cadence®

Voltus-Fi Custom Power Integrity Solution L User Guide

Product Version IC23.1
August 2023

© 2023 Cadence Design Systems, Inc. All rights reserved.
Printed in the United States of America.

Cadence Design Systems, Inc., 2655 Seely Avenue, 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 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.

Cadence is committed to using respectful language in our code and communications. We are also active in the removal and replacement of inappropriate language from existing content. This product documentation may however contain material that is no longer considered appropriate but still reflects long-standing industry terminology. Such content will be addressed at a time when the related software can be updated without end-user impact.

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

© 2023 All Rights Reserved.

© 2023 All Rights Reserved.

© 2023 All Rights Reserved.

© 2023 All Rights Reserved.

© 2023 All Rights Reserved.

© 2023 All Rights Reserved.

© 2023 All Rights Reserved.

© 2023 All Rights Reserved.

1

Product and Licensing Information

© 2023 All Rights Reserved.

Voltus-Fi Custom Power Integrity Solution Product Description

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

Table 1-1 Voltus-Fi Products

Product Name	Abbreviation	Product Number	Description
Voltus-Fi Custom Power Integrity Solution-L	VTS-Fi-L	VTS400	Uses the extracted view to perform IR drop and Electromigration (EM) analysis. It uses the results of the simulation performed using the `vavo-db` or the `psf` data.
Voltus-Fi Custom Power Integrity Solution-XL	VTS-Fi-XL	VTS500	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.
Voltus-Fi Custom Power Integrity Solution-AA	VTS-Fi-AA	VTS501	Supports the following advanced features: ■ Hierarchical EMIR Analysis ■ Hierarchical SHE analysis and modeling for Voltus, which is similar to the PGV Flow.

© 2023 All Rights Reserved.

Voltus-Fi Custom Power Integrity Solution Licenses

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 names and the corresponding license strings.

Table 1-2 Product Name and Corresponding License String

Product Name	License String
Voltus-Fi Custom Power Integrity Solution-L (VTS-Fi-L)	`Voltus_Power_Integrity_Fi_L`
Voltus-Fi Custom Power Integrity Solution-XL (VTS-Fi-XL)	`Voltus_Power_Integrity_Fi_XL`
Voltus-Fi Custom Power Integrity Solution-AA (VTS-Fi-AA)	`Voltus_Power_Integrity_Fi_AA`

Important

With the Voltus-Fi Custom Power Integrity Solution-XL license, you can also run Voltus-Fi Custom Power Integrity Solution-L (VTS-Fi-L).

For more information about the Voltus-Fi-XL product license, see the “Product and Licensing Information” chapter in the *Voltus-Fi Custom Power Integrity Solution XL User Guide*.

Note: Voltus-Fi will be obsolete in the next base IC release. The “next-generation” solution, Voltus-XFi, is available in this release of Virtuoso. It offers significant improvements to the overall EM-IR flow, including a unified use model, simplified simulation setup, and faster loading of results. The Voltus-XFi license is backward compatible and can be used to run Voltus-Fi if needed.

Checking Out Licenses for Voltus-Fi Custom Power Integrity Solution Products

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

© 2023 All Rights Reserved.

Running Voltus-Fi Custom Power Integrity Solution 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 functionality.
- Virtuoso® Analog Design Environment XL (ADE XL)

  Note: For information on licenses for the Virtuoso® Studio Design Environment, see “Virtuoso Studio Design Environment Licensing Setup” in the *Virtuoso Software Licensing and Configuration User Guide*.

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

For extraction, you require a license for the following Cadence tools:

- Quantus QRC version 9.1.2 and above

For simulation, you require a license for one of the following:

- SPECTRE version 10.1 and above
- APS version 10.1 and above
- ViVA version IC6.1.6 and above
- ADE L version IC6.1.6 and above

Voltus-Fi-L Compatibility with MMSIM/SPECTRE and EXT Product Versions

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

Note: From the 16.10 release version of the software, MMSIM is renamed to SPECTRE.

Voltus-Fi-L Version	EXT Versions	MMSIM/SPECTRE Versions
Voltus-Fi-L ICADVM20.1	EXT 20.1x	SPECTRE 19.1.0 ISRx
Voltus-Fi-L ICADVM20.1 ISR15	EXT 20.1x	SPECTRE 19.1.0 ISRx

© 2023 All Rights Reserved.

Voltus-Fi-L Version	EXT Versions	MMSIM/SPECTRE Versions
Voltus-Fi-L ICADVM20.1 ISR16	EXT 20.1x	SPECTRE 19.1.0 ISRx
Voltus-Fi-L ICADVM20.1 ISR17	EXT 20.1x	SPECTRE 19.1.0 ISRx
Voltus-Fi-L ICADVM20.1 ISR18	EXT 21.1x	SPECTRE 19.1.0 ISRx
Voltus-Fi-L ICADVM20.1 ISR19	EXT 21.1x	SPECTRE 20.1.0 ISRx
Voltus-Fi-L ICADVM20.1 ISR20	EXT 21.1x	SPECTRE 20.1.0 ISRx
Voltus-Fi-L ICADVM20.1 ISR21	EXT 21.1x	SPECTRE 21.1.0 ISRx
Voltus-Fi-L ICADVM20.1 ISR22	EXT 21.1x	SPECTRE 21.1.0 ISRx
Voltus-Fi-L ICADVM20.1 ISR23	EXT 21.1x	SPECTRE 21.1.0 ISRx
Voltus-Fi-L ICADVM20.1 ISR24	EXT 21.1x	SPECTRE 21.1.0 ISRx
Voltus-Fi-L ICADVM20.1 ISR25	EXT 21.1x	SPECTRE 21.1.0 ISRx
Voltus-Fi-L ICADVM20.1 ISR26	EXT 21.1x	SPECTRE 21.1.0 ISRx
Voltus-Fi-L ICADVM20.1 ISR27	EXT 21.2x	SPECTRE 21.1.0 ISRx

Voltus-Fi-L Compatibility with Voltus IC Power Integrity Solution

The table below provides the mapping between the Voltus-Fi-L 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.

The power-grid view (PGV) generation flow—Voltus-Fi and Voltus—does not provide backward compatibility with the earlier Voltus versions. It is recommended that you use the version specifications provided below.

Voltus-Fi-L Version	Voltus Version
Voltus-Fi-L ICADVM20.1	Voltus IC Power Integrity Solution 19.14
Voltus-Fi-L ICADVM20.1 ISR15	Voltus IC Power Integrity Solution 19.14
Voltus-Fi-L ICADVM20.1 ISR16	Voltus IC Power Integrity Solution 19.17
Voltus-Fi-L ICADVM20.1 ISR17	Voltus IC Power Integrity Solution 19.17

© 2023 All Rights Reserved.

Voltus-Fi-L Version	Voltus Version
Voltus-Fi-L ICADVM20.1 ISR18	Voltus IC Power Integrity Solution 19.17
Voltus-Fi-L ICADVM20.1 ISR19	Voltus IC Power Integrity Solution 20.62
Voltus-Fi-L ICADVM20.1 ISR20	Voltus IC Power Integrity Solution 20.62
Voltus-Fi-L ICADVM20.1 ISR21	Voltus IC Power Integrity Solution 20.63
Voltus-Fi-L ICADVM20.1 ISR22	Voltus IC Power Integrity Solution 20.63
Voltus-Fi-L ICADVM20.1 ISR23	Voltus IC Power Integrity Solution 20.63
Voltus-Fi-L ICADVM20.1 ISR24	Voltus IC Power Integrity Solution 20.63
Voltus-Fi-L ICADVM20.1 ISR25	Voltus IC Power Integrity Solution 20.63
Voltus-Fi-L ICADVM20.1 ISR26	Voltus IC Power Integrity Solution 20.63
Voltus-Fi-L ICADVM20.1 ISR27	Voltus IC Power Integrity Solution 20.63

© 2023 All Rights Reserved.

2

Getting Started

© 2023 All Rights Reserved.

Product and Installation Information

The Voltus-Fi-L product is launched within the Virtuoso Platform. To use this product, you must have the Virtuoso IC 6.1.8 release version.

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

```
vfibatch -v
```

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

For basic information and procedures required to install Cadence® products, see *Cadence Installation Guide*.

For information about how to configure the Virtuoso® Studio Design Environment, see "Setting up the Virtuoso Software", in *Virtuoso Software Licensing and Configuration User Guide*.

Note: Voltus-Fi will be obsolete in the next base IC release. The "next-generation" solution, Voltus-XFi, is available in this release of Virtuoso. It offers significant improvements to the overall EM-IR flow, including a unified use model, simplified simulation setup, and faster loading of results. The Voltus-XFi license is backward compatible and can be used to run Voltus-Fi if needed.

Supported and Compatible Platform

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

- Linux (only 64-bit)

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:

     ```
     setenv CDS_AUTO_64BIT ALL
     ```


© 2023 All Rights Reserved.

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

```
setenv CDS_AUTO_64BIT spectre:vfibatch
```


© 2023 All Rights Reserved.

Starting the Software

You can run Voltus-Fi-L interactively from the GUI, or you can execute it as a batch command. For details, see the following sections:

- Running Voltus-Fi-L in Batch Mode
- Running Voltus-Fi-L in the GUI Mode

Before running Voltus-Fi-L, ensure that the prerequisites detailed below are met.

PreRequisites for Running Voltus-Fi-L

Ensure that the following are available before running Voltus-Fi-L:

- A simulation run using either the `vavo-db` or the `PSF` data.
- Parasitic resistance and capacitance from the power and ground nets, performed to create an extracted view of the design using Quantus QRC.

  Note: For more information, see the "IR/EM Analysis Flow" chapter.
- An EM rule file that contains the EM reliability rules that need to be specified for EM analysis. For example, the rule specifying different current limits with respect to width, length, and temperature. These rules are specified in the technology file.

  Note: This is required only for EM analysis. For more information about specifying the EM rules, see the "EM Rules Specification" chapter.

Running Voltus-Fi-L in Batch Mode

Voltus-Fi-L can be run as a batch command, which means it runs from Virtuoso in a no-graph mode. Batch mode is typically used to perform the initial pass or fail analysis of the design's power-grid networks.

After running the analysis in batch mode, use the interactive GUI to view and analyze the results of the analysis.

For details of running Voltus-Fi-L in the batch mode, see the following topics in the "Batch Mode Execution" chapter:

- Migrating from Legacy (VPS-L) to Voltus-Fi-L in Batch Mode
- The Command File Support

© 2023 All Rights Reserved.

- Supported VPSL Batch File Commands
- Sample Batch Mode Command File

■ The Configuration File Support

- Sample Configuration Files
- EMIR Control File Options Supported in Voltus-Fi-L

The details of running Voltus-Fi-L in the GUI mode are provided below.

© 2023 All Rights Reserved.

Running Voltus-Fi-L in the GUI Mode

This section covers the details of how to run Voltus-Fi in the GUI mode including all the data requirements. The following topics are covered in this section:

- Data Requirements for the GUI Flow
- Launching Voltus-Fi-L from Virtuoso Schematic Editor L

Data Requirements for the GUI Flow

The following inputs are required for running the extracted view flow in the GUI mode:

- Simulation directory – specifies the path of the Spectre simulation directory containing the `vavo_db` database or the `PSF` database.
- `qrcTechFile`/`ICT` file or `emDataFile` – specifies the path of the Quantus QRC tech file/ICT file/emDataFile.

  Note: The tech file is only required for the EM analysis flow.
- QRC directory – specifies the location of the Quantus QRC run.
- QRC run name – specifies the Quantus QRC run name.
- Layermap file (optional) – specifies the layermap file that provides the mapping between the layer names in the simulation database to the layer names in the technology file. If not specified, the software assumes that the layer names are same.

  Note: The layermap file is only required for the EM analysis flow.
- Hierarchy (optional) – specifies the hierarchy name.
- Pres_cell name (optional) – specifies the presistor cell name.

© 2023 All Rights Reserved.

Launching Voltus-Fi-L from Virtuoso Schematic Editor L

Before viewing the IR drop and EM analysis results in the Voltus-Fi-L GUI, you will be performing the following tasks:

- Starting the Virtuoso Studio Design Environment
- Using the CIW to Open the Virtuoso Schematic Editor L
- Descending into Design Hierarchy
- Launching Voltus-Fi-L

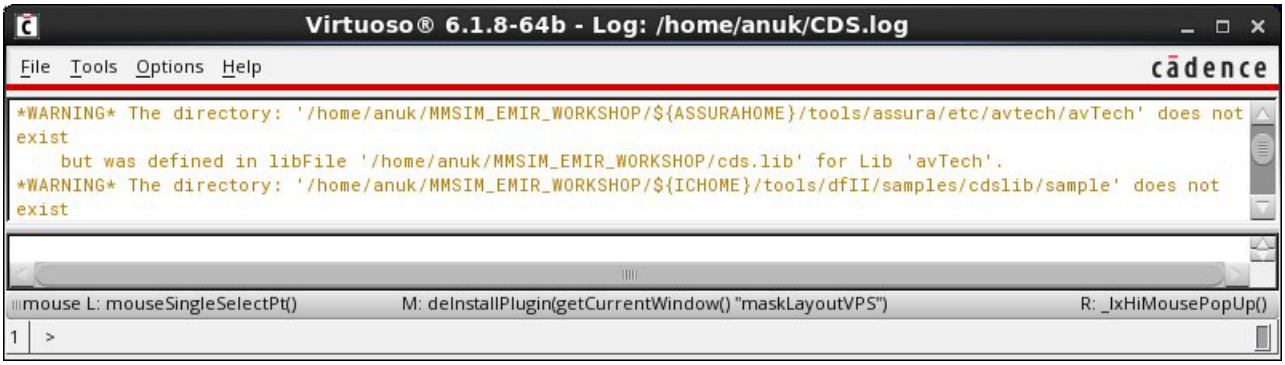
Starting the Virtuoso Studio Design Environment

Once you have set up the Virtuoso Studio Design Environment, you can start Virtuoso by typing the following in the unix, shell, or xterm window:

```
virtuoso &
```

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

Figure 2-1 Command Interpreter Window (CIW)



The title bar of the CIW contains the following information:

- The name of the workbench you are running (for example, `Virtuoso`).
- The path to the log file (CDS.log) that records the ongoing events of the design session. The content of the log file is displayed in the output area.

© 2023 All Rights Reserved.

Using the CIW to Open the Virtuoso Schematic Editor L

You can use the CIW to open the schematic view of the design you want to analyze. From the Main menu, you can either use the Tools menu or the File menu to access your design.

- Using the File Menu
- Using the Tools Menu

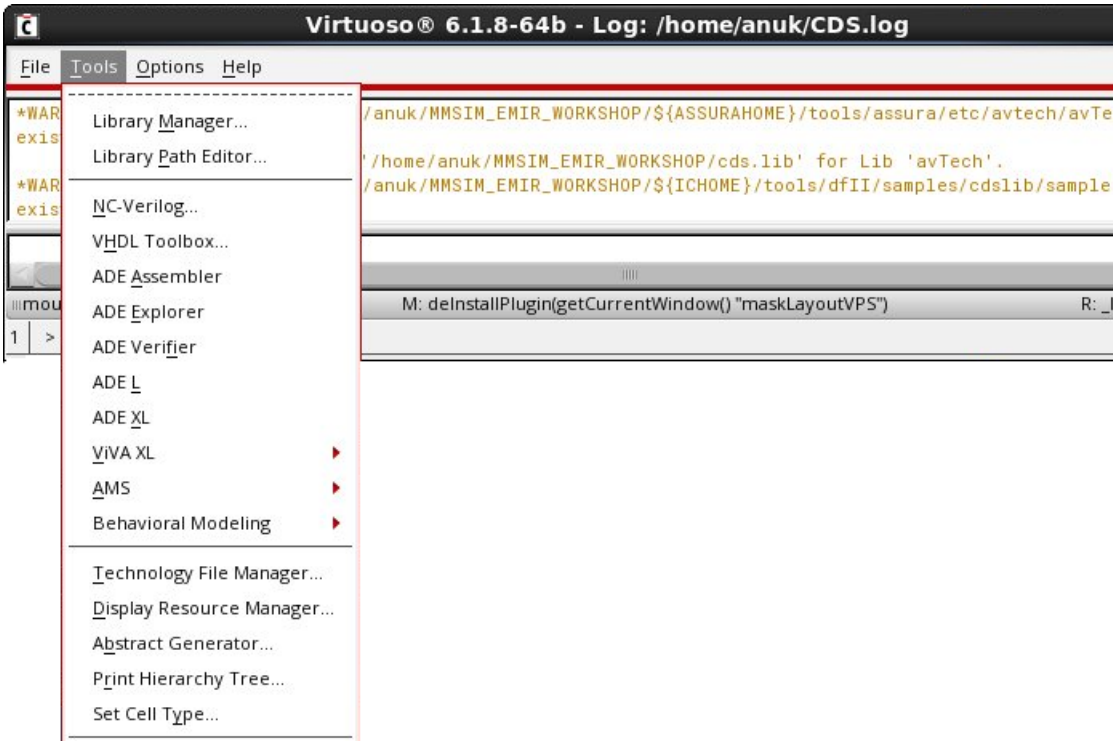
Using the File Menu

For information about how to open the schematic view of Virtuoso from the *File* menu in the CIW, see, "Working with Cellviews", in the *Virtuoso Design Environment User Guide*.

