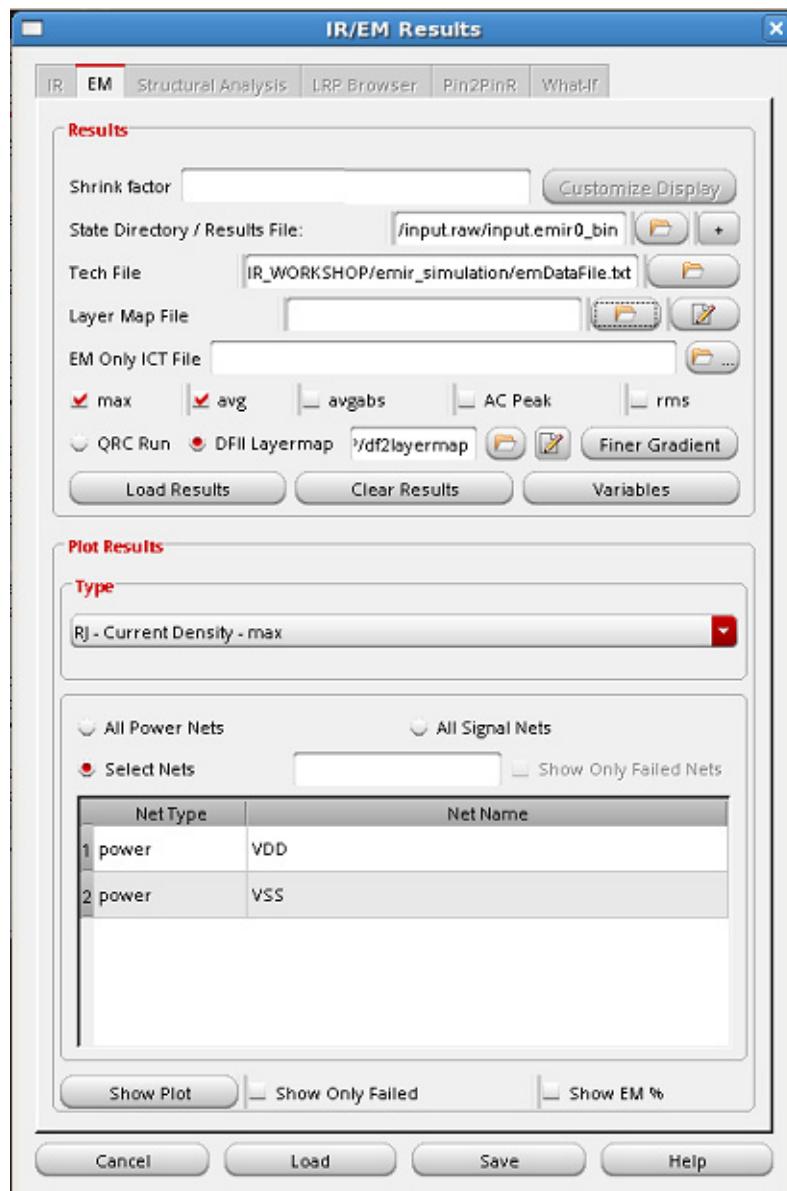
## Voltus-Fi Custom Power Integrity Solution User Guide

### EM Analysis Results

- In the Voltus-Fi-XL console, choose *IR/EM Analysis – Rail Analysis Results*. The IR/EM Results form opens. Click the EM tab to plot the EM analysis results. This form is shown below.

**Note:** Structural analysis is not enabled with EM analysis results.

**Figure 8-1 IR/EM Results Form – EM Tab**



- In the *Results* group box, specify the shrink factor, if any, by which the xDSPF was shrunk in the *Shrink factor* field.

- Click *Customize Display* to select the nets for which you want to view the display. The Customize Display pop-up window opens. You can only customize the display for selected nets before the results are loaded. The *Customize Display* button is disabled once the results are loaded.
- Specify the *State Directory / Results File*, which is a file that stores the results of the APS/XPS simulation. The naming convention of the result file is \*.emir#\_bin.
- Select the + button to specify multiple result files.
- Specify the technology file in the *Tech File* field. The following technology files are supported in the Voltus-Fi-XL flow:
  - The interconnect technology (ICT) file
  - The QRC technology file (qrcTechFile)
  - The EM data file (emDataFile)

**Note:** For both the ICT file and qrcTechfile flow, Voltus-Fi-XL supports the use of encrypted files.

- Specify the *Layer Map File*. This file is used to map the layer names in the simulation database to the layer names in the technology file specified in the *Tech File* field. This is optional and is only required if the layer names in the simulation database are different from the ones in the technology file. You can also create a new layer map file or edit an existing file, using the edit button provided next to the field. For more information about the layer map file, see [Layer Map File](#) in the “File Formats” chapter.

**Note:** If the above layer map file is not provided and you click *Load Results*, a pop-up window with the message, “Map file not specified. EM analysis will run without layer mapping”, opens. Click *OK* to proceed.

- Specify the *EM Only ICT File*, which includes information about process and EM models to be used for EM analysis. This is an optional file that can be provided with the qrcTechFile flow. When provided, the `process` and `em_model` information will be picked up from the specified `EMOnlyICTFile`.
- Specify either the *QRC Run* or the *DFII Layermap*. When either of these options is specified, the plots displayed on the layout show solid shape highlighting. For more information, see [Solid Shape Highlighting](#).
  - Select *QRC Run*. The QRC Run Location form opens. In this form, specify the QRC run details, which include the path to the *QRC Run Directory* and the *QRC Run Name*. This form is shown below:

**Figure 8-2 Specifying the QRC Run**



The display now generated will be based on the shape database generated by QRC. For more information about the setup requirements for the QRC Run flow, see [Solid Shape Highlighting](#).

- Specify the *DFII Layermap*. This layer map file is the APS/XPS-to-DFII layer map file. When this file is specified, the plots displayed on the layout show solid shape highlighting. You can also create a new layer map file or edit an existing file, using the edit button provided next to the field. This layer map file is the same file that is used in structural analysis. For more information about the file format and description, see [DFII Layer Map File](#) in the “File Formats” chapter.

You can also specify this file by using the `vfiDfiiLayerMapFile` environment variable either in the `.cdsenv` or the `.cdsinit` file. For more information, see [Loading the DFII Layer Map File by Default](#) in the “Data Preparation” chapter.

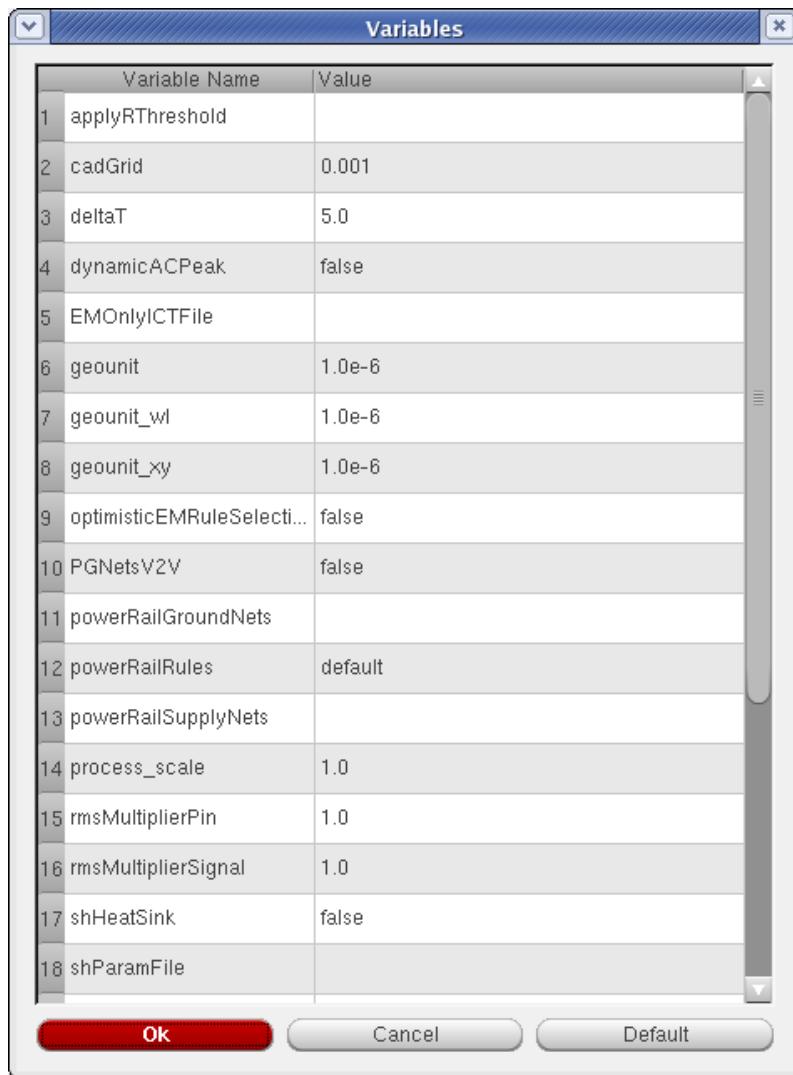
**Note:** If you do not specify this file and click *Load Results*, a pop-up-window opens to verify that you want to proceed without the map file. If you click *Yes*, the plots are drawn using stick display.

- For the DFII layermap flow, click *Finer Gradient* to view the finer gradient for the selected layers. When you click this button, the *Finer Gradient Layer Selection* pop-up window opens. In this window, select the PGDB layers and click *OK*. For details, see [Displaying Finer Color Gradient for Selected Layers in IR and EM Plots](#).

**Note:** Perform this step before loading the EM Results. After loading the results, when you click *Show Plot*, you can see the finer gradient for the selected layers.

- Click *Variables* to open the Variables form. In this form, you can view and/or edit the variables that are being used for EM analysis. Double-click the value of any variable to edit it. After updating the values, click *OK*. To restore the default values, click *Default*. This form is shown below.

**Figure 8-3 The Variables Form**



You can also set above variables in the batch mode by using the [set\\_variable](#) command.

**Note:** For descriptions of the variables that can be set in the above form, see "[Variables](#)".

- Specify the *Type* of EM analysis for which you want to plot results. The available options are *max*, *avg*, *avgabs*, *AC Peak*, and *rms*. You can select more than one analysis type at a time. By default, *max* and *avg* are selected.
  - max**: calculates the current density violations based on the maximum DC current
  - avg**: calculates the current density violations based on the average value of the DC current

## Voltus-Fi Custom Power Integrity Solution User Guide

### EM Analysis Results

---

- avgabs**: calculates the current density violations based on the average value of absolute current
- AC Peak**: calculates the current density violations based on peak AC current  
**Note:** This analysis type is not supported in the `emDataFile` flow.
- rms**: calculates the current density violations based on the root mean square (RMS) value of the AC current

For details of the above analysis types, see [Types of EM Analysis](#).

**Note:** When you specify any of the above analysis types, ensure that the rules for the selected analysis type are available in the technology file specified in the *Tech File* field. If the EM rules are not available, the analysis result file will be empty.

- Click *Load Results* to load the results of the EM Analysis. Once the results are loaded, the drop-down list in the *Plot Results* group box is enabled. The list of plots corresponds to the selected analyses types. In addition to the plots specific to the chosen analysis type, Peak TC and RC plot types are available for all EM analyses.

- TC - Peak Tap Current**: analyzes and reports peak tap currents
- RC - Peak Resistor Current**: analyzes and reports peak resistor currents

You can select from the available plot types. For more information about the different plots, see [Types of EM Analysis Plots](#).

- Click *Clear Results* to clear the results.
- Click *All Power Nets* to view EM plots for all power nets.
- Click *All Signal Nets* to view EM plots for all signal nets.
- Click *Select Nets* to specify the nets for which you want to view EM plots. You can select one or more nets from the list of nets provided in the list box. The list box lists *Net Type* and the corresponding *Net Name*.

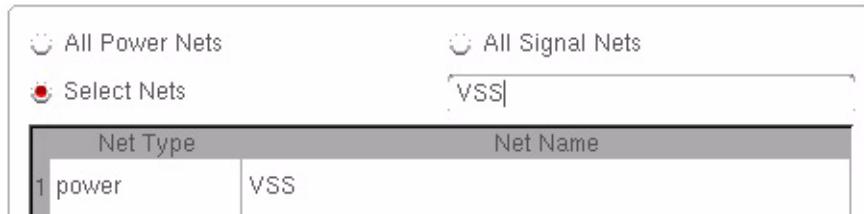
You can search for specific nets for which you want to plot results by typing the net name in the text field provided in the form. The list box is updated to display information about the specified net.

## Voltus-Fi Custom Power Integrity Solution User Guide

### EM Analysis Results

---

**Note:** The net name is case-sensitive. This is shown in the image below.

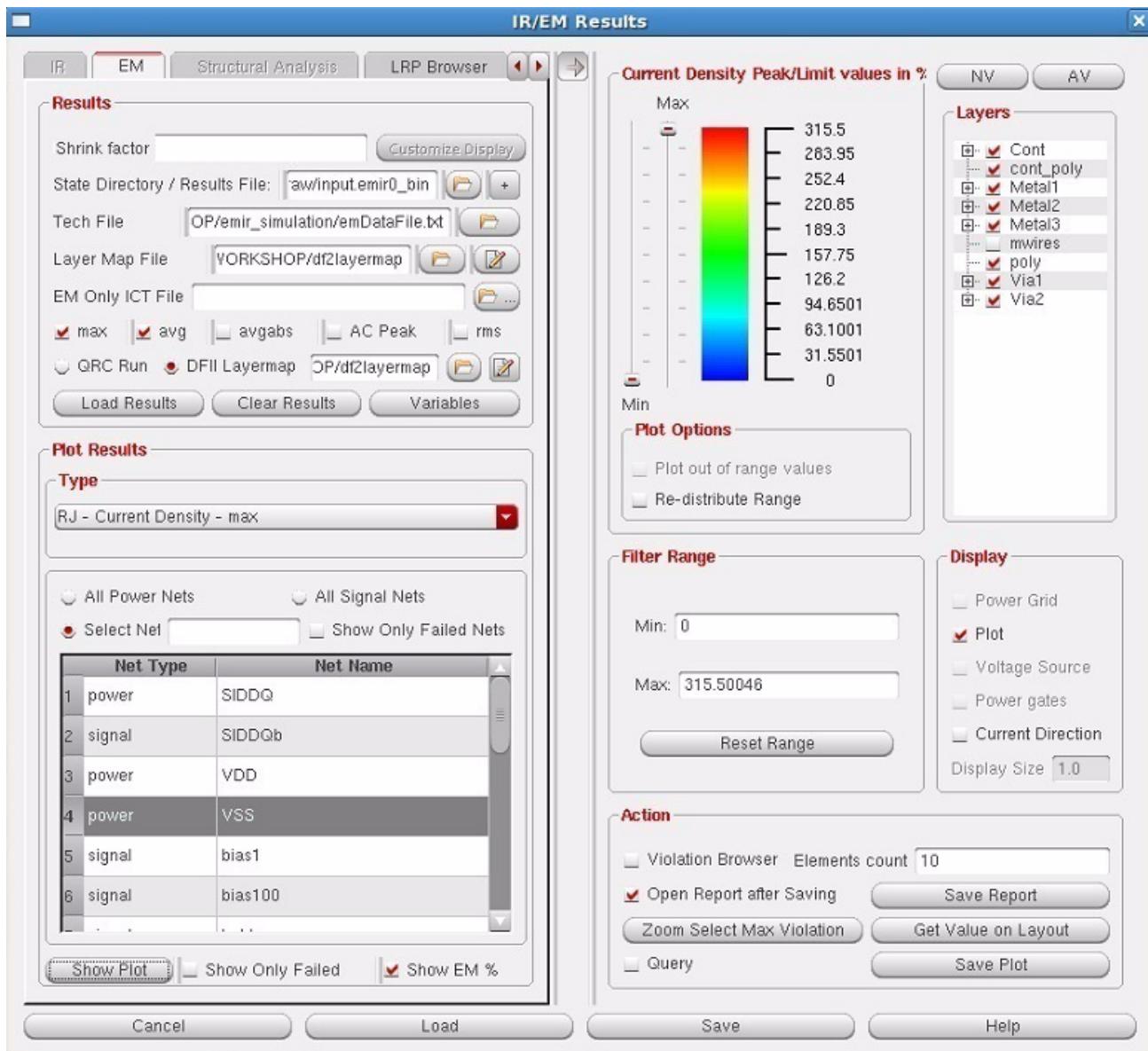


- Click *Show Only Failed* to plot results only for those nets that failed the EM check. Selecting this option will set the minimum value for the slider to 1.
- Click *Show EM %* to display the EM violations as percentages. The *Min - Max* slider range displays the EM violations as a percentage of current density and current density limit ( $J/J_{limit}$ ) values. A percentage of 100 and above indicates EM violations. This is shown in image below.

# Voltus-Fi Custom Power Integrity Solution User Guide

## EM Analysis Results

**Figure 8-4 Viewing EM Violations as Percentages**



- Click *Show Plot* to view the plot on the layout. When you click *Show Plot*, the IR/EM Results form expands to show the available plot display options. In this form, you can customize the EM analysis plot displays. You can use these options to customize the displayed plots.

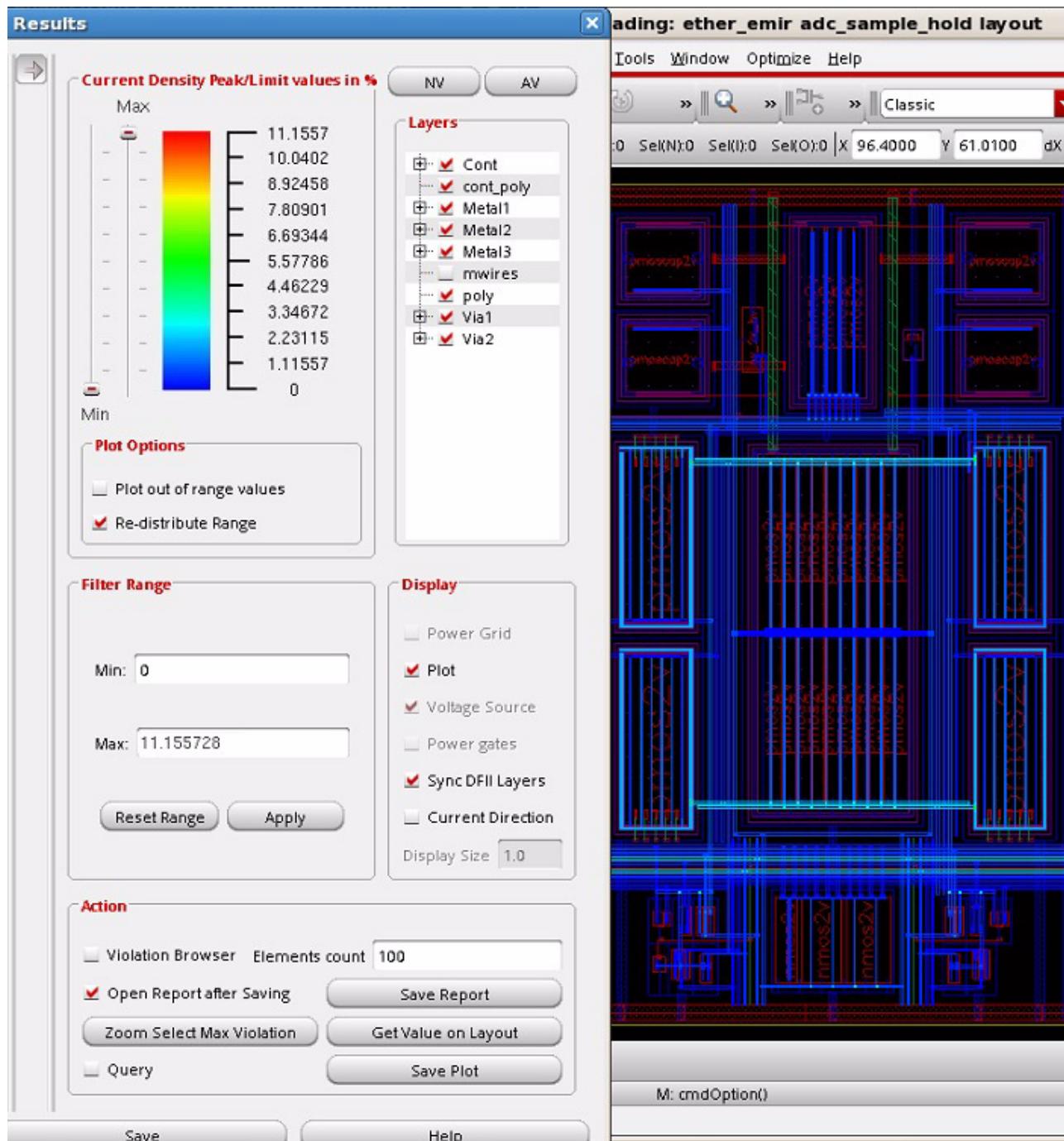
These options are explained in detail in [Specifying the Display Options for IR/EM Plots](#) section in the “IR Drop Analysis Results” chapter.

# Voltus-Fi Custom Power Integrity Solution User Guide

## EM Analysis Results

The figure below shows the display options in the expanded IR/EM Results form and the EM results plot for all signal nets.

**Figure 8-5 Viewing the EM Plot for All Signal Nets**



## Displaying Current Directions for EM Plots

When you load EM results and click *Show Plot*, the expanded IR/EM Results form opens. In this form, there are options provided for displaying the current direction between nodes and for customizing the size of the arrows showing the direction of the flow of current. In the image above, the *Display* group box shows the following options:

- *Current Direction*: Select this option to enable the display of current directions on the layout for any EM plot type.
- *Display Size*: Specify the size of the symbols that are used for displaying the current directions. You can increase or decrease the size of the symbols based on your requirements. The acceptable range is between 0 and 5. If you specify a value greater than 5, the software will set the display size to the maximum possible size, which is 5.

When the *Current Direction* option is enabled, the plot on the layout shows white highlights for the current direction flow of selected nets. The highlight appears on top of the plot highlights. The current direction is available for all the selected nets for the selected plot type. The current direction for each layer can be turned on or off as per the Layer selection specified in the *Layers* group box.

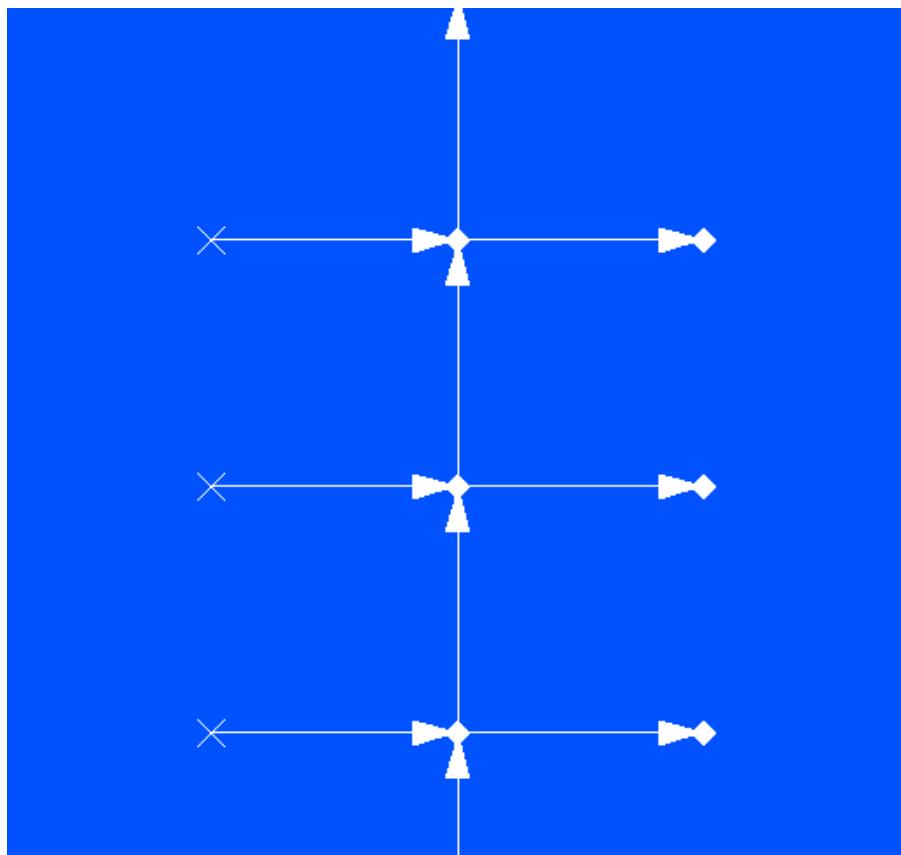
**Note:** The *Min/Max* slider range setting is not honored by the current direction plot. The plot displays current directions for all nodes.

The symbols used to show the direction of currents on the layout are as follows:

- For metal layers, arrows are used to indicate the flow of current in the directions; "E", "W", "N", "S", "NE" and so on. For example, if the direction of current is from West to East, the arrow will be horizontal and will point from left-to-right while if the direction of current is South to North, the arrow will be vertical and will point from bottom-to-top.
- For vias, the direction of the current is from layer -> to layer. A cross symbol (X) is used to show currents flowing down, while a diamond-shaped dot shows currents flowing up.

This is shown in the image below.

**Figure 8-6 Current Flow Direction Representation on the Layout**



## Reporting the Direction of Currents in the EM Result File

You can print information about the direction of current flow for each layer in the EM result file. This current direction is reported in the “Current Direction” column in the report.

You can specify the reporting of current direction in the batch mode in either of the following ways:

- Specify the following in the `vfibatch` command file:

```
set_variable idirn true
```

- Specify the following in the EMIR control file:

```
emirutil idirn=true
```

**Note:** For more information on how to set variables, see “[Variables](#)”. For more information about the available EMIR control file options, see [EMIR Control File Options Supported in Voltus-Fi-XL](#) in the “Data Preparation” chapter.

## Voltus-Fi Custom Power Integrity Solution User Guide

### EM Analysis Results

The direction of current is printed in the following manner:

- For metal layers, the direction of current is printed as “E”, “W”, “N”, “S”, “NE”, and so on. For example, if the report prints the direction of current as “E”, it means the current is flowing from West to East. If the report prints the direction of current as “N”, it means the current is flowing from South to North.
- For vias, the direction of current is printed as “from layer -> to layer”. For example, in below figure, current direction for via2 is metal3->metal2.

### Sample EM Analysis Results File

```
ELECTROMIGRATION ANALYSIS RESULTS
VERSION = 1
BINARY FILE      = ./emir_simulation/aps.direct.raw/input.emir0.bin
RESULTS FILE CREATED = 2013-Nov-18 01:17:39 (2013-Nov-18 09:17:39 GMT)
USER SUPPLIED VALUES:
  RESULTS TYPE      = TRANSIENT
  TRANSIENT START    = 0
  TRANSIENT STOP     = 1.2e-07
  SIM TEMPERATURE   = 27 C
  AVERAGE           = n/a
  AVERAGE ABS        = n/a
  AVERAGE POS        = n/a
  ACPEAK            = n/a
  CUSTOM             = n/a
  RMS                = n/a
  PEAK               = t

----- NET "bias100" -----
max
Pass/Fail % Resistor layer Current Width PathLength I limit X1 Y1 X2 Y2 J/JMAX Res ViaArea No of vias needed width/#via CurrentDirection
pass-75.5 rg1974 metal3 1.61816 2 103.149 6.6 0.25 1.75 0 0 0.245176 0.008084 NA NA 0.490352 S
pass-79.9 rg1973 metal3 1.32752 2 103.149 6.6 0.55 1.75 0.2 1.75 0.20114 0.00972 NA NA 0.40228 E
pass-91.0 rh3592 metal2 0.119003 0.6 89.909 1.32 41.95 11.4 41.7 11.13 0.0901539 0.01622 NA NA 0.0540923 NE
pass-98.7 rn17660 via2 0.0282677 NA 89.909 2.2 6.745 10.95 6.74 10.90 0.0128489 2.66 16900 1 1 metal3->metal2
pass-98.7 rn17662 via2 0.0282677 NA 89.909 2.2 7.025 10.90 7.02 10.90 0.0128489 2.66 16900 1 1 metal3->metal2
```

## Generating EM Reports in Voltus-Fi-XL

You can generate both textual and html reports for EM analysis results in Voltus-Fi-XL from the simulation results by specifying the EMIR control file (`emir.conf`), the EMIR database file, and the output directory.

For more information about generating EM reports in Voltus-Fi-XL, see [Generating EMIR Analysis Reports in Voltus-Fi-XL](#).

For more information about default reports generated for EM analysis and to view a sample EM analysis report, see [Default Reports Generated for EMIR Analysis](#).

## Types of EM Analysis Plots

Following EM Plots are available for viewing and debugging:

- [Tap Current Plots](#)
- [Resistor Current Plots](#)

- RJ – Current Density – max
- JAVG – Current Density – avg
- JABSAVG – Current Density – avgabs
- JACRMS – Current Density – rms

## Tap Current Plots

Analyze and report tap currents. These plots show the current distribution inside the transistor. The power-grid library generation program characterizes the distribution of the current inside the transistor and replaces the devices connected to the power grid with current taps (sink). The relative distribution of the current inside the transistor power-grid view library is based on the device width and length.

If an accurate power-grid view is used for the memories during rail analysis, the tap current plots clearly highlight the low current distribution in the transistor region and high power distribution in the sense amplifier and the decoder circuit of the memory.

There are three types of TC plots available in the drop-down list – **TC - Peak Tap Current**, **TCAVG - Average Tap Current**, and **TCRMS - RMS Tap Current**.

**Note:** The average and RMS tap current plots are only available in the `qrcTechFile` flow.

### *Information Displayed in the Annotation Browser for Tap Current Plot Types*

The EM/IR tab of the Annotation Browser displays the following columns for TC - Peak, TCAVG, and TCRMS plot types:

- Value
- Layers
- Short Message
- Resistance
- Via Area
- Number of Vias
- Needed Number of Vias
- Net

## Voltus-Fi Custom Power Integrity Solution User Guide

### EM Analysis Results

**Note:** For details about the information displayed in the columns of the Annotation Browser, see [Information Displayed in the EM/IR Tab of the Annotation Browser](#) in the “IR Drop Analysis Results” chapter.

The plot shown below displays peak tap currents. The violation range is customized by using the *Min Max* sliders in the IR/EM Results form. This plot is shown below.

**Figure 8-7 Peak Tap Current Plot for All Power Nets – Displaying Total Violation Range**



## Resistor Current Plots

Analyze and report resistor currents. The resistor current (RC) plots show current-flow trends that you either might not have expected or that the current from several power pins is reconverging in the middle of the chip to create high current through some wires.

## Voltus-Fi Custom Power Integrity Solution User Guide

### EM Analysis Results

---

There are three types of RC plots available in the drop-down list – **RC - Peak Resistor Current**, **RCAVG - Average Resistor Current**, and **RCRMS - RMS Resistor Current**.

**Note:** The average and RMS resistor current plots are only available in the qrcTechFile flow.

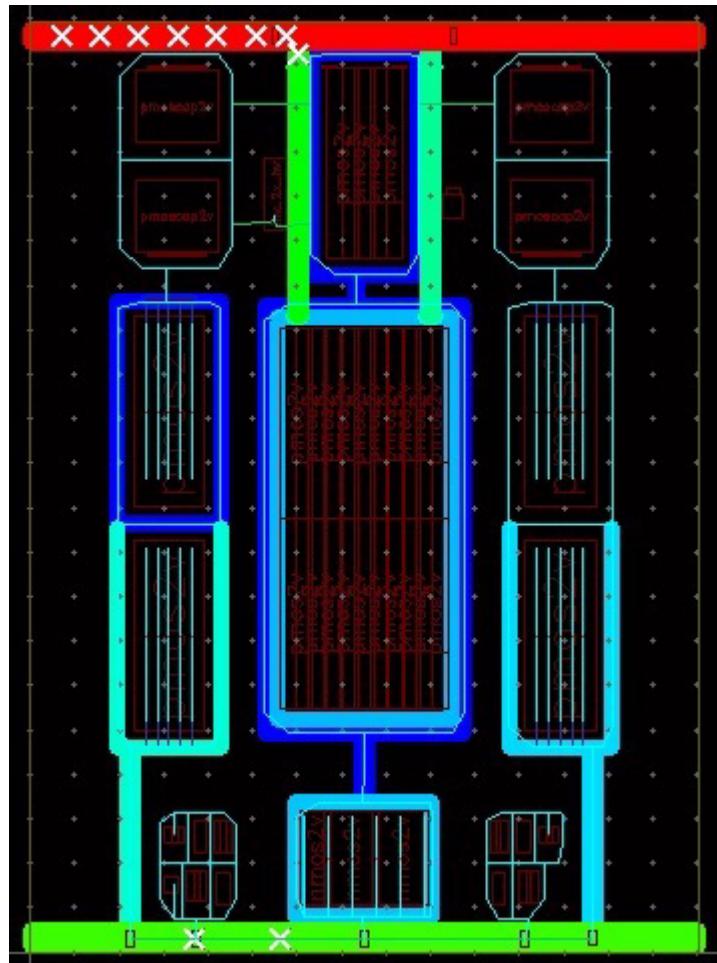
#### ***Information Displayed in the Annotation Browser for Resistor Current Plot Types***

The EM/IR tab of the Annotation Browser displays the following columns for RC - Peak, RCAVG, and RCRMS plot types:

- Value
- Layers
- Short Message
- Needed Width
- Width
- Length
- Resistance
- Via Area
- Number of Vias
- Needed Number of Vias
- Net

The plot shown below displays peak resistor currents. The violation range is customized by using the *Min Max* sliders in the IR/EM Results form.

**Figure 8-8 Peak RC Plot for Specified Power Nets – Displaying the Customized Violation Range**



### RJ – Current Density – max

Analyzes and reports peak current density. It is used to locate high current density areas in the design.

Current density reporting does not plot the current density of the segment but it is based on the ratio of the wire current density to the current density limit that is allowed by the process – ( $J/J_{max}$ ). So, all wires should have a ratio of less than 1.0. You can debug or fix the current density violations that have a ratio of more than 1.0. This ratio is easy to work with because its value is consistent across designs.