Using the Tools Menu

To open the schematic view of the design from the *Tools* menu, choose *Tools – Library Manager.* This is shown below.

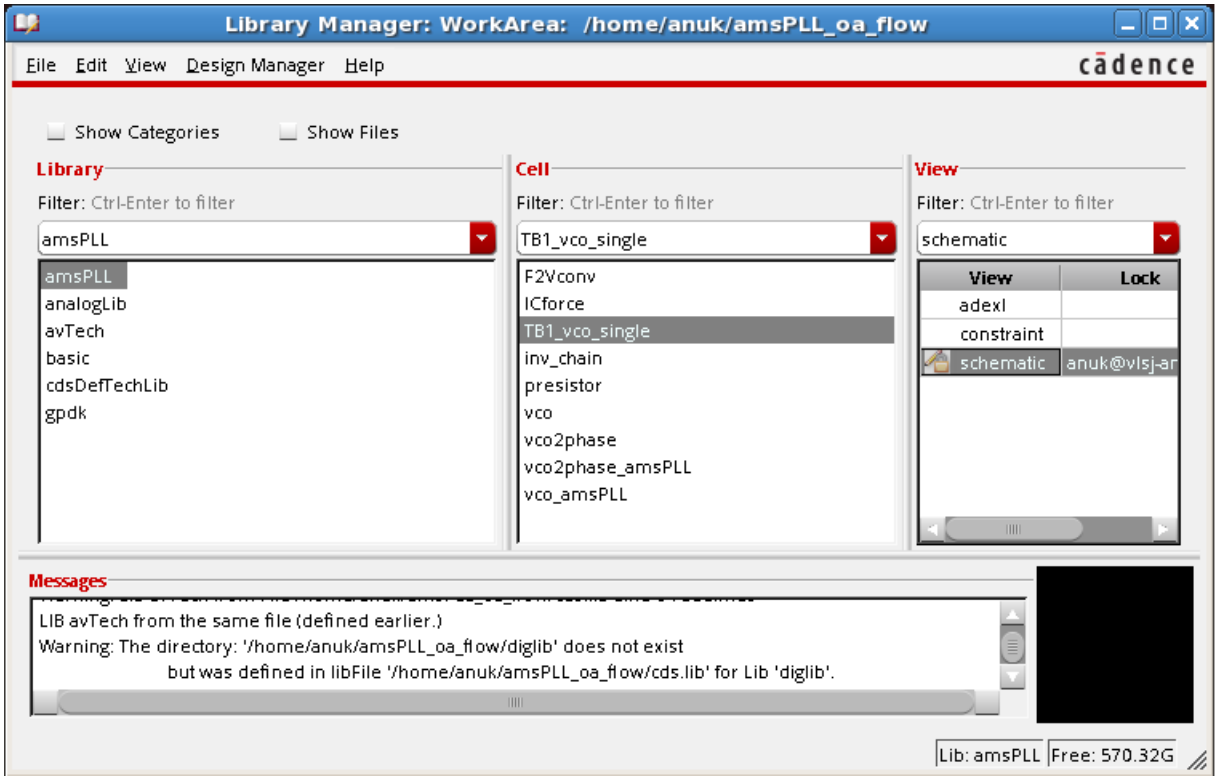
Figure 2-2 Command Interpreter Window – Opening the Library Manager Form



The Library Manager form opens. This is shown below.

© 2023 All Rights Reserved.

Figure 2-3 Library Manager Form



For more information on the *Library Manager*, see the *Cadence Library Manager User Guide*.

Opening a Schematic View in the Library Manager

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).

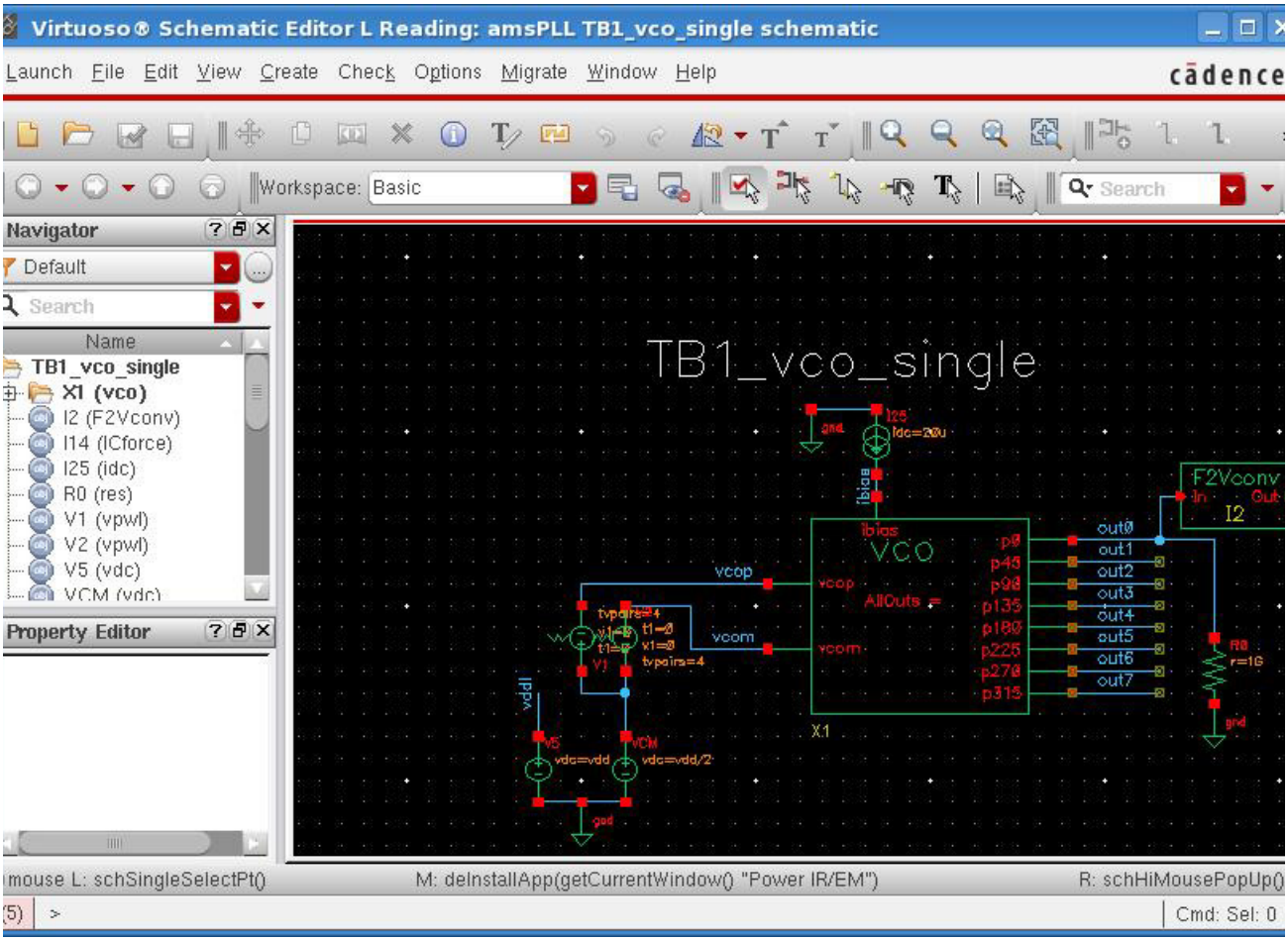
The information provided below is for *Views – Lists* mode.

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*. 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.

Once you have selected the schematic view, doubleclick the view to open the design in Virtuoso Schematic Editor L. This is shown below.

© 2023 All Rights Reserved.

Figure 2-4 Virtuoso Schematic Editor – Displaying the Design



You can also open the schematic view of the design by using the SKILL function, vsaOpenSchematic.

Descending into Design Hierarchy

In the Schematic Editor, before you run Voltus-Fi-L, you need to descend into the design hierarchy. A design hierarchy consists of many levels of a single design. When you descend into the design hierarchy, you view the extracted layout of the design you want to analyze.

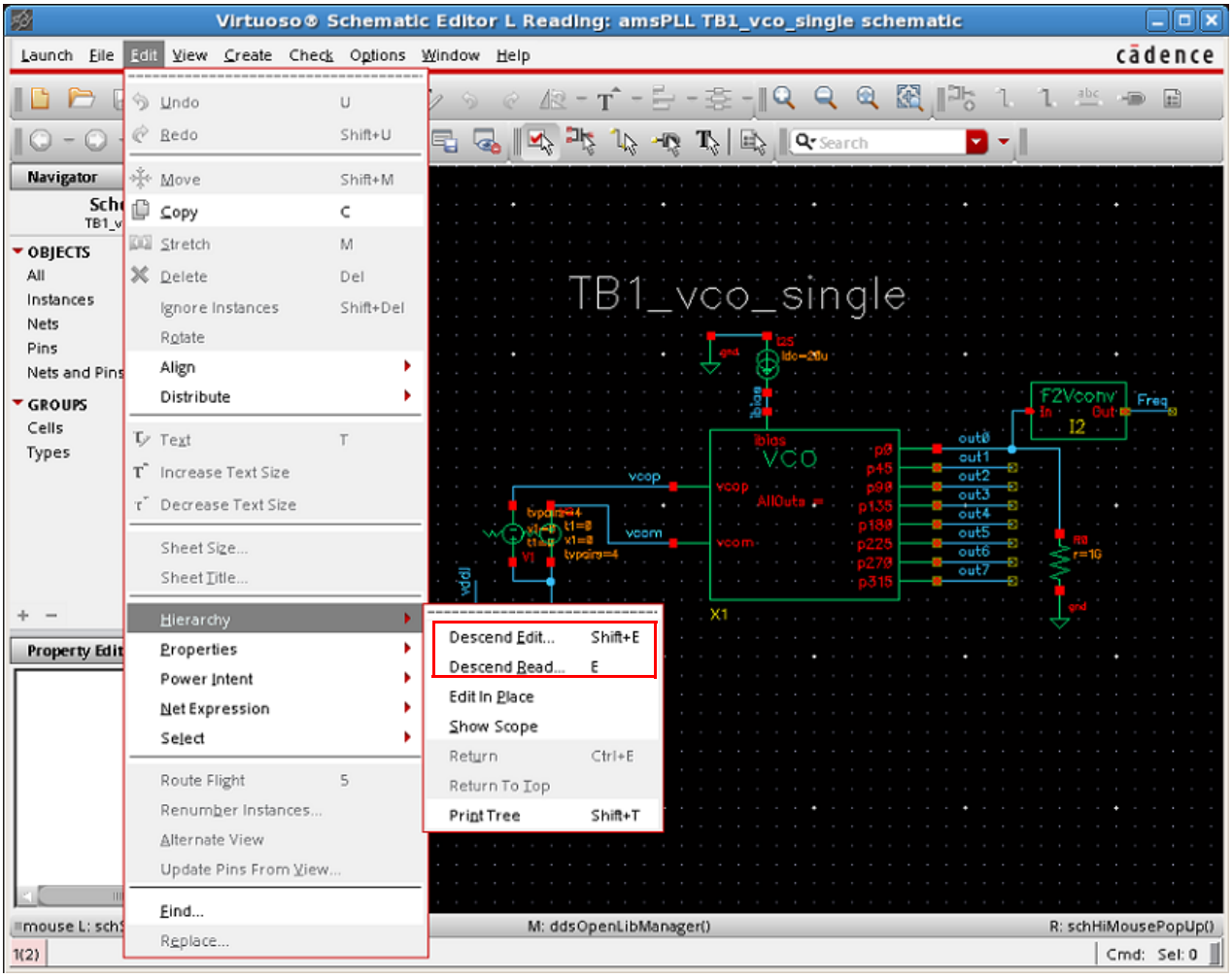
You can descend into the design by using the SKILL function, vsaDescendView.

In the GUI, to descend into the design hierarchy, follow these steps:

- In the *Virtuoso Schematic Editor L*, choose *Edit – Hierarchy –* and *Descend Read*. Choose the *Descend Edit* option to make any change in the extracted layout.

© 2023 All Rights Reserved.

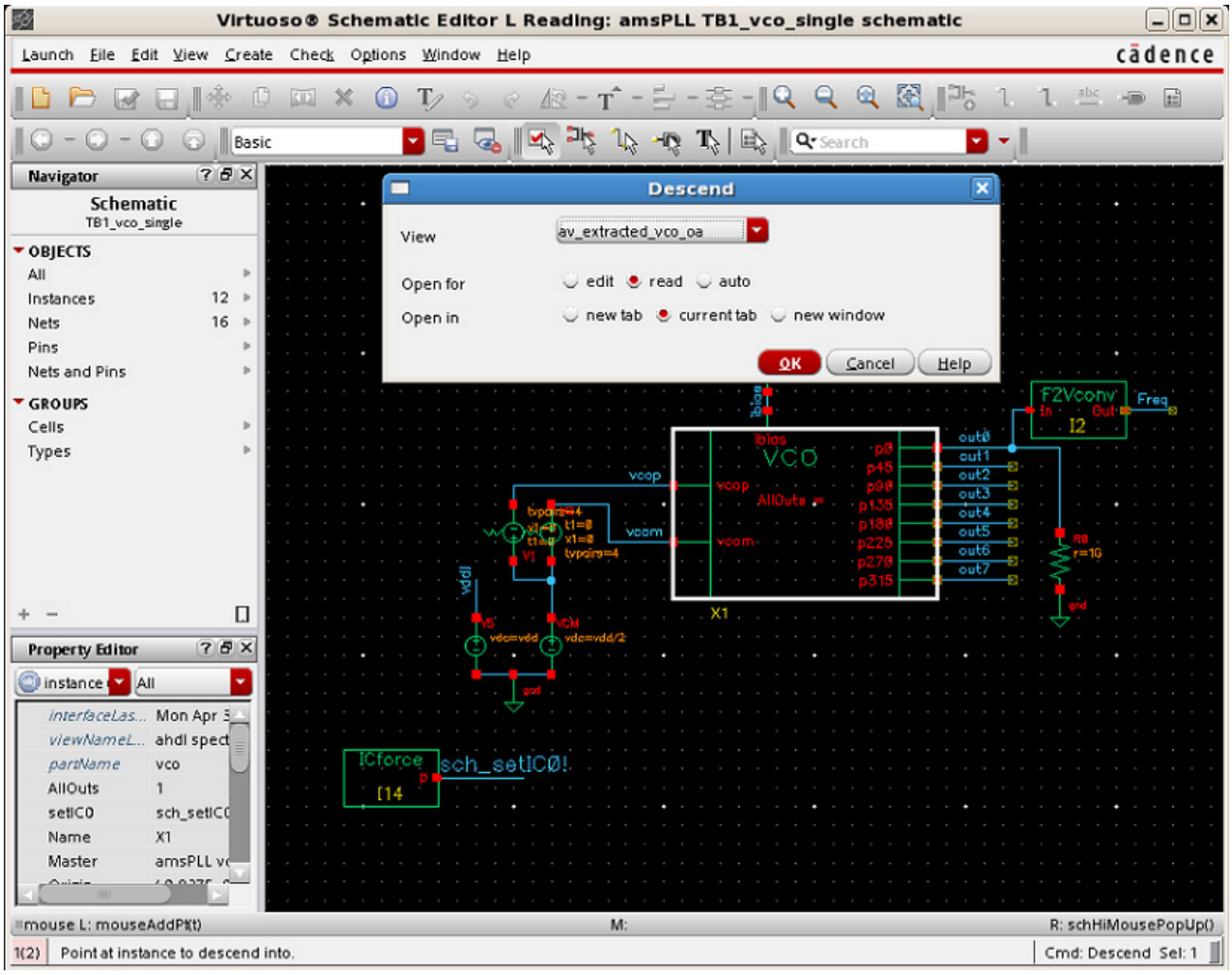
Figure 2-5 Virtuoso Schematic Editor L – Descending into Design Hierarchy



Note: You can also double click the instance to which you want to descend. The Descend form opens.

© 2023 All Rights Reserved.

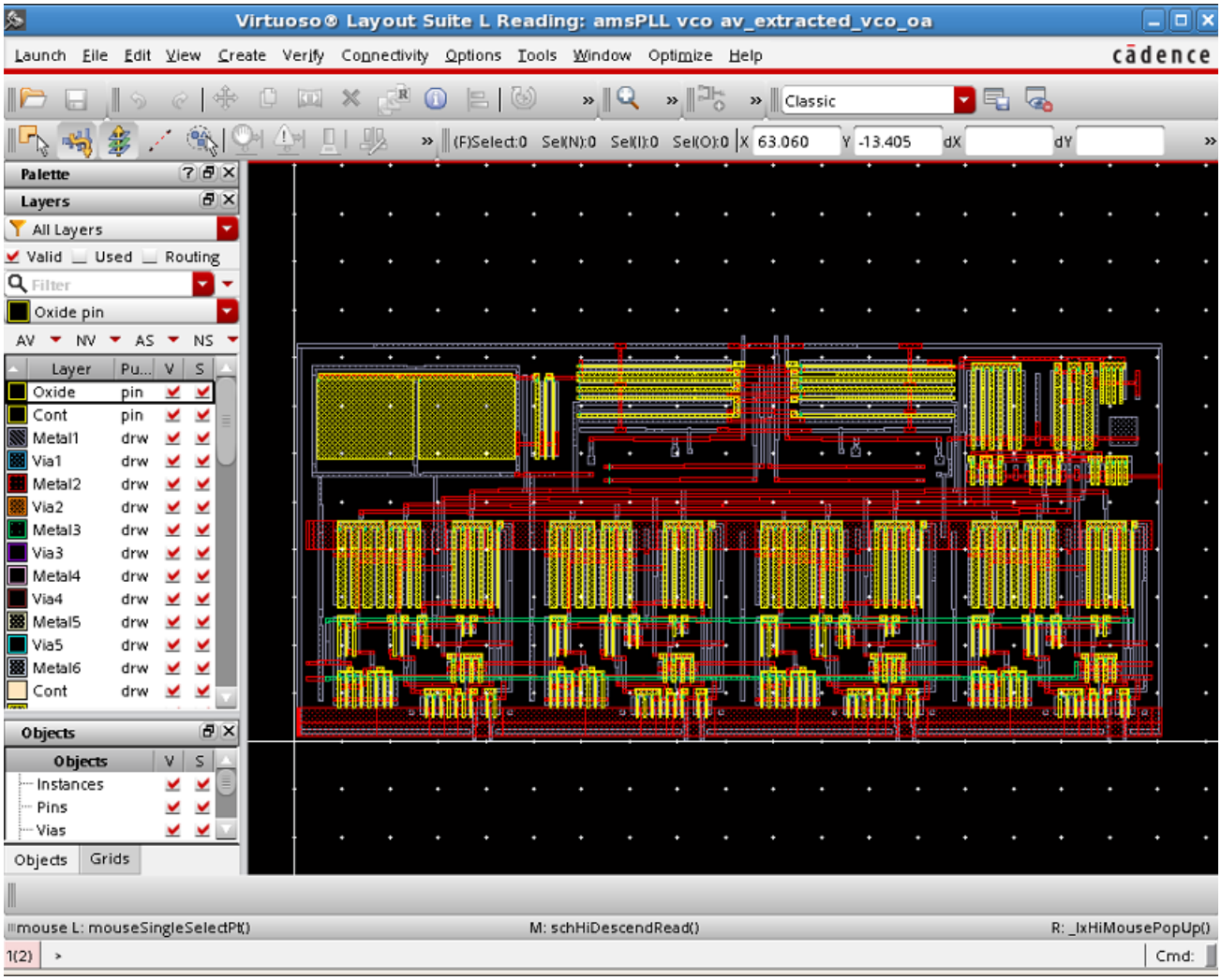
Figure 2-6 Descend Form – Selecting the Design View



- In this form, select the name of the extracted view from the options provided in the *View* cyclic field. In the example shown in the figure above, the extracted view is `av_extracted_vco_oa`.
- Click *read* to open the extracted view for reading. You can select *edit* if you want to modify something in the extracted view.
- Choose where you want to open the extracted view by selecting *Open in – new tab, current tab,* or *new window.*
- Click *OK*.
- The selected view of the instance opens in Virtuoso® Layout Suite L. This is shown in the figure below.

© 2023 All Rights Reserved.

Figure 2-7 Virtuoso Layout Suite L – Displaying the Selected Design View



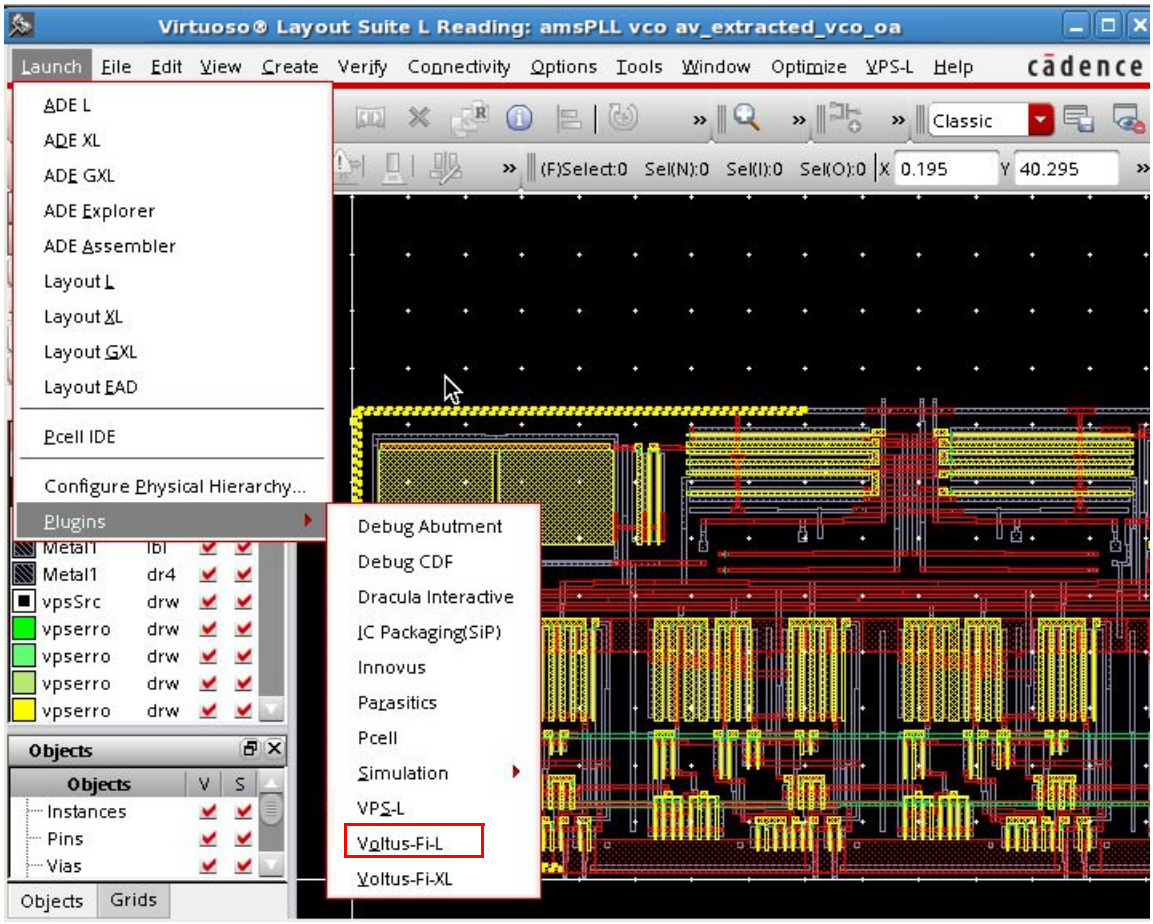
Launching Voltus-Fi-L

After opening the detailed view of the selected instance of the design, open the Voltus-Fi-L GUI from the Main menu of the Virtuoso® Layout Suite by performing the following steps:

- Choose *Launch – Plugins – Voltus-Fi-L*. This is shown below.

© 2023 All Rights Reserved.

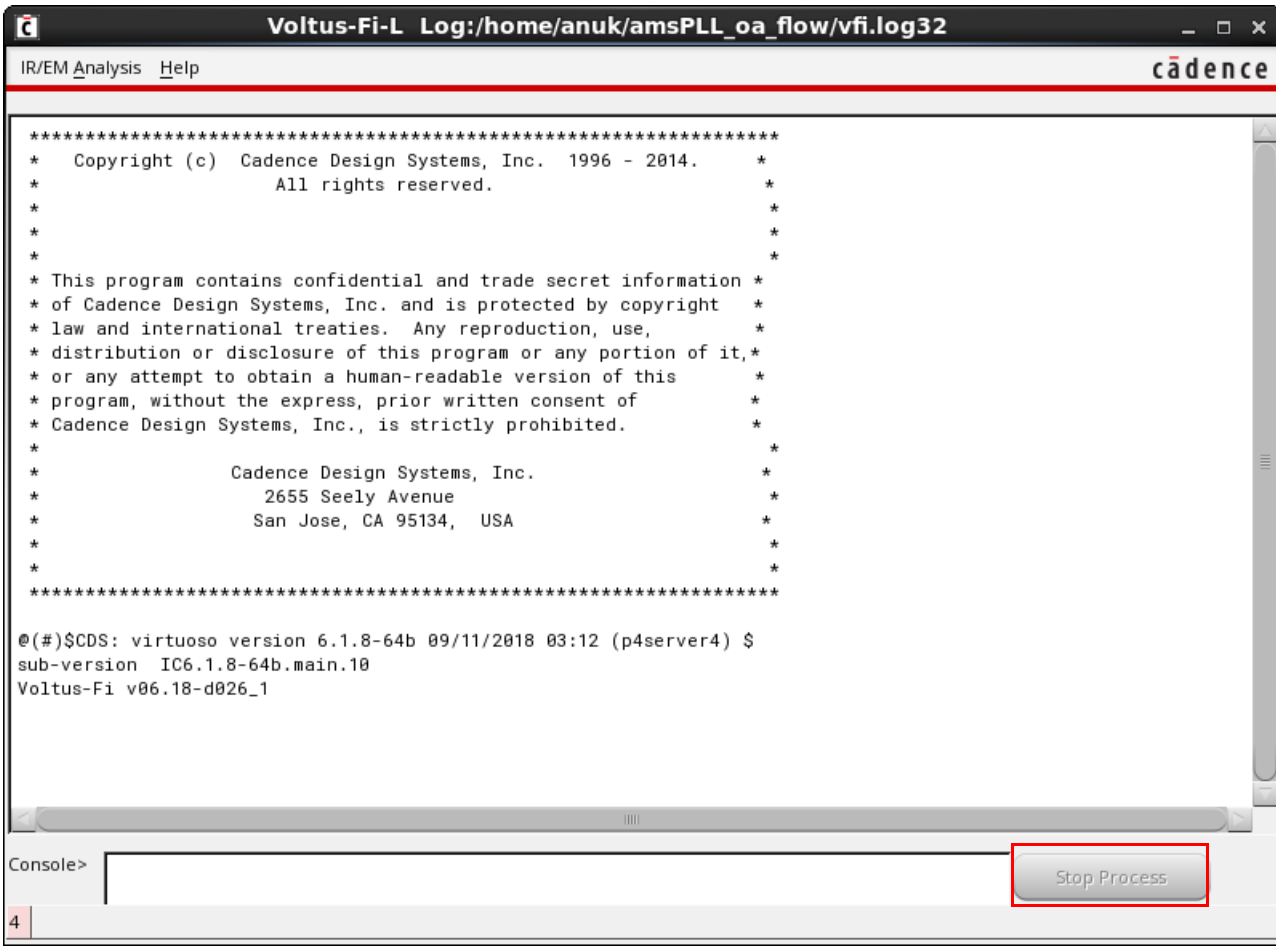
Figure 2-8 Running Voltus-Fi-L from the Layout Suite



- The Voltus-Fi-L console opens. This is shown below.

© 2023 All Rights Reserved.

Figure 2-9 The Voltus-Fi-L Console



- The following main menus of Voltus-Fi-L are available:
  - 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. This shown using a red highlighted text box in the above figure. Click this button to stop a process while it is running.

Log Files for Voltus-Fi-L

When you run Voltus-Fi-L either in the GUI or in the batch mode, a standalone log file is created to record the complete Voltus-Fi-L session. This file is created in addition to the

© 2023 All Rights Reserved.

`CDS.log` file that records the ongoing events of the design session in Virtuoso. This Voltus-Fi-L-specific log file is saved in the following location by default.

working_dir`/vfi.log#`

Example:

working_dir`/vfi.log19`

For each session, a new log file is created, such as `vfi.log1, vfi.log2,` and so on. You can save up to twenty log files, after which the last log file, `vfi.log20`, will be overwritten in subsequent sessions.

Note: If you do not have permissions in the working directory, these files will be saved in your home directory.

A sample log file is shown below.

```
************************************************************

@(#)$CDS: virtuoso version 6.1.7-64b 01/24/2017 20:25 (sjfhw312) $
sub-version  IC6.1.7-64b.main.372
Voltus-Fi v06.17-e282_1
Checkout of license "Voltus_Power_Integrity_Fi_L" was successful
INFO : Loading results for IR plots display

INFO : Loading results done
Time Msg : Time taken 2.00 seconds. User cpu 1.00 seconds. System cpu 1.00 seconds.
Time Msg : Time taken 2.00 seconds. User cpu 1.00 seconds. System cpu 1.00 seconds.
INFO : Loading results for IR plots display success.
Time Msg : Time taken 2.00 seconds. User cpu 1.00 seconds. System cpu 1.00 seconds.
Time Msg (IR plot) : Time taken 0.00 seconds. User cpu 0.00 seconds. System cpu
0.00 seconds.
INFO : Loading results for IR plots display

INFO : Loading results done
Time Msg : Time taken 1.00 seconds. User cpu 1.00 seconds. System cpu 0.00 seconds.
Time Msg : Time taken 1.00 seconds. User cpu 1.00 seconds. System cpu 0.00 seconds.
INFO : Loading results for IR plots display success.
Time Msg : Time taken 1.00 seconds. User cpu 1.00 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 450:1005 by 90265:41155
INFO: layout X-Y are 1450:10790 by 86985:39000
```


© 2023 All Rights Reserved.

```
INFO: layout X-Y are 1450:10790 by 86985:39000
```


© 2023 All Rights Reserved.

Accessing Documentation and Help

You can access the Voltus-Fi-L documentation and help system using the following methods:

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 GUI

The software provides the following two methods to access documentation and help from the GUI:

Select Help on the Main Menu

Click the Help button in the Voltus-Fi-L console. The following submenus are available:

- *Voltus-Fi User Guide*: opens the Voltus-Fi Custom Power Integrity Solution L User Guide.
- *What's New*: opens the Voltus-Fi Custom Power Integrity Solution L What's New chapter in the Virtuoso What's New.
- *About:* provides the version information for the software.