## Voltus-Fi Custom Power Integrity Solution User Guide

### EM Analysis Results

---

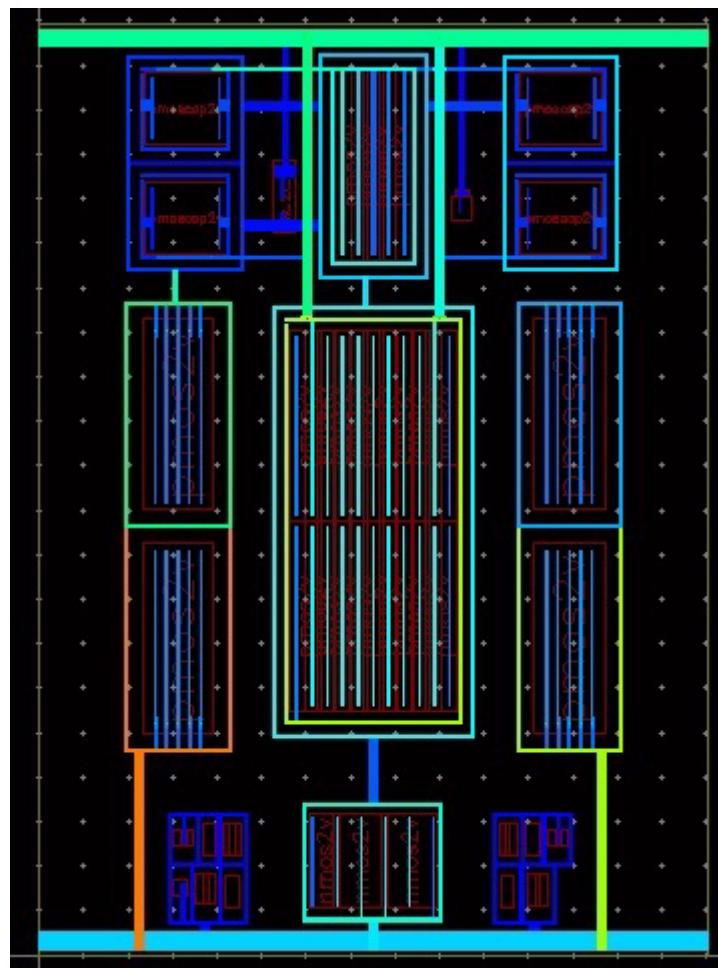
#### ***Information Displayed in the Annotation Browser for Current Density Plot Type***

The EM/IR tab of the Annotation Browser displays the following columns for RJ plot type:

- Value
- Layers
- Short Message
- Density
- Current
- Density Limit
- Needed Width
- Width
- Length
- Resistance
- Via Area
- Number of Vias
- Needed Number of Vias
- Net

This plot is shown below.

**Figure 8-9 RJ Plot for Specified Power Nets – Displaying Customized Violation Range**



### **JAVG – Current Density – avg**

Analyzes and reports the current density violations based on the average currents.

#### ***Information Displayed in the Annotation Browser for Average Current Density Plot Type***

The EM/IR tab of the Annotation Browser displays the following columns for JAVG plot type:

- Value
- Layers
- Short Message

## Voltus-Fi Custom Power Integrity Solution User Guide

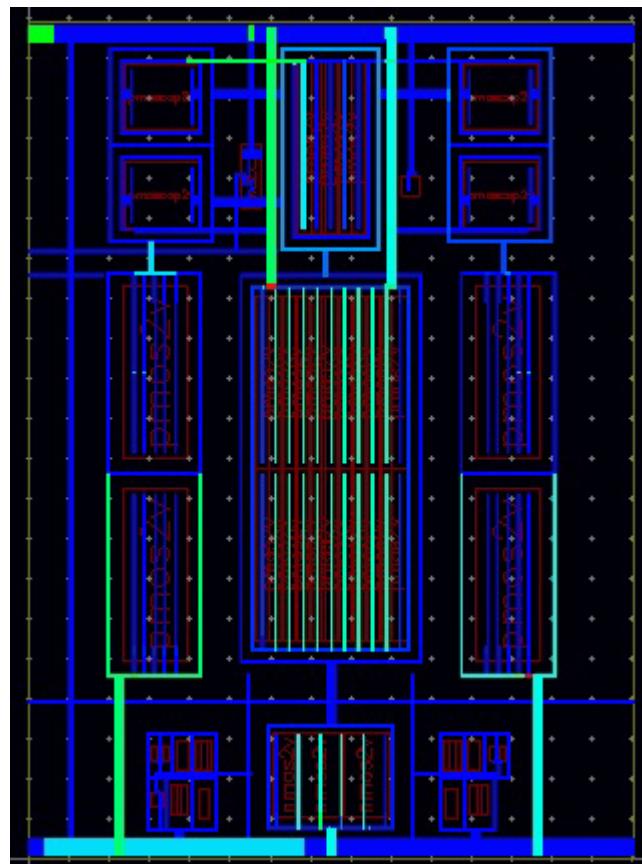
### EM Analysis Results

---

- Density
- Current
- Density Limit
- Needed Width
- Width
- Length
- Resistance
- Via Area
- Number of Vias
- Needed Number of Vias
- Net

The plot below shows the average current density violations for selected nets. The violation range is customized using the *Min Max* sliders in the IR/EM Results form.

**Figure 8-10 JAVG Plot for Selected Nets – Displaying the Customized Violation Range**



### **JABSAVG – Current Density – avgabs**

Analyzes and reports the current density violations based on the average of the absolute current.

#### ***Information Displayed in the Annotation Browser for JABSAVG Plot Type***

The EM/IR tab of the Annotation Browser displays the following columns for JABSAVG plot type:

- Value
- Layers
- Short Message
- Density

## Voltus-Fi Custom Power Integrity Solution User Guide

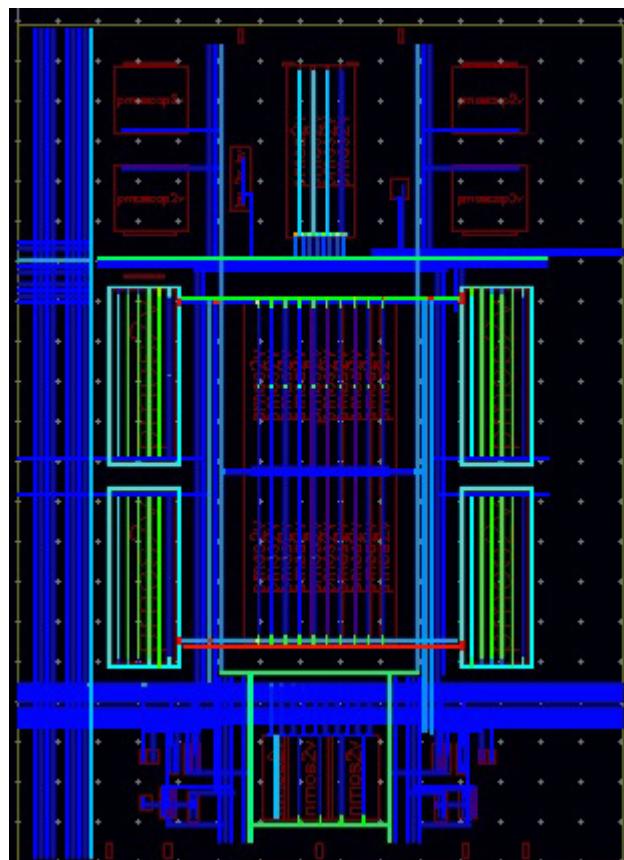
### EM Analysis Results

---

- Current
- Density Limit
- Needed Width
- Width
- Length
- Resistance
- Via Area
- Number of Vias
- Needed Number of Vias
- Net

The plot below shows the average of the absolute current density violations for all the signal nets. The violation range is customized by using the *Min Max* sliders in the IR/EM Results form.

**Figure 8-11 JABSAVG Plot for All Signal Nets – Displaying the Customized Violation Range**



### **JACRMS – Current Density – rms**

Analyzes and reports current density violations based on the root mean square (RMS) value of the AC current.

#### ***Information Displayed in the Annotation Browser for JACRMS Plot Type***

The EM/IR tab of the Annotation Browser displays the following columns for JACRMS plot type:

- Value
- Layers
- Short Message

## Voltus-Fi Custom Power Integrity Solution User Guide

### EM Analysis Results

---

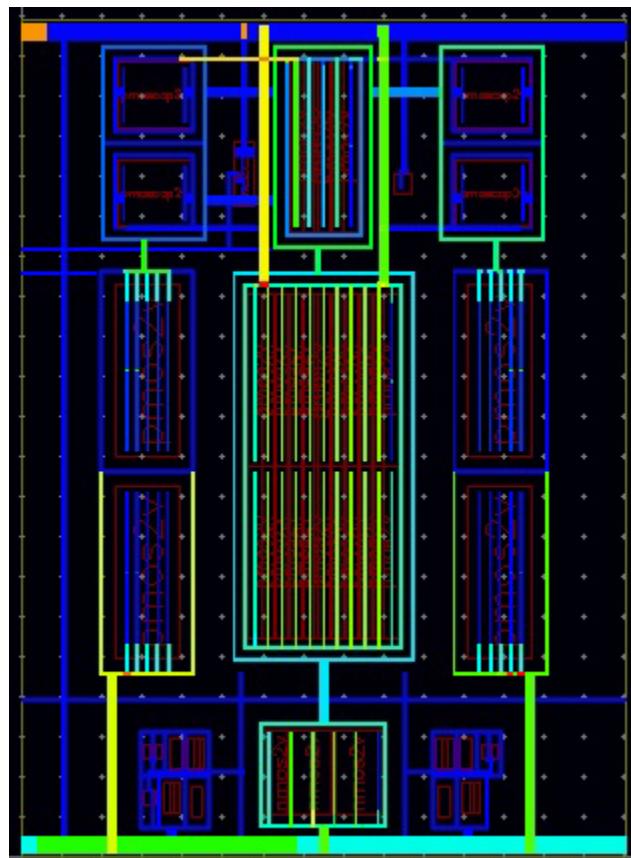
- Density
- Current
- Density Limit
- Needed Width
- Width
- Length
- Resistance
- Via Area
- Number of Vias
- Needed Number of Vias
- Net

The plot below shows the current density violations for all power nets. The violation range is customized by using the *Min Max* sliders in the IR/EM Results form.

## Voltus-Fi Custom Power Integrity Solution User Guide

### EM Analysis Results

**Figure 8-12 JACRMS - Plot – Displaying the Customized Violation Range for All Power Nets**



**Voltus-Fi Custom Power Integrity Solution User Guide**  
EM Analysis Results

---

---

# **Self-Heating Effect Analysis**

---

- [Overview](#) on page 226
- [Data Requirements for SHE Analysis](#) on page 227
- [Compatible Software Versions for SHE Analysis](#) on page 227
- [Flow Overview](#) on page 228
- [Performing SHE Analysis](#) on page 229
  - [Running PVS LVS](#)
  - [Running Quantus QRC Parasitic Extraction](#)
  - [Running Post-Layout Simulation using Spectre® APS](#) on page 230
  - [Viewing SHE Analysis Plots in Voltus-Fi-XL](#) on page 231
- [Types of Self-Heating Analysis Plots](#) on page 232
  - [Thermal-DeltaT Regions](#) on page 233
  - [Thermal-DeltaT Tiles](#) on page 234
  - [RJ\\_SH - Current Density - max \(SH\)](#) on page 235
  - [JAVG\\_SH - Current Density - avg \(SH\)](#) on page 235
  - [JACRMS\\_SH - Current Density - rms \(SH\)](#) on page 235
- [Re-Evaluation of EM Limit Based on dTemp](#) on page 236
- [EMIR Control File Options for Self-Heating Effect Analysis](#) on page 236
- [Running SHE Analysis in the Batch Mode](#) on page 239
- [Generating Reports for SHE Analysis](#) on page 239

## Overview

As semiconductor technology scales down to sub-micron dimensions, power density increases, resulting in temperature increases—self-heating effect (SHE)—in devices. Semiconductor components are sensitive to temperature variations.

The main effects of self-heating are as follows:

- Degraded carrier mobility, which leads to a reduced on-current of transistors and, therefore, slower speed
- Higher interconnect metal resistivity, which results in longer delays
- Increased failure rate and reduced reliability of electronic devices, with secondary effects such as interconnect electromigration

In digital circuits, the key consequences are longer delays due to increased interconnect resistance and reliability issues due to electromigration. Analog circuits are more sensitive to temperature variations, which may degrade matching properties. Therefore, thermal simulations and modeling play an important role in the design of integrated circuits today.

The self-heating effect is driven by front-end-of-line (FEOL) self heat and back-end-of-line (BEOL) self heat.

- The FEOL is the first part of IC fabrication where the individual devices such as transistors, capacitors, and resistors are patterned in the semiconductor. FEOL generally covers everything up to, but not including, the deposition of metal interconnect layers.
- The BEOL is the second part of IC fabrication where the individual devices get interconnected with the wiring on the wafer. Both contribute towards the maximum rise in temperature or deltaT that causes the self-heating effect.

Voltus-Fi-XL supports the SHE analysis flow visualization. You can view the results of the SHE analysis by plotting them on the Virtuoso layout to show the worst deltaT violations. You can then debug your design to reduce the self-heating effect. The SHE analysis flow is detailed in subsequent sections.



Self-heating effect analysis is a limited-access feature in this release. To use this feature, contact your Cadence representative to explain your usage requirements, and make sure the feature meets your needs before deploying it widely.

## Data Requirements for SHE Analysis

The table below lists the inputs that are required for SHE analysis in Voltus-Fi-XL:

**Table 9-1 Data Required for SHE Analysis**

Data	Purpose	Details
Model cards	Simulation	Spice model cards, which support SHE analysis for simulation
Third-party Detailed Standard Parasitic Format (xDSPF) file	Simulation	A detailed standard parasitic extraction file that contains x, y coordinates and l (length), and w (width) values. This file is required post simulation.
iRCX file	EMIR and SHE analysis	A file containing current limits per unit width or area for each process layer. This file is usually available through a foundry. <b>Note:</b> This file should contain the $I_{rms}$ equation.
EMIR control file ( <code>emir.config</code> )	Simulation	A configuration file needed to run EMIR and SHE analysis with Spectre® APS solver run.
<code>param.sh</code> file	SHE analysis	File for deltaT FEOL and BEOL calculations.
XML file	SHE analysis	A file created after simulation and provided as an input file for the Voltus-Fi-XL SHE flow. It contains deltaT information for all devices on the layout.

## Compatible Software Versions for SHE Analysis

The table below lists the compatible software versions that are needed for running the SHE analysis flow:

Product Name	Release Version
PVS	PVS 13.1.1 HF1 and later

## Voltus-Fi Custom Power Integrity Solution User Guide

### Self-Heating Effect Analysis

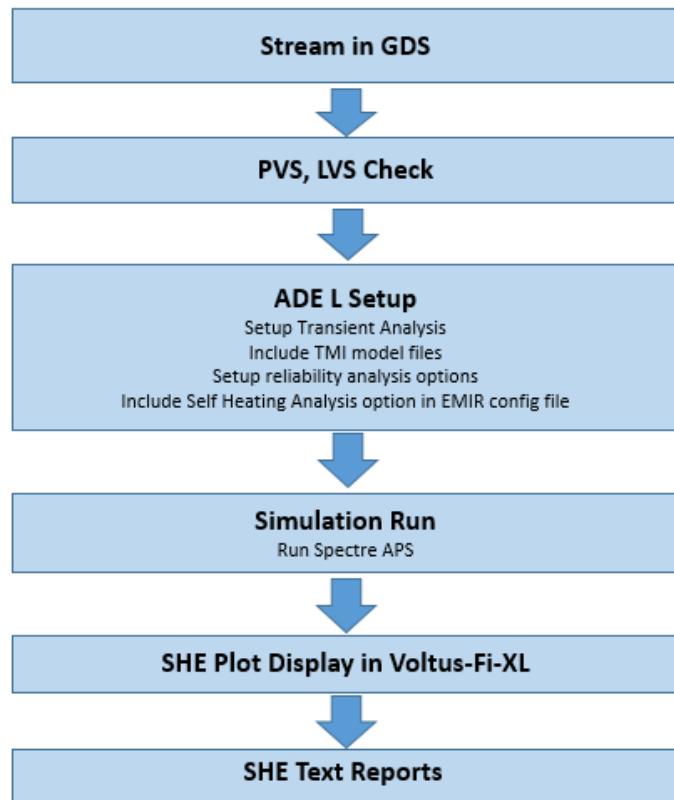
Product Name	Release Version
EXT	EXT 13.2.0 and later
MMSIM	MMSIM 13.1.1 ISR10 and later

**Note:** It is recommended that if any of the above release versions are not available, a later version of the product be used.

## Flow Overview

The diagram below illustrates the PVS LVS – Quantus – Virtuoso® Analog Design Environment L for Netlist and Config file generation – Spectre® APS – Voltus-Fi-XL flow for SHE analysis.

**Figure 9-1 The Self-Heating Effect Flow**



## Performing SHE Analysis

The following key tasks are performed for SHE flow:

- [Running PVS LVS](#)
- [Running Quantus QRC Parasitic Extraction](#)
- [Running Post-Layout Simulation using Spectre® APS](#)
- [Viewing SHE Analysis Plots in Voltus-Fi-XL](#)

These steps are detailed in subsequent sections.

### Running PVS LVS

This is the first step, which involves verifying the layout connectivity and creating an LVS database that is required for the subsequent Quantus parasitic extraction.

### Running Quantus QRC Parasitic Extraction

This is the second step, which involves using the Quantus QRC engine to create the xDSPF file for all the device parameters and interconnect parasitics.

Ensure that the following options are included during the Quantus QRC run in order to generate the xDSPF file suitable for SHE analysis flow:

- `-max_fraction_viacount` is changed to 1
- `-include_parasitic_res_temp_coeff` is changed to "true"
- `-include_parasitic_res_width` is added

**Note:** For SHE Analysis, shape database of metal layers is required. If the QRC command file includes the `parasitic_blocking_device_cells_file` option, then set the environment variable "`DBGRCX_HIERARCHICAL`" before running Quantus QRC. This ensures that the correct shape database is generated.

```
setenv DBGRCX_HIERARCHICAL Y
```

In the main menu of the Virtuoso Layout Suite, choose QRC. You can choose Run PVS – Quantus QRC option. The Quantus QRC Parasitic Extraction Run Form opens. Run extraction.

## Voltus-Fi Custom Power Integrity Solution User Guide

### Self-Heating Effect Analysis

---

For detailed information about all the settings in the Quantus QRC Parasitic Extraction Run form, see the “Quantus QRC Graphical Interface” chapter in *Quantus QRC Extraction Users Manual*.

The output includes the following:

- An xDSPF file, which contains all the parasitic information of the design – all the nets, x- and y-coordinates of transistors, parasitic resistors, and nets connected.
- A Quantus QRC output directory, which contains the shape database.

## Running Post-Layout Simulation using Spectre® APS

Once extraction is done, run post-layout simulation using the EMIR control file or the `emir.config` file to create the SHE analysis reports. Before performing simulation, ensure the following in the ADEL or the Simulation window:

- Setup transient analysis.
- Choose *Setup – Simulation File – Model Files* to add the SHE Models. These models are required to create the `*.dtemp#.xml` file that contains information about the increase in temperature for each device.
- Select *Simulation – Reliability – Setup*. The Reliability form opens. In this form, specify the following:
  - Select *Enable Reliability* to enable reliability simulation.
  - Select *Enable TMI Flow*.
  - In the *self-heating/aging(tmiShe)* cyclic field, select *self-heating only* (*tmiShe=1*). This option enables only the TMI self-heating mode.

**Note:** This option is available in IC616ISR8 and ICADV12.1ISR10 releases with MMSIM13.1ISR10 or above versions.

Now, you can start the simulation either by using *Simulation – Run* option or clicking the “run simulation” button provided in the right panel of the ADEL window.

**Note:** For detailed information about how these options are set, see “Using the Reliability Simulator Interface” chapter in the *Virtuoso® Analog Design Environment L User Guide*.

**Note:** The EMIR control file options required to be specified for SHE analysis are detailed in the [EMIR Control File Options for Self-Heating Effect Analysis](#) section.

## Viewing SHE Analysis Plots in Voltus-Fi-XL

Once the simulation run is complete, the simulation database saved in the `*.emir#_bin` file is loaded into Voltus-Fi-XL to view the SHE analysis plots. The SHE analysis plots are available on the EM tab of the IR/EM Results form.

The steps for populating the EM tab of the IR/EM Results form are detailed in the [Viewing the EM Analysis Results](#) section of the “EM Analysis Results” chapter.

In addition, for the SHE analysis flow, ensure that the following inputs are provided on the EM tab of the IR/EM Results form:

- Enable the *QRC Run*.
- In the QRC Run Location pop-up window, provide the details of the *QRC Run Directory* and *Run Name*.
- Click *Variables* to open the Variables form. In this form, specify the following variables:
  - shParamFile
  - shXMLFile

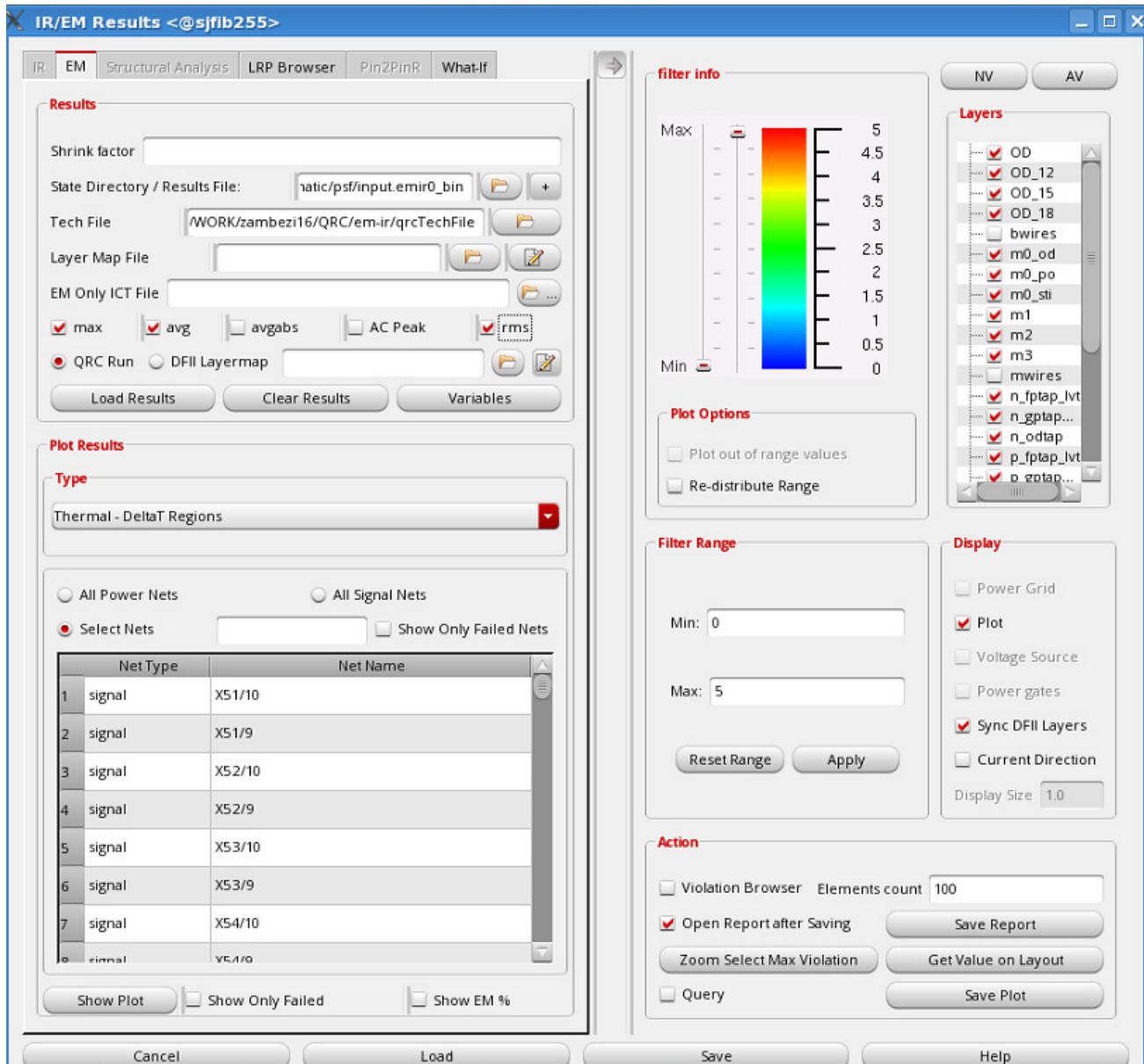
**Note:** For a sample `shParamFile`, see [Self-Heating Effect Analysis File](#) in the “File Formats” chapter.

- Select *rms*. This form is shown below.

# Voltus-Fi Custom Power Integrity Solution User Guide

## Self-Heating Effect Analysis

**Figure 9-2 The IR/EM Results Form – SHE Analysis**

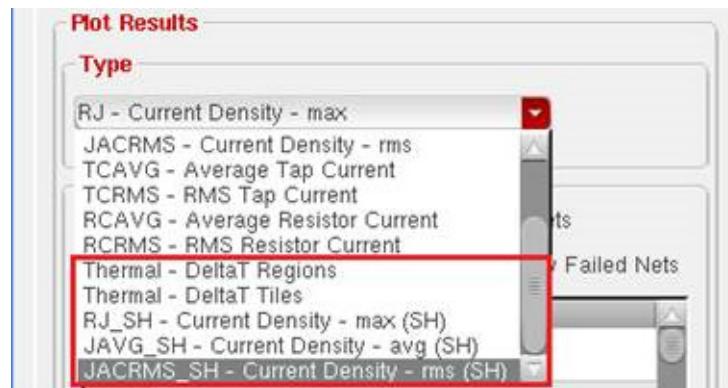


- In the Plot Results group box, the self-heating effect analysis plots are available in the drop-down list. These are detailed in the section below.

## Types of Self-Heating Analysis Plots

The types of self-heating effect analysis plots that are available in the drop-down list are detailed below.

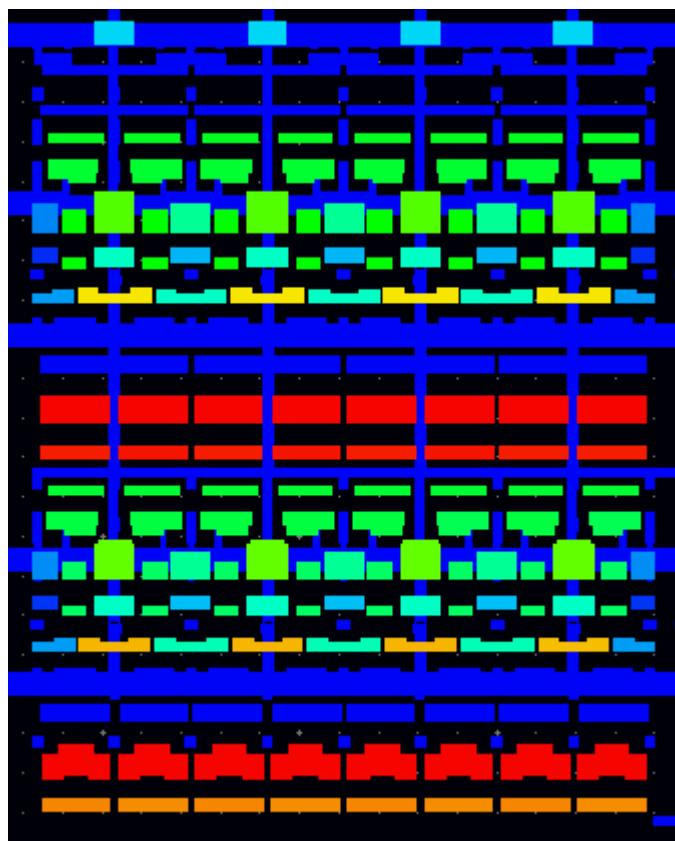
**Figure 9-3 SHE Analysis Plot Types**



## Thermal-DeltaT Regions

Reports the increase in temperature ( $\Delta T$ ) for all OD and metal layers in the entire design on the Virtuoso layout. This plot provides a detailed view of the temperature variations on the design. This is shown in the figure below.

**Figure 9-4 The Thermal-DeltaT Regions Plot**

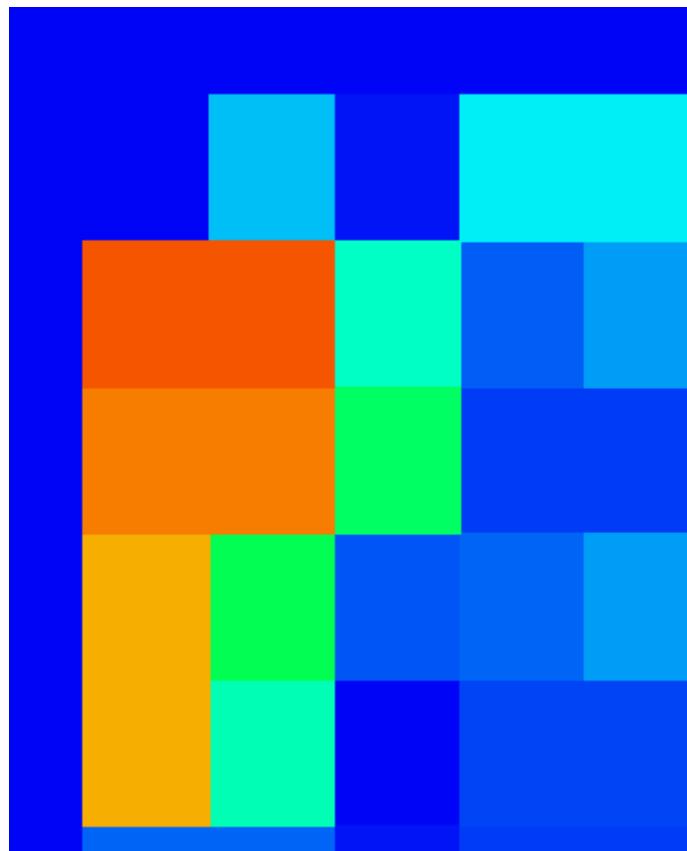


## Thermal-DeltaT Tiles

Reports the increase in temperature ( $\Delta T$ ) per tile based on the number of tiles, specified in the IR/EM Results form, on the Virtuoso layout. For this plot, specify the dimensions for the tiles in the **Tiles** text box on the *EM* tab of the IR/EM Results form. The design on the layout is divided into sections based on the number of tiles that are specified, for example, 15x20.

On the Virtuoso layout, the tiles are colored on the basis of the maximum temperature increase in each tile. This is shown in the figure below.

**Figure 9-5 The Thermal-DeltaT Tiles Plot**



### **RJ\_SH - Current Density - max (SH)**

Analyzes and reports the current density violations based on peak current after performing the SHE analysis.

### **JAVG\_SH - Current Density - avg (SH)**

Analyzes and reports the current density violations based on the average currents after performing the SHE analysis.

### **JACRMS\_SH - Current Density - rms (SH)**

Analyzes and reports current density violations based on the root mean square (RMS) value of the AC current after performing the SHE analysis.

## Re-Evaluation of EM Limit Based on dTemp

The EM effect is re-evaluated based on the new temperature, which is the sum of the simulation temperature and the changed temperature due to SHE. The current limit is calculated based on this new temperature and compared with the actual current flowing through the metal resistor to find the EM pass or fail.

The shEMEffect variable is used to re-evaluate the EM limit based on individual metal BEOL\_T and tile BEOL\_T temperatures. This variable is set using the `set_variable` command in the Voltus-Fi command file:

To re-evaluate EM on individual metal BEOL\_T:

```
set_variable shEMEffect beolT
```

To re-evaluate EM limit based on tile BEOL\_T temperature:

```
set_variable shEMEffect tiles_no.of divisions in x-direction X no.of divisions in y-direction
```

## EMIR Control File Options for Self-Heating Effect Analysis

A complete list of the EMIR control file options that are supported in Voltus-Fi-XL is provided in the table, [EMIR Control File Options Supported in Voltus-Fi-XL](#), in the “Data Preparation” chapter.

In addition, there are some options that are specific to the SHE flow. The table below provides a list of options that are specified in the `emir.config` file for the self-heating effect analysis flow.

**Table 9-2 Supported EMIR Control File Options for Self-Heating Effect Analysis**

Keyword	Option Set	Explanation	Default Value
net	analysis=[selfheating]	Turns on the self-heating effect analysis for the specified device under test (DUT).  <b>Note:</b> Ensure that the Irms option is specified in the control file.	none

# Voltus-Fi Custom Power Integrity Solution User Guide

## Self-Heating Effect Analysis

---

Keyword	Option Set	Explanation	Default Value
	selfheating_skip inst=[notDUT]	Specifies the instances on which SHE analysis will not be performed.	none
	selfheating_skip models=[<models to skip>]	Specifies the models on which SHE analysis will not be performed.	none
emirutil	shParamFile="param.sh"	<p>Specifies the self-heating effect analysis parameter file, which is required for calculating the SHE analysis results.</p> <p>A sample shParamFile format is provided in the “<a href="#">File Formats</a>” chapter.</p>	none
	filter_em_sh_threshold=value	<p>Defines the SHE EM analysis threshold value for the resistors being reported. The value for filter_em_sh_threshold is the J/Jlimit value.</p> <p>filter_em_sh_threshold=1 reports only those resistors that have a J/Jlimit value above 1.</p> <p><b>Note:</b> Specifying this option is equivalent to filtering all resistors that have passed the EM analysis check.</p>	none
	shHeatSink=[false]	<p>When set to true, specifies that heat sink effect should be included in the SHE analysis.</p> <p><b>Note:</b> If this parameter is set to true, ensure that heat sink models are available.</p>	false
	shXMLFile=<file. ]	Specifies the XML file to be used in the SHE analysis flow. This file contains information about the temperature change for every instance.	none

# Voltus-Fi Custom Power Integrity Solution User Guide

## Self-Heating Effect Analysis

---

Keyword	Option Set	Explanation	Default Value
	shEMEffect=[beolT   tiles_no.of divisions in x-direction X no.of divisions in y-direction]	<p>Specifies the basis for the re-evaluation of EM effect.</p> <p>beolT: specifies that re-evaluation will be done on the basis of individual metal BEOL_T.</p> <p><i>tiles_no.of divisions in x-direction X no. of divisions in y-direction</i>: specifies that re-evaluation will be done on the basis of tile BEOL_T.</p> <p>For example: shEMEffect=tiles_10x15</p>	none
	view=[lib=<lib> cell=<cell> view=<layout>]	<p>Specifies the Virtuoso Layout view.</p> <p><b>Note:</b> For the software to recognize the cell, view and library, ensure that "cds.lib" is present in the directory in which EMIR simulation is run.</p>	none
	qrc_output=<svdb_directory>]	<p>Specifies the Quantus QRC output directory.</p> <p>This parameter is required for self-heating effect analysis.</p>	none
	selfheating_report=[type=mos-region   metal filename=<filename> layer=<layername> tile=<x,y>]	<p>Generates the self-heating effect report, which is a text file. You can specify whether the report should contain <u>deltaT</u> information for the specified mos-region or metal and resistor or tile-based by using the relevant parameters.</p> <p>Example:</p> <pre>emirutil selfheating_report=[type=mos-region filename=mos-region-tiles.txt tiles=15x10]</pre>	none

## Running SHE Analysis in the Batch Mode

The syntax of the sequence of commands to run the SHE flow in Voltus-Fi-XL in the batch mode is provided below:

```
set_variable shParamFile param_sh.file
set_variable shXMLFile xml_file
set_variable shEMEffect {beolt | tiles_no.of divisions in x-direction X no.of
divisions in y-direction}

load_view \
-libname lib \
-cellname cell \
-viewname layout

qrc_run_directory svdb_directory
qrc_run_name qrc_run_name

load_em_results \
-tech_file tech_file_name \
[-layer_mapfile layer_map_filename] \
results_file_name

print_em_report \
-net -all_nets \
-type [acrms_sh | avg_sh | peak_sh] \
-filename output_file_name

print_thermal_report \
-type {mos-region | metal} \
-filename output_file_name
```

For details of the above commands, see [Batch Commands for SHE Analysis](#) in the “Batch Mode Execution” chapter.

## Generating Reports for SHE Analysis

The text reports for SHE analysis are only available in the batch mode using the sequence of commands detailed in the above section. The types of reports that are generated for SHE analysis are detailed below:

- Thermal reports containing temperature rise of OD region and metal resistors.