Select Help on a Voltus-Fi-L Form

Click the *Help* button on the bottom right corner of a form. Clicking the *Help* button opens the Voltus-Fi-L user guide entry for the form in the Cadence Help window.

© 2023 All Rights Reserved.

Other Source of Information

You can also get help on Cadence products by selecting *Customer Support* on the *Help* menu. The *Customer Support* submenu provides access to the following Cadence resource:

- Cadence Online Support: Opens Cadence Online Support in your browser.

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

© 2023 All Rights Reserved.

© 2023 All Rights Reserved.

3

Product Overview

© 2023 All Rights Reserved.

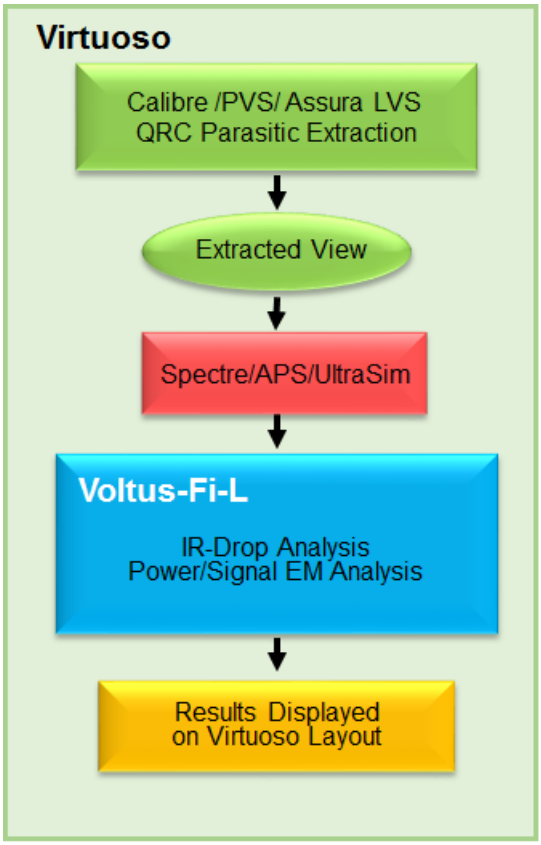
Voltus-Fi-L Product Overview

Voltus-Fi-L provides the ability to perform IR drop and Electromigration (EM) analysis using an extracted view of the design. It uses the results of the simulation performed using the `vavo-db` or the parameter storage format (`PSF`) data.

The target designs include the analog, analog/mixed-signal (AMS) and custom digital designs of varying 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-L 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.

Figure 3-1 Voltus-Fi-L Flow Within the Virtuoso Environment



© 2023
All Rights Reserved.

The product also supports viewing of the IR drop and EM analysis results in the vsaplot utility. This utility opens independently of the Virtuoso environment and provides a high speed of visualization.

Note: Voltus-Fi will be obsolete in the next base IC release. The “next-generation” solution, Voltus-XFi, is available in this release of Virtuoso. It offers significant improvements to the overall EM-IR flow, including a unified use model, simplified simulation setup, and faster loading of results. The Voltus-XFi license is backward compatible and can be used to run Voltus-Fi if needed.

The key product features are described below.

Key Product Features and Description

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

Table 3-1 Product Features – Voltus-Fi-L

Feature	Description
Viewing IR Drop Analysis Results	**Signal net and power-rail IR drop analysis** includes support for the following: ■ Solid shape highlighting ■ Querying results ■ The following plot types; peak and average IR drop, and peak, average, and RMS resistor current plots For details, see the “IR Drop Analysis Results” chapter.

© 2023 All Rights Reserved.

Viewing EM Analysis Results	**Signal net and power-rail EM analysis** includes support for the following: ■ Solid shape highlighting ■ Querying results ■ EM analysis types: peak, average, rms and AC-peak ■ Support for using text-based ICT file for specifying EM rules ■ Advanced EM rules with `qrcTechFile` for 20nm and below technologies. ■ The `emDataFile.txt` file is supported for backward compatibility in 28nm and above technologies. For details, see "EM Analysis Results" chapter.
PGV Generation	**Static and Dynamic PGV generation** feature lets you create power-grid views (PGVs) of an analog design block that can be used in Voltus for mixed-signal analysis. For details, see "Power-Grid View Creation" chapter.
LRP Support	**Least-Resistive Path (LRP) analysis** helps you identify weakly-connected instances in the design during early stages of power planning. It displays the worst IR drop violations and plots the LRP on demand for any node. For details, see "IR Drop Analysis Results" chapter.
Viewing Results in vsaplot	The **vsaplot utility** is used to view the results of IR drop and EM analysis that is performed in Voltus-Fi-L. The key benefits of viewing the results of IR drop and EM analysis in vsaplot are as follows: ■ No dependency on the Virtuoso layout ■ Increased speed of visualization For details, see "Viewing Results in vsaplot or vfiplot" chapter.

© 2023 All Rights Reserved.

4

IR/EM Analysis Flow

© 2023 All Rights Reserved.

Overview

Before performing IR drop and EM analysis in Voltus-Fi-L, you need to generate the extracted view of the design and then simulate this view. This is referred to as the IR/EM Analysis flow.

The IR/EM Analysis flow involves the following:

- Comparing the schematic design with the layout design to verify the logical connectivity of the design. This is done by running either Pegasus or PVS LVS (Layout as compared to Schematic) or Calibre® LVS.
- Extracting the parasitic resistance and capacitance from the power and ground nets and creating an extracted view of the design using Quantus QRC.
- Simulating the extracted view of the design using Spectre or APS simulators.

Once these are completed successfully, the simulation results generated as an output, are used as input in Voltus-Fi-L to view and plot the IR drop and EM analysis results on the Virtuoso layout and generate reports of the analyses to identify and debug regions of high IR drop and EM violations in the design.

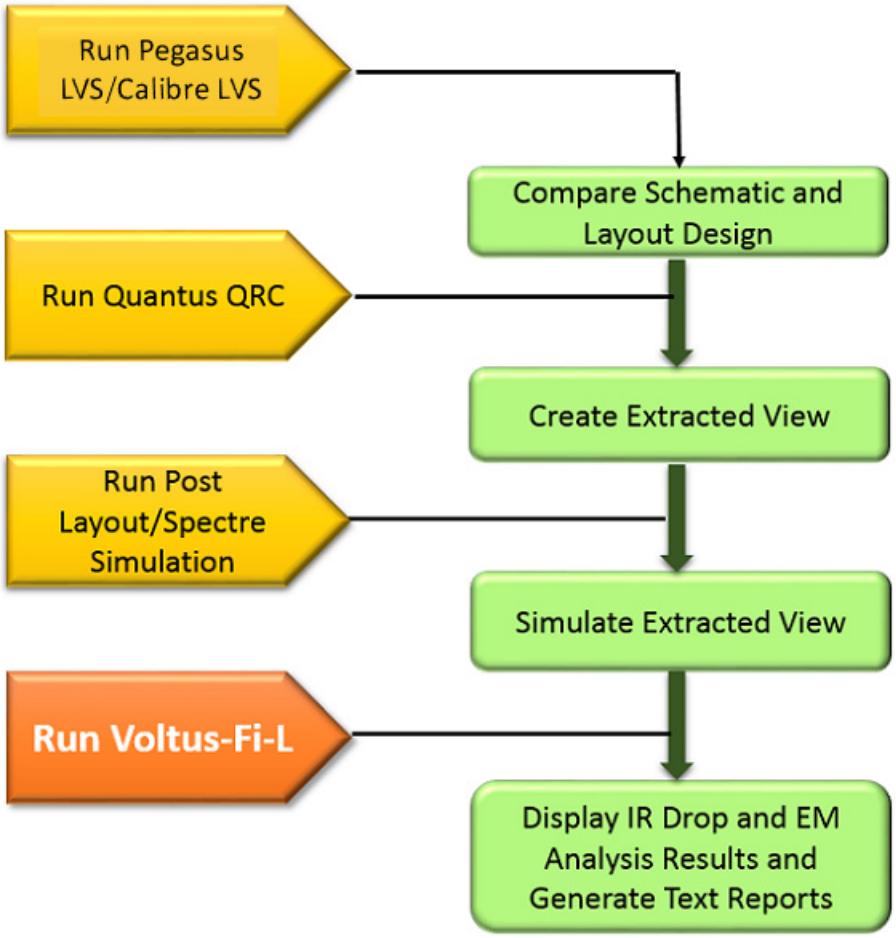
The above tasks are detailed in the following sections:

- Running Pegasus or PVS LVS
- Running Pegasus or PVS Quantus
- Running Extraction with Quantus QRC (Calibre) Interface
- Running a Spectre Simulation
- Running an APS Simulation

This high-level flow of the above tasks is shown below.

© 2023 All Rights Reserved.

Figure 4-1 Voltus-Fi-L Pre-Requisites High-Level Flow



Note: Quantus QRC creates an extracted view that can be simulated in Cadence® Analog Design Environment (ADE). Use Voltus-Fi-L to superimpose the simulation data generated through circuit simulation using Cadence® Spectre® Circuit Simulator or Spectre® Accelerated Parallel Simulator (APS) onto the Virtuoso extracted view.

The following sections describe this flow in detail and explain the options to be set for Pegasus/PVS and Spectre for this analysis and the flow for Voltus-Fi-L.

Running Pegasus or PVS LVS

Before extracting the power grid and the signal nets, run Pegasus or PVS LVS on the layout view of the design you want to analyze. No special modifications are required to be made to the Pegasus or PVS LVS decks.

© 2023 All Rights Reserved.

In the *Virtuoso Layout Suite*, perform the following steps:

- Select *Pegasus* or *PVS* from the Main menu of the Virtuoso Layout Suite.
- Select *Run LVS* from this menu. This is shown below.

Figure 4-2 Run LVS Menu from Pegasus Menu

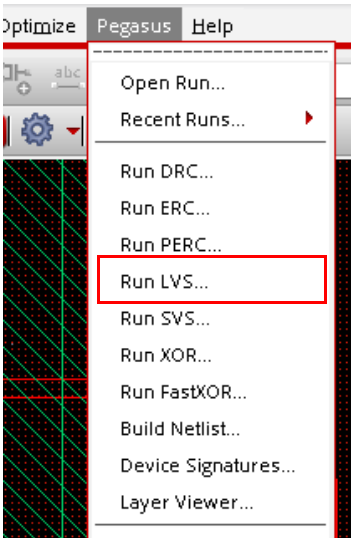
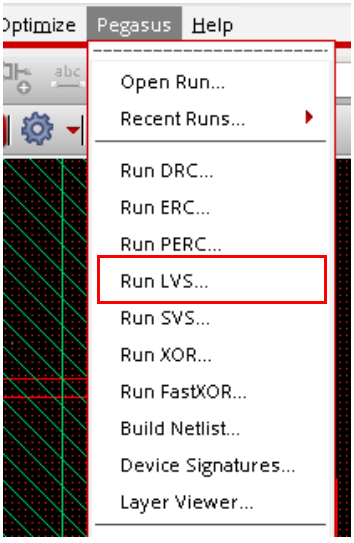


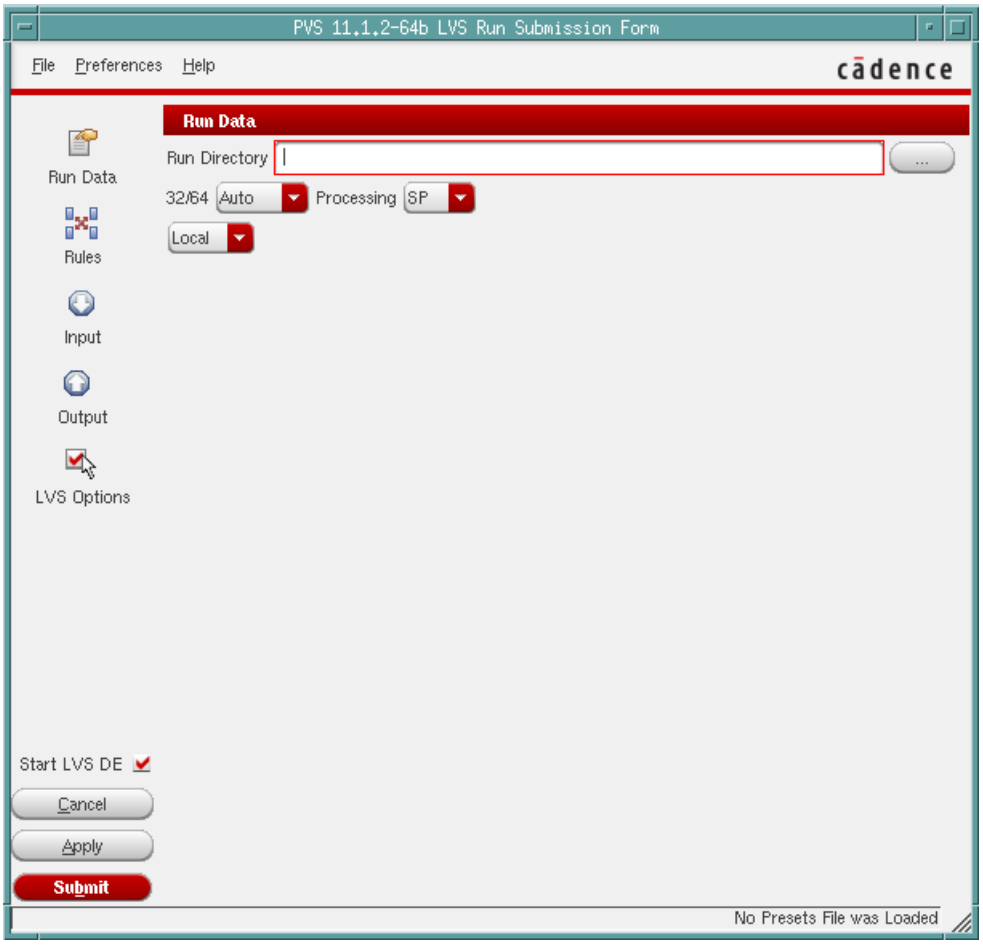
Figure 4-3 Run LVS Menu from PVS Menu



- The Pegasus or PVS LVS Run Submission Form opens. This form is shown below.

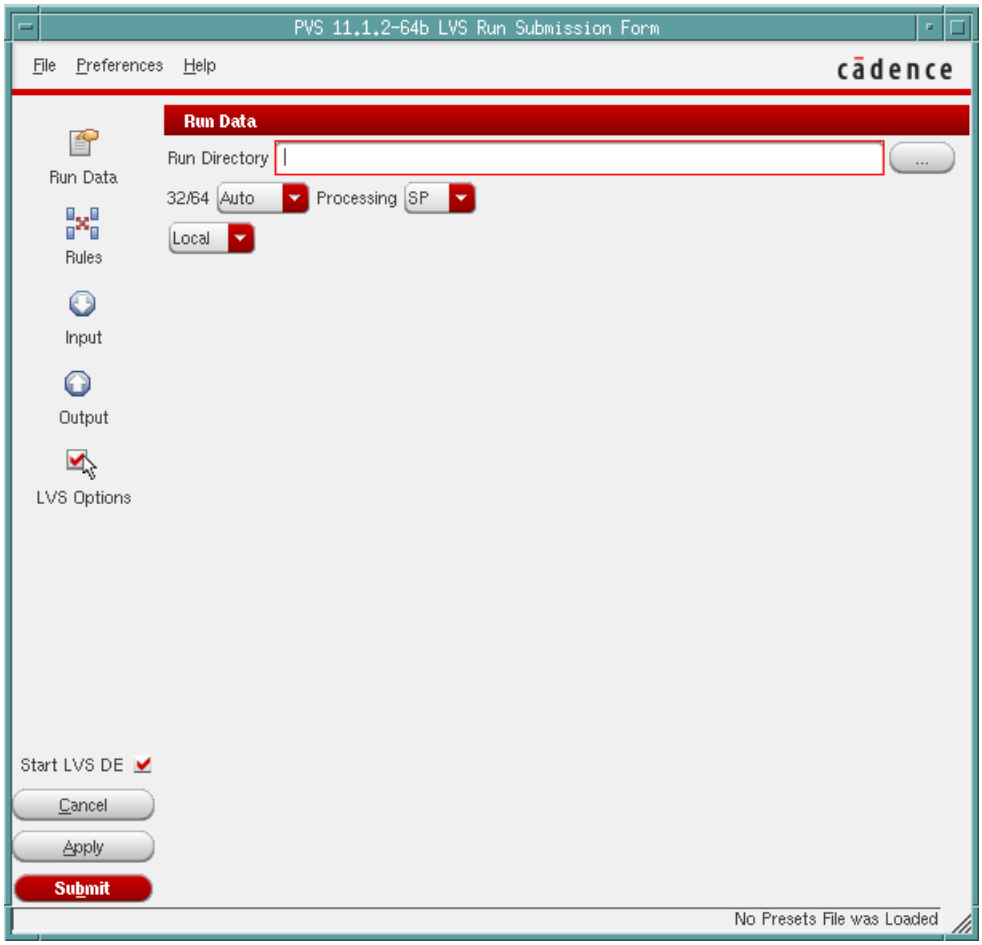
© 2023 All Rights Reserved.

Figure 4-4 Pegasus LVS Run Submission Form



© 2023
All Rights Reserved.

Figure 4-5 PVS LVS Run Submission Form

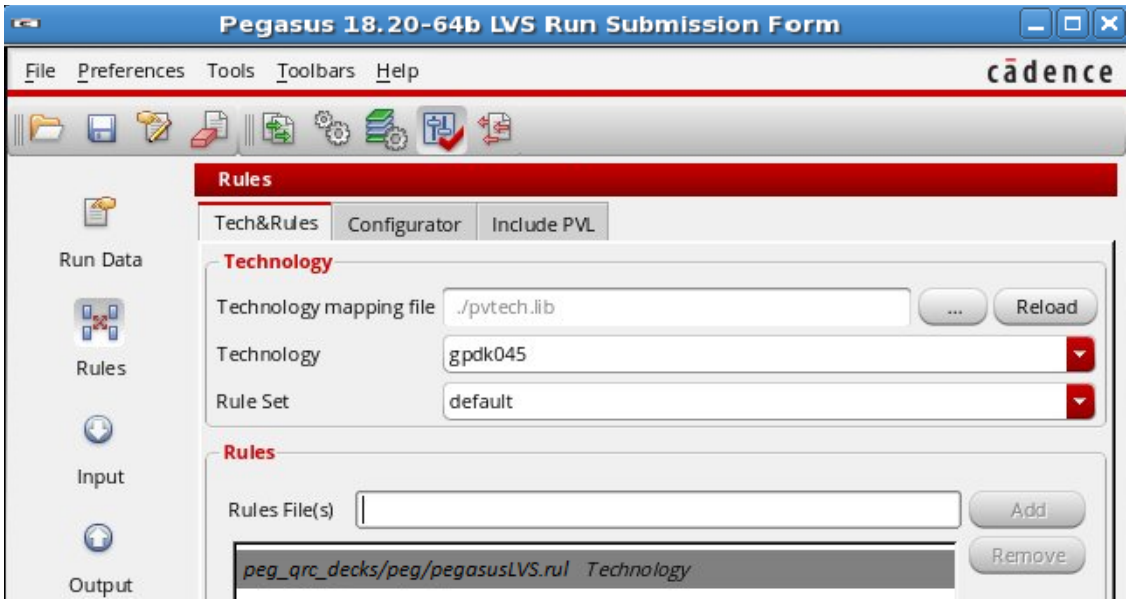


In this form,

- Click *Run Data* and specify the Run Directory path.
- Click *Rules* and specify the rules deck file as shown below.

© 2023
All Rights Reserved.

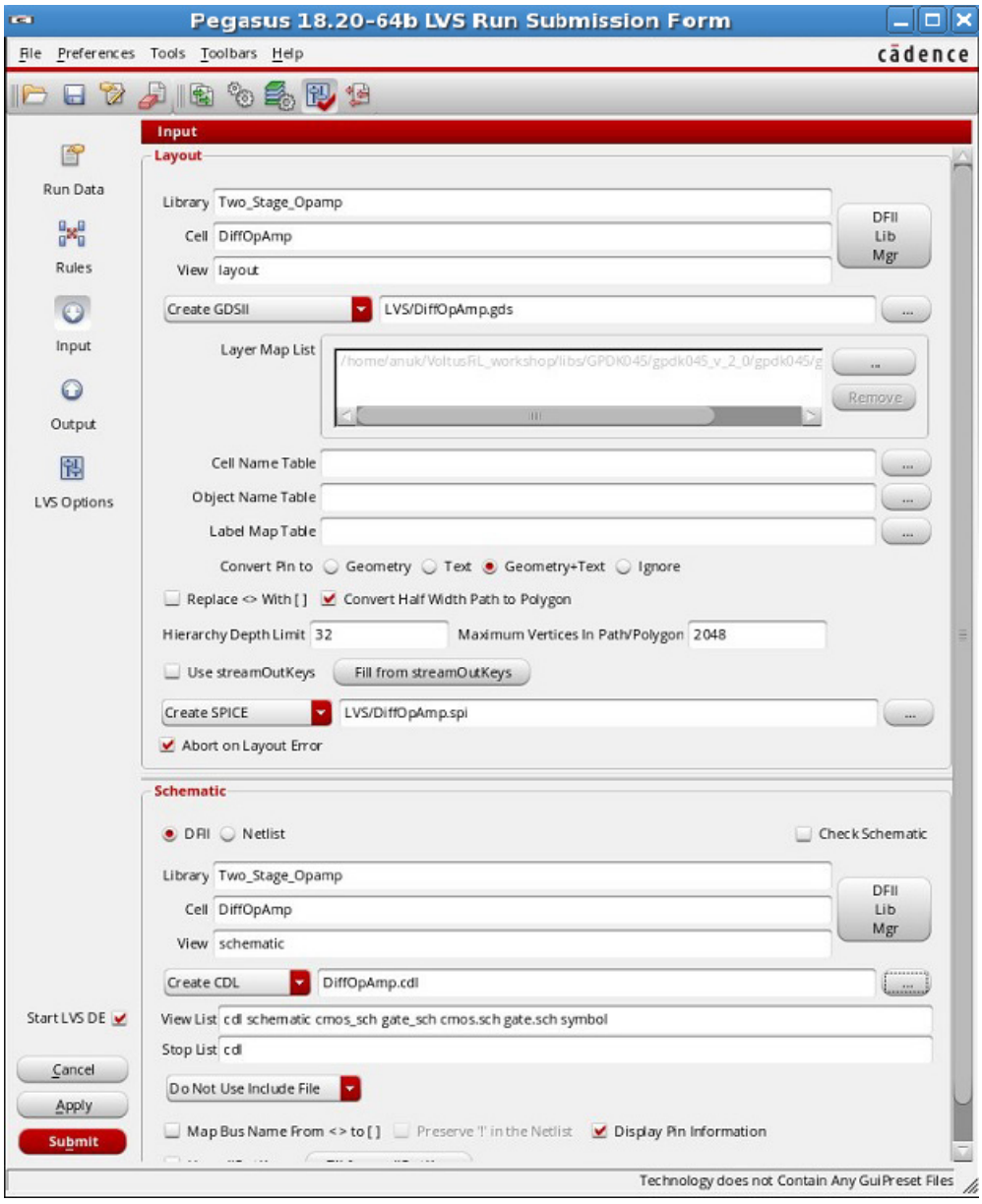
Figure 4-6 Specifying the Rules Deck File



- Select *Input* and provide the input data in the *Layout* and *Schematic* group boxes. The forms for Pegasus are shown below.

© 2023
All Rights Reserved.

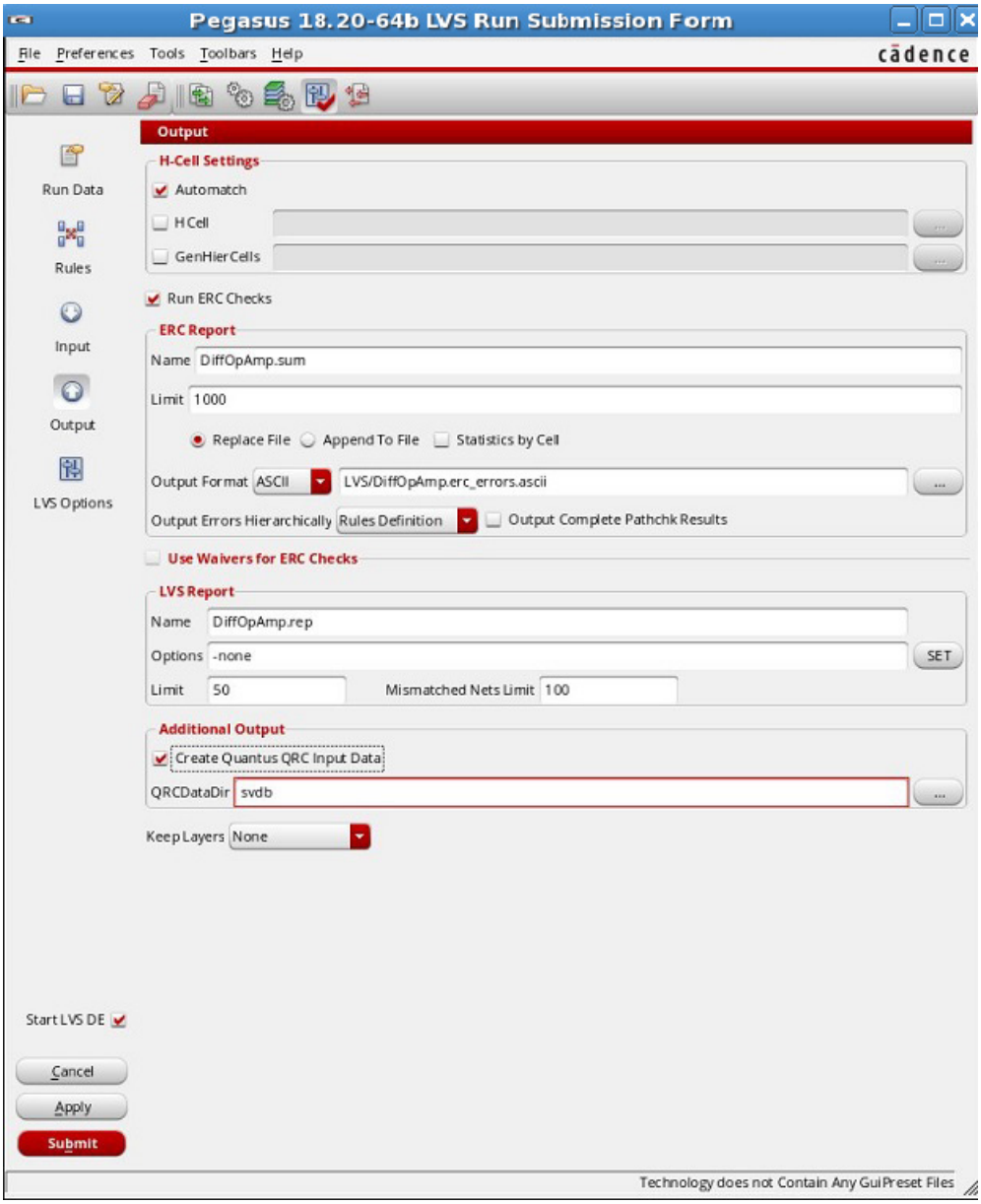
Figure 4-7 Specifying the Input Options



- Select *Output* on the left side of the form and provide the required inputs, as shown below.

© 2023 All Rights Reserved.

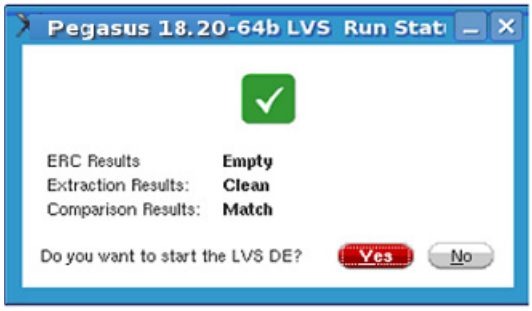
Figure 4-8 Specifying the Output Options in Pegasus



- Click Apply. The Pegasus/PVS LVS run starts. At the end of the LVS run, the following window opens. It displays the Comparison Results as Match.

© 2023 All Rights Reserved.

Figure 4-9 LVS Run Status



- The LVS Run results are stored in the `runLVS` directory.

Note: For detailed information on running Pegasus/PVS LVS, see the *Cadence® Pegasus™ User Guide* or *Cadence PVS User Guide*.

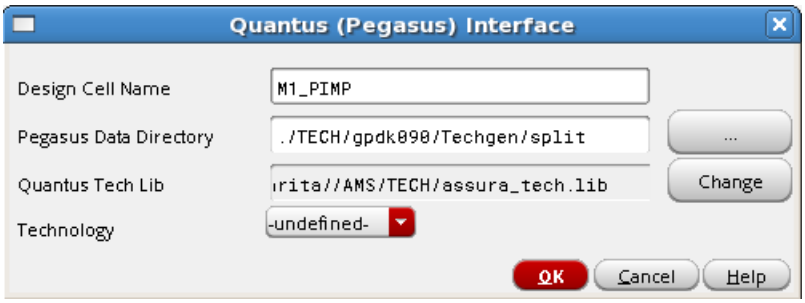
Running Pegasus or PVS Quantus

After the LVS run is complete, the parasitic resistance and capacitance must be extracted from the power and ground nets for power-grid analysis, and from the signal nets. Although, no rule changes are required to run Quantus, some options need to be enabled to generate the extracted view correctly. For more information on running Quantus, see the *Quantus QRC Extraction Users Manual*.

The following steps are detailed for running Pegasus or PVS Quantus:

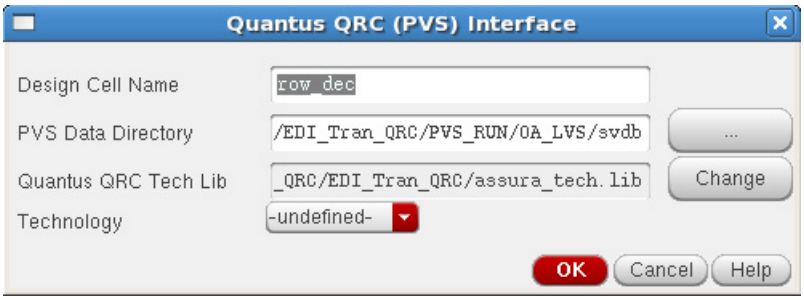
- In the main menu of the *Virtuoso Layout Suite*, choose *QRC – Run Pegasus* or *Run PVS – Quantus*.
- The Quantus (Pegasus) or Quantus QRC (PVS) Interface form opens. This form is shown below.

Figure 4-10 Quantus (Pegasus) Interface Form



© 2023 All Rights Reserved.