# Voltus-Fi Custom Power Integrity Solution User Guide

## Self-Heating Effect Analysis

The format of the OD region report is as follows:

**Figure 9-6 OD Region Report Format**

```
design: design_name
area: (x1 y1) (x2 y2)
report format: <delta_temperature> <od_area>
coordinate unit: um
delta_temperature          od_area
```

The format of the metal resistors report is as follows:

**Figure 9-7 Metal Resistors Report Format**

#	LayerName	Irms(mA)	Alpha	Beta	T_FEOL(C)	T_RMS(C)	T_BEOL(C)	Metal-region
---	-----------	----------	-------	------	-----------	----------	-----------	--------------

- Tiles-thermal reports containing temperature rise of OD region and metal resistors based on the specified tile matrix.

The format of the metal resistors report (tile-based) is as follows:

**Figure 9-8 Metal Resistors Tile-Based Report Format**

```
-----  
DIE_STACK_TEMPERATURE_MAP  
DIE_AREA <x y> <x y> <x y>  
NUMBER_OF_LAYERS #  
  
LAYER <layername>  
TEMPERATURE  
XCOR  x1  x2  x3  x4  x5  x6....xn  
YCOR  y1  y2  y3  y4  y5  y6....yn  
  
END_TEMPERATURE  
END_LAYER  
-----
```

The format of the OD region (tile-based) report is as follows:

## Voltus-Fi Custom Power Integrity Solution User Guide

### Self-Heating Effect Analysis

**Figure 9-9 OD Region Tile-Based Report Format**

```
#Tiles MOS-region report
"lower left x" "lower right y" "upper right x" "upper right y" "number of ROWS" "number of columns"
<deltaT for each tile in the matrix>
```

- The SHE EM reports containing dTemp of each parasitic resistor and EM report (including pass/fail, em\_limit, and needed width) both with and without the self-heating effect. The following reports are generated:
  - acrms\_sh.txt: AC RMS EM report both with and without the self-heating effect
  - avg\_sh.txt: Average EM report both with and without the self-heating effect
  - peak\_sh.txt: Peak EM report both with and without the self-heating effect

The formats of these report file are provided below.

**Figure 9-10 AC RMS EM Report Format**

```
ELECTROMIGRATION ANALYSIS RESULTS (acrms_sh)

Tj = 70
deltaT = 5

| Without SH Effect | With SH Effect |
Resistor layer X Y current width(um)/| %fail curr limit minW | %fail curr limit minW |dT(SH) feoLT beoLT coupled
(mA) viaCount | (mA) (um) | (mA) (um) | (mA) (um) |
```

**Figure 9-11 Average EM Report Format**

```
CTROMIGRATION ANALYSIS RESULTS (avg_sh)

= 70

| Without SH Effect | With SH Effect |
sistor layer X Y current width(um)/| %fail curr limit minW | %fail curr limit minW | Tj(SH) feoLT beoLT
(mA) viaCount | (mA) (um) | (mA) (um) | (um) |
```

**Figure 9-12 Peak EM Report Format**

```
# ELECTROMIGRATION ANALYSIS RESULTS (peak_sh)

# Tj = 70
#
#
# Resistor layer X Y current width(um)/| Without SH Effect | With SH Effect |
# (mA) viaCount | %fail curr limit minW | %fail curr limit minW | Tj(SH) feoLT beoLT
#
```

**Voltus-Fi Custom Power Integrity Solution User Guide**  
Self-Heating Effect Analysis

---

## **Structural Analysis**

---

- [Overview](#) on page 244
- [Checks Performed for Structural Analysis](#) on page 244
  - [Top Via Missing](#) on page 245
  - [Bottom Via Missing](#) on page 246
  - [Missing Via Connections](#) on page 246
  - [Via Coverage Ratio](#) on page 248
  - [Skewed Via Ratio](#) on page 249
- [Running Structural Analysis](#) on page 250
- [Examples of Structural Analysis Results](#) on page 255
  - [Show Unconnected from Top](#) on page 255
  - [Bottom / Top Via Ratio](#) on page 257
  - [Missing Via Connections](#) on page 258

## Overview

A typical electrical IR/EM analysis, which involves extraction of the power network, current calculation, and matrix solve, is able to identify devices that have a higher IR drop than the user-defined threshold.

Debugging the source of IR drop problems is a manual process involving the study of various voltage or current plots and patterns, and then narrowing down the weak links in the power grid. This is an effort-intensive activity.

Voltus-Fi-XL supports structural analysis, which is a shape-based geometrical analysis that is used to quickly identify power-grid weaknesses in designs and mark them on the Virtuoso® layout for you to view and debug. Structural analysis cuts down on the sign-off analysis time. This analysis is available in the GUI mode.

You can perform structural analysis using the results of the IR drop analysis.

## Checks Performed for Structural Analysis

The checks performed in structural analysis work on the basic definition of connectivity—for every metal segment there should be connectivity from the top metal layers to the bottom metal layers unless the metal segment belongs to either the top-most metal layer or to the bottom-most metal layer.

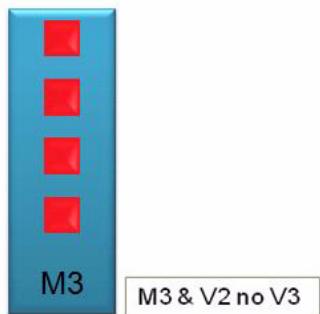
The via connectivity checks are performed on full-metal segments and not on the individual shapes that are part of the metal segments.

The following categories of checks can be performed for routing layers:

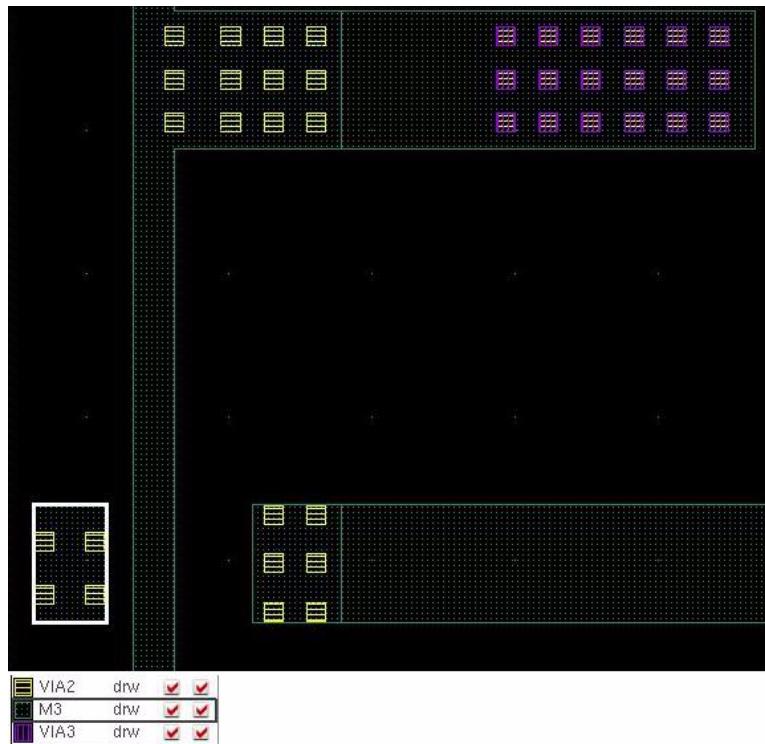
- For unconnected shapes in power net layout
  - [Top Via Missing](#)
  - [Bottom Via Missing](#)
- For overlapping layers
  - [Missing Via Connections](#)
  - [Via Coverage Ratio](#)
- For weakly connected shapes in power net layout
  - [Skewed Via Ratio](#)

## Top Via Missing

This check highlights all metal segments that have no top via or contact shapes overlapping them on layer basis. In the following example, metal layer, M3, has via, V2, but does not have via, V3.

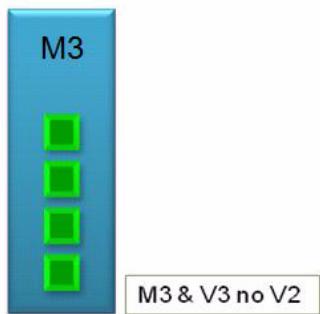


The highlighted shape in the example below has V2 but not V3.

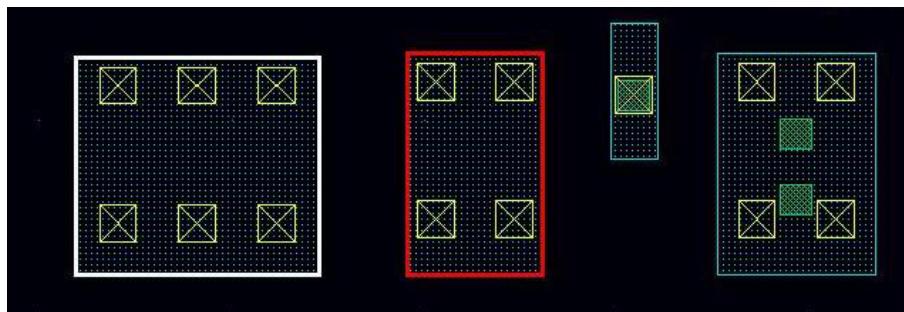


### Bottom Via Missing

This check highlights all metal segments that have no bottom via or contact shapes overlapping them on layer basis. In the following example, metal layer, M3, has via, V3, but does not have via, V2.



The highlighted shapes in the example below have V3 but not V2.

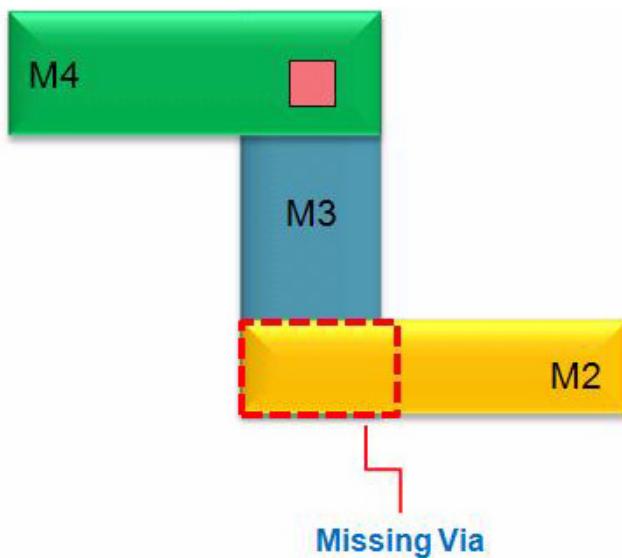


### Missing Via Connections

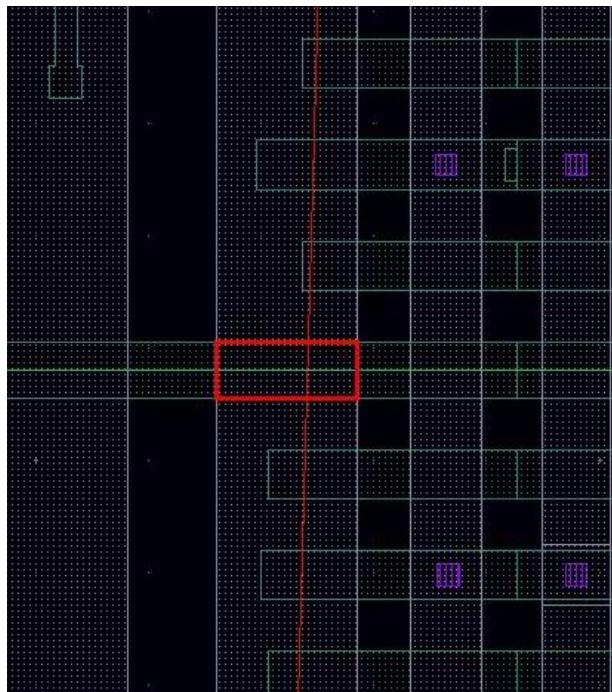
This check highlights overlapping wire segments that have no vias in the overlapping area. In the following example, overlapping area between M4 and M3, has V3 but the overlapping area between M3 and M2 does not have V2.

# Voltus-Fi Custom Power Integrity Solution User Guide

## Structural Analysis

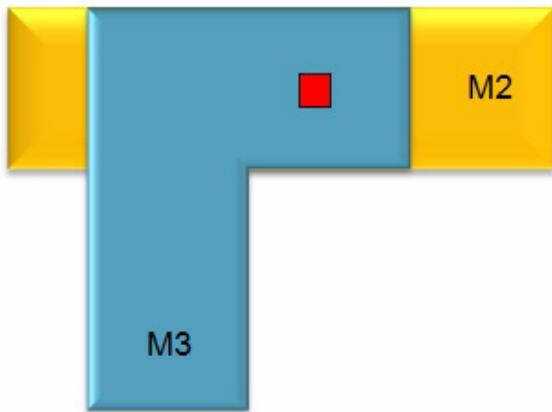


The highlighted overlapping shape of the metal segments in the example below has missing via.

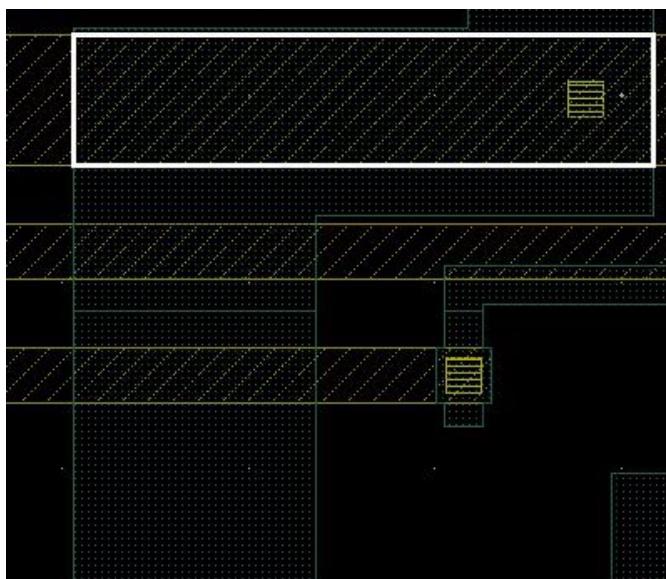


### Via Coverage Ratio

This check highlights segments that have less than user-specified via coverage ratio in the overlapping area. In the following example, the overlapping area between M2 and M3 has a via but the via coverage ratio or the ratio of via area and overlapping area is less than the user-specified ratio.

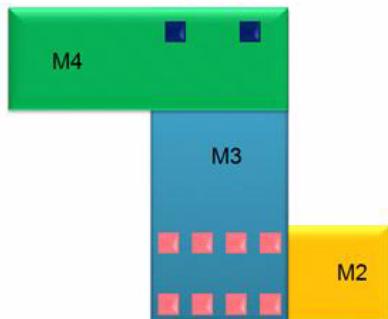


The highlighted shape in the example below has a via coverage ratio less than that specified by the user.

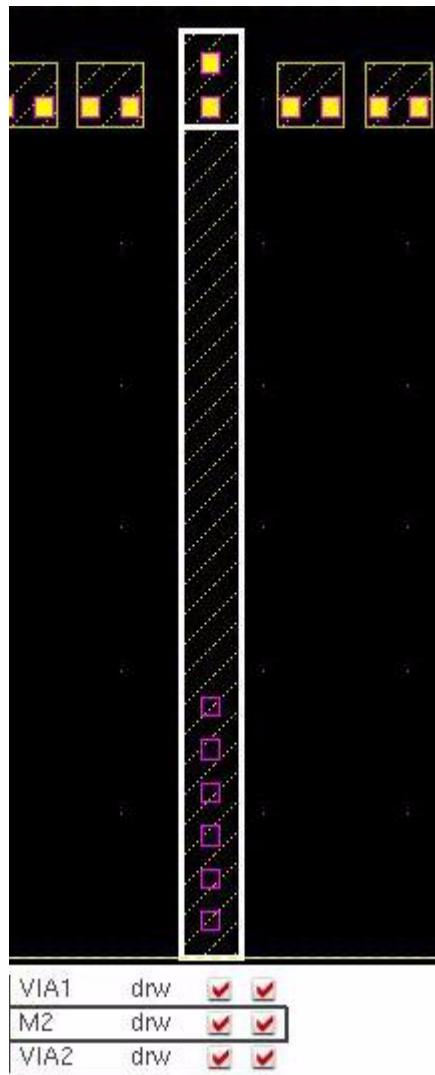


## Skewed Via Ratio

This check highlights segments in which the ratio of top and bottom connecting vias is skewed. In the following example, the ratio of top/bottom via is 2:8, which is skewed.



The highlighted shape in the example below has a skewed top/bottom via ratio.



For this check, you can choose to:

- Highlight wire segments that have a top/bottom via ratio higher than the specified ratio
- Highlight wire segments that have a bottom/top via ratio higher than the specified ratio

## Running Structural Analysis

To perform structural analysis, you first need to load the IR drop analysis results. Before you run structural analysis, ensure the following:

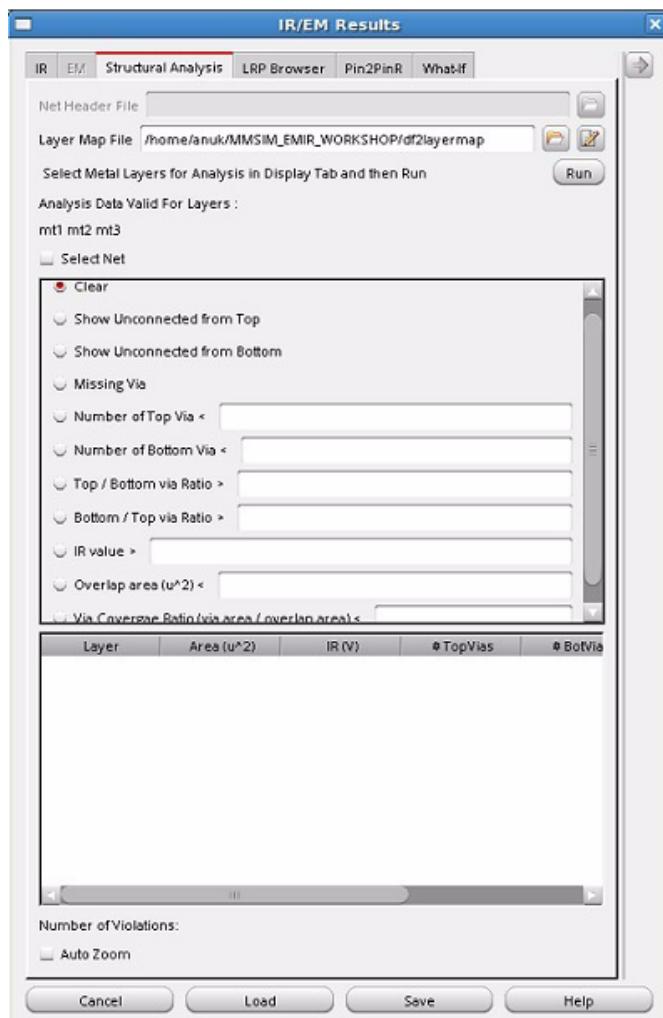
# Voltus-Fi Custom Power Integrity Solution User Guide

## Structural Analysis

- In the *Layers* group box in the expanded IR/EM Results form, the layers for which you want to view the results of the structural analysis are selected
- The list of selected layers include metal layers because structural analysis is performed only for metal layers

In the IR/EM Results form, select the *Structural Analysis* tab. The form shown below opens.

**Figure 10-1 IR/EM Results Form – Structural Analysis Tab**



- Select the *Layer Map File*, which is the APS/XPS-to-DFII layer map file. You can also create a new layer map file or edit an existing file, using the edit button provided next to the field. An example of the layer map file is provided below.

### Example: APS/XPS to DFII Layer Map File Format

```
#<type> <extraction layer name> <dfII layer name>
```

```
metal    metal7    M7
via      VIA5     VIA6
metal    metal6    M6
via      VIA5     VIA5
metal    metal5    M5
via      VIA4     VIA4
metal    metal4    M4
via      VIA3     VIA3
metal    metal3    M3
```



### Important

The order of layers in the layer map file is important. The layer map file must have layers listed in a top-to-bottom order.

- Click *Run*. The names of layers for which data is available will appear under the field titled, *Analysis Data Valid For Layers*.

**Note:** You can only perform structural analysis on one net at a time. So, if you are using the IR drop analysis results for performing structural analysis, ensure that only one net is selected on the IR tab. If you select *All Power Nets*, *All Signal Nets*, or multiple nets in the *Select Nets* option on the IR tab, the software give the following pop-up error message.



- Click *Select Net* to select the net on the Virtuoso layout.
- To view the data, the options provided for different check types are enabled. You can select the specific check you want to perform for the selected layers. The available options are listed below.
  - Click *Clear* to clear any existing highlights on the layout.
  - Click *Show Unconnected from Top* to highlight all unconnected metal segments that have no top via/contact shapes overlapping them on layer basis. For this check, the result browser shows the following information:

**Figure 10-2 Result Browser for Show Unconnected from Top Check**

Layer	Area ( $\mu^2$ )	IR (V)	# TopVias	# BotVias	Ratio
METAL_2	5118.98	0.955203	0	2830	0
METAL_2	5118.98	0.957325	0	2980	0

- Click *Show Unconnected from Bottom* to highlight all unconnected metal segments that have no bottom via/contact shapes overlapping them on layer basis.
- Click *Missing Via* to highlight overlapping metal segments that have missing vias in the overlapping area. For this check, the result browser will display only layer and area information. This is shown below.

**Figure 10-3 Result Browser for Missing Via Connections Check**

Layer	Area ( $\mu^2$ )
METAL_4 - METAL_3	0.6174
METAL_4 - METAL_3	0.6174
METAL_4 - METAL_3	0.6174

- Specify the value in the *Number of Top Via* < field to highlight metal segments that have less than the specified number of top vias.
- Specify the value in the *Number of Bottom Via* < field to highlight metal segments that have less than the specified number of bottom vias.
- Specify the value in the *Top / Bottom via Ratio* > field to highlight metal segments that have more than the specified top to bottom via ratio.
- Specify the value in the *Bottom / Top via Ratio* > field to highlight metal segments that have more than the specified bottom to top via ratio.
- Specify the value in the *IR value* < field to highlight segments that have less than the specified IR drop values.

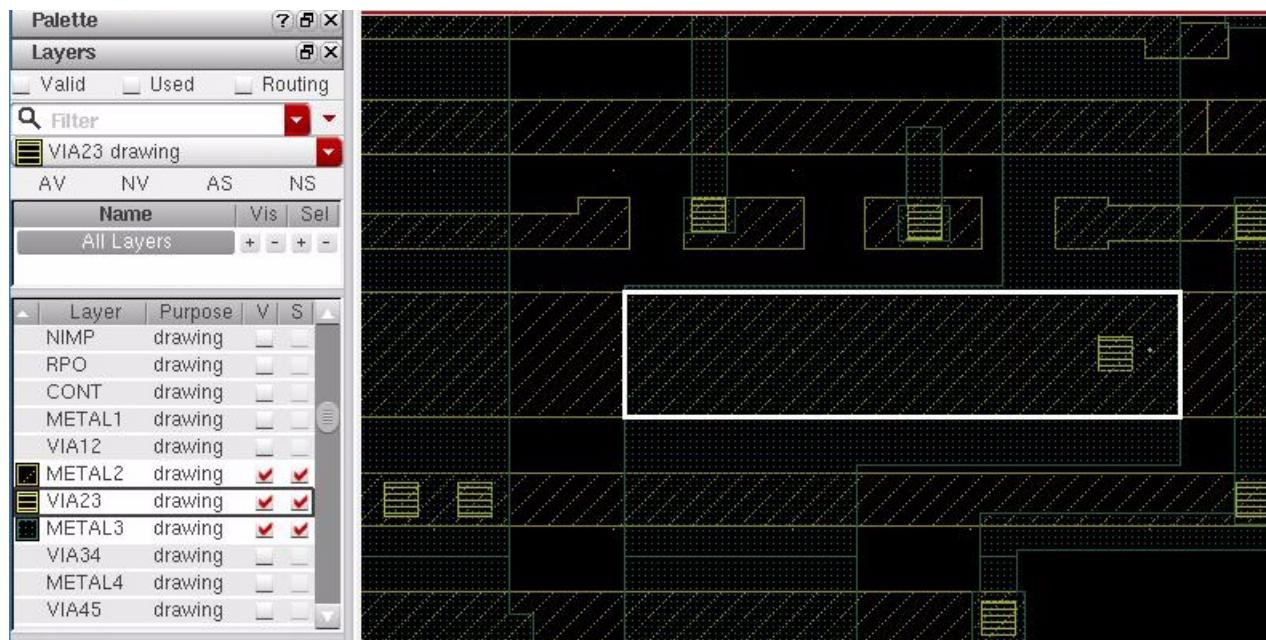
**Note:** This check is performed at the segment level. For segments with multiple shapes, the software picks the worst-case IR values.

- Specify the value in the *Overlap area ( $\mu^2$ )* < field to highlight segments that have an overlapping area less than the specified overlapping area.

- Specify the value in the *Via Coverage Ratio (via area / overlap area) <* field to highlight segments that have less than the specified via coverage ratio in the overlapping area. This check is used to highlight segments that have a via in the overlapping area but the via coverage is less as compared to the size of the overlapping area.

For example, in the image below, the overlapping area of metal layers, M2 and M3, has a via, Via23. However, the ratio of via area to overlapping area is less than the user-specified ratio of 0.3. So, the overlapping area is highlighted.

**Figure 10-4 Via Coverage Ratio**



- You can view the violations for the selected check type in the result browser.

**Figure 10-5 Result Browser**

Layer	Area ( $\mu\text{m}^2$ )	IR (V)	# TopVias	# BotVias	Ratio
METAL_1	0.991	0.942184	0	4	0
METAL_1	0.991	0.942581	0	4	0
METAL_1	0.991	0.943152	0	4	0

Number of Violations :234

- The following information is available for the selected metal layers for all checks except the missing via check:
  - *Layer* – layer name for which information is provided
  - *Area (u^2)* – area of the segment or layer
  - *IR(V)* – worst IR drop in the node
  - *#TopVias* – number of top vias
  - *#BotVias* – number of bottom vias
  - *Ratio* – ratio of top-to-bottom or bottom-to-top vias
- **Note:** Double-click the column headings of the result browser to list the results in either an ascending or descending order.
- The *Number of Violations* for the selected check type is reported below the result browser.
- Click *Auto Zoom* to zoom into the violations for the specified check.
- To change the layers for which you want to perform structural analysis, select or deselect the layers in the *Layers* group box in the expanded IR/EM Results form and click *update*.

## Examples of Structural Analysis Results

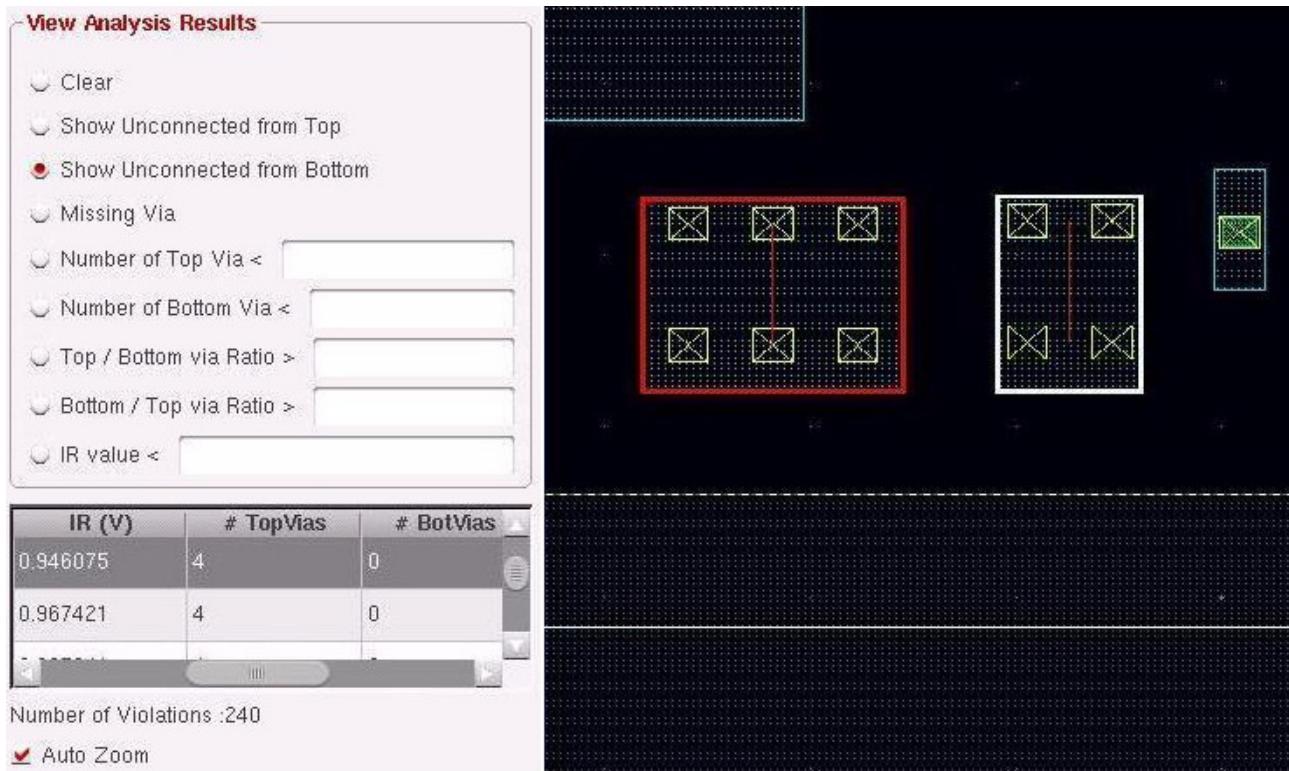
Provided below are examples for the following checks:

- Show Unconnected from Top
- Show Unconnected from Bottom
- Bottom / Top Via Ratio
- Missing Via Connections

### Show Unconnected from Top

The figure below shows an example of structural analysis results in the Virtuoso layout when *Show Unconnected from Top* view is selected in the *View Analysis Results* field. The metal segment (*m2*), shown in the below figure, does not have any top vias.

#### Example 10-1 View Analysis Results – Show Unconnected from Top



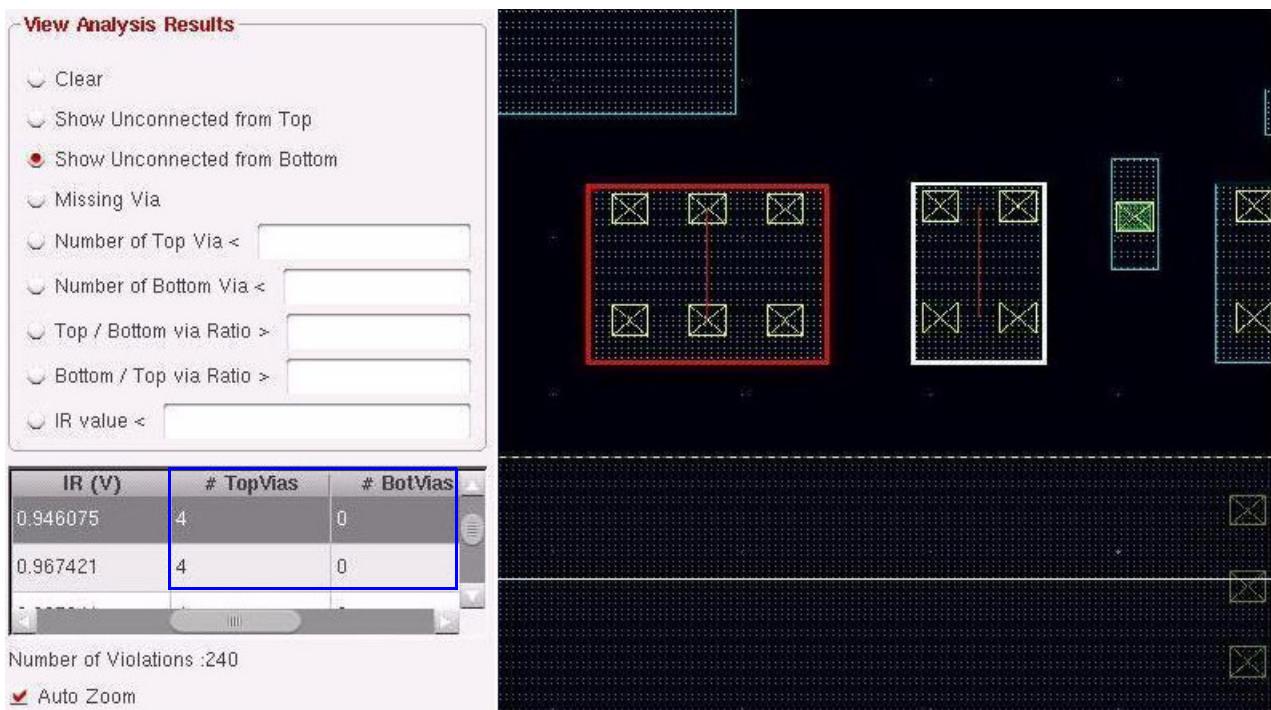
## Show Unconnected from Bottom

The figure below shows an example of structural analysis results in the Virtuoso layout when *Show Unconnected from Bottom* view is selected in the *View Analysis Results* field. The metal segment (Metal1\_1), shown in the below figure, has only top vias. It does not have any bottom vias.

### Example 10-2 View Analysis Results – Show Unconnected from Bottom

# Voltus-Fi Custom Power Integrity Solution User Guide

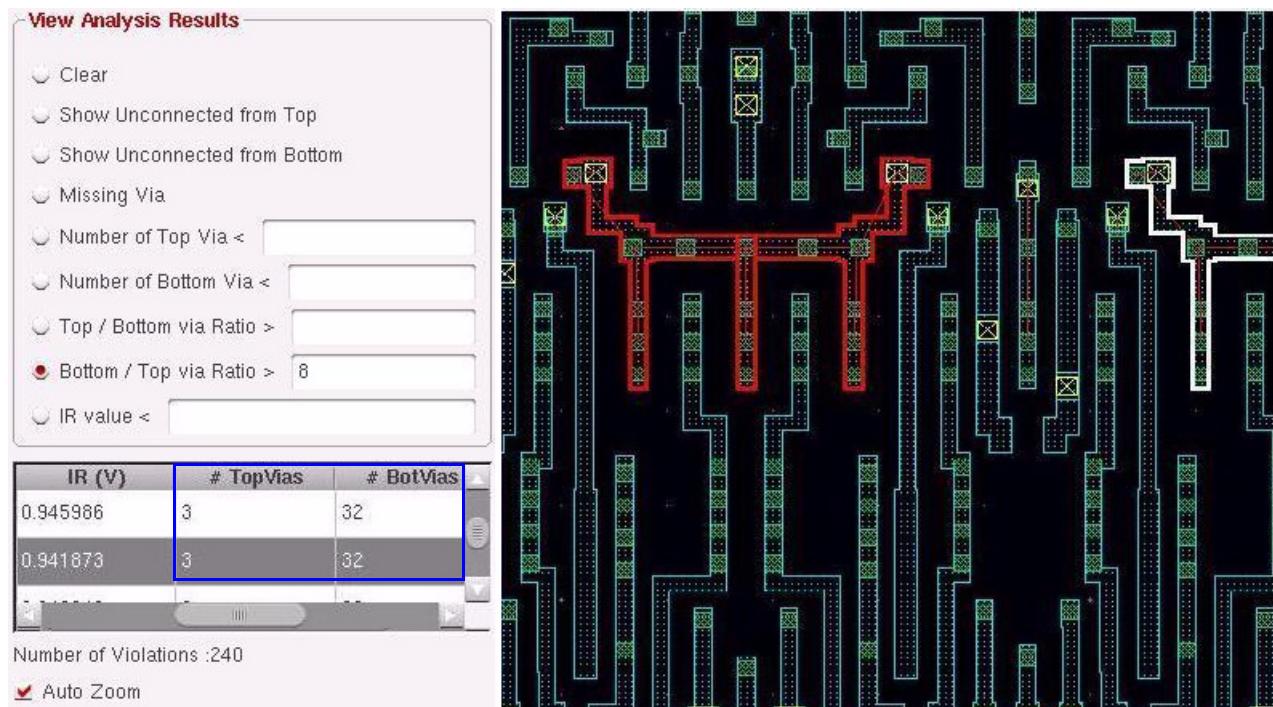
## Structural Analysis



### Bottom / Top Via Ratio

The figure below shows an example of structural analysis results in Virtuoso layout when *Bottom / Top via Ratio>* is selected in the *View Analysis Results* field. In this example, the user-specified bottom/top via ratio is 8. The metal segment (`Metal1_1`) has 3 top vias and 32 bottom vias. The bottom to top via ratio is 32:3, which is greater than the user-specified ratio of 8.