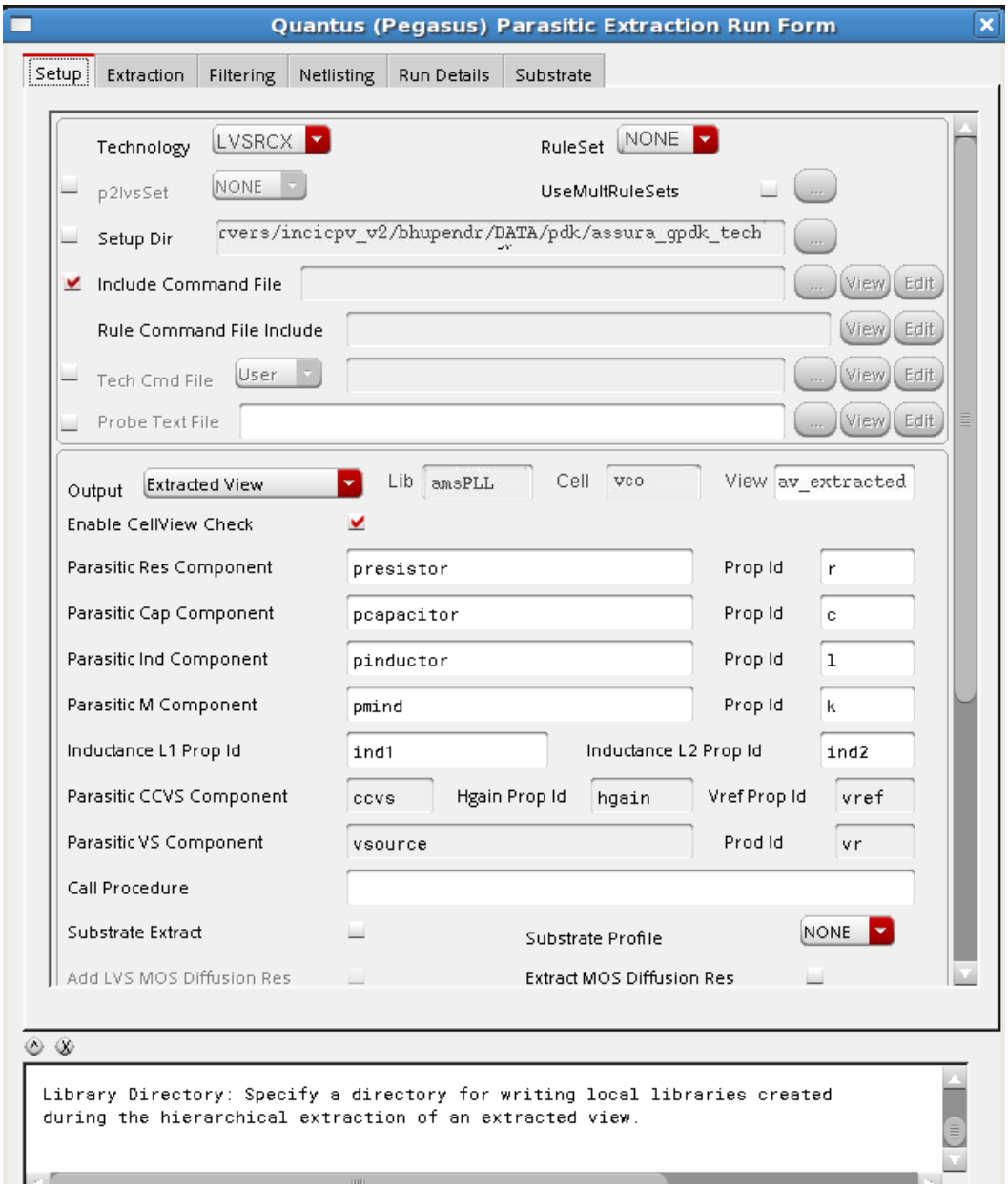
Figure 4-11 Quantus QRC (PVS) Interface Form



- Specify the *Pegasus/PVS Data Directory* and the *Quantus/Quantus QRC Tech Lib.*
- Click *OK*.
- The Quantus (Pegasus)/Quantus QRC (PVS) Parasitic Extraction Run Form opens. In this form, select the *Setup* tab.The form shown below opens.

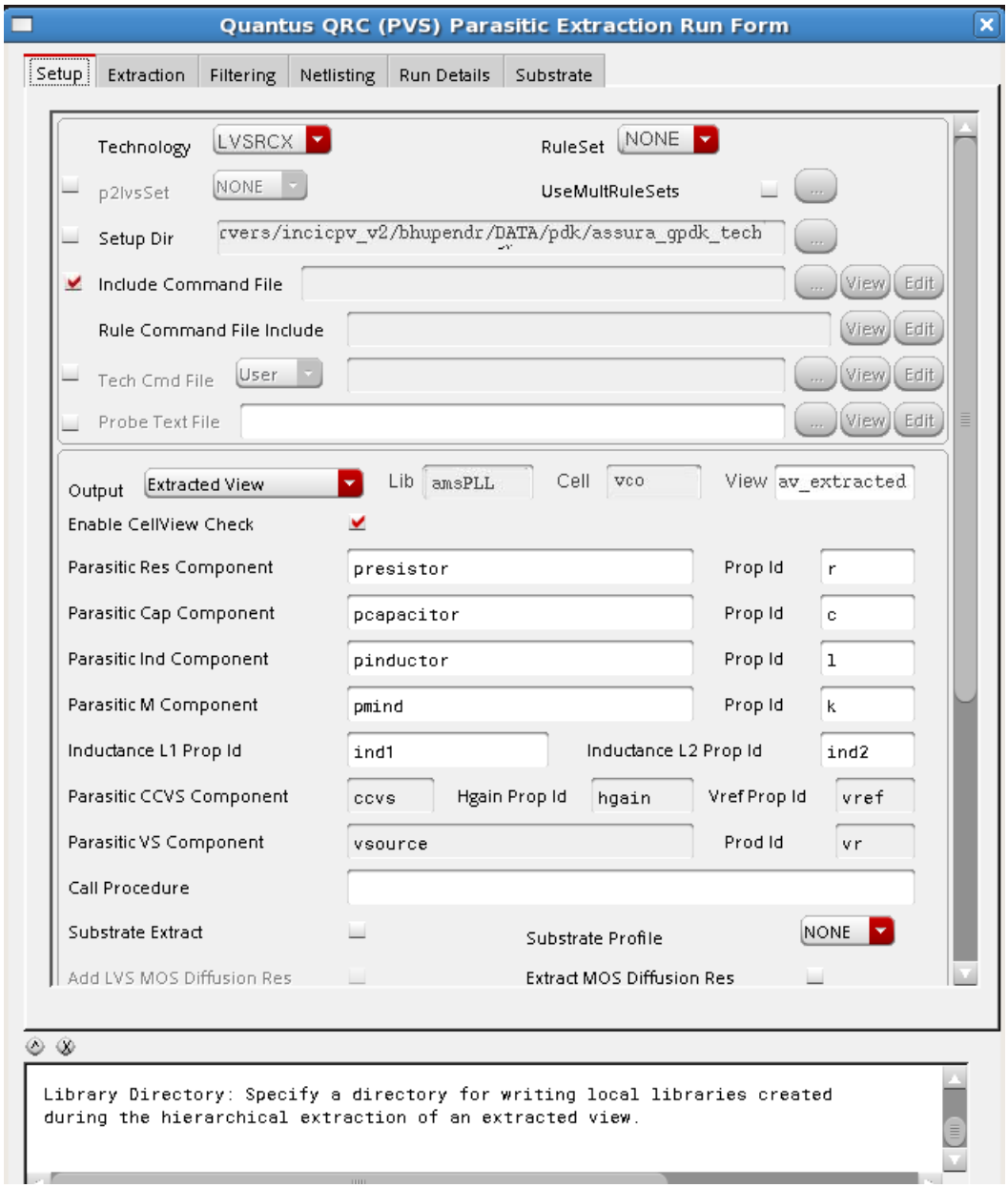
© 2023 All Rights Reserved.

Figure 4-12 Quantus (Pegasus) Parasitic Extraction Run Form – Setup Tab



© 2023 All Rights Reserved.

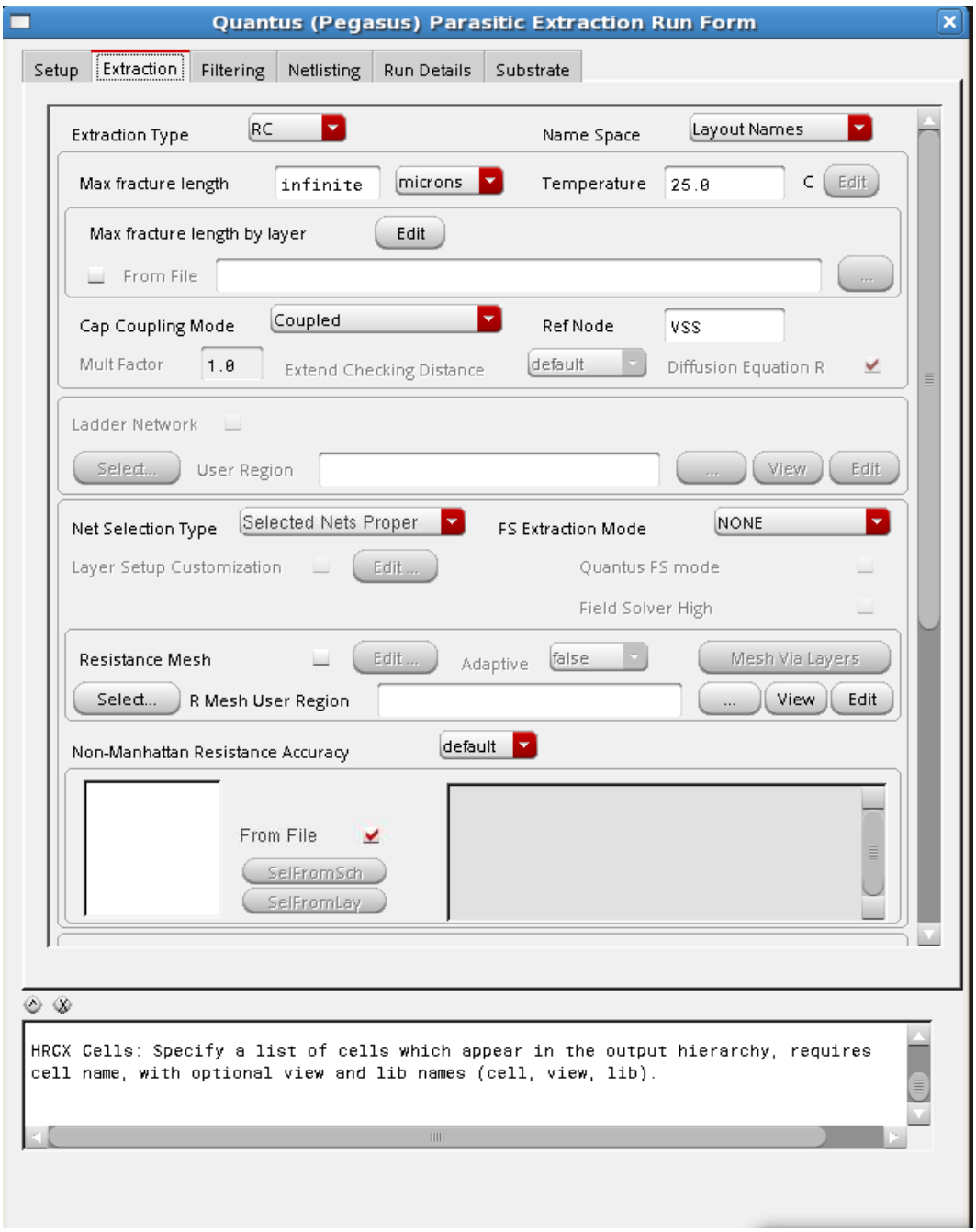
Figure 4-13 Quantus QRC (PVS) Parasitic Extraction Run Form – Setup Tab



- Ensure that the *Output* is set to *Extracted View*.
- Select the *Extraction* tab. The form shown below opens.

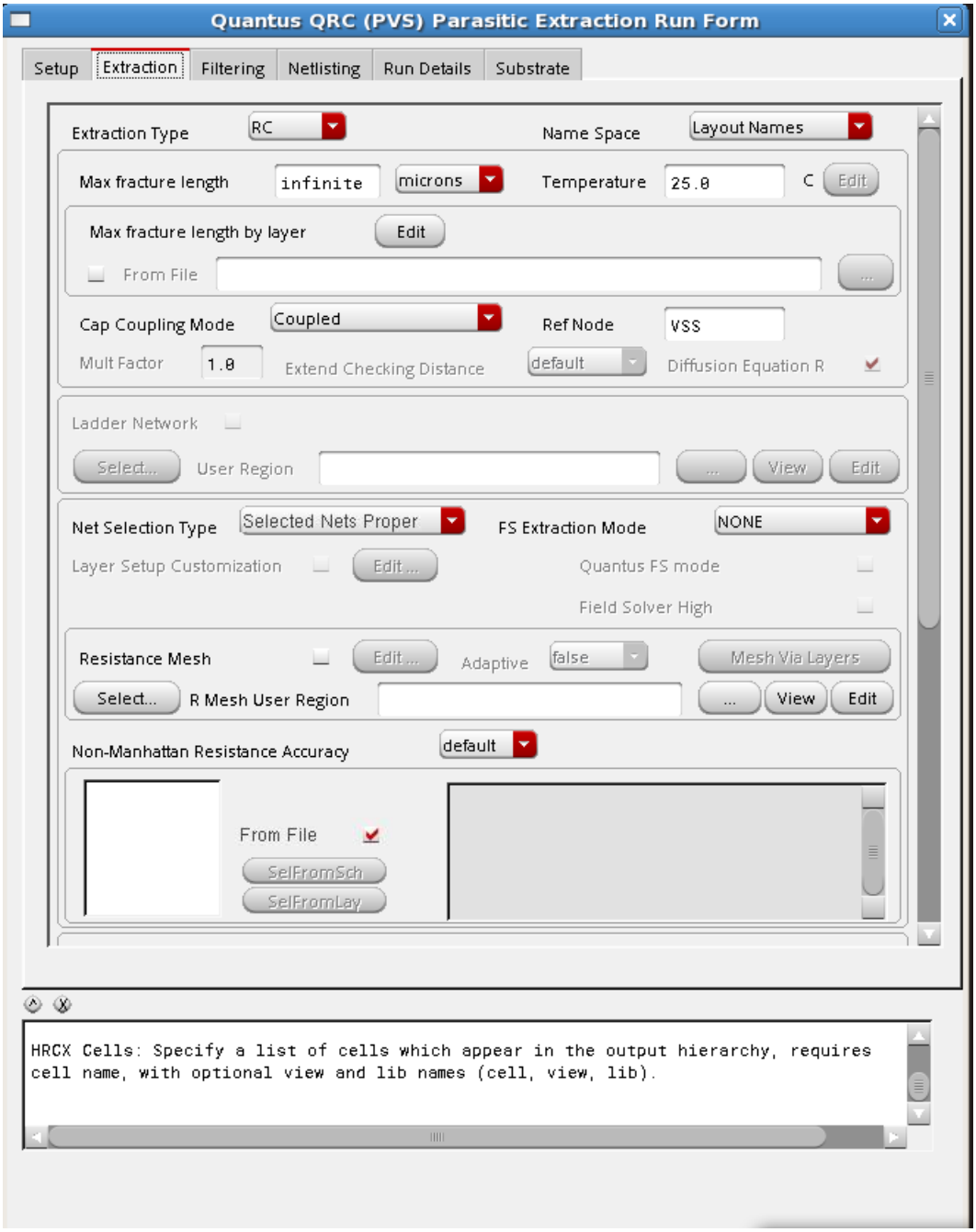
© 2023
All Rights Reserved.

Figure 4-14 Quantus (Pegasus) Parasitic Extraction Run Form – Extraction Tab



© 2023
All Rights Reserved.

Figure 4-15 Quantus QRC (PVS) Parasitic Extraction Run Form – Extraction Tab



- In the *Extraction Type* cyclic field, specify either resistance and capacitance (*RC*) or only resistance (*R only*). In the above image, RC is selected, which means that both resistance and capacitance are being extracted. When *Extraction Type* selected is *R Only*, the simulation time is short. While selecting the extraction type, consider the type of simulation being run.

© 2023 All Rights Reserved.