### Example 10-3 View Analysis Results – Bottom / Top Via Ratio



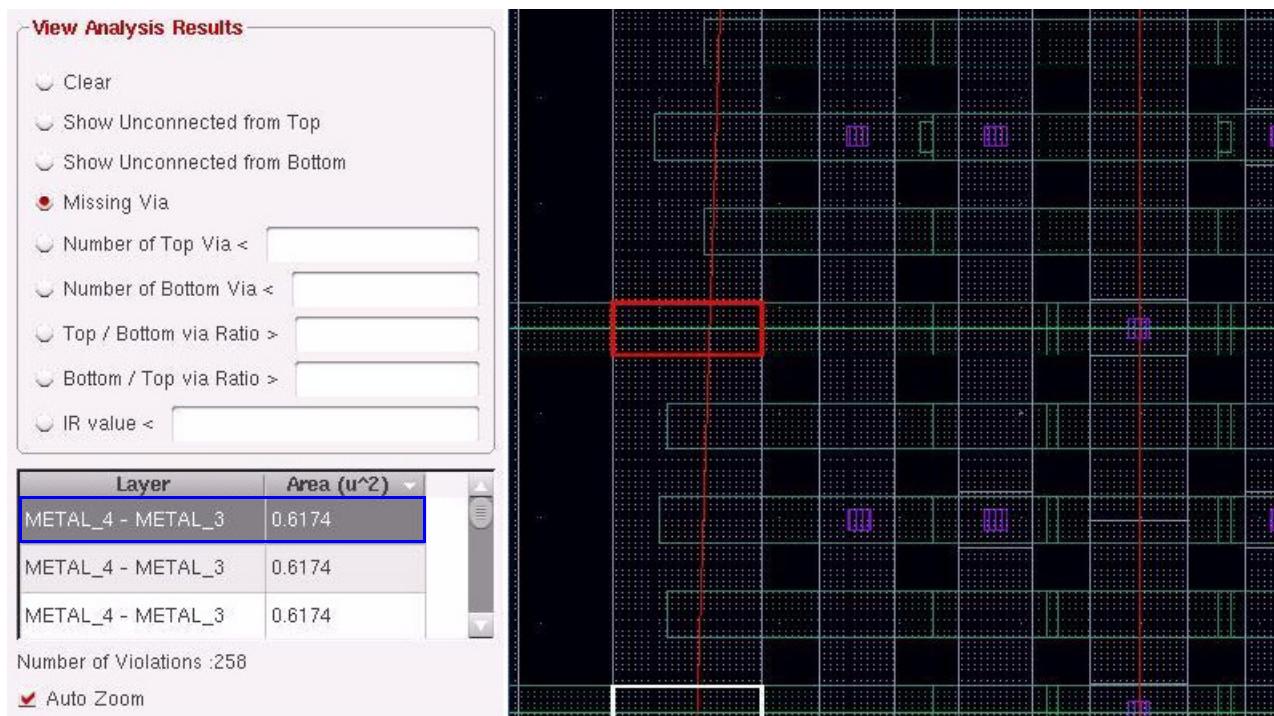
## Missing Via Connections

In this example, the overlapping area of metal segments Metal\_4 and Metal\_3, which is highlighted in red, has a missing via, v3. The result browser shows the overlapping layer names and the overlapping area of the two segments. This is shown in the image below.

### Example 10-4 View Analysis Results – Missing Via Connections

# Voltus-Fi Custom Power Integrity Solution User Guide

## Structural Analysis



## **Voltus-Fi Custom Power Integrity Solution User Guide**

---

## **What-If EMIR Analysis**

---

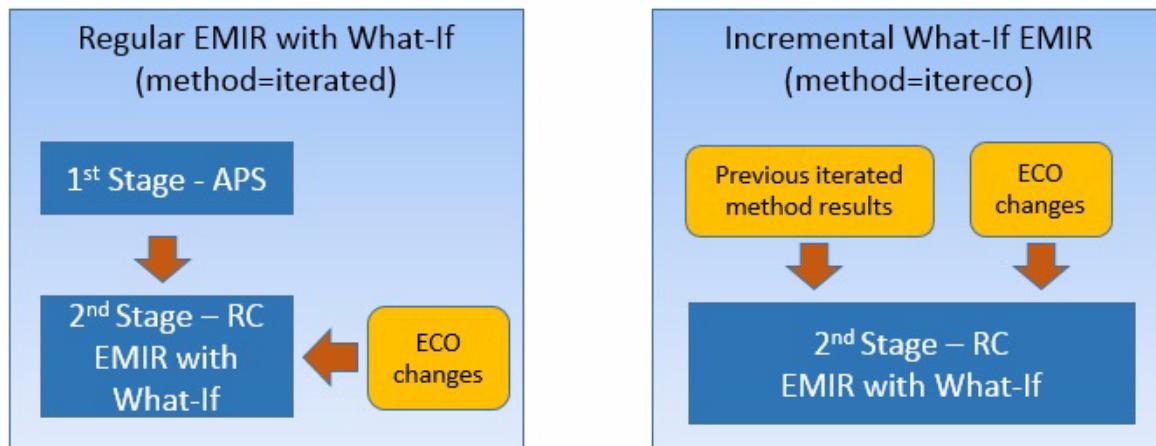
- [Overview](#) on page 262
- [ECO Commands Supported in Voltus-Fi-XL](#) on page 263
- [Performing What-If Analysis in Voltus-Fi-XL](#) on page 264
- [Pre-Filling the Run Simulation Form](#) on page 272
- [Output](#) on page 273
- [Options for Performing What-If Analysis](#) on page 273

# Overview

The What-If analysis feature lets you analyze the impact of potential layout changes, without actually implementing these changes in the layout, and re-extracting the xDSPF file. These changes are also called the ECO changes.

The What-If flow is supported in the regular “iterated” method EMIR flow, and in an incremental “itereco” method EMIR flow, which reuses the circuit simulation results from a previous iterated mode EMIR analysis. This is shown in the figure below.

**Figure 11-1 The What-If Flow**



The What-If changes are defined in an ECO file, which is included in the EMIR control file (`emir.config`). All ECO changes are relative to the original xDSPF content. In one What-If simulation, all changes defined in the ECO file are applied. These are then analyzed together in the second stage of the EMIR analysis.

Multiple incremental ECO runs are supported. For each run, the changes defined in the ECO file are applied to the original xDSPF content. Changes from previous ECO runs are not considered. In the incremental What-If analysis, only the nets with ECO changes are analyzed.

Other EMIR methods, such as “method=direct”, are not supported when performing What-If analysis.

All ECO changes for a specific What-If run are defined in one ECO file.

## ECO Commands Supported in Voltus-Fi-XL

The following ECO commands are supported in Voltus-Fi-XL:

- add\_pin net=<net\_name> name=[node1 node2 ...]
- delete\_pin net=<net\_name> name=[node1 node2 ...]
- add\_cap net=<net\_name> cap=[node1 cvalue1 node2 cvalue2 ...]
- add\_res net=<net\_name> node=<node1 node2> r=<rvalue>
- delete\_res net=<net\_name> name=<resistor1 resistor2 ...>
- scale\_res net=<net\_name> name=<resistor1 resistor2 ...> scale = <scalefactor>

The parameters listed above are defined as follows:

- net\_name : name of the net as defined in the xDSPF | \*net statement. There is no support for wildcards, same as that in the xDSPF file.
- node1 , node2 : name of pin, sub, or tap node. There is no support for wild cards, same as that in the xDSPF file.
- cvalue1, cvalue2 : the value of capacitance to be added. Negative value is supported for reducing net capacitance. It is a floating number.
- rvalue ... value: value of resistance to be added. This is a floating number.
- resistor1, resistor2 : the instance name of resistors to be deleted, or scaled.
- scalefactor : the scaling factor. This is a floating number.

An example ECO file (`emir.eco`) is as follows:

```
add_pin net=[VDD] name=[VDD#9]
del_pin net=[bias1] name=[Rj3652]
scale_res net=[VDD] name=[Rg1 Rg2 Rg3 Rg4] scale=0.5
delete_res net=[VDD] name=[Rg5]
add_res net=[VDD] node=[VDD#80 VDD#81] r=10
add_cap net=[VDD] cap=[VDD#80 10e-15 VDD#81 5e-15]
```

Alternatively, all ECO commands can also be used with the prefix, `eco_` in the `emir.config` file directly. For example,

```
eco_add_pin net=[VDD] name=[VDD#9]
```

When performing a regular iterated method EMIR analyses with What-If, the only change required in the `emir.config` file is the reference to the ECO file.

```
solver method=iterated  
eco file="--emir.eco"
```

The following lines need to be defined in the `emir.config` file to enable an incremental What-If EMIR analysis.

```
solver method=itereco inputwf="./input.emirtap.pwl"  
eco file="emir.eco"
```

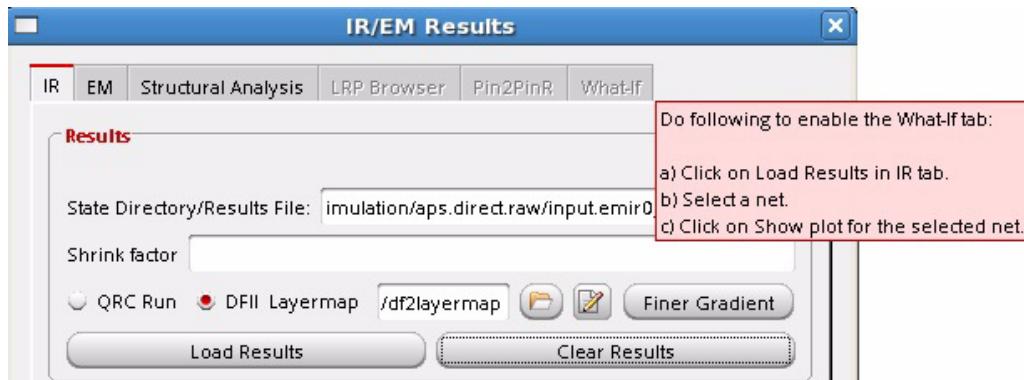
## Performing What-If Analysis in Voltus-Fi-XL

The What-If flow in Voltus-Fi-XL GUI is as follows:

- ECO commands are specified in the Voltus-Fi-XL GUI and the ECO file is generated.
- Voltus-Fi-XL calls Spectre to perform the EMIR analysis. Fully accurate EMIR simulation is performed.
- Only nets that are changed in the ECO statement are changed.

To perform What-If analysis, first load the IR drop analysis results, select the net and click *Show Plot*. In the IR/EM Results form, the *What-If Analysis* tab gets enabled. The software provides support in the form of tips about how to enable the *What-If* tab when you hover over the tab name. This is shown below. It also provides help to fill out information in the various fields in the tab. This tool tip appears when you hover over the fields in the form.

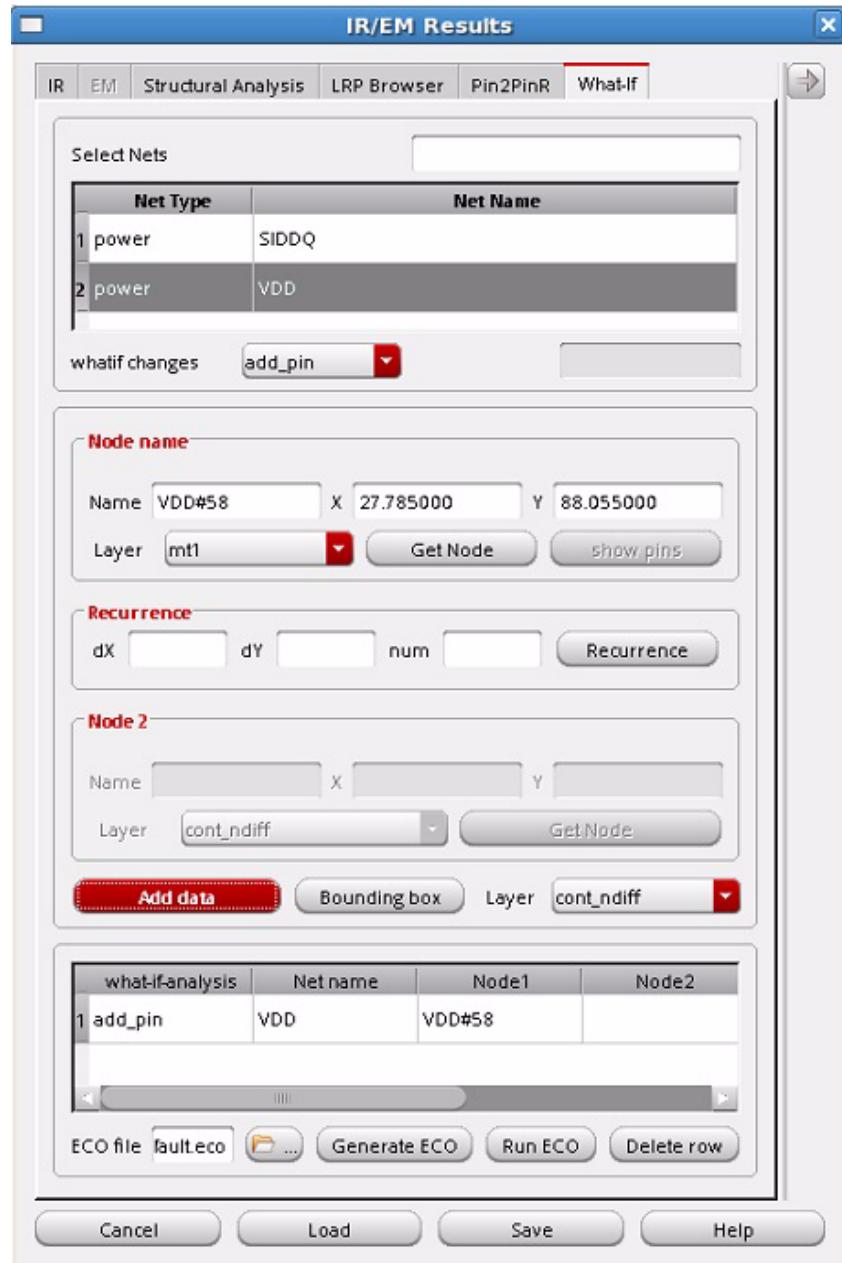
**Figure 11-2 Tool Tips for What-If analysis Tab of the IR/EM Results Form**



# Voltus-Fi Custom Power Integrity Solution User Guide

## What-If EMIR Analysis

Figure 11-3 IR/EM Results Form – What-If Tab



In this form, provide the following information:

- Select the nets on which you want to perform What-If analysis. In the text field provided with the *Select Nets* field, enter the full net name. The text field also accepts regular expression. For example, if you enter V\*, the table will list all nets whose names start with V, for example VDD and VSS .

- From the *whatif changes* drop-down list, select the ECO command you want to run. The following commands are available:

- add\_pin
- delete\_pin
- add\_cap
- scale\_res
- add\_res
- delete\_res

**Note:** For each of the above options, there are some changes in the fields enabled in the form. For details, see [Options for Performing What-If Analysis](#).

In the *Node name* group box, provide the information for the node on which the What-If analysis is to be performed.

- Select the *Layer* on which you want to perform the analysis from the drop-down list. By default, this field is populated with the name of the layer selected in the Display form. If multiple layers are selected in the Display form, all the layer names will appear in the drop-down list.
- Click *Get Node* and then select any point on the specified layer in the layout. The nearest node on the specified layer is identified for the whatif changes. The node name and its x- and y- coordinates are populated in the three text boxes; *Name*, *X*, and *Y*, respectively.

In the *Recurrence* group box, provide the following information:

- Click *Recurrence* to add pins in recurrence in both horizontal and vertical directions. The starting point is the node that was specified using the *Get Node* button in the previous step. This option is only applicable when *add\_pin* ECO command is selected. For this, perform the following tasks:
  - Specify *dX*, or delta-X, which is the horizontal spacing between the nodes that will be used for adding pins.
  - Specify *dY*, or delta-Y, which is the vertical spacing between the nodes that will be used for adding pins.
  - Specify *num*, which is the number of nodes with the specified spacing, both horizontally and vertically.

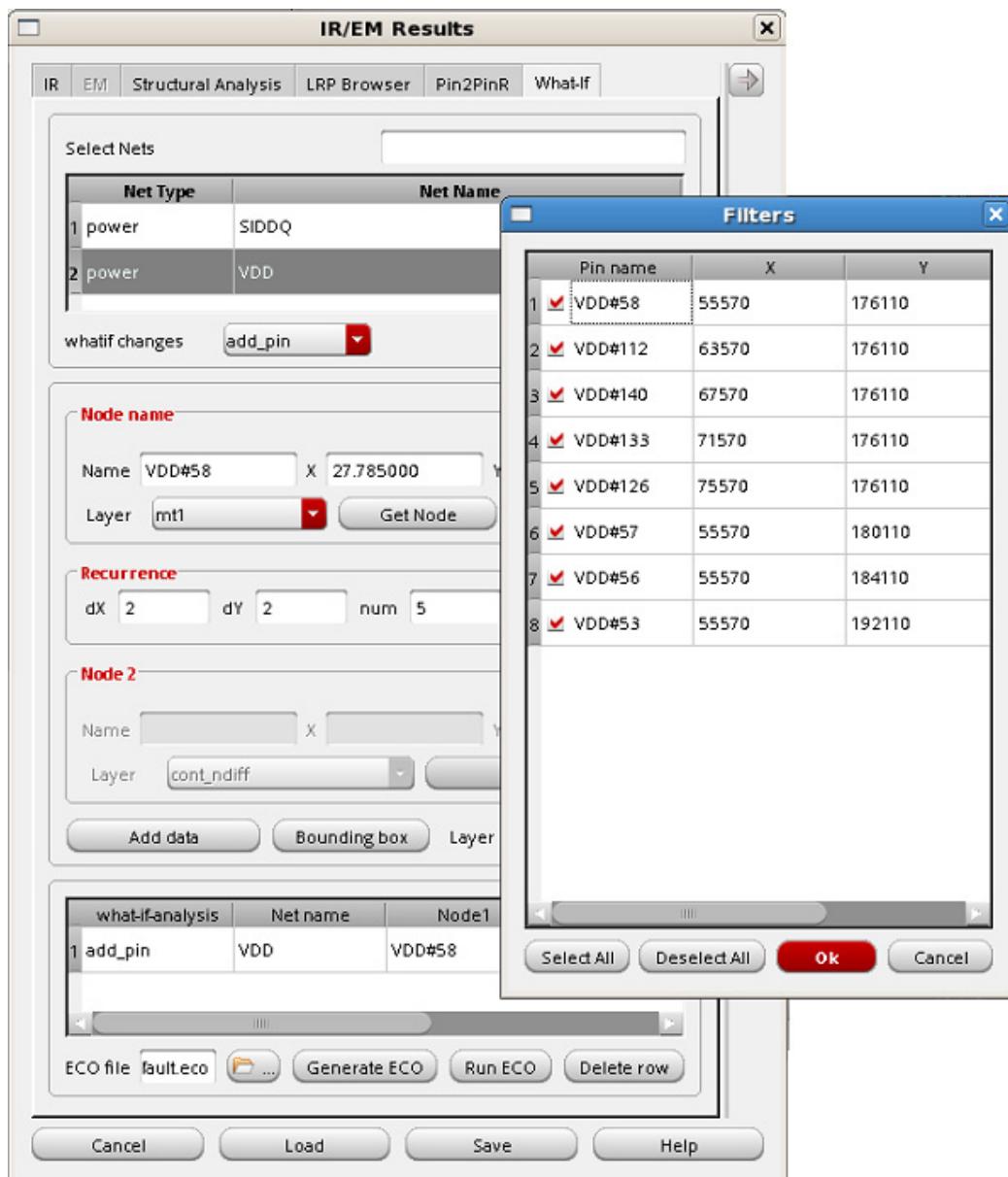
When you click *Recurrence*, the Filters pop-up window opens. It displays a table with names of nodes that will be used for adding pins along with their x- and y-

# Voltus-Fi Custom Power Integrity Solution User Guide

## What-If EMIR Analysis

coordinates. When you click *OK*, the selected nodes will be used to add pins. This information is then populated in the table provided in the What-If form. This is shown below.

**Figure 11-4 Adding Multiple Pins for What-If Analysis**



- In the *Node-2* group box, provide the same information as above for the second node. This field is only enabled when *add\_res* is specified in the *whatif changes* drop-down list.

**Note:** You can choose a node either on the same layer or on a different layer.

- Click *Add data* to add the location of the nodes, one row at a time, in the table provided below this button. You can add multiple pairs of coordinates in the table. The table provides information about the type of What-If analysis, the net name, information about the nodes, and the analysis data.
- Specify the *Layer* on which you want to create the bounding box.
- Click *Bounding box* and then select an area on the Virtuoso layout. Depending on the what-if analysis type, nodes and resistors will be selected from the layout. For example, if `delete_res` ECO command is selected in the *whatif changes* drop-down list, resistors will be selected for deletion. The nodes and resistors are selected on the basis of the specified layer name. These nodes will be listed in the Filters pop-up window. When you click *OK*, all the selected nodes will be used for deleting resistors.

If `add_pin` command is selected in the *whatif changes* drop-down list, then all nodes inside the rectangle, for the specified layer, will be selected. These nodes will be listed in the Filters pop-up window. When you click *OK*, all the selected nodes will be used for adding pins. Their names and the corresponding net names will be added in the table in the What-If form.

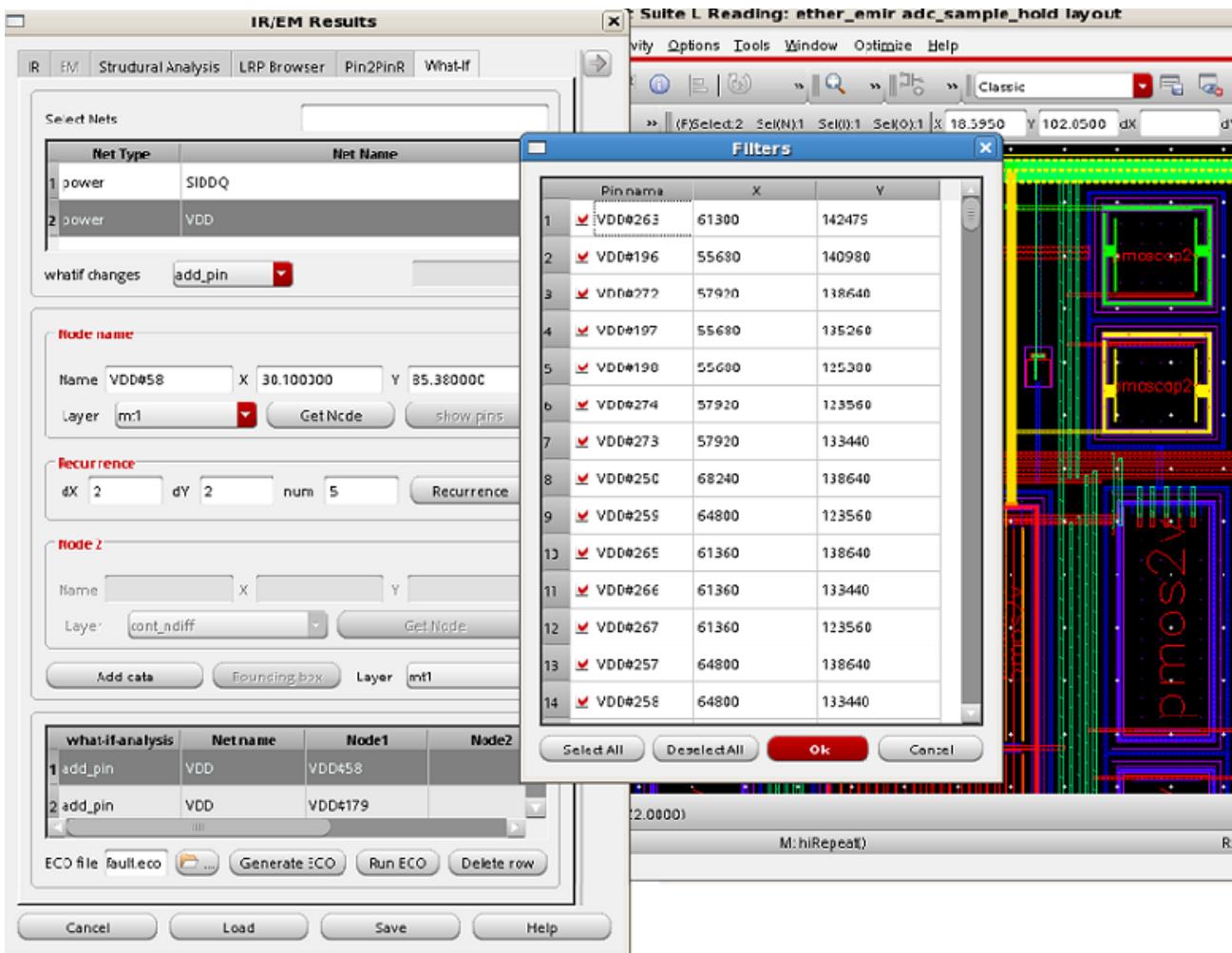
**Note:** The Bounding Box can only be used when `add_pin`, `add_cap`, `delete_res`, and `scale_res` ECO commands are selected in the *whatif changes* drop-down list. An example of using the Bounding Box for adding pins is shown below.

# Voltus-Fi Custom Power Integrity Solution User Guide

## What-If EMIR Analysis

---

**Figure 11-5 Using the Bounding Box Option to Add Pins**



- Specify the path and name of the ECO file to be generated in the *ECO file* field.
- Click *Delete row* to delete the data added in the table. For this, first select the row for which you want to delete the data and then click *Delete row*.
- Click *Generate ECO* to generate the ECO file with the specified filename in the specified directory. The file opens in a pop-up window. A sample ECO file is shown below.

## **Figure 11-6 Sample ECO File**

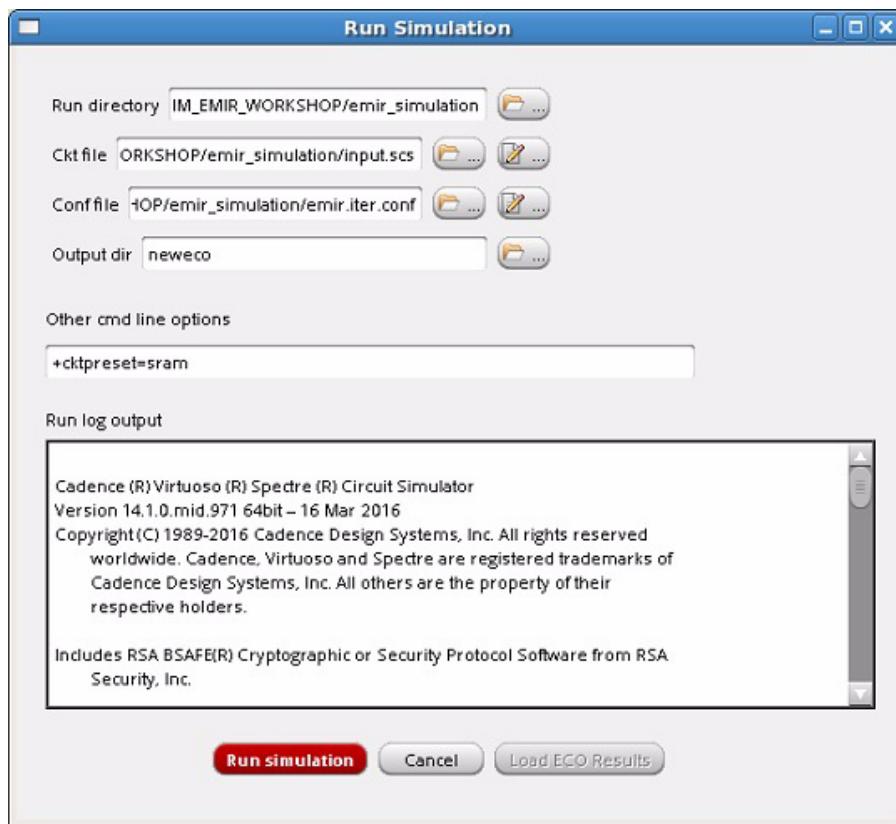
- Click *Run ECO* to run the simulation with the specified What-If ECO changes. When you click *Run ECO*, the Run Simulation form opens. The fields in this form are filled in automatically with information from the Spectre run. For details of how the form is pre-filled by the software, see [Pre-Filling the Run Simulation Form](#).

In this form, check all the pre-filled information to ensure that it is correct. This form is shown below.

## Voltus-Fi Custom Power Integrity Solution User Guide

### What-If EMIR Analysis

**Figure 11-7 The Run Simulation Form**



In this form, the following information is pre-filled:

- The name of the simulation *Run directory*
- The name of the input *Ckt file*
- The name of the simulation configuration file in the *Conf file* field

**Note:** You can edit the *Ckt file* and the *Conf file* by using the edit buttons provided against these options.

- The path of the output directory for the What-If Spectre run in the *Output dir* field, and
- The Spectre command line parameters in the *Other cmd line options* field, for example, if Spectre® XPS is used, then +spice +xps and +cktpreset=sram is specified and if Spectre® APS is used, then +spice +aps is specified

**Note:** For details of command line options, see “Post-Layout Simulation” chapter in the *Virtuoso® Spectre® Circuit Simulator and Accelerated Parallel Simulator User Guide*.

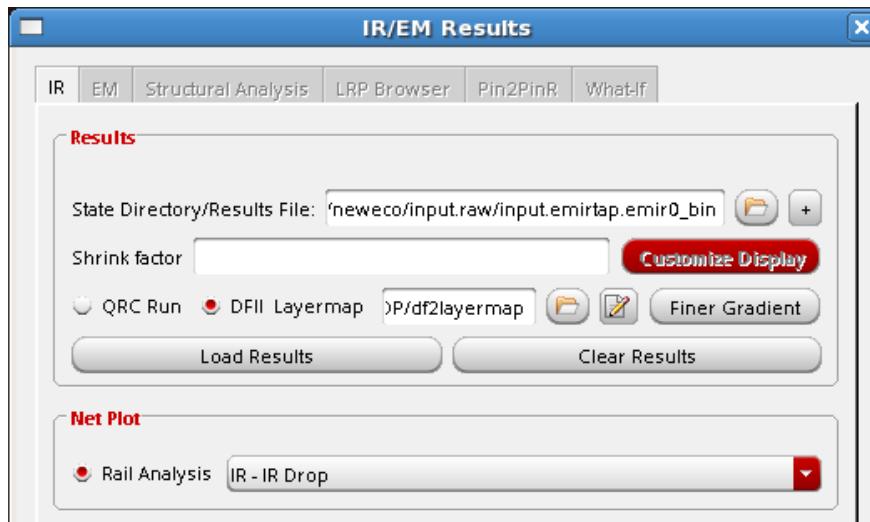
- ❑ Click *Run Simulation*. Once the simulation run is complete, you will get a pop-up telling you that the run is successful. The details of the run are provided in the Run log output.

Errors, if any, are written in the log. You can fix the errors and run the simulation again.

- ❑ Once the simulation run is complete, the *Load ECO Results* button is enabled. This button is used to load the simulation results file that includes the What-if ECO changes. When you click this button, the Run Simulation form closes and the IR/EM Results form opens. In this tab, the new results file is loaded, which includes information about only those nets for which what-if ECO changes have been made.

You can now generate IR/EM plots as per the changes made in ECO simulation. This is shown below.

**Figure 11-8 IR/EM Results Form – Results File with What-If ECO Changes**



## Pre-Filling the Run Simulation Form

The Run Simulation form that opens when you click the *Run ECO* button in the *What-If* analysis tab, is pre-filled with information that is already available from the Spectre run. To ensure that the form is pre-filled, the following information must be available:

- The emir bin file that contains the simulation results from the original Spectre run
- The conf file generated during the original Spectre run
- The log file generated during the original Spectre run

The software performs below steps to extract the data from the original Spectre run for pre-filling the Run Simulation form.

1. The software locates the log file generated by Spectre. The "xps.log" or the "aps.log" file is located either in the same directory as the emir bin file or in the parent directory of the emir bin file.
2. The software parses the ".log" file and searches for the "command line" used in the original Spectre run. The command line contains the "ckt file" and "command line options" used in the original Spectre run.
3. It then creates a directory parallel to the directory containing the "ckt file" for the What-If Spectre run. The new directory is created with the name, "whatIfRunDir". This will contain the output of the What-If Spectre run.
4. The software searches for the "conf" file generated after the original Spectre run in the emir bin file directory and then creates a new "conf" file for the What-If spectre run, in the directory "whatIfRunDir" and performs the following tasks in the newly created "conf" file:
  - Copies the data from the "conf" file from the original Spectre run
  - Changes the solver method from "iteronly" to "itereco"
  - Appends the ECO file, "eco file:ECO\_filename" to the conf file

Now, when you click *Run Simulation*, the simulation runs using the above newly-generated conf file that includes information about the ECO options.

## Output

The output of the above simulation run with ECO changes is a simulation database (\*.emir#\_bin). You can use this simulation database for running the EMIR analysis.

For this, load this simulation database—with ECO changes—when loading the IR drop analysis results on the IR tab of the EM/IR Results form.

The subsequent EMIR analysis will be performed using this simulation database.

## Options for Performing What-If Analysis

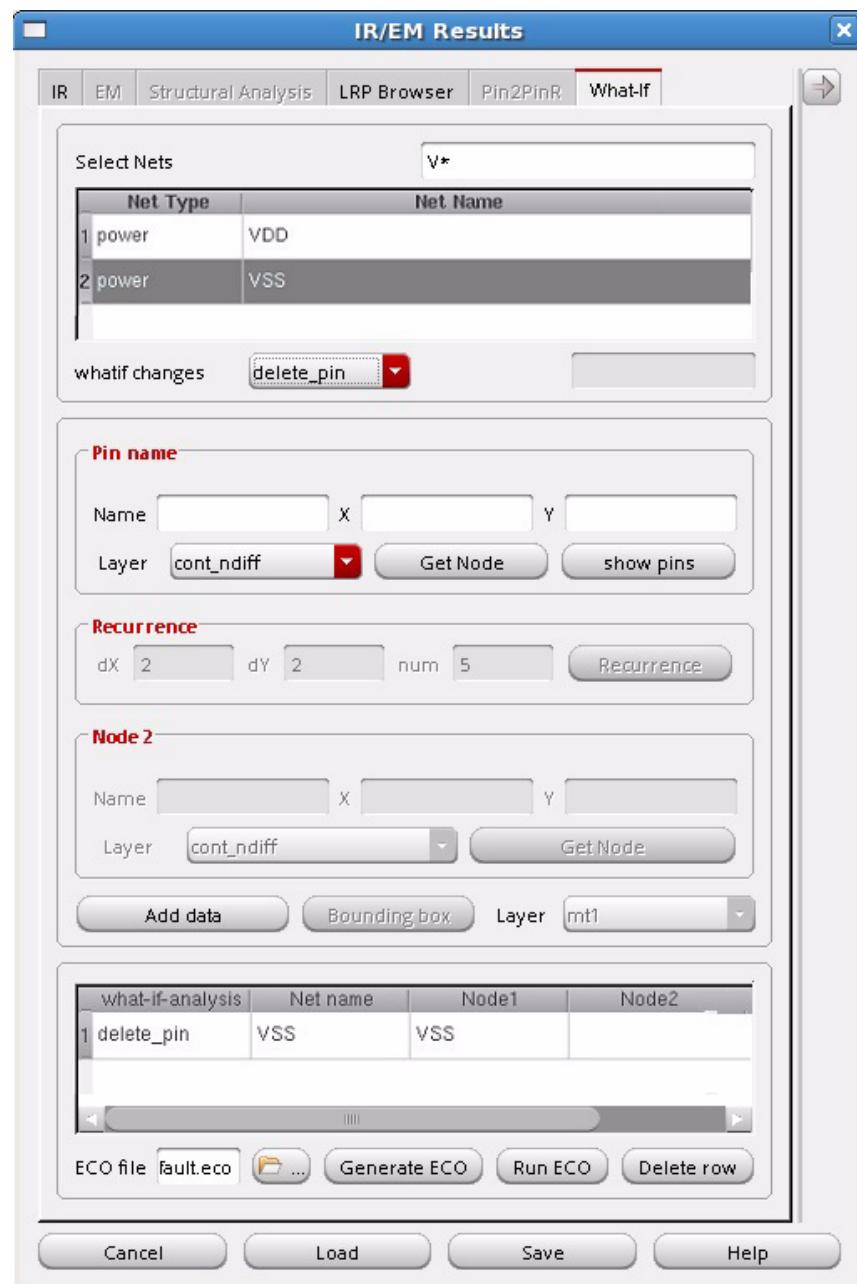
When you select the ECO command that you want to run in the *whatif changes* drop-down list, there are some changes in the options available on the What-If tab. These are detailed below.

- For `add_pin`, the following flows are supported for What-If analysis:
  - You can select a node by retrieving the x- and y- coordinates information on the layout. This node will be used for adding pins.
  - You can add multiple nodes at specified distances from an identified node at once by using the *Recurrence* option provided in the What-If Analysis form. These nodes are used to add pins.
  - You can create a bounding box on the layout for a specified layer. All nodes within the bounding box or the rectangle will be used to add pins.
- The fields on the What-If tab are the same as described above and shown in [Figure 11-3](#) on page 265. For details, see [Performing What-If Analysis in Voltus-Fi-XL](#).
- For `delete_pin`, the following options are available in the What-If form:
  - The *Node name* group box is replaced with the *Pin name* group box. This is shown below.
  - The *show pins* button is enabled. Click this button to view the list of pins on the selected layer. The Filters pop-up opens. The pin name and the x- and y- coordinates of the pin are provided in the pop-up. Select the row and click *OK*. The pin name and its coordinates appear in the table. This is shown below.

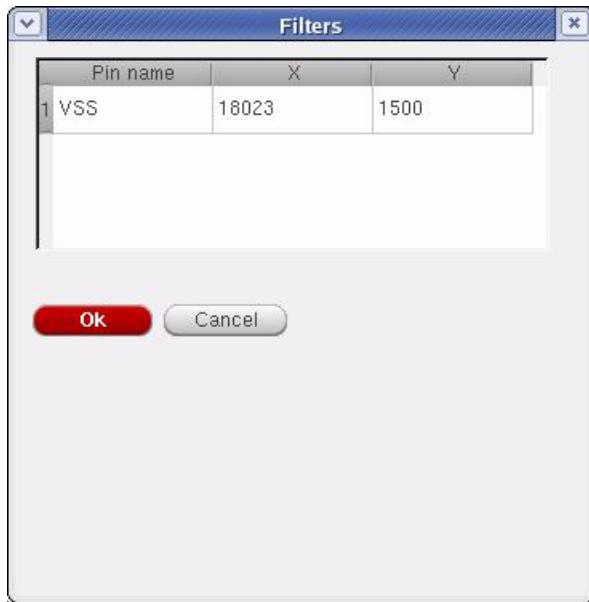
# Voltus-Fi Custom Power Integrity Solution User Guide

## What-If EMIR Analysis

**Figure 11-9 Options for delete\_pin What-If Analysis**



**Figure 11-10 The Filters Pop-Up Window**

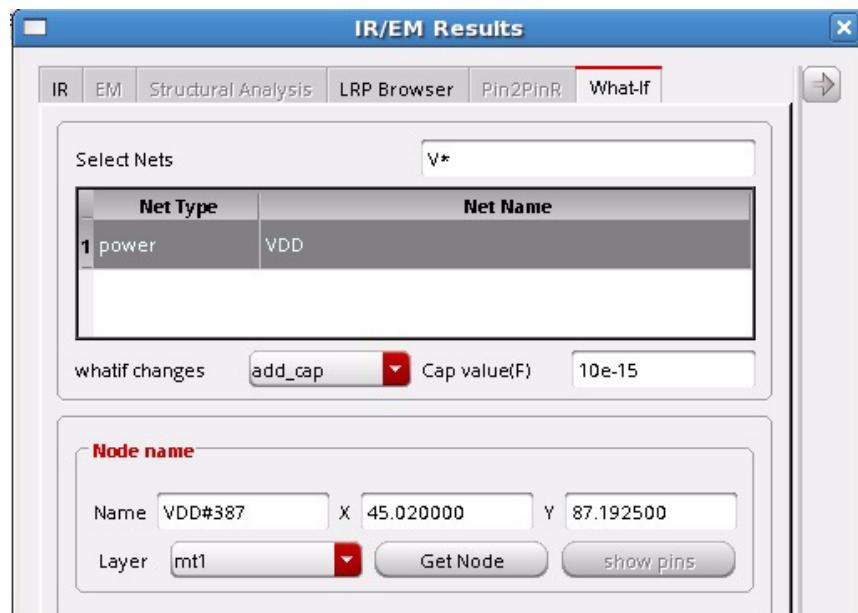


- For `add_cap`, the following options are available in the What-If form:
  - Specify the *Cap value(F)*, which is the capacitance value for the analysis in femtofarads ( $fF$ ).
  - Provide the node information in the same manner as that detailed for `add_pin` option.

## Voltus-Fi Custom Power Integrity Solution User Guide

### What-If EMIR Analysis

**Figure 11-11 Options for add\_cap What-If Analysis**



- For `scale_res`, the following options are available in the What-If form:
  - Specify the *Scale factor(Ohm)* for the analysis.
  - The *Node name* group box is replaced with the *Resistor* group box. Provide the *Resistor* information in the same manner as the node information detailed for the `add_pin` option.

**Figure 11-12 Options for scale\_res What-If Analysis**

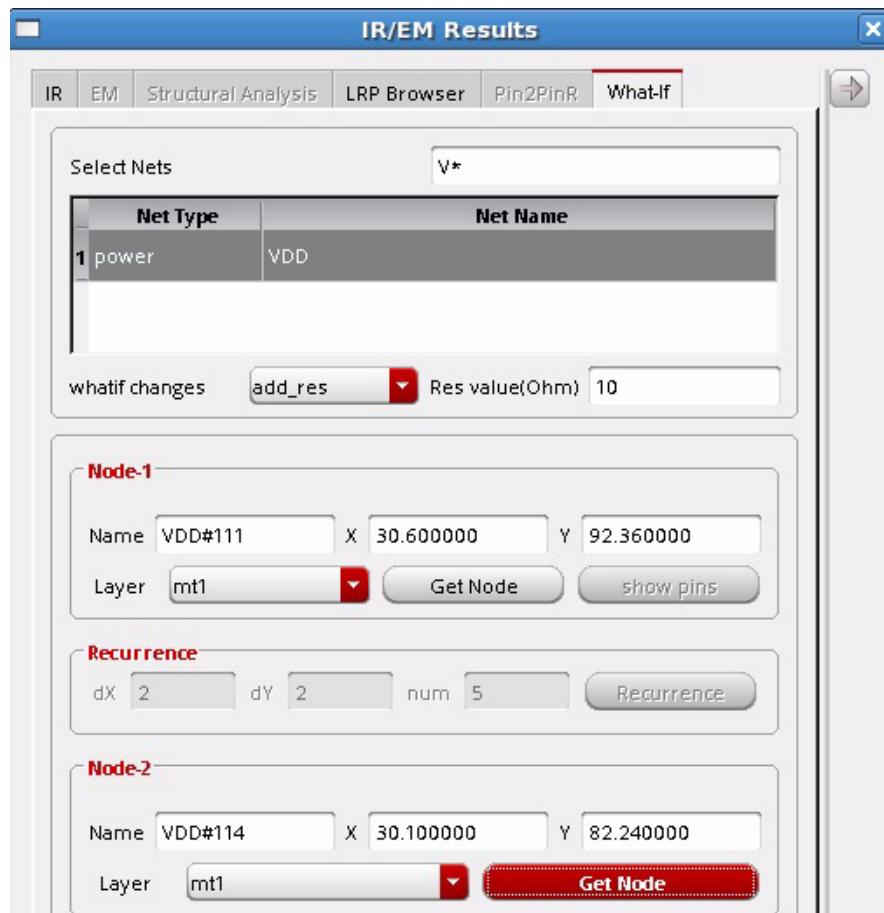


- For add\_res, the following options are available in the What-If form:
  - Specify the *Res value (Ohm)*, which is the resistance value for the analysis.
  - Both the Node group boxes , *Node-1* and *Node-2* are enabled for this option. You can provide the resistor information for multiple nodes.

## Voltus-Fi Custom Power Integrity Solution User Guide

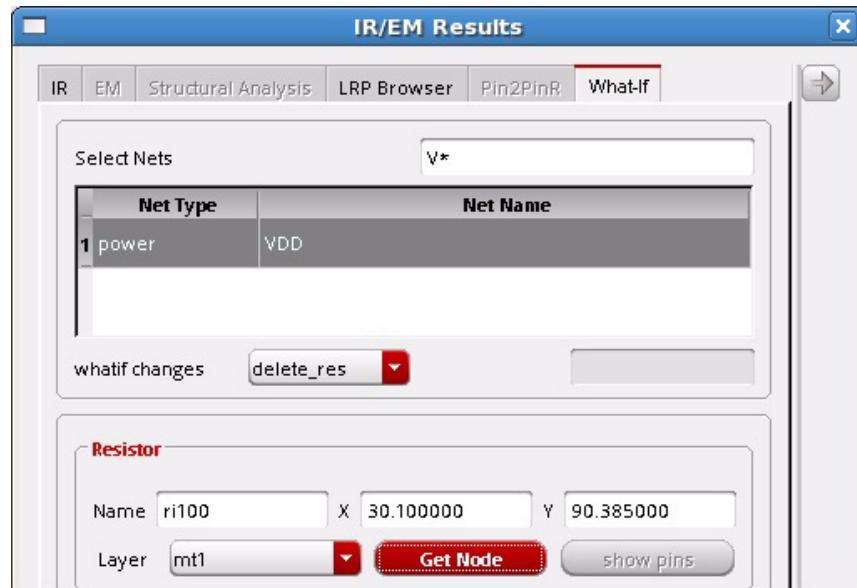
### What-If EMIR Analysis

**Figure 11-13 Options for add\_res What-If Analysis**



- For `delete_res`, the following options are available in the What-If form:
  - The *Node name* group box is replaced with the *Resistor* group box. Provide the *Resistor* information in the same manner as the node information detailed for the `add_pin` option.

**Figure 11-14 Options for delete\_res What-If Analysis**



## **Power-Grid View Generation**

---

- [Overview](#) on page 282
- [Data Requirements and Flow for PGV Creation](#) on page 282
- [Open Access \(OA\)-Based Library Generation](#) on page 286
- [Generating Power-Grid Views](#) on page 286
- [Outputs of PGV Generation](#) on page 289
- [Verifying the PGVs](#) on page 290

## Overview

Increasing operating frequency and density of IC designs increases the need for IR drop and Electromigration (EM) power-grid analysis. Power-grid signoff analysis is provided by Voltus in the digital domain, and Voltus-Fi-XL in the analog domain.

Voltus-Fi-XL provides the ability to create power-grid views (PGVs) of an analog design block that can be used in Voltus for mixed-signal analysis. PGVs contain the following:

- A model view of the design
- Information about the power ports of the cell or block
- Information about the internal power grid, intrinsic capacitance, and the tap current distribution within the cell or block

PGVs provide an accurate characterization of capacitance and currents, and power-grid extraction. Voltus uses these views to model power rail and power distribution information for each instance of the cell in the design for dynamic or static full-chip power-grid analysis.

**Note:** Both Voltus and Voltus-Fi-XL support PGV generation only for power pins.



The power-grid views generated using the Voltus-Fi-XL 6.1.7 ISR1 version of software are compatible only with the 15.21 and later versions of Voltus.

For more information about PGVs, refer to the “Power-Grid Library Generation” chapter in *Voltus IC Power Integrity Solution User Guide*.

You can generate PGVs both in the GUI and batch command mode. Information about generating PGVs in the GUI is covered in subsequent sections.

For generating PGVs using the batch mode, see [Batch Commands for PGV Generation](#) in the “Batch Mode Execution” chapter.

## Data Requirements and Flow for PGV Creation

Power-grid view generation in Voltus-Fi-XL depends upon the simulator-generated database and current files. For generating PGVs in Voltus-Fi-XL, ensure the following while performing simulation using Spectre® APS/XPS.

- Enable the `emir` mode. The EMIR analysis is run like a regular APS/XPS simulation with the additional `+emir` option. For details, see [Simulation Requirements and Setup](#) in the “Data Preparation” chapter.
- Generate the current files for all the power nets for which PGVs are being generated. This is done by specifying UTI commands in the Spectre input file.

The syntax of the command is as follows:

```
uti0 uti signal=net_name start=start_time  
cycles=number_of_cycles_for_this_measure intervals=number_of_points  
clockcycle=clock_period method=[0|1|2][0|1|2][0|1|2]  
filename=filename_prefix_for_this_measure termflag=0 Namemangling=0 eisopt  
save_gnd_caps=1
```

The syntax contains the following parameters:

`signal=signal_name`

Specifies the net name for which current is to be calculated.

**Note:** Ensure that the net name is specified along with the instance hierarchy name. For example, if your instance hierarchy name is `IO`, then specify the signal name as `IO.VDD` for the `VDD` net.

`start=start_time`

Specifies the start time from which the current will be written in the current file.

`cycles=number_of_cycles_for_this_measure`

Specifies the number of clock cycles to be captured in the current file. By default, one cycle is captured.

`intervals=number_of_points`

Specifies the number of time points or measurement intervals within a clock cycle.

`clockcycle=clock_period`

Specifies the length of the clock cycle or the total duration for which the current information is being written in the current file. You must specify the unit.

`method=[0|1|2][0|1|2][0|1|2]`

Specifies the method that the simulator uses in post-processing clock analysis data.

[`0|1|2`] The number in the first position, (`0`), specifies the method of peak current calculation.

[`0|1|2`] The number in the second position, (`1`), specifies the method of average current calculation.

## Voltus-Fi Custom Power Integrity Solution User Guide

### Power-Grid View Generation

---

[012] The number in the third position, (2), specifies the method of RMS current calculation.

0: Reports the maximum out of each interval over all periods.

1: Reports the average out of each interval over all periods.

2: Does not calculate peak, average, or RMS current.

The default is 212, which is average current calculation.

- ❑ `filename=filename_prefix_for_this_measure`

Specifies the prefix for the current files that are being generated.

**Note:** For Voltus-Fi-XL, ensure that the prefix provided is "vps". For example, your filename could be `vps_VDD.ptiavg`.

- ❑ `termflag`

Specifies the terminals which will be output.

- ❑ `Namemangling`

Specifies namemangling. When set to 0, specifies that names should not be mangled.

Here is an example of this command.

```
uti0 uti signal="IO.VDD" start=0n cycles=1 intervals=133 clockcycle=2ns  
method="212" filename="vps" termflag=0 Namemangling=0
```

- In the Spectre input file, specify `eisopt save_gnd_caps=1` to include the grid capacitances in the EMIR database so that they can be used in power-grid view generation.

The following table shows the required inputs for generating power-grid views in Voltus-Fi-XL.

**Table 12-1 Inputs Required for PGV Generation**

Type of Inputs	Details
Simulation database	It is required as input for performing EMIR analysis in Voltus-Fi-XL.
QRC Techfile	It is required for reading the technology data for PGV generation.
LEF DEF file	It includes the LEF information for the technology data. It is required for port connectivities.

# Voltus-Fi Custom Power Integrity Solution User Guide

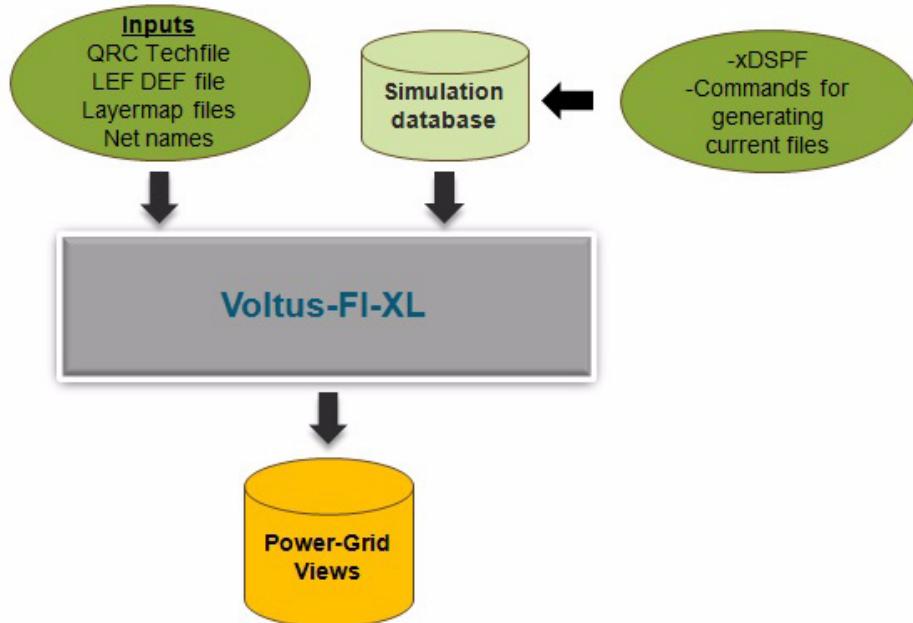
## Power-Grid View Generation

---

Type of Inputs	Details
Macro LEF file	It specifies the LEF information for the design. <b>Note:</b> In case the technology data and design data are in the same LEF file, you do not need to specify the Macro LEF file separately.
OA Library	It specifies the OpenAccess (OA) database library name for PGV generation.
PGDB layermap file	It provides mapping between the layer names in the xDSPF that is written in the simulation EMIR database and the layer names in the technology file.
LEF DEF layermap file	It provides mapping between the layer names in the LEF file to the layer names in the technology file.
Net names	It is required for pin information.

The following diagram illustrates the flow of PGV generation in Voltus-Fi-XL:

**Figure 12-1 Power-Grid View Generation Flow in Voltus-Fi-XL**



## Open Access (OA)-Based Library Generation

Voltus-Fi-XL lets you specify an OpenAccess (OA) database for PGV Generation. You can either specify the LEF technology file/Macro LEF file or the OA libraries. The OA library files contain information from both the LEF technology file and the Macro LEF file. The layermap file is automatically created from the specified OA file. Alternatively, you can specify the layermap file.

In the GUI, the OA libraries can be specified in the *OA Libs* field of the Create Power Grid Views form. This is shown in the subsequent section.

In the batch mode, the OA library can be specified by using the `-oa_lib` parameter of the `create_pgv` command.

## Generating Power-Grid Views

Voltus-Fi-XL creates PGVs for both static and dynamic rail analysis. The software creates PGVs for static rail analysis along with the PGVs for dynamic rail analysis by using the average values of the current waveforms. It does not create PGVs for static rail analysis only.

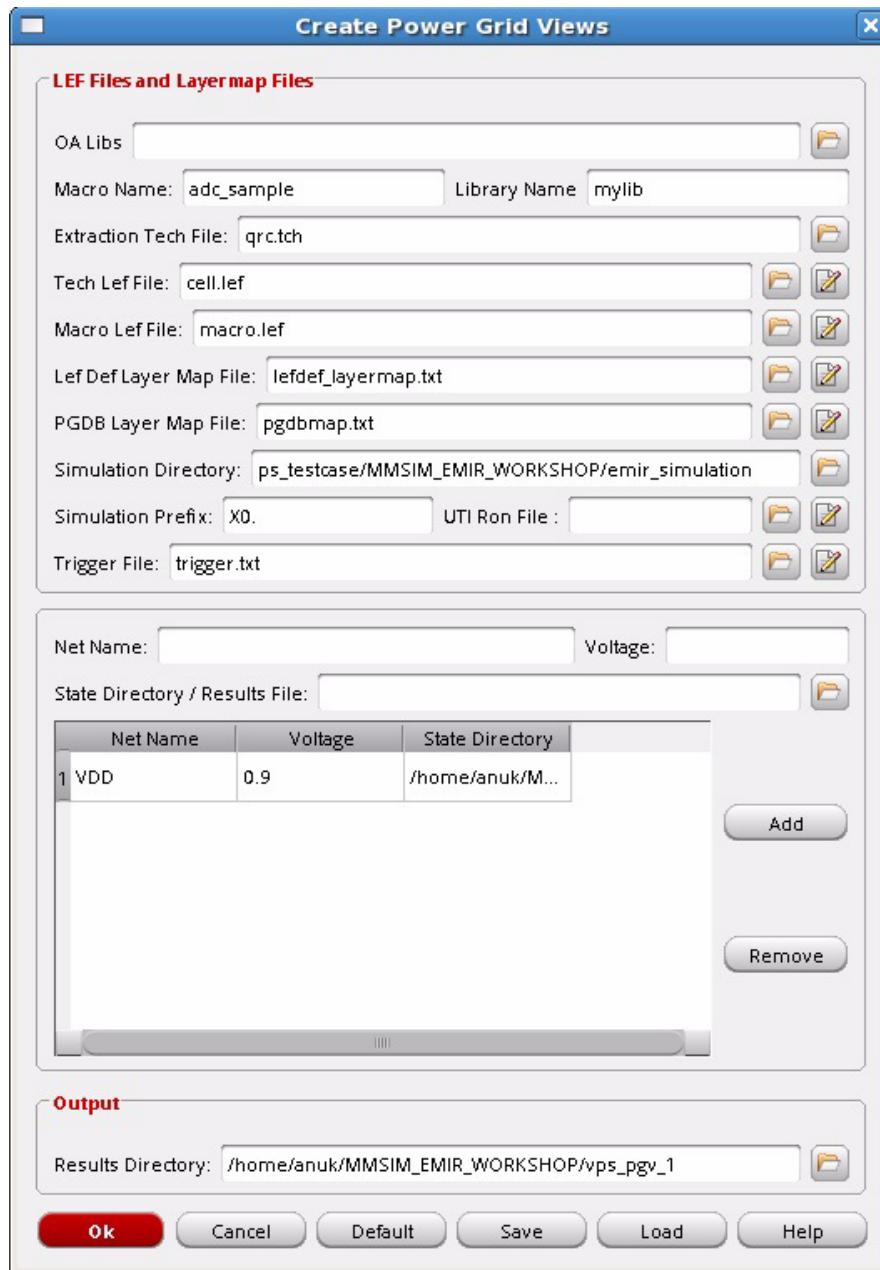
Follow these steps to create power-grid views in Voltus-Fi-XL:

- Choose *IR/EM Analysis – Create Power Grid View*. The Create Power Grid Views form opens. This form is shown below.

## Voltus-Fi Custom Power Integrity Solution User Guide

### Power-Grid View Generation

**Figure 12-2 Create Power Grid Views Form**



- In the *LEF Files and LayerMap Files* group box, specify the various filenames.
    - Specify the *OA Libs*, which is the name of the OA database library.
- Note:** If this option is specified, you do not need to specify the *Tech Lef File/Macro Lef File* options.

## Voltus-Fi Custom Power Integrity Solution User Guide

### Power-Grid View Generation

---

- Specify the *Macro Name*, which is the top cell name.
- Specify the *Library Name*, which is the name of the library. This option is used to create a library with a name different than that of the macro. By default, a library is created with the same name as that of the macro.
- Specify the *Extraction Tech File*, which provides the name of the technology file that will be used for reading the technology data.
- Specify the *Tech Lef File*, which includes the LEF information for the technology data.
- Specify the *Macro Lef File*, which *includes the* LEF information for the design.

**Note:** In case the technology data and design data are in the same LEF file, use any of the two options, *Tech Lef File* or *Macro Lef File* to specify the filename.

- Specify the *Lef Def Layer Map File*, which provides the mapping between LEF layer names and technology layer names.
- Specify the *PGDB Layer Map File*, which provides the mapping between xDSPF layer names that are written in the simulation database, and the technology layer names.

**Note:** If an xDSPF layer name is missing in this file, the PGVs will not contain data for that layer.

**Note:** For more information about the LEF layermap file and PGDB layermap file formats, see [LEF Layermap File](#) and [PGDB Layermap File](#) in the “File Formats” chapter.

The Tech Lef file, the Macro Lef file, the Lef Def Layer Map file, and the PGDB Layer Map file can be created and edited using the edit buttons provided against each of these fields.

- Specify the *Simulation Directory*, which contains the current (\*.ptiavg) files.
- Note:** The PGV generation fails if this directory does not contain the current files for all the nets for which the PGVs are being generated.
- Specify the *Simulation Prefix*, which is the prefix used for creating an instance while performing the simulation. Ensure that the prefix is specified along with the hierarchy separator.

For example, you can specify “`XO.`” or “`IO.`”. In these examples, “`XO`” or “`IO`” are prefixes and “`.`” is the hierarchy separator. You can check the prefix used during simulation in your simulator input file.

## Voltus-Fi Custom Power Integrity Solution User Guide

### Power-Grid View Generation

---

- ❑ Specify the *UTI Ron File* that is generated by Spectre. This file contains multiple on-resistance (Ron) values, that is minimum, maximum, and average, for powergates. A sample file is shown below:

D-Term-Net	S-Term-Net	Ron_avg(Ohm)	Ron_max(Ohm)	Ron_min(Ohm)	Power-Gate
outp	VDD	1268.34	1302.38	1230.52	MPM3
outp	VDD	1268.03	1302.06	1230.2	MPM3@39
outp	VDD	1268	1302.04	1230.18	MPM3@40
outp	VDD	1268.03	1302.07	1230.21	MPM3@41

If this file is not specified, Voltus-Fi-XL uses the Ron values from the MMSIM simulation database (\*.emir#\_bin file).

- ❑ Specify the *Trigger File*, which is a text file that is used during dynamic current simulation to run a detailed analysis for determining current distribution. For detailed information about the trigger file format and content, see [Format and Contents of Trigger File](#) in the “File Formats” chapter.
- ❑ Specify the *Net Name* and the corresponding *Voltage*.
- ❑ Specify the *State Directory / Results File*, which is the simulation result file. The naming convention of the simulation result file is, \*.emir#\_bin. For example, your result file could be named, xps.emir0\_bin.
- ❑ Click *Add* to add the net to the list.
- ❑ Click *Remove* to remove a net from the list.
- In the *Output* group box, specify the *Results Directory*, which is the output directory for the power-grid views. By default, the power-grid views are stored in the work directory.  
**Note:** This option lets you save different outputs without overwriting the existing ones. However, if the output is stored in the default directory, then the new output file created while generating the power-grid views in the current run overwrites the existing one.
- Click *OK*.  
**Note:** The option to save the configuration and load it for populating fields is available in this form.

## Outputs of PGV Generation

Following are the outputs of PGV generation in Voltus-Fi-XL:

- A binary database that contains the following:

- Geometric views of the cell
- Port information
- Power-grid views
- Text report and summary file with detailed information about the power-grid views.

The output of PGV generation in Voltus-Fi-XL is used in *Voltus* for dynamic power calculation and static and dynamic IR drop analysis. For more information, see “Power-Grid Library Generation” chapter in *Voltus IC Power Integrity Solution User Guide*.

## Verifying the PGVs

After generating the PGVs, you can check whether the PGVs are correctly generated or not. Listed below are a couple of issues that you may encounter while generating PGVs, and the information to help resolve them.

### ■ No current information in the PGVs

If there is no current information in the PGVs, check your log file for a warning titled, VOLTUS\_LGEN-3065. This warning is issued when there is a mismatch between taps in the current files and the simulation database. If all taps fail, the current value in the PGV will be zero. To check the names and number of taps in the current file, you can use Itaputil. Itaputil is a Cadence utility that either reports characteristics of tap current files or extracts portions of the data and places them into a new tap current file.

For example, if your current file is vps\_VDD.ptiavg, you can use the following command to list the tap names in this file:

```
itaputil list vps_VDD.ptiavg > names
```

**Note:** To get help on the complete list of available commands in Itaputil, type the following in the xterm or shell:

```
itaputil -h
```

### ■ The current file does not have any waveforms

If the current file does not have any waveforms, it could be because of an error in the UTI command used for generating the current file. For resolving this issue, use Itaputil to check the tap names in the current file and to dump the current data in the file.

## **Batch Mode Execution**

---

- [Overview](#) on page 293
- [Batch Command for xDSPF Generation](#) on page 294
  - [extract\\_xdspf](#) on page 294
- [Batch Commands for Static Current Analysis](#) on page 300
  - [ground\\_pins](#) on page 301
  - [power\\_pin](#) on page 302
  - [run\\_static\\_simulation](#) on page 303
  - [setup\\_simulation](#) on page 306
- [Batch Commands for IR Reports](#) on page 308
  - [load\\_ir\\_results](#) on page 308
  - [print\\_ir\\_report](#) on page 308
  - [print\\_rlp\\_report](#) on page 312
  - [pin\\_2\\_pin\\_res](#) on page 313
- [Batch Commands for EM Reports](#) on page 315
  - [load\\_em\\_results](#) on page 315
  - [print\\_em\\_report](#) on page 316
- [Batch Commands for SHE Analysis](#) on page 320
  - [load\\_view](#) on page 320
  - [print\\_thermal\\_report](#) on page 321
- [Batch Commands for PGV Generation](#) on page 323
  - [create\\_pgv](#) on page 323

## **Voltus-Fi Custom Power Integrity Solution User Guide**

### Batch Mode Execution

---

- ❑ [set mmsim pgv nets](#) on page 326
  - [Batch Commands for Querying](#) on page 328
    - ❑ [search\\_res\\_id](#) on page 328
    - ❑ [res\\_id\\_info](#) on page 329
    - ❑ [search\\_node\\_id](#) on page 332
    - ❑ [node\\_id\\_info](#) on page 333

## Overview

You can run Voltus-Fi-XL in batch mode from the UNIX window, shell, or xterm in the following way:

```
vfibatch -cmd <cmd_file>
```

Where:

<cmd\_file> is a file with a list of batch commands that define the input data, control the execution of the analysis, and specify the output location.

**Note:** For detailed descriptions of the batch commands used in Voltus-Fi-XL, see the following sections:

- [Batch Command for xDSPF Generation](#)
- [Batch Commands for Static Current Analysis](#)
- [Batch Commands for IR Reports](#)
- [Batch Commands for EM Reports](#)
- [Batch Commands for SHE Analysis](#)
- [Batch Commands for PGV Generation](#)
- [Batch Commands for Querying](#)

**Note:** You can also use the emirreport command, from the simulation hierarchy, to generate textual and html reports for IR and EM analysis. For details, see Generating Reports in Voltus-Fi XL in the “IR Drop Analysis Results” chapter.

## Batch Command for xDSPF Generation

You can generate the xDSPF file from Quantus QRC in the Voltus-Fi-XL flow using the below command:

### **extract\_xdspf**

```
extract_xdspf
    -signalEM {true | false}
    [-coloring {true | false}]
    [-PGNets list_of_pg_nets]
    [-subcktPinOrder pin_order_file]
    -process {n10pg | n10 | n20 | n16 | n18 | others}
    -techDir tech_dir_name | -techLib tech_lib_file_name
    [-techName technology_name]
    -temperature value
    [-multiCpu number_of_cpus]
    -inputType {pvs | calibre}
    -inputDir input_dir_name
    -RunName input_run_name
    -outputDir output_dir_name
    [-outputDspf output_dspf_filename]
    [-customExtract extract_Commands_File]
    [-groundNet ground_net_name]
    [-blockingCellFile parasitic_blocking_device_cell_names_file]
    [-subNodeChar sub_node_char]
    [-compression {true | false}]
    [-reduction {true | false}]
    [-commentSubCkt {true | false}]
    [-subConductorModel {true | false}]
    [-fractureViaCnt via_count_number]
    [-arrayViaSpacing {auto | value}]
```

Generates the xDSPF file using Quantus from within Voltus-Fi-XL.

**Note:** Before running this command, ensure that the Quantus hierarchy is available as the tool will call the Quantus binary from this path and also that the PVS or Calibre® run is performed successfully.

### **Parameters**

-arrayViaSpacing {auto | value}

## Voltus-Fi Custom Power Integrity Solution User Guide

### Batch Mode Execution

---

Specifies the via array spacing, which is the distance, floating point micron value, that will be used as the maximum distance criteria for grouping vias within the same array.

The default value of via array spacing is 0.35 times the via size for N10, N16, and N20 processes. For other processes, the default is *auto*, in which case the software sets the via spacing automatically. You can specify either *auto* or a floating value for this option.

*Default:* *auto*

This is an optional parameter.

`-blockingCellFile parasitic_blocking_device_cell_names_file`

Specifies the file containing parasitic blocking device cell names, if any. Parasitic extraction for these devices will be skipped.

This is an optional parameter.

`-coloring {true | false}`

When set to *true*, this parameter enables color-aware EM analysis. This is an optional parameter.

`-commentSubCkt {true | false}`

When set to *true*, this parameter will comment out the top sub circuit statement in the xDSPF file.

This is an optional parameter.

`-compression {true | false}`

When set to *true*, specifies that the xDSPF file should be generated in a compressed format.

This is an optional parameter.