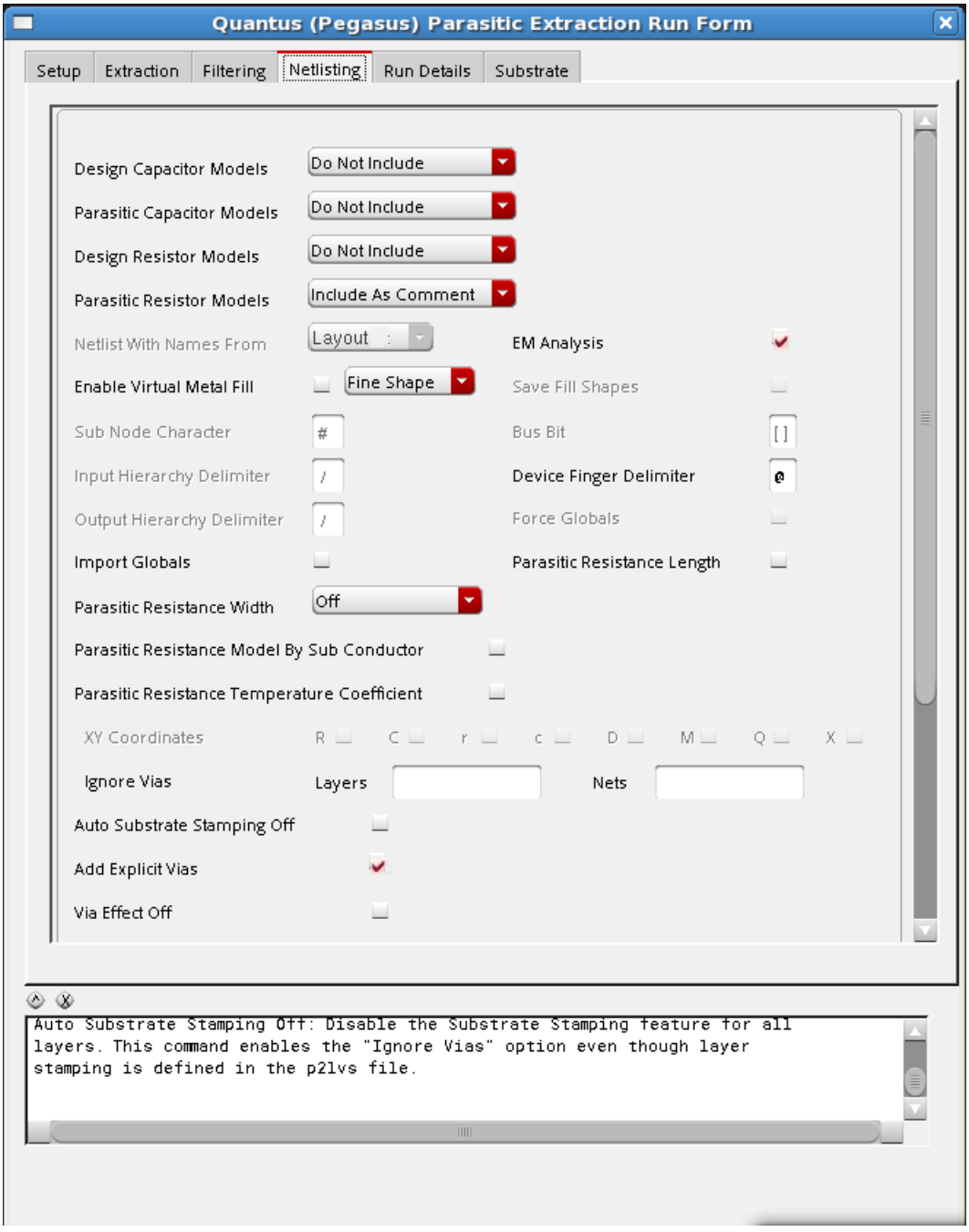
For example, to run a DC operating point (DCOP) simulation, select *R only* as the *Extraction Type* because the capacitance values are not required.

Note: The extraction time in Quantus can be reduced by selecting specific nets of interest. To do this, perform the following steps:

- Select *Net Selection Type – Selected Nets Proper*.
- Select the *Netlisting* tab. The form shown below opens.

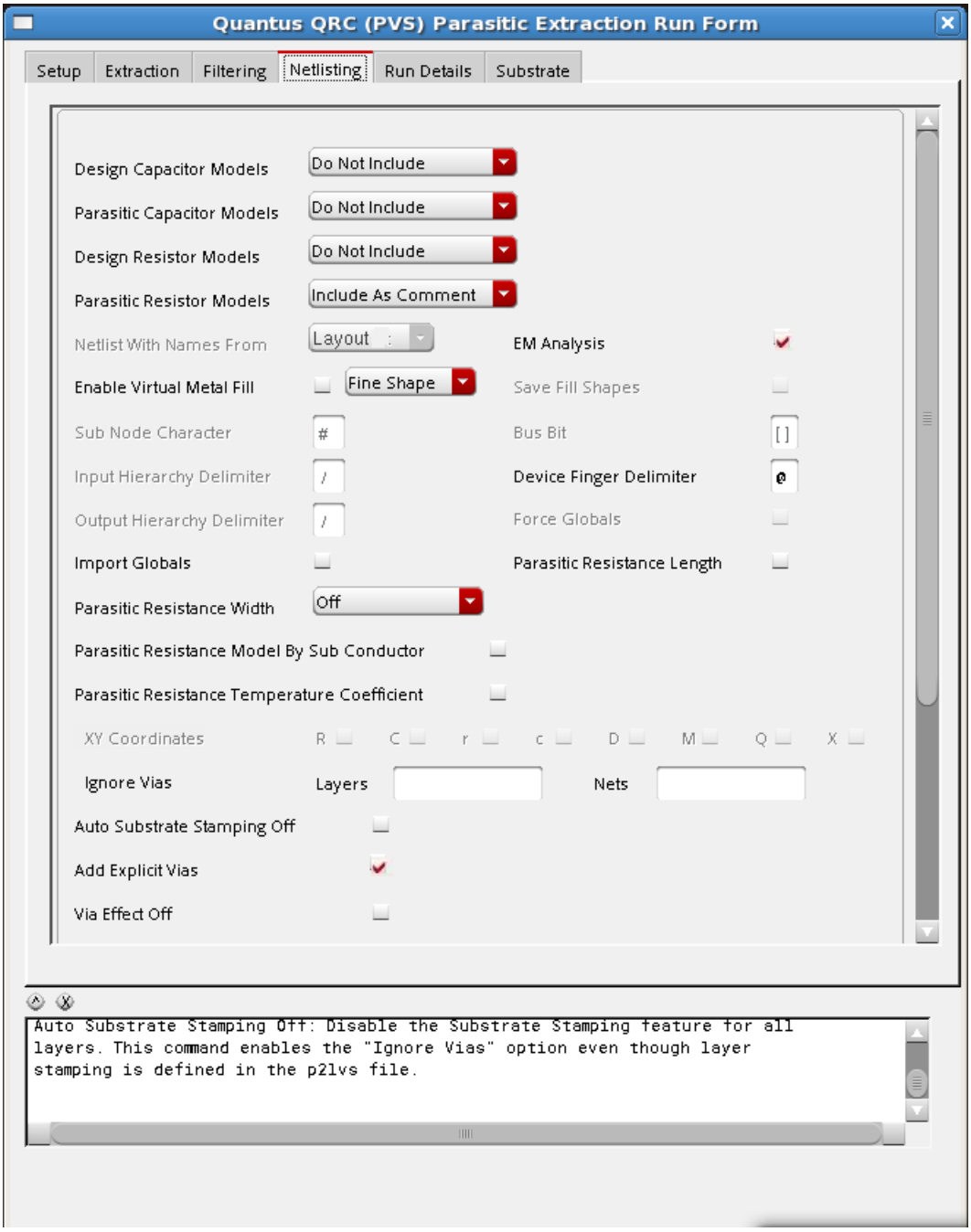
© 2023 All Rights Reserved.

Figure 4-16 Quantus (Pegasus) Parasitic Extraction Run form – Netlisting Tab



© 2023
All Rights Reserved.

Figure 4-17 Quantus QRC (PVS) Parasitic Extraction Run form – Netlisting Tab



- Click *EM Analysis* to activate the extraction options specific to EM analysis during the Quantus QRC run.
- Click *Add Explicit Vias* to prevent the merging of via resistance into the net resistance.

© 2023
All Rights Reserved.

- In the *Parasitic Resistance Width* cyclic field, specify *Drawn*. The parasitic resistance width option annotates the extracted view with the width of parasitic resistors, which is required to compute the Current Density.
- Click *Parasitic Resistance Temperature Coefficent* if you want to store TC1 and TC2 parameters in the extracted view. This is useful when you want to perform IR/EM analysis at different temperatures.
- Deselect *Import Globals* to enable Quantus QRC to extract resistance from power and ground nets.

  Note: The Electromigration analysis requires the parasitic resistors to be present in the `extracted_view`. To ensure this, the *Parasitic Resistors Models* switch must be set to "`Include Model`" or "`Include as Comment`". If "`Include Model`" is selected, you will be required to update the `emDataFile.txt` to match the model names.
- Click *Apply*. The Quantus extraction run will start. At the end of the extraction run, the following window opens to show the status of the run.

Figure 4-18 Quantus QRC Run Status

Important

To proceed to the next step, simulating the extracted view, ensure that the extracted view in Quantus QRC is successfully created.

Running Extraction with Quantus QRC (Calibre) Interface

Quantus QRC accepts a Calibre database as input for extraction in place of Pegasus or PVS LVS.

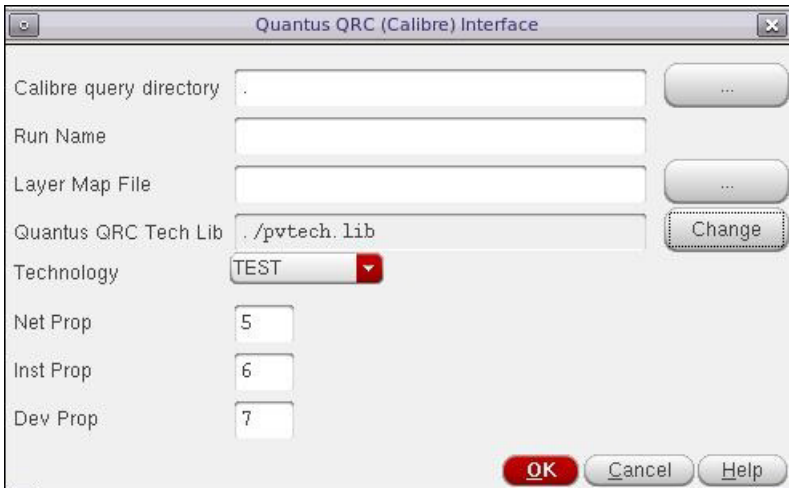
© 2023 All Rights Reserved.

The extracted view includes a standard DFII (design framework II) extracted view, which has parasitic properties backannotated onto a layout design for simulation purposes. The extracted view can be simulated using third-party simulation tools like Spectre or APS (Accelerated Parallel Simulator) to interpret the Quantus QRC output effectively, and use the simulation results as input to Voltus-Fi-L.

In the main menu of the *Virtuoso Layout Suite*, perform the following steps:

- Choose *QRC – Run Calibre-Quantus QRC*.
- The Quantus QRC (Calibre) Interface form opens. This form is shown below.

Figure 4-19 Quantus QRC (Calibre) Interface Form



For more information on how Quantus QRC reads the Calibre input data and performs extraction, see “Running Quantus QRC with Calibre® Input” in the *Quantus QRC Extraction Users Manual.*

Running a Spectre Simulation

After running extraction, the next step is to use an existing testbench and simulate the extracted view within the Virtuoso® Analog design Environment (ADE). The results of the simulation are stored in the simulation directory. This directory is specified when loading the IR drop and EM analysis results in Voltus-Fi-L.

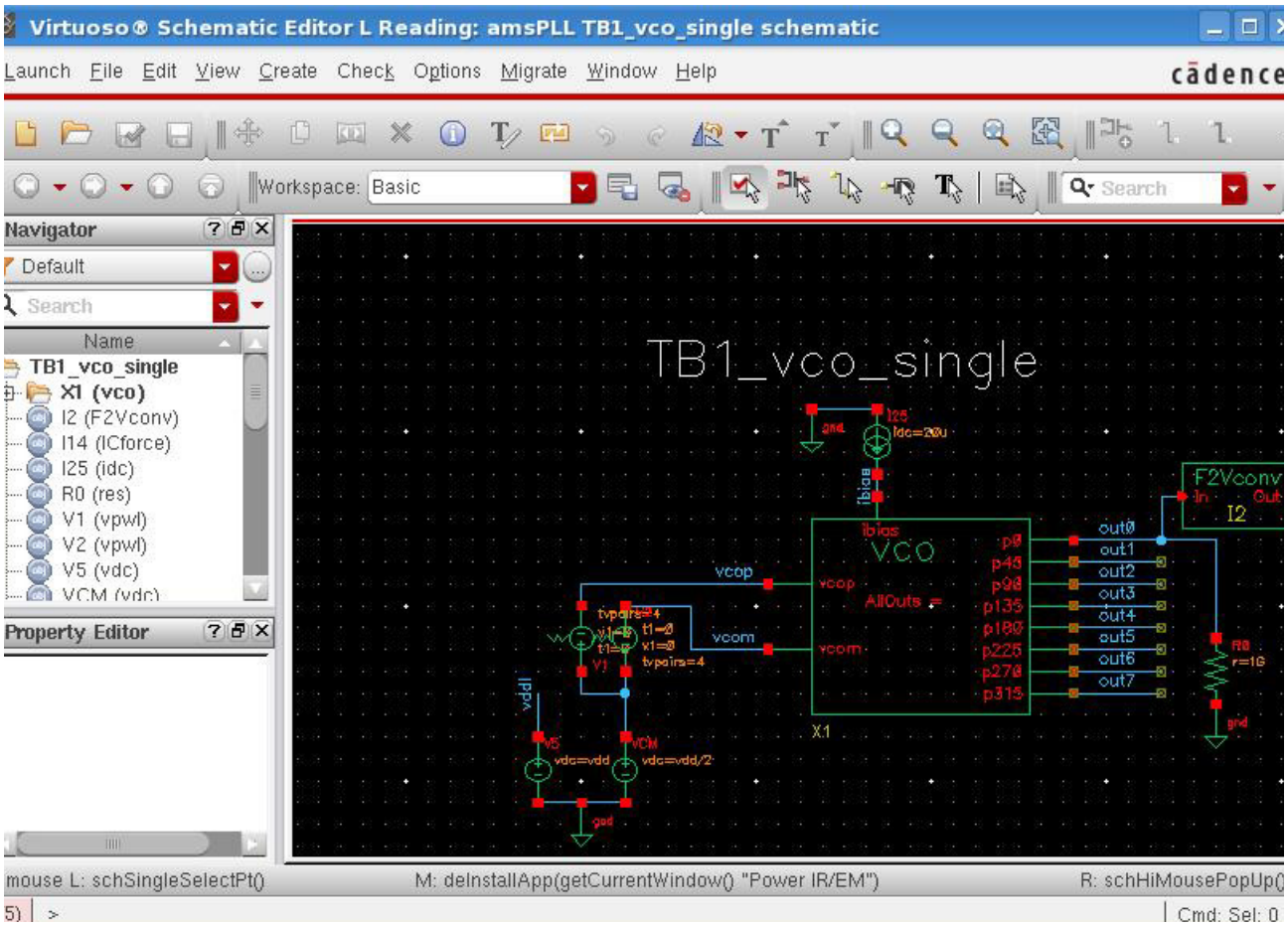
You can use Cadence® Spectre® Circuit Simulator. This simulation environment saves the voltages on all the nodes in the design. You must run a transient or a DC operating point (DCOP) simulation to perform IR Drop analysis or electromigration analysis.

© 2023 All Rights Reserved.

Note: For Voltus-Fi-L, the Spectre simulation is supported only with the 64-bit version of the Spectre software.

An example of a testbench in Virtuoso® Schematic Editor is shown below.