`-customExtract extract_Commands_File`

## Voltus-Fi Custom Power Integrity Solution User Guide

### Batch Mode Execution

---

Specifies the file with user-customized extract commands. When this file is specified, the Voltus-Fi-generated CCL will not have any extract commands and the commands from the customized extract commands file will be added as is in the Voltus-Fi-generated CCL.

A sample customized extract command file is as follows:

```
extract \
    -selection "all" \
    -type "rc_coupled"
```

**Note:** You can create customextract file by using either single or multiple extract settings. Ensure that the commands are in the Quantus QRC CCL file syntax as this file is added as is in the Voltus-Fi-generated CCL.

This is an optional parameter.

For information about the format of the Voltus-Fi-generated CCL file and its example, see [Common Command Language \(CCL\) File](#) in the “File Formats” chapter.

**-fractureViaCnt** *via\_count\_number*

Specifies the fracture via count to be used for dividing pseudo vias into segments during extraction.

*Default:* 1

**-groundNet** *ground\_net\_name*

Specifies the name of the ground net on which extraction is to be performed.

*Default:* 0

This is an optional parameter.

**-inputDir** *input\_dir\_name*

Specifies the directory in which the PVS or Calibre data is stored. This is a required parameter.

**-inputType** {pvs | calibre}

Specifies whether the input data is from the PVS or Calibre tool. This is a required parameter.

**-multiCpu** *number\_of\_cpus*

## Voltus-Fi Custom Power Integrity Solution User Guide

### Batch Mode Execution

---

Specifies the number of CPUs to be used for distributed processing during the Quantus (QRC) extraction run.

This is an optional parameter. By default, single CPU is used.

`-outputdir output_dir_name`

Specifies the directory in which the xDSPF file that is generated will be saved. This is a required parameter.

`-outputDspf -output_dspf_filename`

Specifies the name of the xDSPF file that is generated. This is an optional parameter. If the file name is not specified, the software generates the file, *<runName>.dspf* by default.

`-PGNets list_of_pg_nets`

Provides the list of PG nets, separated by spaces, for which the xDSPF file is to be generated. This is an optional parameter. It is only specified when the `-signalEM` parameter is set to `false`.

`-process {n10pg | n10 | n20 | n16 | n18 | others}`

Specifies the process node.

`n10pg`: for N10 process node with PG rules

`n10`: for N10 process node

`n20`: for N20 process node

`n16`: for N16 process node

`n18`: for N18 process node

`others`: for N20 and above process nodes

`-reduction {true | false}`

When set to `true`, this parameter enables the `reduce_i_cards` option while generating the xDSPF file.

This is an optional parameter.

`-RunName input_run_name`

Specifies the name of the PVS or Calibre run. This is a required parameter.

`-signalEM {true | false}`

## Voltus-Fi Custom Power Integrity Solution User Guide

### Batch Mode Execution

---

Specifies whether the names of signal nets should be included in the xDSPF file or not. If signal nets are not to be included in the EM analysis, set this parameter to `false`.

`-subConductorModel {true | false}`

When set to true, specifies that parasitic resistor models for conductor layers should be included during extraction. This is used to support different EM rules for poly layers in EM analysis.

This is an optional parameter.

`-subNodeChar sub_node_char`

Specifies the character that will be used for subnode identification.

Default: #

This is an optional parameter.

`-subcktPinOrder pin_order_file`

Specifies the name of the sub-circuit pin order file. The pin order is specified in the xDSPF file using the .SUBCKT and .ENDS statements. You can use this option to provide a file with a different pin order that will override the order specified using the default .SUBCKT statement in the xDSPF file.

This is an optional parameter.

`-techDir tech_dir_name | -techLib tech_lib_file_name`

Use either the `-techDir` parameter to specify the name of the directory containing the tech file or use the `-techLib` parameter to specify the path of the technology file that Quantus will use for generating the xDSPF file.

One of the two parameters must be specified.

`-techName technology_name`

Specifies the technology file name to be used for extraction. This parameter is optional if the `-techDir` parameter has been specified.

`-temperature value`

Specifies the temperature at which the extraction will be performed. It is specified in degree Celsius.

This is a required parameter.

## Example

- The following command generates the xDSPF file, test\_lib.dspf in the vfi\_quantus1 directory:

```
extract_xdspf \
    -signalEM true \
    -subcktPinOrder ./your_subckt_pin_order_file
    -process n20 \
    -techLib pvs_tech.lib \
    -techName n20tech \
    -temperature 25 \
    -inputType pvs \
    -inputDir test_lib/pvs \
    -RunName test_lib \
    -outputdir vfi_quantus1 \
    -compression true \
    -subNodeChar ":" \
    -groundNet "vss" \
    -customExtract myextract.txt
```

## Batch Commands for Static Current Analysis

You can setup and run static current analysis in the Voltus-Fi-XL flow using the following commands:

- ground\_pins
- power\_pin
- run\_static\_simulation
- setup\_simulation

## **ground\_pins**

`ground_pins gnd_pin_list`

Specifies the names of the virtual ground pins to which the SPICE subcircuit terminal will be connected.

### **Parameters**

`gnd_pin_list`      Specifies the names of the virtual ground pins.

### **Example**

The following command is used to specify the names of two ground pins, VSS and VSSA:

`ground_pins VSS VSSA`

## **power\_pin**

```
power_pin  
  -pin_name pin_name  
  -voltage voltage_value
```

Specifies the power pin and its corresponding voltage value. To specify multiple power pins, use this command multiple times.

### **Parameters**

**-pin\_name *pin\_name*** Specifies the name of the virtual power pin to which the SPICE subcircuit terminal should be connected.

**-voltage *voltage\_value***  
Specifies the voltage value of the power pin.

### **Example**

The following commands are used to specify the names of two power pins, VDD and VDDA, and their corresponding voltage values:

```
power_pin \  
  -pin_name VDD \  
  -voltage 0.9  
power_pin \  
  -pin_name VDDA \  
  -voltage 1.1
```

## **run\_static\_simulation**

```
run_static_simulation
  -method {Ipeak | Iavg}
  -nets net_name_list
  [-freq freq_value]
  [-global_activity number]
  [-activity_file file_name]
  [-vcd vcd_file_name]
  [-clock_file clock_file_name]
  [-average_power power_value_in_W]
  [-power_file power_file_name]
```

Runs static current analysis.

### **Parameters**

**-activity\_file file\_name**

Specifies the path of the activity file, which contains the activity factor on any node of the circuit.

This is an optional parameter. It is only used for the average static current analysis.

**-average\_power power\_value\_in\_W**

Specifies the average power of the design. The default unit is  $\text{W}$  (Watts). If the value is in any other unit, it must be specified. For example,  $20\text{mW}$ .

You can specify a single value or multiple values depending upon whether the same or a different power is to be used for the specified nets. This is shown in the examples below.

This is an optional parameter. This is only used for the average static current analysis.

**-clock\_file clock\_file\_name**

Specifies the name of the clock file, which contains information about the clock nodes, subcircuit definitions, and subcircuit instance names and their frequencies.

This is an optional parameter. This is only used for the average static current analysis.

**-freq freq\_value**

Specifies the value of the dominant frequency of the design.  
The unit is in mHz (Mega Hertz).

This is an optional parameter. This is only used for the average static current analysis.

`-global_activity number`

Specifies the average number of times all the unset nodes switch in a clock cycle.

This is an optional parameter. This is only used for average static current analysis.

`-method {Ipeak | Iavg}`

Specifies whether the peak or average static analysis will be performed.

This is a required parameter.

`-nets net_name_list`

Specifies the list of nets for which current analysis is to be performed.

This is a required parameter.

`-power_file power_file_name`

Specifies the name of the average power file, which contains the average power for specific sub circuits.

This is an optional parameter. This is only used for the average static current analysis.

`-vcd vcd_file_name`

Specifies the vcd file, which contains information about the number of transitions for each net.

This is an optional parameter. This is only used for the average static current analysis.

## Examples

- The following command is used to run the average static current analysis.
  - In this example, the same power value is assigned to the two nets, VDD and VSS:  
`run_static_simulation \`

## Voltus-Fi Custom Power Integrity Solution User Guide

### Batch Mode Execution

---

```
-method Iavg \
-nets {VDD VSS} \
-freq 300 \
-global_activity 0.25 \
-average_power 20mW
```

- In this example, different power values are assigned to the three nets, VDD\_1 , VDD\_2 , and VSS\_1:

```
run_static_simulation \
-method Iavg \
-nets {VDD_1 VDD_2 VSS_1} \
-freq 300 \
-global_activity 0.25 \
-average_power {10mW 8mW 18mW}
```

**Note:** The above command is only valid if you provide power values for all the specified nets.

- The following command is used to run the peak static current analysis:

```
run_static_simulation \
-method Ipeak \
-nets {VDD VSS}
```

### Related Topic

For more information about the activity file, clock file, vcd file, and the power file formats, see [Static Current Analysis Files](#) in the “File Formats” chapter.

## **setup\_simulation**

```
setup_simulation
  -spice_netlist spice_file_name
  [-simulation_temperature temp_value]
  [-hierarchy_separator char_name]
  [-topcell cell_name]
  [-output_directory dir_name]
```

Sets up the options for static current analysis.

### **Parameters**

**-hierarchy\_separator *char\_name***

Specifies the hierarchical separator in the circuit file. This is an optional parameter. This is specified only if the hierarchy separator is other than “/”.

**-output\_directory *directory\_name***

Specifies the name of the output directory where the current database will be saved.

This is an optional parameter. If this parameter is not specified, the output is stored in the current working directory.

**-simulation\_temperature *temp\_value***

Specifies the temperature, in degree Celsius, at which the simulation will be performed.

This is an optional parameter.

**-spice\_netlist *spice\_file\_name***

Specifies the SPICE netlist file, which is the DSPF file.

This is a required parameter.

**-topcell *cell\_name***

Specifies the name of the design. This is an optional parameter.

By default, the tool populates the top cell name from that specified in the layout.

## **Voltus-Fi Custom Power Integrity Solution User Guide**

### Batch Mode Execution

---

#### **Example**

The following command is used to set up simulation:

```
setup_simulation \
-spice_netlist design.spice \
-simulation_temperature 100 \
-topcell designA \
-output_directory ./vps_simulation_1
```

## Batch Commands for IR Reports

You can load and print the results for IR drop analysis in the Voltus-Fi-XL flow using the following commands:

- [load\\_ir\\_results](#)
- [print\\_ir\\_report](#)

### **load\_ir\_results**

```
load_ir_results  
  -results_file_name
```

Loads the results of IR drop analysis in the Voltus-Fi-XL flow.

#### **Parameter**

*-results\_file\_name*

Specifies the directory in which the IR drop analysis results should be loaded.

#### **Example**

- The following command loads the results for IR drop analysis that are stored in the `xps.emirtap.emir0_bin` file:

```
load_ir_results \  
 ./abc/raw.out/xps.emirtap.emir0_bin
```

### **print\_ir\_report**

```
print_ir_report  
  -net {all_power | netname}  
  -type {ir | iv | rc | rcavg | rcrms | reffective}  
  [-threshold threshold_value]  
  -filename output_file_name
```

Prints the results of IR drop analysis in the Voltus-Fi-XL flow. You can generate IR reports for all the power nets or for specific nets.

## Voltus-Fi Custom Power Integrity Solution User Guide

### Batch Mode Execution

---

**Note:** Before running this command, ensure that the IR drop results are loaded using the `load_ir_results` command.

#### Parameters

`-filename output_file_name`

Specifies the name of the IR drop analysis report file. This is a required parameter.

`-net {all_power | netname}`

Specifies whether the IR drop analysis report is to be printed for all the power nets or for the specified net.

Select `all_power` to print the IR report for all the power nets.  
Select `netname` to print the IR report for the specified net.

This is a required parameter.

`-threshold threshold_value`

Specifies that resistors or nodes that have a threshold IR drop ratio value greater than the specified value will be reported.

This is an optional parameter.

*Default:* The report prints results for all resistors or nodes.

`-type {ir | iv | rc |rcavg | rcrms | reffective}`

Specifies the analysis type for which the report is being generated.

`ir` – specifies that the report is generated for IR drop analysis.

`iv` – specifies that the report is generated for transistor-based supply voltage data analysis.

`rc` – specifies that the report is generated for peak resistor current density analysis.

`rcavg` – specifies that the report is generated for average resistor current density analysis.

`rcrms` – specifies that the report is generated for RMS resistor current density analysis.

`reffective` – specifies that the report is generated for effective resistance analysis.

This is a required parameter.

### **Examples:**

- The following command prints the IR drop analysis report for all the nets for `ir` analysis in the `all_ir.txt` file:

```
print_ir_report \
-net all_power \
-type ir \
-filename all_ir.txt
```

- The following command prints the IR drop analysis report for the VDD net for `iv` analysis in the `vdd_iv.txt` file:

```
print_ir_report \
-net VDD \
-type iv \
-filename vdd_iv.txt
```

- The following command prints the IR drop analysis report for the VSS net for `rc` analysis in the `vss_rc.txt` file:

```
print_ir_report \
-net VSS \
-type rc \
-filename vss_rc.txt
```

## **Voltus-Fi Custom Power Integrity Solution User Guide**

### Batch Mode Execution

---

- The following command prints the IR drop analysis report for all power nets for rcrms analysis in the `all_power_rcrms.txt` file:

```
print_ir_report \
-net all_power \
-type rcrms \
-filename all_power_rcRMS.txt
```

## **print\_rlp\_report**

```
print_rlp_report  
  -net netname  
  -filename output_file_name  
  -tap tapname
```

Prints the RLRP analysis report in the Voltus-Fi-XL flow.

**Note:** Before running this command, ensure that the IR drop results are loaded using the `load_ir_results` command.

### **Parameters**

`-filename output_file_name`

Specifies the name of the RLRP analysis report file. This is a required parameter.

`-net netname`

Specifies the name of the net for which the RLRP report is to be printed. This is a required parameter.

`-tap tapname`

Specifies the instances or tap-nodes of the specified net for which the LRP values are to be printed. You can specify multiple tap nodes for the specified net as shown in the example below. This is a required parameter.

### **Example:**

- Use the following commands to load the IR drop analysis results and print the RLRP analysis report for the tap nodes specified for the TVDD net in the `RLRP.rpt` file:

```
load_ir_results ./abc/raw.out/xps.emirtap.emir0_bin  
print_rlp_report -net TVDD -filename RLRP.rpt -tap MavD7_1_unmatched##  
print_rlp_report -net TVDD -filename RLRP.rpt -tap MavD7_2_unmatched##
```

## pin\_2\_pin\_res

```
pin_2_pin_res
  -xdspf filename
  -x1 value
  -y1 value
  -x2 value
  -y2 value
  -layer1 layernname
  -layer2 layernname
  -net netname
```

This command is used to calculate the effective resistance between any two nodes (pins, tap nodes, or subnodes), either on the same layer or on different layers of the same net. Use this command after the `load_ir_results` command.

**Note:** All parameters of this command are required parameters.

### Parameter

-xdspf <i>filename</i>	Specifies the location of the xDSPF file to be used for the Static Power Grid Solver (SPGS) flow of Spectre.
-x1 <i>value</i>	Specifies the x co-ordinate for node 1 in microns.
-y1 <i>value</i>	Specifies the y co-ordinate for node 1 in microns.
-x2 <i>value</i>	Specifies the x co-ordinate for node 2 in microns.
-y2 <i>value</i>	Specifies the y co-ordinate for node 2 in microns.
-layer1 <i>layernname</i>	Specifies the name of the layer for node 1.
-layer2 <i>layernname</i>	Specifies the name of the layer for node 2.
-net <i>netname</i>	Specifies the name of the net on which the two nodes are located.

### Example:

- Use the following commands to load the IR results and specify the nodes between which you want to calculate effective resistance.

```
load_ir_results \
./abc/raw.out/xps.emirtap.emir0_bin

pin_2_pin_res \
-xdspf abc/raw.out/abc_sample.dspf \
```

## **Voltus-Fi Custom Power Integrity Solution User Guide**

### Batch Mode Execution

---

```
-x1 83.40 \
-y1 34.205 \
-x2 83.40 \
-y2 35.205
-layer1 mt1 \
-layer2 mt2 \
-net VDD
```

## Batch Commands for EM Reports

You can load and print the results for EM analysis in Voltus-Fi-XL using the following commands:

- [load\\_em\\_results](#)
- [print\\_em\\_report](#)

### **load\_em\_results**

```
load_em_results
  -tech_file tech_file_name
  [-em_only_ict_file ict_file_name]
  [-layer_mapfile layer_map_filename]
  results_file_name
```

Loads the results of EM analysis in the Voltus-Fi-XL flow.

**Note:** The `qrcTechFile` name and layer map file must be specified if the layer names in the `qrcTechFile` are different from those in the simulation database.

### Parameters

`-em_only_ict_file ict_file_name`

Specifies the name of the ICT file with process and EM model information that will be used for EM analysis. This is an optional parameter. For a sample EM only ICT file, see the [“File Formats” chapter](#).

`-layer_mapfile layer_map_filename`

Specifies the name of the layer map file that provides the mapping between the layer names used in the technology file and the simulation database.

This is an optional parameter. It is used only when the layer names in the `qrcTechFile` are different from those in the simulation database.

`results_file_name`

Specifies the name of the file which contains the EMIR simulation results data. This is a required parameter.

`-tech_file -tech_file_name`

Specifies the name of the `qrcTechFile` or the `emDataFile` used for EM analysis. This is a required parameter.

## Example

- The following command loads the results of EM analysis stored in the `xps.emirtap.emir0_bin` file, and when the layer map file is `contactmapfile`:

```
load_em_results \
-tech_file qrcTechFile \
-layer_mapfile contactmapfile \
-layer_mapfile contactmapfile /xps.emirtap.emir0_bin
```

## print\_em\_report

```
print_em_report
  -net {all_power | all_signal | all_nets | netname}
  -type {javg | jmax | jabsavg | jacpeak | jacrms | rc | rcavg | rcrms | tc |
  tcavg | tcrms | acrms_sh | avg_sh | peak_sh}
  [-threshold threshold_value]
  -filename output_file_name
  [-rule custom_em_rule_name]
```

Prints the EM analysis reports in the Voltus-Fi-XL flow.

**Note:** Before running this command, ensure that the EM results are loaded using the `load_em_results` command.

## Parameters

`-filename output_file_name`

Specifies the name of the EM analysis report file. This is a required parameter.

`-net {all_power | all_signal | all_nets | netname}`

## Voltus-Fi Custom Power Integrity Solution User Guide

### Batch Mode Execution

---

Specifies that the EM analysis report is to be generated for one of the following:

`all_power`: prints the report for all power nets

`all_signal`: prints the report for all signal nets

`all_nets`: prints the report for all nets

`netname`: prints the report for the specified net

This is a required parameter.

`-rule custom_em_rule_name`

Specifies the custom EM rule for which the report will be created.

This is an optional parameter.

`-threshold threshold_value`

Specifies that resistors or nodes that have a threshold value of EM ratio above the specified value will be reported.

This is an optional parameter.

*Default:* The report prints the results for all resistors or nodes.

`-type {javg | jmax | jabsavg | jacpeak | jacrms | rc | rcavg | rcrms | tc | tcavg | tcrms | acrms_sh | avg_sh | peak_sh}`

Specifies the analysis type for which the report is being generated.

`javg` – specifies that the report is generated for the average current density analysis data.

`jmax` – specifies that the report is generated for the peak current density analysis data.

`jabsavg` – specifies that the report is generated for the average absolute current density analysis data.

`jacpeak` – specifies that the report is generated for the AC peak current density analysis data.

`jacrms` – specifies that the report is generated for the RMS current density analysis data.

`rc` – specifies that the report is generated for the peak resistor current density analysis data.

`rcavg` – specifies that the report is generated for the average resistor current density analysis data.

`rcrms` – specifies that the report is generated for the RMS resistor current density analysis data.

`tc` – specifies that the report is generated for the peak tap current density analysis data.

`tcavg` – specifies that the report is generated for the average tap current density analysis data.

`tcrms` – specifies that the report is generated for the RMS tap current density analysis data.

`acrms_sh` – specifies that the report is generated for AC RMS current analysis data with self-heating effect.

`avg_sh` – specifies that the report is generated for average current analysis data with self-heating effect.

`peak_sh` – specifies that the report is generated for peak current analysis data with self-heating effect.

**Note:** The above three types of analyses are only applicable in the SHE flow.

This is a required parameter.

**Examples:**

- The following command prints the EM analysis report for all the power nets for javg analysis in the all\_power\_javg.txt file:

```
print_em_report \
-net all_power \
-type javg \
-filename all_power_javg.txt
```

- The following command prints the EM analysis report for all the signal nets for jmax analysis in the all\_signal\_jmax.txt file:

```
print_em_report \
-net all_signal \
-type jmax \
-filename all_signal_jmax.txt
```

- The following command prints the EM drop analysis report for the VDD net for jabsavg analysis in the vdd\_jabsavg.txt file:

```
print_em_report \
-net VDD \
-type jabsavg \
-filename vdd_jabsavg.txt
```

- The following command prints the EM drop analysis report for the VDD net for tcrms analysis in the vdd\_tcrms.txt file:

```
print_em_report \
-net VDD \
-type tcrms \
-filename vdd_tcrms.txt
```

## Batch Commands for SHE Analysis

You can print the results for self-heating effect analysis (SHE) in Voltus-Fi-XL using the following commands:

- [load\\_view](#)
- [print\\_thermal\\_report](#)

### **load\_view**

```
load_view  
  -libname lib  
  -cellname cell_name  
  -viewname view_name
```

Specifies the layout to be used for SHE analysis in the Voltus-Fi-XL flow.

#### **Parameters**

`-cellname cell_name`

Specifies the name of the cell. This is a required parameter.

`-libname lib`

Specifies the name of the library. This is a required parameter.

`-viewname view_name`

Specifies the layout view for which SHE analysis is to be performed. This is a required parameter.

#### **Example:**

The following command opens the layout view for cell1 of library, test\_lib:

```
load_view  
  -libname test_lib \  
  -cellname cell1 \  
  -viewname layout
```

## **print\_thermal\_report**

```
print_thermal_report
  -type {mos-region | metal}
  [-tiles tile_matrix]
  -filename output_file_name
```

Prints the results of SHE analysis in the Voltus-Fi-XL flow.

### **Parameters**

`-filename output_file_name`

Specifies the name of the file in which the SHE analysis results will be printed. This is a required parameter.

`-tiles tile_matrix`

Specifies the OD regions or metal resistors and their temperature change after self-heating effect in tile format. The tile matrix is specified as follows:

The design on the layout is divided into sections based on the tile matrix provided, which is the number of divisions in the x-direction multiplied by the number of divisions in the y-direction.

For example:

`-tiles 10x15`

This is an optional parameter.

`-type {mos_region | metal}`

When `mos-region` is specified, the shape of OD regions and their temperature change after self-heating effect are printed in the report.

When `metal` is specified, the shape of metal resistors and their temperature change after self-heating effect are printed in the report.

This is a required parameter.

## Example

- The following sequence of commands will print the results of SHE analysis for OD regions in a tile format in the `mos-region-tiles.txt` file:

```
load_em_results \
-tech_file qrcTechFile
-layer_mapfile contactmapfile /input.emir0_bin

print_em_report \
-net all_nets \
-type acrms_sh \
-filename ./acrms_sh.txt

print_thermal_report \
-type mos-region \
-tiles 15x10 \
-filename ./mos-region-tiles.txt
```

## Batch Commands for PGV Generation

You can create power-grid views in Voltus-Fi-XL flow using the following commands:

- create\_pgv
- set\_mmsim\_pgv\_nets

### **create\_pgv**

```
create_pgv
    -macro_name macro_name
    [-lib_name name_of_library]
    -tech_file tech_file_name
    [-tech_lef_file tech_lef_file_name]
    -type {static | dynamic}
    [-macro_lef_file macro_lef_file_name]
    -sim_dir simulation_dir
    [-lef_layer_map lef_layer_map_file_name]
    [-oa_lib list_of_OA_libraries]
    -pgdb_layer_map pgdb_layermap_file_name
    [-trigger_file trigger_file_name]
    [-output_directory directory]
    [-uti_Ron_file Ron_report_file_dumped_by_Spectre]
```

Creates power-grid views. The power-grid views contain information about the power ports of a transistor or block, the internal power grid, and the tap current distribution within a transistor or block.

### **Parameters**

*-lef\_layer\_map lef\_layer\_map\_file\_name*

# Voltus-Fi Custom Power Integrity Solution User Guide

## Batch Mode Execution

---

Specifies the mapping between LEF layer names and technology layer names. This is a required parameter.

The syntax of the layermap file is as follows:

```
#type    technology_layer_name  lefdef  lef_layer_name
```

**Example:**

```
# -----  
metal   METAL_1           lefdef  M1  
via     VIA_1             lefdef  VIA1  
metal   METAL_2           lefdef  M2  
via     VIA_2             lefdef  VIA2
```

where “type” can be poly/via/metal/diff.

**-lib\_name** *name\_of\_library*

Specifies the name of the library. This is specified if you want to create a library with a name different than that of the macro.

This is an optional parameter.

By default, a library is created with the same name as that of the macro.

**-macro\_lef\_file** *macro\_lef\_file\_name*

Specifies the LEF information for the design.

**Note:** If the technology data and design data are in the same LEF file, use any of the two options, **-tech\_lef\_file** or **-macro\_lef\_file** to specify the filename. In this case, it is an optional parameter.

**-macro\_name** *macro\_name*

Specifies the name of the macro. This is a required parameter.

**-oa\_lib** *list\_of\_OA\_libraries*

Specifies the OpenAccess (OA) database library name for PGV generation. When this option is specified, the layermap file is automatically generated. However, you can also specify the layermap file by using the **-lef\_layer\_map** parameter.

**Note:** The parameters, **-tech\_lef\_file**/**-macro\_lef\_file** and **-oa\_lib** are mutually exclusive.

This is an optional parameter.

**-output\_directory** *directory*

## Voltus-Fi Custom Power Integrity Solution User Guide

### Batch Mode Execution

---

Specifies the name of the output directory for power-grid view generation. This is an optional parameter. By default, the power-grid views are stored in the work directory.

`-pgdb_layer_map pgdb_layermap_file_name`

Specifies the mapping between the xDSPF file layer names that are written in the simulation database and the technology layer names. This is a required parameter.

The syntax of the layermap file is as follows:

```
# type  technology_layer_name  pgdb  xDSPF_layer_name
```

**Example:**

```
# -----  
poly  POLYCIDE          pgdb  poly  
via   CONT              pgdb  pl1co  
via   CONT              pgdb  odCont1  
via   CONT              pgdb  odCont2  
metal  METAL_1          pgdb  metal1  
via   VIA_1             pgdb  VIA1  
metal  METAL_2          pgdb  metal2  
via   VIA_2             pgdb  VIA2  
diff   OD                pgdb  nwires  
diff   OD                pgdb  mwires
```

where “type” can be poly/via/metal/diff.

**Note:** Ensure that all the xDSPF file layer names are mapped to the technology layer names. You can map multiple xDSPF file layer names to a single technology layer name. This is shown in the above example.

`-sim_dir simulation_dir`

Specifies the simulation directory for the current data. This is a required parameter.

`-tech_file tech_file_name`

Specifies the technology file to be used for reading the technology data. This is a required parameter.

`-tech_lef_file tech_lef_file_name`

Specifies the LEF information for the technology data.

This is an optional parameter. This parameter is not specified when the OA libraries are specified.

`-trigger_file trigger_file_name`

Specifies the trigger file that is used during dynamic current simulation to run a detailed analysis for determining current distribution.

This is an optional parameter.

```
-type {static | dynamic}
```

Specifies the type of analysis for which you want to generate power-grid views. This is a required parameter.

```
-uti_Ron_file Ron_report_file_dumped_by_Spectre
```

Specifies the UTI-Ron file that contains multiple on-resistance (Ron) values, that is minimum, maximum, and average (min/avg/max) values, for powergates.

This is an optional parameter. If this file is not specified, Voltus-Fi-XL uses the Ron values from the MMSIM simulation database (\*.emir#\_bin file).

## Example

- The following command is used to create power-grid views:

```
create_pgv \
-macro_name design1 \
-lib_name ether \
-tech_file RCgen.tch \
-type dynamic \
-tech_lef_file tech.lef \
-macro_lef_file mac.lef \
-sim_dir pgv-sim-132 \
-lef_layer_map lefdef.layermap.txt \
-pgdb_layer_map pgdpmap.txt \
-trigger_file trigger_vps.txt \
-output_directory ./vps_pgv_1
```

## set\_mmsim\_pgv\_nets

```
set_mmsim_pgv_nets
-net net_name
-voltage voltage_value
-results_file result_file_name
-prefix prefix_name
```

Specifies the nets for which you want to create power-grid views in the MMSIM flow. You can use this command to specify multiple power-grid view nets at a time.

## Parameters

`-net net_name`

Specifies the name of the net. This is a required parameter.

`-prefix prefix_name`

Specifies the name of the prefix used while performing the simulation. This is a required parameter.

`-results_file result_file_name`

Specifies the simulation result file to be used. The naming convention of the simulation result file is, \*.emir#\_bin. This is a required parameter.

`-voltage voltage_value`

Specifies the net's voltage. This is a required parameter.

## Example:

- The following command is used to create power-grid views for nets VSS and VDD:

```
set_mmsim_pgv_nets
-net {VDD VSS}
-voltage {1.1 0}
-results_file /emirtap.emir0_bin
-prefix XO
```

## Batch Commands for Querying

You can query resistors and nodes in Voltus-Fi-XL flow using batch commands. For this, first search for specific resistors and nodes and then query specific information about these resistors and nodes using batch commands.

You can also specify these commands in a `tcl` file (`conf.tcl`) and specify the file in the EMIR control file by using the following `emirutil` command as follows:

```
emirutil postTclCmdFile=[conf.tcl]
```

For details, see [Using the emirutil Command in the EMIR Control File](#) in the “EM Analysis Variables” chapter.

Listed below are the batch commands that can be specified for querying specific data.

- [search\\_res\\_id](#)
- [res\\_id\\_info](#)
- [search\\_node\\_id](#)
- [node\\_id\\_info](#)

### **search\_res\_id**

```
search_res_id
  -x x_location
  -y y_location
  -layer extracted_layer_name
  [-net net_name]
```

Specifies the location and other information of the resistor to be queried.

#### **Parameters**

`-layer extracted_layer_name`

Specifies the name of the extracted layer on which the resistor is located. This is a required parameter.

`-net net_name`

Specifies the name of the net. This is an optional parameter.

## Voltus-Fi Custom Power Integrity Solution User Guide

### Batch Mode Execution

---

**-x *x\_location***

Specifies the X-co-ordinate of the resistor to be queried. This is a required parameter.

**-y *y\_location***

Specifies the Y-co-ordinate of the resistor to be queried. This is a required parameter.

#### **Example:**

- The following command is used to query a resistor at the specified co-ordinates on metal layer, m1:

```
search_res_id  
-x 4.78  
-y 9.75  
-layer m1
```

## **res\_id\_info**

```
res_id_info  
-id res_id  
[-type  
{default/a/w/l/La/Lu/Lb/Wu/Wb/delta_T/Lv2v_rms/Tj/Tlife/Td/thickness/r/Iavg/  
Ipeak/Irms/Iabs_avg/Ipos_avg/Ineg_avg/Iavg_max/Ipeak_max/Irms_max/Iacpeak_ma  
x/T/N/name/layer/netName/node1/node2/type/x/y/current_direction/feolT/beolT/  
beta/alpha/alpha_connecting/alpha_overlapping}]
```

Specifies the information to be queried for the specified resistor (pRes).

#### **Parameters**

**-id *res\_id***

Specifies the pRes ID for which the information is to be queried. This is a required parameter.

**-type**

Specifies the type of information to be queried for the pRes. You can specify any number of options detailed below. The query will output the values of the specified options for the pRes.

# Voltus-Fi Custom Power Integrity Solution User Guide

## Batch Mode Execution

---

**default:** When this option is specified, the software returns a set of values for the specified pRes in a specific format. An example of the default output of this command is provided below:

```
name : rx3
layer : m1
netName : VG
node1 : VG#2
node2 : XX1/M0#g
type : Metal
x : 4.780
y : 9.784
current_direction : UnKnown
w : 0.02
l : 0.1
Iavg : 3.41e-10
Ipeak : 9.25e-07
Irms : 1.28e-07
Iavg_max : 6.6e-05
Ipeak_max : 0
Irms_max : 0
Iacpeak_max : 0
delta_T : 5
Td : 0.000315
r : 0.0315
rmsLv2v : 1.8e+308
Iavg_rule : (w*3) * 1.1 * 1 : (l <= 5)
```

**a:** Retrieves the area of the pRes.

**w:** Retrieves the width of the pRes.

**l:** Retrieves the blech length of the pRes.

**La:** Retrieves the length of the pseudo vias.

**Lu:** Retrieves the upper metal blech length.

**Lb:** Retrieves the bottom metal blech length.

**Wu:** Retrieves the upper metal width.

**Wb:** Retrieves the lower metal width.

# Voltus-Fi Custom Power Integrity Solution User Guide

## Batch Mode Execution

---

`delta_T`: Retrieves the `delta_T` value. For more information, see `delta_T` in the “[Variables](#)” chapter.

`Lv2v_rms`: Retrieves the distance between vias for the RMS analysis.

`Tj`: Retrieves the temperature to be used for EM analysis.

`Tlife`: Retrieves the lifetime for which the EM analysis was performed.

`Td` : Retrieves the time duration in micro second or total “On Time” period.

`thickness`: Retrieves the thickness of the layer.

`r`: Retrieves the duty ratio.

`Iavg`: Retrieves average current value.

`Ipeak`: Retrieves peak current value.

`Irms`: Retrieves RMS resistor current value.

`Iabs_avg`: Retrieves absolute of average current value.

`Ipos_avg`: Retrieves average current value in the positive direction.

`Ineg_avg`: Retrieves average current value in the negative direction.

`Iavg_max`: Retrieves the maximum average current density.

`Ipeak_max`: Retrieves the maximum peak current density.

`Irms_max`: Retrieves the maximum RMS current density.

`Iacpeak_max`: Retrieves the maximum AC peak current density.

`T`: Retrieves the total time of EM analysis.

`N`: Retrieves the number of vias.

`name`: Retrieves the name of the pRes.

`layer`: Retrieves the layer name.

`netName`: Retrieves the net name.

`node1`: Retrieves the name of the first node connected to the pRes.

**node2:** Retrieves the name of the second node connected to the pRes.

**type:** Retrieves the type of layer, which may be metal, via, or pseudo via.

**x:** Retrieves the X-co-ordinate of the pRes.

**y:** Retrieves the Y-co-ordinate of the pRes.

**current\_direction:** Retrieves the current direction flow between nodes 1 and 2 of the resistors.

**Example:**

- The following set of commands are used to query the specified resistor on metal layer, m1 , for the specified information:

**Step1:** Search resistor id:

```
set rid [search_res_id -x 3.75 -y 8.35 -layer m1]
```

**Step2:** Get specific information from resistor id:

```
puts [res_id_info -id $rid -type Iacpeak_max]
```

## **search\_node\_id**

```
search_node_id  
-x x_location  
-y y_location  
-layer extracted_layer_name  
[-net net_name]
```

Specifies the information required to search for the node to be queried.

### **Parameters**

**-layer extracted\_layer\_name**

Specifies the name of the extracted layer on which the node is located. This is a required parameter.

**-net net\_name**

Specifies the name of the net. This is an optional parameter.

*-x x\_location*

Specifies the X-co-ordinate for the node to be searched. This is a required parameter.

*-y y\_location*

Specifies the Y-co-ordinate for the node to be searched. This is a required parameter.

**Example:**

- The following command is used to search for a node at the specified co-ordinates on metal layer, m2:

```
search_res_id  
-x 3.75  
-y 8.35  
-layer m2
```

**node\_id\_info**

```
node_id_info  
-id node_id  
[-type {default/name/x/y/IRavg/IRpeak}]
```

Specifies the information to be retrieved for the specified resistor.

**Parameters**

*-id node\_id*      Specifies the node ID for which the information is to be retrieved. This is a required parameter.

*-type*

Specifies the type of information to be retrieved for the node. You can specify from the options detailed below using this parameter.

## Voltus-Fi Custom Power Integrity Solution User Guide

### Batch Mode Execution

---

**default:** When this option is specified, the software returns a set of values for the specified node in a specific format. An example of the default output of this command is provided below:

name	:	VG#2
x	:	4.780
y	:	9.834
IRavg	:	8.54e-07
IRpeak	:	0.00224
netName	:	VG

**name:** Retrieves the name of the node.

**x:** Retrieves the X-co-ordinate for the node.

**y:** Retrieves the Y-co-ordinate for the node.

**IRavg:** Retrieves the average voltage drop for the node.

**IRpeak:** Retrieves the peak voltage drop for the node.

#### **Example:**

- The following set of commands are used to query the specified node on metal layer, m2 , for the specified information:

##### **Step1:** Search node id:

```
set nid [search_node_id -x 3.75 -y 8.35 -layer m2]
```

##### **Step2:** Get specific information from node id:

```
puts [node_id_info -id $nid -type IRavg]
```

## **Environment Variables**

---

- Setting Up the Voltus-Fi-XL Environment Variables on page 336
  - VFI\_LIC\_QUEUE on page 336
  - VOLTUSFI\_DELTA\_T on page 336
  - VOLTUSFI\_DYNAMIC\_AC\_PEAK on page 336
  - VOLTUSFI\_SPLIT\_ACDC\_RULES on page 337

## Setting Up the Voltus-Fi-XL Environment Variables

Environment variables are set using the `setenv` command before invoking the Voltus-Fi-XL software.

The following environment variables can be set for Voltus-Fi-XL:

- [VFI\\_LIC\\_QUEUE](#)
- [VOLTUSFI\\_DELTA\\_T](#)
- [VOLTUSFI\\_DYNAMIC\\_AC\\_PEAK](#)
- [VOLTUSFI\\_SPLIT\\_ACDC\\_RULES](#)

### **VFI\_LIC\_QUEUE**

```
setenv VFI_LIC_QUEUE time_in_seconds
```

Specifies the duration for which the software should wait for the availability of the Voltus-Fi-XL license. By default, the software waits for 24 hours or 86400 seconds.

**Example:**

```
setenv VFI_LIC_QUEUE 1000
```

### **VOLTUSFI\_DELTA\_T**

```
setenv VOLTUSFI_DELTA_T value
```

Specifies the deltaT value for EM analysis. It is used to specify maximum rise in temperature in degree Celsius, caused due to Joule Heating. It must be a positive value.

If you do not specify the rise in temperature, the software considers a default temperature of 10 degree Celsius. The variable, `deltaT`, can also be specified using either the `set_variable` command or in the Variables form. For details, see “[Variables](#)”.

**Example:**

```
setenv VOLTUSFI_DELTA_T 15
```

### **VOLTUSFI\_DYNAMIC\_AC\_PEAK**

```
setenv VOLTUSFI_DYNAMIC_AC_PEAK true | false | multiplePeak
```

## Voltus-Fi Custom Power Integrity Solution User Guide

### Environment Variables

---

Specifies that the current density violations should be calculated based on the peak AC current for metal lines, vias, and contacts. It is applied to periodic AC or pulsed DC signals. Valid values are `true`, `false`, and `multiplePeak`. For details, see [dynamicACPeak](#).

#### Example:

```
setenv VOLTUSFI_DYNAMIC_AC_PEAK true
```

## **VOLTUSFI\_SPLIT\_ACDC\_RULES**

```
setenv VOLTUSFI_SPLIT_ACDC_RULES true | false
```

Specifies different rules for EM analysis of power and signal nets. The default value is `false`. For details, see [splitACDCRules](#). You can also see [Rules for Specifying EM Analysis Type for Power and Ground Nets](#) in the “Data Preparation” chapter.

#### Example:

```
setenv VOLTUSFI_SPLIT_ACDC_RULES true
```

# **Voltus-Fi Custom Power Integrity Solution User Guide**

## Environment Variables

---

## **Variables**

---

- [Overview](#) on page 341
- [Using the set\\_variable Command in the Voltus-Fi-XL Command File](#) on page 341
- [Using the emirutil Command in the EMIR Control File](#) on page 342
- [Setting the Configuration Variable in the Batch Mode](#) on page 342
- [Using the Variables in the Voltus-Fi-XL GUI](#) on page 343
  - [applyRThreshold](#) on page 344
  - [consolidatedReport](#) on page 344
  - [deltaT](#) on page 344
  - [disableLayoutScale](#) on page 344
  - [dynamicACPeak](#) on page 345
  - [geounit](#) on page 346
  - [geounit\\_wl](#) on page 346
  - [geounit\\_xy](#) on page 346
  - [optimisticEMRuleSelection](#) on page 346
  - [powerRailSupplyNets](#) on page 346
  - [powerRailRules](#) on page 347
  - [powerRailGroundNets](#) on page 347
  - [process\\_scale](#) on page 347
  - [reportUnmatchedRuleResistor](#) on page 348
  - [rmsMultiplierPin](#) on page 348
  - [rmsMultiplierSignal](#) on page 348

## Voltus-Fi Custom Power Integrity Solution User Guide

### Variables

---

- ❑ [shEMEffect](#) on page 348
- ❑ [shHeatSink](#) on page 348
- ❑ [shParamFile](#) on page 349
- ❑ [shXMLFile](#) on page 349
- ❑ [splitACDCRules](#) on page 349
- ❑ [Tj](#) on page 349
- ❑ [Tlife](#) on page 349
- ❑ [useEncryptedWidth](#) on page 349
- ❑ [useRecoveryFactor](#) on page 350
- ❑ [viaCountFromResistance](#) on page 350

## Overview

This chapter provides a list of variables that you can specify in Voltus-Fi-XL. You can set these variables both in the GUI and in the batch mode.

In the GUI, you can set these variables in the Variables form, which opens when you click *Variables* in the EM tab of the IR/EM Results form. For details, see [Viewing the EM Analysis Results](#) in the EM Analysis Results chapter.

In the batch mode, you can set these variables using the `set_variable` and `emirutil` commands.

Information about how these variables can be set in both the batch mode and in the GUI is provided in the following sections:

- [Using the set\\_variable Command in the Voltus-Fi-XL Command File](#)
- [Using the emirutil Command in the EMIR Control File](#)
- [Setting the Configuration Variable in the Batch Mode](#)
- [Using the Variables in the Voltus-Fi-XL GUI](#)

## Using the `set_variable` Command in the Voltus-Fi-XL Command File

When using the `vfibatch` command to run Voltus-Fi-XL in batch mode, you can set the variables using the `set_variable` command. The syntax of the command and an example is provided below.

### **`set_variable`**

`set_variable var_name var_value`

Sets the user-specified value for the variable in the batch mode.

### **Parameters**

`var_name`

Specifies the name of the variable for which you want to set the value.

`var_value`      Specifies the value that you want to set for the variable.

**Example:**

The following command sets the value of the variable, `dbu` to 10000:

```
set_variable dbu 10000
```

## Using the emirutil Command in the EMIR Control File

When using the EMIR control file to generate textual reports of EMIR results in Voltus-Fi-XL in batch mode, you can set the variables using the `emirutil` command in the control file. The command syntax and example is provided below.

```
emirutil <var_name=var_value>
```

**Example:**

The following commands set the values of variables, `deltaT`, `dbu`, and `dynamicACPeak` to 10, 500, and `false`, respectively.

```
emirutil deltaT=10.0  
emirutil dbu=500  
emirutil dynamicACPeak=false
```

For details of the supported EMIR control file options, see [EMIR Control File Options Supported in Voltus-Fi-XL](#) in the “Data Preparation” chapter.

## Setting the Configuration Variable in the Batch Mode

You can set the configuration variable, `dbu` or `cadGrid` using either the `set_variable` command or the `emirutil` command.

The configuration variable, `dbu`, specifies the number of database units (DBU) per micron for the layout view. The default value of `dbu` is 1000.

The `cadGrid` variable specifies the resolution of the software in terms of width and length and the unit of measurement is micrometer or micron. The default value of `cadGrid` is 0.001.

**Note:** The value of the `dbu` variable is 1/`cadGrid`. If both `dbu` and `cadGrid` are specified, the `dbu` variable value is given priority.

The above variables are specified only in the batch mode when the view information is not specified in either the `vfibatch` command file or in the `emir` configuration file. They are not required to be specified when running the tool in the GUI mode as it is read from the open view. If this variable is not specified, the tool uses the default value.

For more information about how this variable is set in the Virtuoso Layout, see “Layout L Forms” chapter in the *Virtuoso® Layout Suite L User Guide*.

## Using the Variables in the Voltus-Fi-XL GUI

The following variables can be specified using the Variables form that can be accessed from the EM tab of the IR/EM Results form:

- [applyRThreshold](#)
- [consolidatedReport](#)
- [deltaT](#)
- [disableLayoutScale](#)
- [dynamicACPeak](#)
- [geounit](#)
- [geounit\\_wl](#)
- [geounit\\_xy](#)
- [optimisticEMRuleSelection](#)
- [powerRailSupplyNets](#)
- [powerRailRules](#)
- [powerRailGroundNets](#)
- [process\\_scale](#)
- [reportUnmatchedRuleResistor](#)
- [rmsMultiplierPin](#)
- [rmsMultiplierSignal](#)
- [shEMEffect](#)
- [shHeatSink](#)

- shParamFile
- shXMLFile
- splitACDCRules
- T<sub>i</sub>
- T<sub>life</sub>
- useEncryptedWidth
- useRecoveryFactor
- viaCountFromResistance

The above variables are described in detail in the subsequent section.

### **applyRThreshold**

Specifies the threshold ratio “r” for AC-Peak analysis. The default value of the threshold ratio is 0 . 05.

### **consolidatedReport**

When this variable is set to `true`, consolidated EM and IR reports are generated for all available bin files. The default value of this variable is `false`.

### **deltaT**

Specifies the maximum rise in temperature in degree Celsius, caused due to Joule heating. It must be a positive value.

If the rise in temperature is not specified, Voltus-Fi-XL considers a default temperature of 5 degrees Celsius. This is an ideal condition wherein the design must be EM free.

`deltaT` can be used in defining the current density limit: `Irms`. For example:

```
em_jmax_ac_rms EQU (w - 0.004) * 1.24 * deltaT
```

### **disableLayoutScale**

Disables the layout scale factor defined in the ICT file. The `layout_scale` factor is the ratio of the scaled dimensions divided by the drawn dimensions. The ICT file is defined based on the

scaled dimensions. Quantus applies the layout\_scale factor to the input design during extraction so that the design dimensions and the ICT file dimensions are consistent.

When the disableLayoutScale variable is set to true, the layout\_scale factor specified in the ICT file is ignored in the Voltus-Fi-XL EM analysis flow. By default, this variable is set to false.

## **dynamicACPeak**

Specifies that the current density violations should be calculated based on the peak AC current for metal lines, vias, and contacts. It is applied to periodic AC or pulsed DC signals. Valid values are true, false, and multiplePeak. When the multiplePeak option is specified, the software considers multiple peaks for the AC peak calculations.

$$I_{peak\_ac} = I_{peak\_dc} / (r)^{1/2}$$

where duty ratio “r” is:

$$r = I_{abs-avg} / I_{peak\_dc}, \text{ by default (“false”)}$$

And

$$r = T_d / T_{total}, \text{ when the dynamicACPeak variable is set.}$$

$$T_d = \text{time duration in micro second or total “On Time” period, where } abs(I) > max(abs(I)) / 2.0 \text{ during transient analysis}$$

$$T_d = \text{the time duration of maximum peak (when dynamicACPeak is set to “true”)}$$

$$T_d = \text{sum of time durations of different peaks, } T_{d1} + T_{d2} + T_{dN} \dots \text{ (when dynamicACPeak is set to “multiplePeak”)}$$

$$T_{total} = \text{total transient time}$$

**Note:** The software replaces the value of “r” with the value of applyRThreshold if the value of “r” is < than that of applyRThreshold. This is because a small “r” value results in an unreasonable increase in the  $I_{peak\_ac}$  limit. To avoid this scenario, use the applyR keyword to reset the value of “r” in the ICT file or change the value of the applyRThreshold variable, either in the Variables form or in the batch mode.

## **geounit**

Defines the scaling of all DSPF geometry parameters (W, L, X, Y). It applies only to the \* | NET section, and not to the instance section. The default value is 1.0e-6. The unit of this variable is meter.

## **geounit\_wl**

Defines the scaling of DSPF geometry parameters W and L. It applies only to the \* | NET section, and not to the instance section. The default value is 1.0e-6. The unit of this variable is meter.

**Note:** If the geounit\_wl and geounit options are specified together, the geounit\_wl option takes precedence over the geounit option.

## **geounit\_xy**

Defines the scaling of DSPF geometry parameters X and Y. It applies only to the \* | NET section, and not to the instance section. The default value is 1.0e-6. The unit of this variable is meter.

**Note:** If the geounit\_xy and geounit options are specified together, the geounit\_xy option takes precedence over the geounit option.

## **optimisticEMRuleSelection**

Specifies whether optimistic or pessimistic values are selected for the specified rules. This is applicable when multiple rules are selected after all conditions match.

By default, the variable is set to false. This means that the pessimistic value is selected.

## **powerRailSupplyNets**

Specifies the supply nets to be used for power-rail analysis. The available options are *list of supply nets* and "" (empty string).

- "" (empty string) : This is the default option. The software automatically identifies supply nets.
- *list of supply nets*: Specifies the list of supply nets to be used for power-rail analysis. These nets will override the nets automatically identified by the software.

**Note:** This variable accepts wildcards. For example:

```
powerRailSupplyNets="vdd vddp*"
```

## powerRailRules

Specifies rules for power-rail analysis. The available options are, `off`, `default`, and `advance`.

- `off`: The software ignores all EM rules specified using the `power_rail` keyword.
- `default`: Enables power-rail analysis. This is the default option.
- `advance`: Specifies the following:
  - EM rules will not be used for metal island
  - Stack vias will be considered as part of terminal vias
  - Current direction check will not be performed for terminal vias.

## powerRailGroundNets

Specifies the ground nets to be used for power-rail analysis. The available options are *list of ground nets* and `" "` (empty string).

- `" "` (empty string): This is the default option. The software automatically identifies ground nets.
- *list of supply nets*: Specifies the list of ground nets to be used for power-rail analysis. These nets will override the nets automatically identified by the software.

**Note:** This variable accepts wildcards. For example:

```
powerRailGroundNets="vss vssp*"
```

## process\_scale

Specifies the value by which the width and length values will be multiplied before the EM rules are applied. The default value of this variable is 1. In the GUI, you can set this variable in the .

In the batch mode, you can set this variable in the `vfibatch` command file by using the following command:

```
set_variable process_scale 0.8
```

You can also set this variable in the EMIR control file by using the following command:

```
emirutil process_scale=0.8
```

### **reportUnmatchedRuleResistor**

When set to `true`, specifies that resistors that do not match any EM rule for the specified analysis in the ICT file will be reported in the `"#.rpt_unmatch"` file. The default value of this variable is `true`. Also, the summary report file, `summary.rpt`, will report a count of such resistors in a separate column. For example, `"iacpeak_num_unmatch"`.

If you do not want to create the `#.rpt_unmatch` file, set this variable to `false`. You can also set this variable in the config file in the following manner:

```
emirutil reportUnmatchedRuleResistor=false
```

For details, see [EMIR Control File Options Supported in Voltus-Fi-XL](#).

### **rmsMultiplierPin**

Specifies the RMS relaxation factor for EM analysis of power grids. The default value is `1.0`.

### **rmsMultiplierSignal**

Specifies the RMS relaxation factor for EM analysis of signal nets. The default value is `1.0`.

### **shEMEffect**

Specifies the basis for the re-evaluation of EM effect. Valid values are:

- `beolT`: specifies that re-evaluation will be done on the basis of individual metal `BEOL_T`.
- `tiles_no.of divisions in x-direction X no. of divisions in y-direction`: specifies that re-evaluation will be done on the basis of tile `BEOL_T`.

### **shHeatSink**

When set to `true`, specifies that heat sink effect should be included in the SHE analysis. By default, this variable is set to `false`.

## **shParamFile**

Specifies the self-heating effect (SHE) analysis parameter file, which is required for plotting the SHE analysis plots. This file is used for deltaT FEOL calculation with finger and fin effect considerations.

## **shXMLFile**

Specifies the XML file to be used in the SHE analysis flow. This file is created by model cards with supporting self-heating as an input file for the final deltaT FEOL value calculation during simulation.

## **splitACDCRules**

Specifies different rules for EM analysis of power and signal nets. The default value is `false`. For details, see [Rules for Specifying EM Analysis Type for Power and Ground Nets](#) in the “Data Preparation” chapter.

## **Tj**

Specifies the temperature to be used for EM analysis. The default value is `Tsim` or the temperature used while performing simulation.

## **Tlife**

Specifies the lifetime for which the EM analysis will be performed. The default unit for this variable is `years`. You can also specify the value in `hours`. The tool will convert it into years. For example:

`Tlife=20,000 hrs`

## **useEncryptedWidth**

Specifies whether encrypted (silicon) width should be used or not. The default value of this variable is `false`, in which case, the software does not use the encrypted width from the DSPE. When set to `true`, the software uses the encrypted width but does not report both the width and the minimum width.

## **useRecoveryFactor**

Specifies whether or not layer-wise recovery factor is to be used for calculating the average current density.

If the recovery factor is specified, then the average current density calculation is as follows:

```
iavg=max(|iavgpos|,|iavgneg|)- recovery_factor*min(|iavgpos|,|iavgneg|)
```

If not specified, then

```
iavg=iavgpos+iavgneg
```

where `iavgpos` is the average current in the positive direction and `iavgneg` is the average current in the negative direction

## **viaCountFromResistance**

Specifies the number of vias when third-party DSPF with missing via count (`$N`) is used for EM analysis. The default value of this variable is `false`. To understand the scenario in which this variable is used, see [Appendix A – Third-Party DSPF](#).

---

## **SKILL Functions**

---

- [Overview](#) on page 352
- [Voltus-Fi-XL SKILL Functions](#) on page 352

## Overview

This section provides syntax, description, and examples for the Cadence® SKILL functions associated with the Voltus™-Fi Custom Power Integrity Solution-XL (Voltus-Fi-XL) flow.

Only the functions documented in this chapter are supported for public use. Any other functions, and undocumented aspects of the functions described below, are private and subject to change or removal at any time.

## Voltus-Fi-XL SKILL Functions

The following types of functions are provided in Voltus-Fi XL:

- [vsaCreateMarker](#)
- [vsaOpenLayout](#)
- [vsaLoadNets](#)
- [vsaLoadSecondaryLayers](#)
- [vsaSetEMIRConfig](#)
- [vsaSetEMResultsFile](#)
- [vsaSetEMLayerMapFileName](#)
- [vsaSetEMTechFileName](#)
- [vsaSetEMTypes](#)
- [vsaSetDfiiLayerMapFileName](#)
- [vsaSetQRCData](#)
- [vsaSelectFailedNets](#)
- [vsaSetEMOnlyICTFile](#)
- [vsaSetEMPlot](#)
- [vsaLoadEM](#)
- [vsaSetIRResultsFile](#)
- [vsaSetIRThreshold](#)
- [vsaSetIRPlot](#)

## **Voltus-Fi Custom Power Integrity Solution User Guide**

### SKILL Functions

---

- [vsaSetLRPShortedLayers](#)
- [vsaLoadIR](#)

## vsaCreateMarker

```
vsaCreateMarker(  
    x x_coordinate  
    y y_coordinate  
    message  
)  
=> t / nil
```

### Description

Creates a point marker with the specified attributes. When this marker is created successfully, it appears in the *Misc* tab of the Annotation Browser.

### Arguments

x	Specifies the x-axis coordinate for the marker.
y	Specifies the y-axis coordinate for the marker.
message	Specifies the message associated with the marker.

### Values Returned

t	Marker was created at the specified coordinates.
nil	Returns nil when the marker is not created.

### Example

```
vsaCreateMarker(34.45 56.67 "Marker1")  
=> t
```

The above example code shows how to create Marker1 at the specified x-axis and y-axis coordinates.

## vsaOpenLayout

```
vsaOpenLayout(  
    libname  
    cellname  
    viewname  
)
```

### Description

Opens the design on the Virtuoso layout for running Voltus-Fi-XL.

### Arguments

<i>libname</i>	Specifies the library name for the design to be opened on the layout.
<i>cellname</i>	Specifies the cell name for the design to be opened on the layout.
<i>viewname</i>	Specifies the view name for the design to be opened on the layout.

### Values Returned

None.

### Example

```
vsaOpenLayout("testlib" "testcell" "layout")
```

The above example code shows how to specify the library name, cell name, and view name of the design you want to open on the layout.

## vsaLoadNets

```
vsaLoadNets(  
    netname  
)
```

### Description

Specifies the nets to be loaded in the Voltus-Fi-XL GUI.

### Argument

<i>netname</i>	Specifies the name of the net to be loaded in the Voltus-Fi XL GUI. You can specify multiple net names separated by a space.
----------------	--

### Values Returned

None.

### Example

```
vsaLoadNets( "VDD VSS" )
```

The above example code shows how to specify the net names for the nets, VDD and VSS, that you want to load in the Voltus-Fi XL GUI.

## **vsaLoadSecondaryLayers**

```
vsaLoadSecondaryLayers()
```

### **Description**

Enables the loading of data for all layers in the Voltus-Fi-XL GUI.

### **Argument**

None.

### **Values Returned**

None.

### **Example**

```
vsaLoadSecondaryLayers()
```

The above example code shows how to enable the loading of data for all layers in the Voltus-Fi XL GUI.

## vsaSetEMIRConfig

```
vsaSetEMIRConfig(  
    config_filename  
)
```

### Description

Loads the EMIR configuration file to be used for EMIR analysis in Voltus-Fi-XL. This file is specified before calling the `vsaLoadEM()` function.

### Argument

<i>config_filename</i>	Specifies the name of the EMIR configuration file to be loaded for EMIR analysis.
------------------------	---

### Values Returned

None.

### Example

```
vsaSetEMIRConfig("abc/emir.conf.variables")  
vsaLoadEM()
```

The above example code shows that the `emir.conf.variables` file will be loaded for EMIR analysis.

## vsaSetEMResultsFile

```
vsaSetEMResultsFile(  
    binfilename  
)
```

### Description

Specifies the simulation results file to be used for viewing the EMIR analysis results in Voltus-Fi-XL. This file contains the simulation database generated using the Spectre simulator.

### Argument

<i>binfilename</i>	Specifies the name of the simulation results file (emir_bin) to be used for loading and displaying the EMIR analysis results in Voltus-Fi-XL.
--------------------	---

### Values Returned

None.

### Example

```
vsaSetEMResultsFile("xps.raw/xps.emirtap.emir0_bin")
```

The above example code shows that the name of the results file specified for loading and displaying EMIR analysis results in Voltus-Fi-XL is xps.emirtap.emir0\_bin.

## **vsaSetEMLayerMapFileName**

```
vsaSetEMLayerMapFileName(  
    layermapfile  
)
```

### **Description**

Specifies the name of the layermap file to be used for the EM technology file. This file provides the mapping between layer names in the simulation database and those in the technology file. This function is only used for EM plots.

### **Argument**

<i>layermapfile</i>	Specifies the name of the layermap file. This file provides the mapping between layer names in the simulation database and those in the technology file.
---------------------	--

### **Values Returned**

None.

### **Example**

```
vsaSetEMLayerMapFileName( "contactmapfile" )
```

The above example code shows that the layermap file name is specified as contactmapfile.

## **vsaSetEMTechFileName**

```
vsaSetEMTechFileName(  
    techfilename  
)
```

### **Description**

Specifies the name of the EM technology file that contains the EM rule information to be used for plotting EM results in Voltus-Fi-XL.

### **Argument**

<i>techfilename</i>	Specifies the name of technology file with EM rules to be used for viewing EM results in Voltus-Fi-XL. You can specify <code>qrcTechFile</code> , <code>emDataFile</code> , or <code>ICT</code> file as input for performing EM analysis.
---------------------	---

### **Values Returned**

None.

### **Example**

```
vsaSetEMTechFileName( "qrcTechFile" )
```

The above example code shows that the technology file name is specified as `qrcTechFile`.

## vsaSetEMTypes

```
vsaSetEMTypes(  
    list (imax iavg irms iacpeak iabsavg)  
)
```

### Description

Specifies the types of EM analyses for which you want to plot EM results in Voltus-Fi-XL. At least one analysis type must be selected.

### Argument

list (imax iavg  
irms iacpeak  
iabsavg)

Specifies the types of EM analyses for which you want to plot EM results. The list can include the following types of analyses:

imax: calculates the current density violations based on maximum DC current

iavg: calculates the current density violations based on the average value of DC current

irms: calculates the current density violations based on the root mean square (RMS) value of AC current

iacpeak: calculates the current density violations based on peak AC current

iabsavg: calculates the current density violations based on the average value of absolute current

### Values Returned

None.

### Example

```
vsaSetEMTypes(list("imax" "iavg" "iabsavg"))
```

The above example code shows that imax, iavg, and iabsavg types of EM analyses are to be performed.

## **vsaSetDfiiLayerMapFileName**

```
vsaSetDfiiLayerMapFileName(  
    df2layermapfile  
)
```

### **Description**

Specifies the DFII layermap file name that provides the mapping between the extractor-generated xDSPF file layer names and the DFII layer names.

### **Argument**

<i>df2layermapfile</i>	Specifies the name of the DFII layermap file that provides the mapping between the xDSPF file layer names and the DFII layer names.
------------------------	---

### **Values Returned**

None.

### **Example**

```
vsaSetDfiiLayerMapFileName( "df2layermap" )
```

The above example code shows that the name of the DFII layermap file is `df2layermap`.

## vsaSetQRCData

```
vsaSetQRCData(  
    rundir  
    runname  
)
```

### Description

Sets the QRC inputs to enable solid shape highlighting for EMIR plots displayed on the Virtuoso layout.

### Arguments

<i>rundir</i>	Specifies the QRC run directory name.
<i>runname</i>	Specifies the run name used while running QRC.

### Values Returned

None.

### Example

```
vsaSetQRCData("./qrc_temp" "testrun")
```

The above example code shows that QRC run directory, `qrc_temp` and run name, `testrun` will be used to enable solid shape highlighting.

## **vsaSelectFailedNets**

vsaSelectFailedNets()

### **Description**

Specifies that EM analysis results for only those nets that failed the EM check should be plotted. It enables the *Show Only Failed* option in the GUI.

### **Arguments**

None.

### **Values Returned**

None.

### **Example**

```
vsaSelectFailedNets ()
```

The above example code shows the EM analysis results for only those nets that fail the EM check will be plotted.

## **vsaSetEMOnlyICTFile**

```
vsaSetEMOnlyICTFile(  
    filename  
)
```

### **Description**

Specifies the name of the ICT file to be used for specifying the EM rules for plotting EM analysis results.

### **Argument**

*filename*                    Specifies the name of the ICT file.

### **Values Returned**

None.

### **Example**

```
vsaSetEMOnlyICTFile("./myICT.txt")
```

The above example code shows that `myICT.txt` file will be used for specifying the EM rules.

## vsaSetEMPlot

```
vsaSetEMPlot(  
    plot_type  
)
```

### Description

Specifies the default plot for EM analysis results in Voltus-Fi-XL.

### Argument

<i>plot_type</i>	Sets the default plot for EM analysis results. You can specify any one of the following plot types:  RJ: sets the default plot to DC current density analysis plot  JAVG: sets the default plot to average current density analysis plot  JACPEAK: sets the default plot to AC peak current density analysis plot  JABSAVG: sets the default plot to average absolute current density analysis plot  JACRMS: sets the default plot to RMS current density analysis plot  TC: sets the default plot to peak tap current density analysis plot  RC: sets the default plot to peak resistor current density analysis plot
------------------	--

### Values Returned

None.

### Example

```
vsaSetEMPlot( "JAVG" )
```

## **Voltus-Fi Custom Power Integrity Solution User Guide**

### SKILL Functions

---

The above example code shows that the default plot for EM analysis results will be average current density analysis data ( $J_{AVG}$ ) plot.

## **vsaLoadEM**

`vsaLoadEM()`

### **Description**

Specifies that the EM analysis results should be loaded. Before specifying this function, ensure that the EM results file and the EM technology file are specified.

### **Arguments**

None.

### **Values Returned**

None.

### **Example**

```
vsaOpenLayout("testlib" "testcell" "layout")
vsaSetEMResultsFile("./xps.raw/xps.emirtap.emir0_bin")
vsaSetEMTechFileName("qrcTechFile")
vsaSetEMLayerMapFileName("contactmapfile")
vsaSetEMTypes(list("imax" "iavg" "irms" "iacpeak" "iabsavg"))
vsaSetDfiiLayerMapFileName("df2layermap")
vsaLoadEM()
```

The above example code shows the set of functions specified for loading the EM analysis results. The `vsaSetEMResultsFile` and the `vsaSetEMTechFileName` functions are required to be specified before specifying `vsaLoadEM`.

## vsaSetIRResultsFile

```
vsaSetIRResultsFile(  
    binfilename  
)
```

### Description

Specifies the results file to be used for viewing the EMIR analysis results in Voltus-Fi-XL.

### Argument

<i>binfilename</i>	Specifies the name of the <code>emir_bin</code> file to be used for loading and displaying the EMIR analysis results in Voltus-Fi-XL.
--------------------	---

### Values Returned

None.

### Example

```
vsaSetIRResultsFile("xps.raw/xps.emirtap.emir0_bin")
```

The above example code shows that the name of the results file to be used for loading and displaying EMIR analysis results in Voltus-Fi-XL is `xps.emirtap.emir0_bin`.

## vsaSetIRThreshold

```
vsaSetIRThreshold(  
    threshold_value  
)  
=> nil
```

### Description

Sets the threshold value for viewing IR drop violations.

### Argument

*threshold\_value*      Specifies the threshold value for viewing the IR drop violations in Voltus-Fi-XL.

### Value Returned

nil                        Returns `nil` when the value is not specified.

### Example

```
vsaSetIRThreshold(0.01)
```

The above example code sets the threshold value to 0.01 volts and all IR drop values below 0.01 volts are filtered while plotting the IR drop results.

## vsaSetIRPlot

```
vsaSetIRPlot(  
    plot_type  
)
```

### Description

Specifies the default plot for IR drop analysis results in Voltus-Fi-XL.

### Argument

*plot\_type* Sets the default plot for IR drop analysis results.  
Specify one of the following plot types:

- IR: sets the default plot to peak IR drop analysis plot
- IRAVG: sets the default plot to average IR drop analysis plot
- RC: sets the default plot to peak current density violation plot
- IV: sets the default plot to transistor-based supply voltage data plot
- Reffective: sets the default plot to effective resistance analysis plot
- PI: sets the default plot to that for currents across resistors when power gates are connected
- PV: sets the default plot to that for IR drop across power-switch instances

### Values Returned

None.

### Example

```
vsaSetIRPlot("Reffective")
```

## **Voltus-Fi Custom Power Integrity Solution User Guide**

### SKILL Functions

---

The above example code shows that the default plot for IR drop analysis will be effective.

## **vsaSetLRPShortedLayers**

```
vsaSetLRPShortedLayers(  
    layer1, layer2,...  
)
```

### **Description**

Specifies the names of layers to be shorted for the least-resistive path (LRP) analysis.

### **Argument**

<i>(layer1, layer2,....)</i>	Specifies the names of layers to be shorted for LRP analysis.
----------------------------------	--

### **Values Returned**

None.

### **Example**

```
vsaSetLRPShortedLayers("M1, OD")
```

The above example code shows that layers, M1 and OD will be shorted for LRP analysis.

## vsaLoadIR

```
vsaLoadIR()
```

### Description

Specifies that the IR drop analysis results should be loaded. Before specifying this function, ensure that the IR drop results file name is specified.

### Arguments

None.

### Values Returned

None.

### Example

```
vsaOpenLayout("testlib" "testcell" "layout")
vsaSetIRRResultsFile("./xps.raw/xps.emirtap.emir0_bin")
vsaSetDfiiLayerMapFileName("df2layermap")
vsaLoadIR()
```

The above example code shows the set of functions specified for loading the IR drop analysis results. The `vsaSetIRRResultsFile` function is required to be specified before specifying `vsaLoadIR`.

# **Voltus-Fi Custom Power Integrity Solution User Guide**

## **SKILL Functions**

---

## **File Formats**

---

- [Static Current Analysis Files](#) on page 378
  - [Activity File](#) on page 379
  - [Clock File](#) on page 380
  - [Value Change Dump \(VCD\) File](#) on page 381
  - [Power File](#) on page 384
- [Rail Analysis Files](#) on page 385
  - [ICT File](#) on page 386
  - [EM Only ICT File](#) on page 389
  - [EM Data File](#) on page 391
  - [Layer Map File](#) on page 393
  - [DFII Layer Map File](#) on page 394
- [Self-Heating Effect Analysis File](#) on page 395
  - [shParamFile](#) on page 395
- [Power-Grid View Creation Files](#) on page 397
  - [Format and Contents of Trigger File](#) on page 397
  - [LEF Layermap File](#) on page 400
  - [PGDB Layermap File](#) on page 401
- [Common Command Language \(CCL\) File](#) on page 402

## Static Current Analysis Files

This section provides information about the files required for performing average static current analysis in Voltus-Fi-XL using Thunder.

- [Activity File](#)
- [Clock File](#)
- [Value Change Dump \(VCD\) File](#)
- [Power File](#)

## Activity File

A file that contains the activity factor on any node of the circuit. This file is used for average current analysis. In the GUI, you can specify the activity file in the *Advanced* tab of the *Setup and Run Current Analysis* form.

**Sample file name:** activity.txt

**Command:** run\_static\_simulation -activity\_file *file\_name*

### Format Description:

The activity file has the following format:

```
file      :rows EOF                      //a file consists of rows of information
rows      :ε                           //an empty line
         | comment                   //a comment line
         | line                     //a specification line
comment   :^[\t]*\.* EOL                //a line starting with * is a comment
line      :name FA_value               //a node with transitional frequency
```

- *Name* can be the name of any node in the circuit.
- *FA\_value* specifies the default nodal activity to be used. It is given as an activity factor. A value of 0.20 would be 20% of the default frequency.

**Example:** This is a sample fa\_file for node activities. The column on the left shows the node names and the column on the right shows the activity.

```
out01    2
out02    3
out03    4
```

## Clock File

A file that contains information about the clock nodes, subcircuit definitions, and subcircuit instance names and their frequencies. This file is only used for average static current analysis. In the GUI, you can specify the average power file in the *Advanced* tab of the *Setup and Run Current Analysis* form.

**Sample file name:** clk.txt

**Command:** run\_static\_simulation -clock\_file *clock\_file\_name*

**Format:**

```
file: rows EOF //a file consists of rows
rows: ε //an empty line
      | comment //a comment
      | line //a specification line
comment:^[\t]*\*.+EOL //a line starting with a *
line :CLK name freq_value //a clock node specification
      | INST name freq_value //a subcircuit instance specification
      | INST name freq_value activity //a subcircuit instance with activity
      | CELL name freq_value //a subcircuit definition specification
      | CELL name freq_value activity //a subcircuit definition with activity
```

**Format Description:**

CLK, INST, and CELL are keywords.

- If you use the CLK keyword, the node name is treated as the root of a clock running at the given frequency.
- If you use the INST keyword, all input and output nodes—that is, node names on the instance card—are set to toggle at the specified frequency. .
- If you use the CELL keyword, all instances of the cell are processed instead of a single instance.

CLK has precedence over INST, and INST overrides CELL information to resolve redundant specifications.

**Example:** The following example specifies the CLK keyword in column 1, clock name in column 2, and the frequency value in column 3.

```
clk      clk_internal      120M
```

## Value Change Dump (VCD) File

A file that contains information about the number of transitions for each net. This file is only used for average static current analysis. In the GUI, you can specify the VCD file in the *Advanced* tab of the *Setup and Run Current Analysis* form.

**Sample file name:** vcd.txt

**Command:** run\_static\_simulation -vcd vcd\_file\_name

### VCD Values

Within the VCD file, these VCD values have the following meanings during driving:

- 0  
Forces the associated node to logic low.
- 1  
Forces the associated node to logic high.
- X  
Forces the associated node to logic unknown.
- Z  
Releases the associated node.

### VCD Command Extensions

The following extensions of the VCD format are supported. You can use these command extensions in the comment section of a VCD file to specify node characteristics.

Use these extensions to specify properties such as input voltages and rise and fall times of nodes listed in the VCD file.

- \*scope  
Sets the scope of the nodeset command by using the scope command. *Scope\_string* is prepended to the nodes listed in the nodeset command. The syntax is as follows:  
`*scope scope_string *end`
- \*nodeset  
Groups nodes in the VCD file into node sets. You can then set properties of the nodes in the node set by using the params command on the node set. The syntax is as follows:

# Voltus-Fi Custom Power Integrity Solution User Guide

## File Formats

---

```
*nodeset set_number [node_name]+ *end
```

### ■ \*params

Assigns parameters to sets of nodes by using the `params` command. The syntax is as follows:

```
*params set_number <vin0=voltage> <vin1=voltage> <vinx=voltage><tir=time>
<tif=time> *end
```

### Example:

Following is a sample VCD file named `vcd`.

```
$date
    Thurs Mar 07 15:09:59 2013
$end
$version
    Cadence 1.0
$end
$timescale
    1ns
$end

$scope module TOP $end

$var wire      1 !      A[7] $end
$var wire      1 "      A[6] $end
$var wire      1 #      A[5] $end
$var wire      1 $      A[4] $end
$var wire      4 %      A[3:0] $end
$upscope $end

$comment

*scope TOP *end
*nodeset 1 A[7:0] *end
*params 1 vin0=0v vin1=5v vinx=2.5v tir=1ns tif=1ns *end

$enddefinitions $end

#0
1!
```

## **Voltus-Fi Custom Power Integrity Solution User Guide**

### File Formats

---

```
1"  
1#  
1$  
b1111 %  
#50  
0!  
0"  
0#  
0$  
b0000 %  
#50  
1$  
b0001 %  
#50  
1#  
0$  
b0010 %  
#50  
1$  
b0011 %  
#50  
1"  
0#  
0$  
b0100 %
```

## Power File

A file that contains the average power for specific sub circuits. This file is only used for average static current analysis. In the GUI, you can specify the average power file in the *Advanced* tab of the *Setup and Run Current Analysis* form.

**Sample file name:** power.txt

**Command:** run\_static\_simulation -power\_file power\_file\_name

### Format and Example

The average power file has the format shown in the following example:

```
CELL C_MUX2_TG 2.0e-4  
Xxev | xeu | xem | wpy | sdh 1.0e-3
```

The CELL keyword specifies that the name following it is a subcircuit definition and that all instances of the cell dissipate the same average power. If the file does not contain the CELL keyword, the name is an instance of a subcircuit dissipating the specified average power.

## Rail Analysis Files

This section lists the rail analysis file formats.

- [ICT File](#)
- [EM Only ICT File](#)
- [EM Data File](#)
- [Layer Map File](#)
- [DFII Layer Map File](#)

## ICT File

The ICT file specifies the EM reliability rules for EM analysis. This is a text-based file, which means you can use any text editor to enter information in this file.

Following is a sample ICT file.

```
process "abcd" {
    background_dielectric_constant 1
    temp_reference 25
    em_tref 110
    em_output_wlt drawn
    em_variables w delta_T
    em_conductor_unit mA/um
    em_via_area_unit mA
    em_via_unit mA
    simulation {
        manual_simulation_points {
            metal_layer "mt1" {
                simulation_widths 0.3 0.6 1.5 3
                simulation_spacings 0.3 0.6 1.5 3
            }
            metal_layer "mt2" {
                simulation_widths 0.3 0.6 1.5 3
                simulation_spacings 0.3 0.6 1.5 3
            }
        }
        manual_simulation_combinations {
            combination_layers "mt1" "mt2"
            separation_values 0 1 2 3
        }
    }
    conductor "mt1" {
```

## Voltus-Fi Custom Power Integrity Solution User Guide

### File Formats

---

```
min_spacing          0.3
min_width           0.3
height              1.05
thickness            0.53
resistivity          0.1
gate_forming_layer  false
thermal_conductivity 0.39
em_model {
    em_jmax_dc_avg EQU w*1.0   jmax_factor 50 1.1 110 1.0 125 0.92 L > 5
    em_jmax_dc_avg EQU w*3.0   jmax_factor 50 1.1 110 1.0 125 0.92 L <= 5
    em_jmax_dc_peak EQU w*1.5  jmax_factor 50 1.1 110 1.0 125 0.92 L > 5
    em_jmax_dc_peak EQU w*3.5  jmax_factor 50 1.1 110 1.0 125 0.92 L <= 5
}
conductor "mt2" {
    min_spacing          0.3
    min_width           0.3
    height              2.38
    thickness            0.53
    resistivity          0.1
    gate_forming_layer  false
    thermal_conductivity 0.39
    em_model {
        em_jmax_dc_avg EQU 2.0*w*1.0   jmax_factor 50 1.1 110 1.0 125
0.92 L > 5
        em_jmax_dc_avg EQU 2.0*w*3.0   jmax_factor 50 1.1 110 1.0 125
0.92 L <= 5
        em_jmax_dc_peak EQU 2.0*w*1.5  jmax_factor 50 1.1 110 1.0 125
0.92 L > 5
        em_jmax_dc_peak EQU 2.0*w*3.5  jmax_factor 50 1.1 110 1.0 125
0.92 L <= 5
    }
}
via "Via1" {
```

## Voltus-Fi Custom Power Integrity Solution User Guide

### File Formats

---

```
top_layer          "mt2"
bottom_layer       "mt1"
contact_resistance 1.1
thermal_conductivity 0.39

em_model {
    em_vcwidth 0.12
    em_jmax_dc_avg 1.0 jmax_factor 50 1.1 110 1.0 125 0.97
    em_jmax_dc_peak 1.5 jmax_factor 50 1.1 110 1.0 125 0.97
}
}
```

**Note:** For more information about the ICT commands and keywords specified in the ICT file, see “Creating the ICT File” chapter in *QRC Techgen Reference Manual*.

## EM Only ICT File

The EM only ICT file provides the process and EM model information for EM analysis. This is a text-based file, which means you can use any text editor to enter information in this file.

Following is a sample EM only ICT file.

```
process "gpdk090" {
    background_dielectric_constant 1.000000
    temp_reference 25
    layout_scale 0.9
#!    use_silicon_density true
    em_tref 110
    em_output_wlt drawn
    em_variables w delta_T
    em_conductor_unit mA
    em_via_area_unit mA
    em_via_unit mA
}

conductor "mt1" {
    em_model {
        em_jmax_dc_avg EQU 100 * 1* ( w - 0.008 ) jmax_factor 100 2.077 105 1.434 110
        1.000 115 0.704 120 0.500 125 0.358 L > 9 W < 0.45
        em_jmax_dc_avg EQU 150 * 1.227 * ( w - 0.008 ) jmax_factor 100 2.077 105 1.434
        110 1.000 115 0.704 120 0.500 125 0.358 L <= 9 L > 4.5 W < 0.45
        em_jmax_dc_avg EQU 200 * 1.227 * ( w - 0.008 ) jmax_factor 100 2.077 105 1.434
        110 1.000 115 0.704 120 0.500 125 0.358 L > 9 W >= 0.45
        em_jmax_dc_avg EQU 200 * 1.227 * ( w - 0.008 ) jmax_factor 100 2.077 105 1.434
        110 1.000 115 0.704 120 0.500 125 0.358 L <= 9 L > 4.5 W >= 0.45
        em_jmax_dc_avg EQU 400 * 1.227 * ( w - 0.008 ) jmax_factor 100 2.077 105 1.434
        110 1.000 115 0.704 120 0.500 125 0.358 L <= 4.5
        em_jmax_dc_peak EQU 25.0 * ( w - 0.008 )
        em_jmax_ac_peak EQU 25.0 * ( w - 0.008 )
        em_jmax_dc_rms EQU sqrt( 18.58 * delta_T * ( w - 0.008 )^2 * ( w - 0.008 +
        0.246 ) / ( w - 0.008 + 0.0443 ) )
```

## Voltus-Fi Custom Power Integrity Solution User Guide

### File Formats

---

```
    em_jmax_ac_rms EQU sqrt( 18.58 * delta_T * ( w - 0.008 )^2 * ( w - 0.008 +  
0.246 ) / ( w - 0.008 + 0.0443 ) )
```

```
}
```

```
}
```

**Note:** For more information about the ICT commands and keywords specified in the ICT file, see “Creating the ICT File” chapter in *QRC Techgen Reference Manual*.

## EM Data File

An EM data input file specifies the technology information, such as current density limits, and provides a mapping between the layers for highlighting.

**Note:** The EM data file syntax is case-sensitive. The file must start with parenthesis.

Following is a sample EM data file

```
(  
; this file is case sensitive  
routingLayers = ("Poly" "Metal1" "Metal2" "Metal3" "Metal4" "Metal5" "Metal6")  
viaLayers = ("Cont" "Nimp" "Pimp" "Via1" "Via2" "Via3" "Via4" "Via5")  
viaWidthList = ((("Nimp" 0.2) ("Pimp" 0.2) ("Cont" 0.2) ("Via1" 0.2)  
("Via2" 0.2) ("Via3" 0.2) ("Via4" 0.2) ("Via5" 0.2))  
xrefLayers = (  
("POLYcont" ("Cont" "Cont"))  
("NSDcont" ("Nimp" "Nimp"))  
("PSDcont" ("Pimp" "Pimp"))  
("poly" ("Poly" "Poly"))  
("mt1" ("Metal1" "Metal1"))  
("mt2" ("Metal2" "Metal2"))  
("mt3" ("Metal3" "Metal3"))  
("mt4" ("Metal4" "Metal4"))  
("mt5" ("Metal5" "Metal5"))  
("mt6" ("Metal6" "Metal6"))  
("Via2NoCapInd" ("Via2" "Via2"))  
("Via2" ("Via2" "Via2"))  
)  
avgCurrentDensSpecList = (  
(nil layer "Poly" minW 0.0 maxW -1.0 minL 0 maxL 5 currentDensity ((1.0 , 110)))  
(nil layer "Metal1" minW 0.0 maxW -1.0 currentDensity ((1.1 , 110)))  
(nil layer "Metal2" minW 0.0 maxW -1.0 currentDensity ((1.2 , 110)))  
(nil layer "Metal3" minW 0.0 maxW -1.0 currentDensity ((1.3 , 110)))  
(nil layer "Metal4" minW 0.0 maxW -1.0 currentDensity ((1.4 , 110)))  
(nil layer "Metal5" minW 0.0 maxW -1.0 currentDensity ((1.5 , 110)))  
(nil layer "Metal6" minW 0.0 maxW -1.0 currentDensity ((1.6 , 110)))  
(nil layer "Cont" minW 0.0 maxW -1.0 res 1.0 currentDensity ((2.0 , 110)))  
(nil layer "Nimp" minW 0.0 maxW -1.0 res 1.1 currentDensity ((2.1 , 110)))  
(nil layer "Pimp" minW 0.0 maxW -1.0 res 1.2 currentDensity ((2.2 , 110)))  
(nil layer "Via1" minW 0.0 maxW -1.0 res 1.3 currentDensity ((2.3 , 110)))  
(nil layer "Via2" minW 0.0 maxW -1.0 res 1.4 currentDensity ((2.4 , 110)))
```

## Voltus-Fi Custom Power Integrity Solution User Guide

### File Formats

---

```
(nil layer "Via3" minW 0.0 maxW -1.0 res 1.5 currentDensity ((2.5 , 110)))
(nil layer "Via4" minW 0.0 maxW -1.0 res 1.6 currentDensity ((2.6 , 110)))
(nil layer "Via5" minW 0.0 maxW -1.0 res 1.7 currentDensity ((2.7 , 110)))
)
peakCurrentDensSpecList = (
(nil layer "Poly" minW 0.0 maxW -1.0 currentDensity ((1.1 , 110)))
(nil layer "Metall1" minW 0.0 maxW -1.0 currentDensity ((1.2 , 110)))
(nil layer "Metal2" minW 0.0 maxW -1.0 currentDensity ((1.3 , 110)))
(nil layer "Metal3" minW 0.0 maxW -1.0 currentDensity ((1.4 , 110)))
(nil layer "Metal4" minW 0.0 maxW -1.0 currentDensity ((1.5 , 110)))
(nil layer "Metal5" minW 0.0 maxW -1.0 currentDensity ((1.6 , 110)))
(nil layer "Metal6" minW 0.0 maxW -1.0 currentDensity ((1.7 , 110)))
(nil layer "Cont" minW 0.0 maxW -1.0 res 1.0 currentDensity ((2.1 , 110)))
(nil layer "Nimp" minW 0.0 maxW -1.0 res 1.1 currentDensity ((2.2 , 110)))
(nil layer "Pimp" minW 0.0 maxW -1.0 res 1.2 currentDensity ((2.3 , 110)))
(nil layer "Vial" minW 0.0 maxW -1.0 res 1.3 currentDensity ((2.4 , 110)))
(nil layer "Via2" minW 0.0 maxW -1.0 res 1.4 currentDensity ((2.5 , 110)))
(nil layer "Via3" minW 0.0 maxW -1.0 res 1.5 currentDensity ((2.6 , 110)))
(nil layer "Via4" minW 0.0 maxW -1.0 res 1.6 currentDensity ((2.7 , 110)))
(nil layer "Via5" minW 0.0 maxW -1.0 res 1.7 currentDensity ((2.8 , 110)))
)
```

## **Layer Map File**

The layer map file or map file provides the mapping between the layer names in the simulation database to those specified in the technology files (`qrcTechFile` and `ICT` file) for the EM analysis flow.

**Sample file name:** contactmapfile.txt

### **Format:**

RCX keyword simulation DB layername ICT keyword technology file layername

### **Format Description:**

RCX specifies the keyword for the simulation database.

simulation DB layername specifies the layer names in the simulation database that are to be mapped to the technology file layer names.

ICT specifies the keyword for the technology file.

technology file layername specifies the layer names in the technology file that map to those in the simulation database.

### **Example:**

```
<RCX keyword> <simulation DB layername> <ICT keyword> <technology file layername>
RCX metal1 ICT mt1
RCX metal2 ICT mt2
RCX metal3 ICT mt3
RCX metal4 ICT mt4
RCX pl1co ICT poly
RCX VIA1 ICT via1
```

## DFII Layer Map File

The APS/XPS-to-DFII layer map file provides the mapping between the layer names in the extractor-generated xDSPF file to the DFII layer names.

Multiple extraction layers can be mapped to the same DFII layer. This layer map file is also used for performing structural analysis.

**Sample file name:** df2layermap.txt

**Format:**

```
type      extraction_layer_name      dfII_layer_name
```

**Format Description:**

`type` specifies the type of layer; for example, metal, via, or poly.

`extraction_layer_name` specifies the name of the layer in the extractor-generated xDSPF file that is to be mapped to the DFII layer.

`dfII_layer_name` specifies the name of the DFII layer to which the extractor-generated layer name is being mapped.

**Example:**

```
#<type> <extraction_layer_name> <dfII_layer_name>
via      via3          Via3
metal    mt3          Metal3
via      via2_out_capind  Via2
metal    mt2          Metal2
via      via1          Via1
metal    mt1          Metal1
via      cont_ndiff    Cont
via      cont_pdiff    Cont
```



The order of layers in the layer map file is important. The layer map file must have layers listed in the top-to-bottom order.

## **Self-Heating Effect Analysis File**

This section lists the self-heating effect analysis parameter file, which is required for plotting the self- heating effect analysis plots.

### **shParamFile**

This file is the self-heating effect analysis parameter file, which is required for plotting the self- heating effect analysis plots.

**Sample file name:** param.sh

**Example:**

```
self_heat_parameters {  
  
    K_SH_Scale      = 1.23  
  
    beta_c1         = 0.0012  
    beta_c2         = 0.0023  
    beta_c3         = 0.0034  
  
    device_layers {  
        layer "OD" {  
            Rth              = 1000  
            finger_effect   = 2.0 * (1-exp(-0.3 * finger_num))  
            fin_effect      = 1.0 - ( 0.018 * (10 - fin_num) )  
        }  
        layer "OD_2" {  
            Rth              = 500  
            finger_effect   = 2.0 * (1-exp(-0.4 * finger_num))  
            fin_effect      = 1.0 - ( 0.019 * (30 - fin_num) )  
        }  
    }  
    conductor_layers {  
        layer "M1" {  
            alpha_connecting = 0.50  
            alpha_overlapping = 0.40  
        }  
        layer "M2" {  
            alpha_connecting = 0.45  
        }  
    }  
}
```

## **Voltus-Fi Custom Power Integrity Solution User Guide**

### File Formats

---

```
    alpha_overlapping = 0.35  
}  
}
```

## Power-Grid View Creation Files

This section lists the power-grid view creation file formats.

- [Format and Contents of Trigger File](#)
- [LEF Layermap File](#)
- [PGDB Layermap File](#)

### Format and Contents of Trigger File

The file format of the trigger file is:

```
CELL <cellname1>
MODE_NAME <mode1>
CONDITIONAL_INPUT <conditional input statement for cellname1>
CONDITIONAL_PIN <pin_name> (rise| fall | both)
SIM_START_TIME <time>
SIM_STOP_TIME <time>
END_MODE
END
```

where,

"|" denotes boolean OR operation

The table below lists the contents of the trigger file and provides their descriptions.

Trigger File Content	Description
<b>#Memory cell name</b>	
CELL <cellname1>	Name of the memory cell
<b>#Mode and conditional details for trigger</b>	
MODE_NAME <mode1>	Mode name and conditional input statement for the cell.
CONDITIONAL_INPUT <conditional input statement for cellname1>	

Trigger File Content	Description
CONDITIONAL_PIN <pin_name> (rise   fall   both)	Conditional pin statement for the cell. Signal pin for which the tool should rise or fall.
<b>#&lt;Other mode independent parameters&gt;</b>	
SIM_START_TIME <time>	SIM_START_TIME: Specifies the simulation start time. The default unit is in nanoseconds (ns).
SIM_STOP_TIME <time>	SIM_STOP_TIME: Specifies the simulation stop time. The default unit is in nanoseconds (ns).
<b>#End cell section</b>	
END_MODE	End cell section

### Example

```

CELL adc_sample
# MODE_NAME READ

MODE READ
CONDITIONAL_INPUT = ( WEN & !CEN )
CONDITIONAL_PIN CLK { RISE }
SIM_START_TIME 35ns
SIM_END_TIME   50ns
END_MODE

MODE WRITE
CONDITIONAL_INPUT = ( !WEN & !CEN )
CONDITIONAL_PIN CLK { RISE }
SIM_START_TIME 15ns
SIM_END_TIME   30ns
END_MODE

MODE STBY
CONDITIONAL_INPUT = ( CEN )
CONDITIONAL_PIN CLK { RISE }
SIM_START_TIME 80ns
SIM_END_TIME   120ns

```

## **Voltus-Fi Custom Power Integrity Solution User Guide**

### File Formats

---

END\_MODE

END

where,

" ! " denotes low state of the signal

" & " denotes boolean AND operation

## LEF Layermap File

A layer mapping file is required for the Rail Analysis library generation flow. The LEF layermap file contains the mapping information to map the layer names in the LEF file to the layer names in the technology file.

In the GUI, you can specify this file in the *Create Power Grid Views* form.

**Sample file name:** lefdef.layermap

**Command:** create\_pgv -lef\_layer\_map lef\_layer\_map\_file\_name

**Format:**

```
type      technology_layer_name      lefdef  lef_layer_name
```

**Format Description:**

**type** specifies the type of library layer, for example, metal or via.

**technology\_layer\_name** specifies the name of the library layer to map to the LEF layer.

**lefdef** specifies the keyword for the LEF layer mapping.

**lef\_layer\_name** specifies the name of the LEF layer to map to the library layer.

**Example:**

type	technology_layer_name	lefdef	lef_layer_name
metal	METAL_1	lefdef	M1
via	VIA_1	lefdef	VIA1
metal	METAL_2	lefdef	M2
via	VIA_2	lefdef	VIA2

where “type” can be poly/via/metal/diff.

## PGDB Layermap File

A layer mapping file is required for the Rail Analysis library generation flow. The PGDB layermap file contains the mapping information to map the xDSPF layer names that are written in the simulation database to the layer names in the technology file.

In the GUI, you can specify this file in the *Create Power Grid Views* form.

**Sample file name:** pgdb.map

**Command:** create\_pgv pgdb\_layer\_map pgdb\_layermap\_file\_name

**Format:**

```
type      technology_layer_name      pgdb      xDSPF_layer_name
```

**Format Description:**

**type** specifies the type of library layer. It can be poly/via/metal/diff.

**technology\_layer\_name** specifies the name of the library layer to map to the xDSPF layer name.

**pgdb** specifies the keyword for the xDSPF layer mapping.

**xDSPF\_layer\_name** specifies the xDSPF layer name.

**Example:**

```
type      technology_layer_name      pgdb      xDSPF_layer_name
# -----
poly      POLYCIDE                  pgdb      poly
via       CONT                      pgdb      pl1co
via       CONT                      pgdb      odCont1
via       CONT                      pgdb      odCont2
metal    METAL_1                   pgdb      metall1
via       VIA_1                     pgdb      VIA1
metal    METAL_2                   pgdb      metall2
via       VIA_2                     pgdb      VIA2
diff     OD                        pgdb      nwires
diff     OD                        pgdb      mwires
```

## Common Command Language (CCL) File

The common command language (CCL) file or the Quantus command file is used to specify the various commands and options that define the extraction run. This command file defines the various input files and libraries that make up the design, the output format, and the desired parasitics to extract from the design. The format and example of a CCL file generated in Voltus-Fi XL is provided below.

### Format:

Lines in the CCL file are in the following general tcl format:

```
command {argument_list}
```

where *argument\_list* is a list of name-value pairs.

For more information about the Quantus QRC command file or CCL format, see the “QRC Command Files” chapter in the *Quantus QRC Extraction Users Manual*.

### Example:

An example of the Voltus-Fi-generated CCL file is below. The customized extract command file, myextract.txt, is also shown below:

The myextract.txt is as follows:

```
extract \
    -selection "all" \
    -type "rc_coupled"
```

The Voltus-Fi-generated CCL file used to extract xDSPF is as follows:

```
xtract \
    -selection "all" \
    -type "rc_decoupled"

extraction_setup \
    -analysis "em" \
    -array_vias_spacing "0.35" \
    -max_fracture_length infinite \
    -max_fracture_length_unit "MICRONS" \
    -max_fracture_via_count 1 \
    -max_via_array_count_by_layer "Via1 1" "Via2 1" "Via3 1" "Via4 1" "Via5 4" \
    "Via6 4" "Via7 4" "Via8 4" "Via9 4" "Via10 4" "Via11 4" \
    -net_name_space "SCHEMATIC"

hierarchical_extract \
    -split_feedthrough_pins "true"
```

# Voltus-Fi Custom Power Integrity Solution User Guide

## File Formats

---

```
filter_res \
    -min_res 0.001 \
    -remove_dangling_res false

filter_cap \
    -exclude_floating_nets "true" \
    -exclude_floating_nets_limit 2000

output_db \
    -type dspf \
    -subtype extended \
    -add_explicit_vias true \
    -busbit_delimiter "\[\]" \
    -disable_instances false \
    -hierarchy_delimiter "/" \
    -include_cap_model "false" \
    -include_res_model "false" \
    -include_parasitic_cap_model "false" \
    -include_parasitic_res_model "comment" \
    -include_parasitic_res_length "true" \
    -include_parasitic_res_width false \
    -include_parasitic_res_width_drawn true \
    -output_xy \
    "CANONICAL_RES" \
    "PARASITIC_RES" \
    -netlist_coupling_values "single" \
    -add_bulk_terminal false \
    -sub_node_char "#" \
    -device_finger_delimiter "@" \
    -include_parasitic_res_temp_coeff "true" \
    -reduce_i_cards "true"

output_setup \
    -directory_name "vfi_quantus101" \
    -file_name "vfi_quantus101/testrun.dspf" \
    -net_name_space "SCHEMATIC" \
    -temporary_directory_name "vfi_temp" \
    -compressed false

process_technology \
    -technology_library_file "tech.lib" \
    -technology_name "tech" \
    -temperature 25.000000

input_db \
    -type pvs \
    -directory_name "testdir" \
    -hierarchy_delimiter "/" \
    -run_name "testrun"
```

## **Voltus-Fi Custom Power Integrity Solution User Guide**

### File Formats

---

---

## **Appendix A – Third-Party DSPF**

---

### **Scenario – Missing Number of Vias**

The DSPF files are created using the parasitic extraction tools and their content and format is heavily dependent on the extraction tool and the extraction runset.

In the EM analysis flow, after you calculate the current density of a resistor, you can compare it with the current density limits from the `qrcTechFile`, the `ICT` file, or the `emDataFile` that you have provided in order to locate violations. The current density limit is width-based for the metal or diffusion layers and area-based for via layers. The value of `$L` or `$W` is used to select the current density limit for metal layers and the value of `$A` is used to select the current density limit for via layers. In case of a via array, the total area of the via specified in the DSPF file is divided by the value of `$N`, which is the number of vias, in order to obtain the area of a single via, and the current density related to that area is selected from the technology file to check the EM violations.

You can view two examples below, one of a QRC-extracted DSPF file and the other of a third-party extracted DSPF file.

#### **Example 18-1 QRC-Extracted DSPF**

```
Rk173 AVSS#200 AVSS#201 0.020247 $metall1_conn $L=0.17 $W=0.54 $X=13.301 $Y=0.43  
Rw174 AVSS#44 AVSS#45 0.250000 $Via1 $A=0.0098 $N=2 $X=18.545 $Y=16.18
```

#### **Example 18-2 Third-Party Extracted DSPF**

```
Rg1 XR1#PLUS A#2 1.209331 $metall1 $L=0.56205 $W=0.1071 $X=5.582 $Y=0.13  
Ro31 net06 net06#7 12.500000 $VIA1 $A=0.003969 $X=23.085 $Y=0.135
```

The number of vias (`$N`) information is missing in the third-party DSPF file example. The EMIR analysis performed using such a third-party DSPF file will lead to erroneous results because the correct EM rules will not be picked from the `ICT` file or from the `emDataFile`.

As in the case of the example shown above, the resistance value, area, and co-ordinates for via resistor are given but the number of vias or `$N` is not mentioned. Without this information, the software is unable to calculate the area of a single via and check the current density limit related to that via area.

## Resolving the Missing \$N Issue in Third-Party DSPF

To resolve this problem of “missing number of vias” or \$N information in the third-party DSPF file, use the following method:

- Use one of the following options in the batch mode:

- Set the following variable in the vfibatch command file before running the load\_em\_results command:

```
set_variable viaCountFromResistance true
```

Alternatively, include the following option in the EMIR control file before running simulation:

```
emirutil viaCountFromResistance=[true]
```

**Note:** Specifying the viaCountResistance variable using the set\_variable command will generate EM results (taking care of \$N) during the batch mode run. Specifying the viaCountResistance variable using the emirutil command will generate EM results (taking care of \$N) in the report file \*rpt\_em, during Spectre simulation itself. The file, \*rpt\_em is created along with the simulation result file, \*.emir#\_bin file.

- Include the “area\_resistance” of vias in the ICT file. The format is as follows:

```
area_resistance <resistance value 1> <area 1> <resistance value 2>  
<area 2> .... <resistance value N> <area N>
```

OR

- Include the "contact\_resistance" of vias in the ICT file. The resistance of a single via is equal to contact\_resistance.

**Note:** If no area\_resistance value is matching, the software will use contact\_resistance.

The software will calculate the number of vias (without \$N) from the following:

- Resistance value of the via array resistor (from the DSPF)
- Resistance value of a single via, which is equal to the contact\_resistance (from the ICT file)

OR

Resistance value of all vias, which is equal to area\_resistance (from the ICT file)

## Index

---

### A

AC-Peak [199](#)  
analysis  
    static [101](#)  
Average (avg) [198](#)  
Avg-Abs (avgabs) [200](#)

### B

batch commands [294](#), [300](#), [308](#)  
    set\_variable [341](#)  
Batchmode  
    batch commands EM [315](#), [316](#)  
    batch commands generate DSPF [294](#)  
    batch commands IR [308](#), [312](#), [313](#)  
    batch commands PGV [323](#), [326](#), [328](#)  
    batch commands query [328](#), [329](#), [332](#),  
        [333](#)  
    batch commands SHE [321](#)  
    batch commands Static Current  
        Analysis [300](#)  
    batch commands xDSPF [294](#)

### C

Cadence Help, launching [50](#)  
create\_pgv [323](#)  
current density [198](#)

### D

data preparation [55](#)  
data requirements  
    EM rule file requirements [78](#)  
        EM data file [91](#)  
        EM rules in techfile [78](#)  
    setup for results visualization [76](#)  
    simulation requirements [61](#)  
    xDSPF requirements [60](#)

### E

Electromigration analysis [282](#)  
EM analysis  
    current directions for EM plots [209](#)  
    reporting current direction [210](#)  
    types of EM analysis [198](#), [199](#), [200](#)  
    viewing the EM analysis results [200](#)  
EM rules in techfile [78](#)  
    jmax parameters [79](#)  
    limit-based parameters [78](#)  
extract\_xdspf [294](#)

### F

file formats  
    pgv creation files [397](#)  
        LEF layermap file [400](#)  
        PGDB layermap file [401](#)  
        trigger files [397](#)  
    rail analysis files [385](#)  
        DFII layermap file [394](#)  
        EM data file [391](#)  
        EM Only ICT file [389](#)  
        ICT file [386](#)  
        Layer Map file [393](#)  
    self-heating effect files [395](#)

### G

ground\_pins [301](#)

### H

help, launching Cadence Help [50](#)

### I

IR drop  
    calculating effective resistance between  
        two nodes [178](#)  
    IR drop analysis results [125](#)

power gate support [118](#)  
solid shape highlighting [122](#)  
static EMIR analysis [119](#)  
static power grid solver [120](#)  
IR drop analysis results  
    displaying EMIR results [135](#)  
    querying EMIR results [139](#)  
    viewing EMIR violations in the Annotation browser [135](#)

other documentation [15](#)  
VPS documentation [15](#)  
Reports  
    default reports [183](#)  
    reports using EMIR control file [182](#)  
res\_id\_info [329, 333](#)  
RMS [198](#)  
run\_static\_simulation [303](#)

## L

load\_em\_results [315](#)  
load\_ir\_results [308](#)

## O

overview  
batchmode [293](#)  
EM analysis results [198](#)  
IR drop analysis results [117](#)  
pgv generation [94, 282](#)  
static current analysis [102](#)  
structural analysis [244](#)  
variables [341](#)  
Voltus-Fi-XL product [52](#)  
what-if analysis [262](#)

## P

Peak (max) [198](#)  
PGV generation  
    data requirements and flow [282](#)  
    flow [286](#)  
    outputs [289](#)  
    verification [290](#)  
pin\_2\_pin\_res [313](#)  
power\_pin [302](#)  
power-grid view [52, 282](#)  
print\_em\_report [316](#)  
print\_ir\_report [308](#)  
print\_rrlp\_report [312](#)  
print\_thermal\_report [321](#)

## R

related documents [14](#)

## S

search\_node\_id [332](#)  
search\_res\_id [328](#)  
set\_mmsim\_pgv\_nets [326](#)  
setup\_simulation [306](#)  
Spectre  
    direct EMIR analysis [56](#)  
    Iterated EMIR analysis [56](#)  
static  
    current analysis [101](#)  
    EM analysis [101](#)  
    rail analysis [101](#)  
    static current analysis [102](#)  
structural analysis  
    checks [244](#)  
        bottom via missing [246](#)  
        missing via connections [246](#)  
        skewed via ratio [249](#)  
        top via missing [245](#)  
        via coverage ratio [248](#)  
examples [255](#)  
    bottom/top via ratio [257](#)  
    missing via connections [258](#)  
    unconnected from bottom [256](#)  
    unconnected from top [255](#)  
    flow [250](#)

## T

Types of EM plots [211](#)  
    current density - avgabs [219](#)  
    current density - rms [221](#)  
    current density average [217](#)  
    current density max [215](#)  
    resistor current [213](#)  
    tap current [212](#)  
Types of IR plots [155](#)  
    IR drop [156](#)  
    least-resistive path (lrp) [166](#)

LRP analysis in power-gated designs [175](#)  
powergate current (pi) [160](#)  
powergate voltage (pv) [163](#)  
REffective (REff) [165](#)  
resistor current (rc) [156](#)  
transistor voltage (iv) [159](#)

## V

Variables [343](#)  
applyRThreshold [344](#)  
consolidatedReport [344](#)  
dynamicACPeak [345](#)  
geounit [346](#)  
geounit\_xy [346](#)  
powerRailGroundNets [347](#)  
powerRailRules [347](#)  
powerRailSupplyNets [346](#)  
process\_scale [347](#)  
rmsMultiplierPin [348](#)  
shParamFile [349](#)  
Tj [349](#)  
Tlife [349](#)  
useEncryptedWidth [349](#)  
useRecoveryFactor [350](#)  
viaCountFromResistance [350](#)  
variables  
optimisticEMRuleSelection [346](#)  
shEMEffect [348](#)  
UnmatchedRuleResistor [348](#)  
Voltus-Fi  
licenses [19](#)  
products [18](#)  
Voltus-Fi-XL  
flow overview [56](#)  
VPS  
data requirements [59](#)  
license string [19](#)  
license strings [19](#)  
product features [52](#)

## X

xDSPF Generation  
data requirements and flow [94](#)

## **Voltus-Fi Custom Power Integrity Solution User Guide**

---