Figure 4-20 Testbench in Virtuoso Schematic Editor

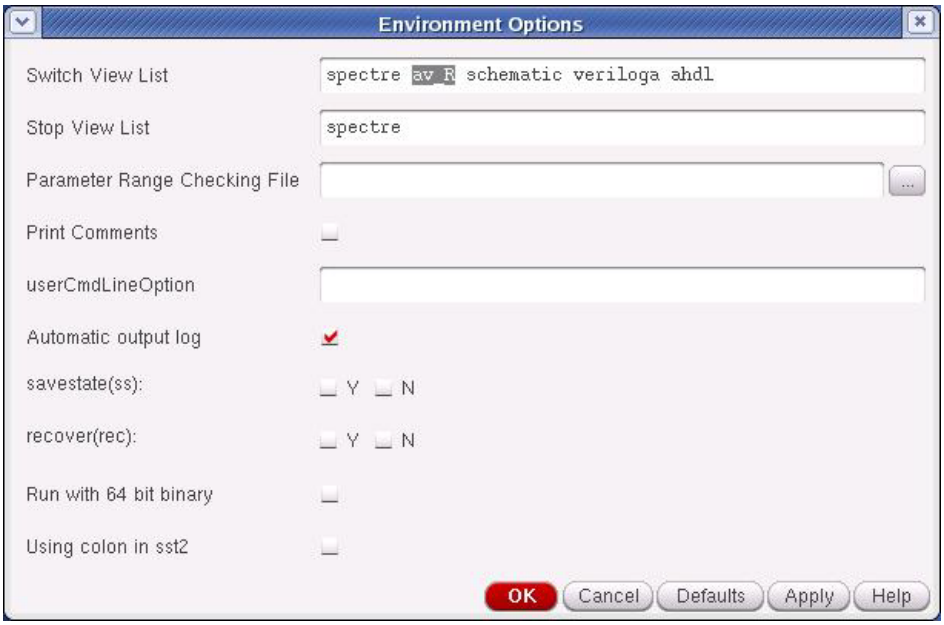


Follow these steps to run Spectre simulation for Voltus-Fi-L:

- In the Environment Options form, specify the extracted view in the simulation in the *Switch View List* field. You can also use config views. This is shown in the figure below.

© 2023 All Rights Reserved.

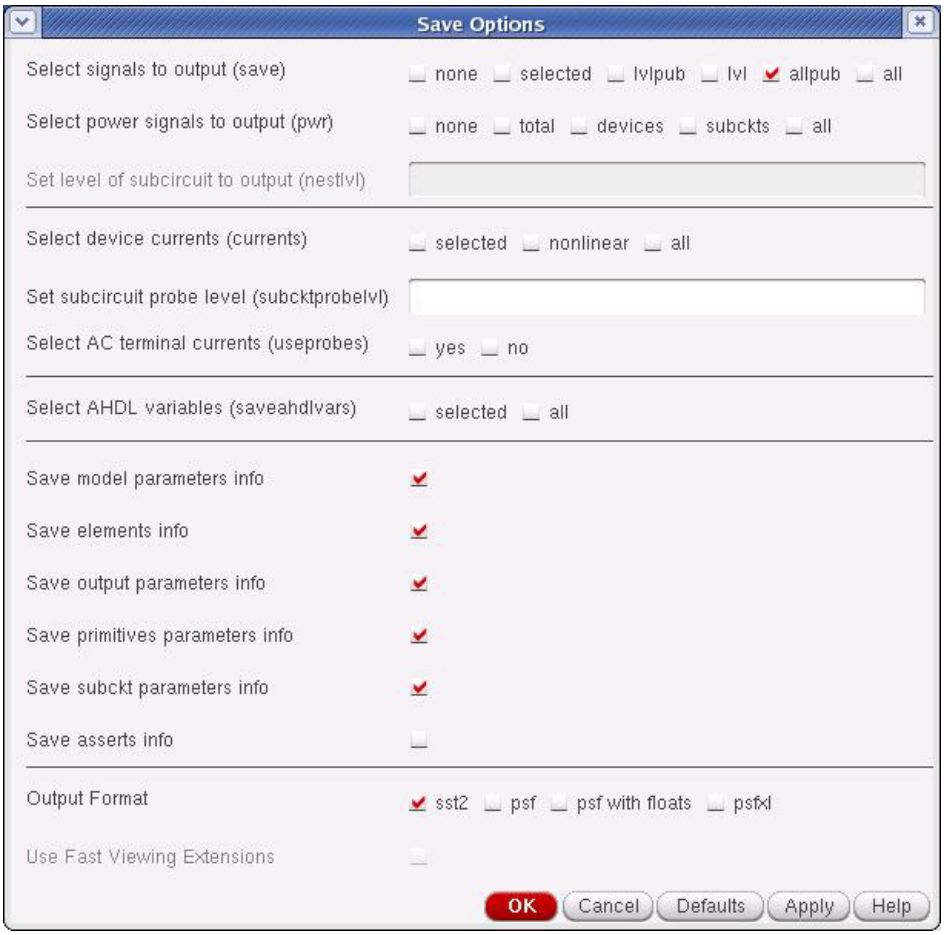
Figure 4-21 Environment Options Form



- In the Save Options form, ensure that the *allpub* option is selected. The Options dialog box is activated by the *Outputs – Save All* command. Only the node voltages are saved during the Spectre simulation. You do not need to create any `measure` statements. This is shown in the figure below.

© 2023 All Rights Reserved.

Figure 4-22 Save Options Form



Before proceeding to the next step, ensure that Spectre simulation is completed successfully. It is useful to save the Spectre simulation state for later use, if required. The Voltus-Fi-L Reader can read results in `SST2` database format as well as `PSF/PSF-XL` waveform format.

Running an APS Simulation

You can perform simulation using the Virtuoso® APS. These simulation results can be used as input to Voltus-Fi-L instead of the simulation results from Virtuoso® Spectre simulator, if required. Virtuoso® APS is a next-generation SPICE simulator that provides high performance, high capacity circuit simulation with full Cadence® Spectre® Circuit Simulator accuracy. APS achieves maximum simulation performance by enabling multi-threading on multi-core and multi-CPU shared memory systems. This simulator lets you simulate large pre- and post-layout designs faster than the Spectre simulator.

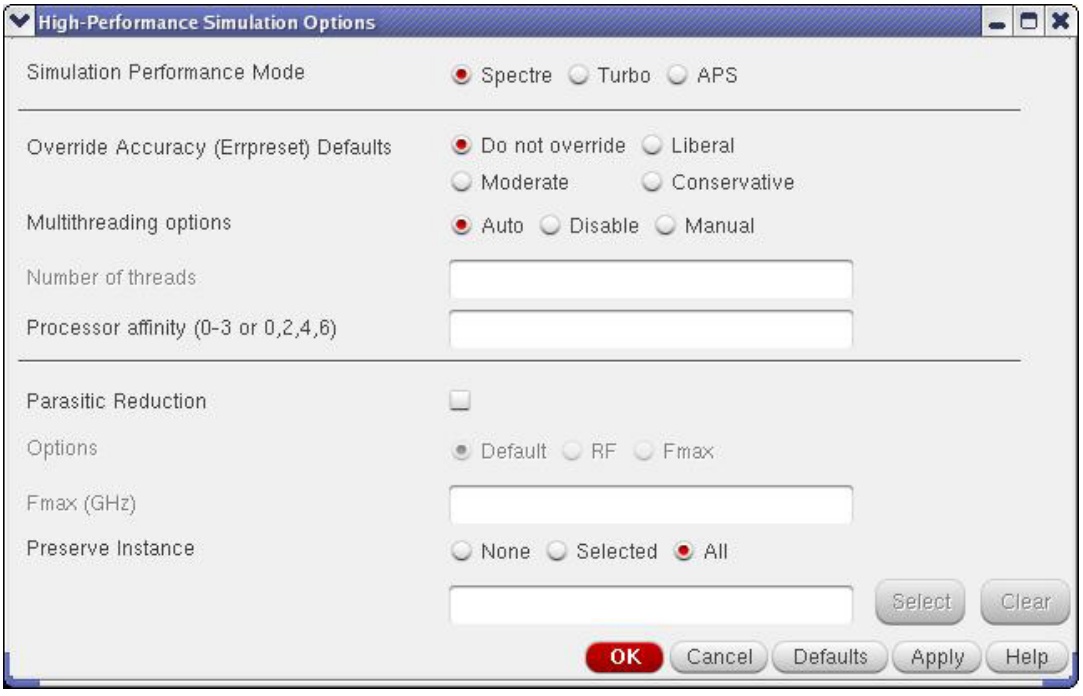
© 2023 All Rights Reserved.

The APS use model is identical to that of Spectre, with the same netlist syntax, device model, analyses, features, and output format support.

Follow these steps to run the APS simulation for Voltus-Fi-L:

- In the *Environment Options* form, specify the extracted view in the simulation in the *Switch View List* field. You can also specify config views. This is shown above in the "Running a Spectre Simulation" section.
- In the *Save Options* form, ensure that the *allpub* option is selected. The Save Options dialog box is activated by the *Outputs – Save All* command. This is shown above in the "Running a Spectre Simulation" section. Only the node voltages are saved during the APS simulation. No `measure` statements need to be created.
- In the High-Performance Simulation Options form, ensure that *All* is selected as *Preserve Instance*. This is shown in the figure below.

Figure 4-23 High-Performance Simulation Options Form



Before proceeding to the next step, ensure that the APS simulation is completed successfully. It is useful to save the APS simulation state for later use, if required. Voltus-Fi-L reader can read results in the `SST2` database and the `PSF/PSF-XL` waveform formats.

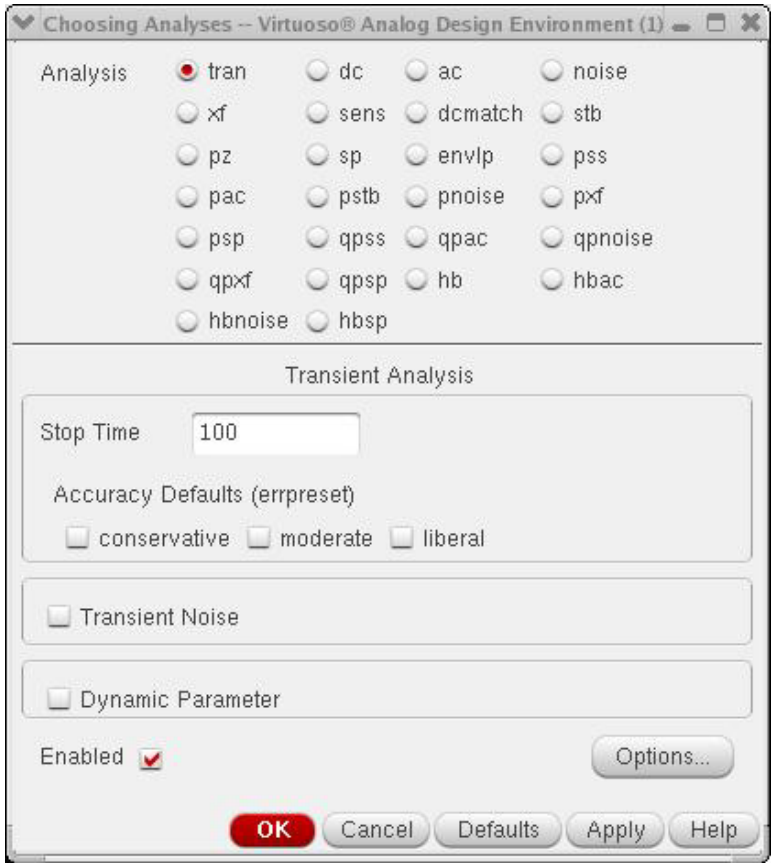
For very large designs, APS can write the simulation data in a Voltus-Fi-L-specific format. Average, peak, and RMS analyses can be performed faster by using this data.

© 2023 All Rights Reserved.

For example, for generating the EMIR output during transient analysis, follow these steps:

- In the Virtuoso® Analog Design Environment, select *Analysis – Choose*. The Choosing Analyses form opens.
- Select *tran* as the *Analysis* type. This form is shown below.

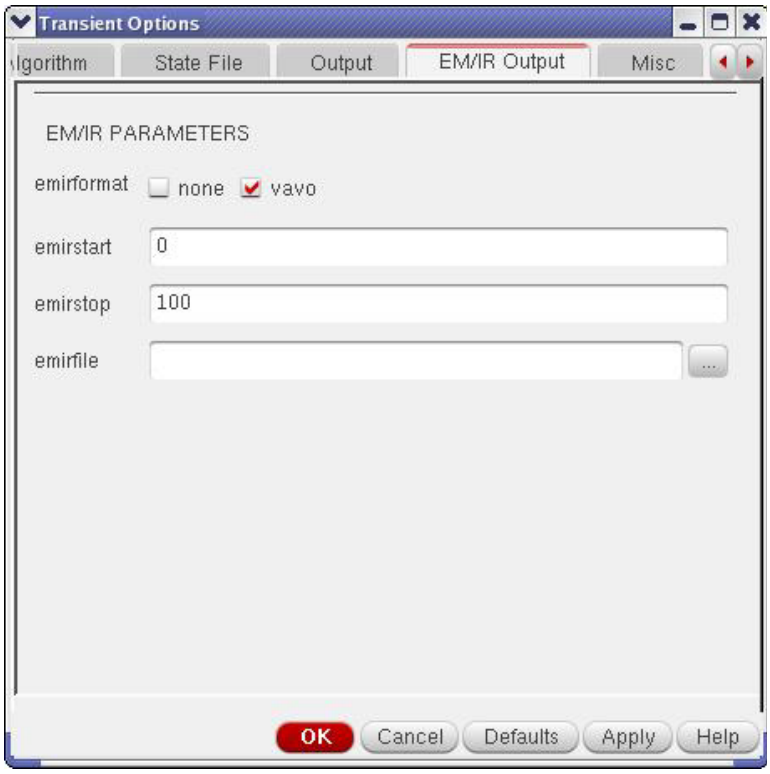
Figure 4-24 Choose Analyses Form



- In this form:
  - Select the *Enabled* check box.
  - Click *Options*. The Transient Options form opens. This form is shown below.

© 2023 All Rights Reserved.

Figure 4-25 Transient Options Form



- In the Transient Options form, select the *EM/IR Output* tab. On this tab:
  - Click *vavo* as the *emirformat*.
  - Specify the *emirstart* time and *emirstop* time.
  - Specify the *emirfile.*
  - Click *OK*.

The syntax for generating EMIR output during transient analysis is as follows:

```
Name tran [emirformat=vavo|none] [emirstart=time] [emirstop=time]
[emirfile=dbfileName]
```

Where:

Name	The name of the transient analysis.
`emirformat`	Turns on the `vavo.db` file output capability. Possible values are `vavo` and `none`. Default is `none`.

© 2023 All Rights Reserved.

`emirstart/emirstop`	Specifies the time window start and stop times. The default of start and stop times are same as the start and stop times for transient analysis, respectively.
`emirfile`	Specifies the name of the EM/IR output database file. Default is `tranName_emir_vavo.db`. This file is saved in the raw directory.

Example:

```
tran1 tran stop=15s errpreset=moderate emirformat=vavo emirfile="testDB"
emirstart=1s emirstop=15s
```

Calculates the required information for EM and IR analysis and saves it in the `testDB.db` file.

Note: To generate the `emirformat` database file version `5.0`, set the environment variable, `EMIR_VAVO_DB 5.0`. This database file version significantly reduces the simulation runtime and it is supported with the following APS simulator versions:

- MMSIM121_ISR16 and above
- MMSIM131_ISR2 and above

© 2023 All Rights Reserved.

© 2023 All Rights Reserved.

5

Environment Variables

© 2023 All Rights Reserved.

Setting Up the Voltus-Fi-L Environment Variables

The environment variables detailed below can be set for Voltus-Fi-L before invoking the software.

VFI_EXTVIEW_UNIT_FACTOR

```
setenv VFI_EXTVIEW_UNIT_FACTOR value
```

Specifies the factor to convert the length and width in the `extview.tmp` file into `um` for Voltus-Fi-L. If the `extview.tmp` contains length and width in meters, then the factor should be specified as `1e06`.

```
setenv VFI_EXTVIEW_UNIT_FACTOR 1e06
```

If the `extview.tmp` contains length and width in `um`, then the factor should be specified as `1`.

```
setenv VFI_EXTVIEW_UNIT_FACTOR 1
```

Note: This environment variable is optional. There is no default value. It is specified in the following scenarios:

- With Quantus version 15.1 or later, the software assumes that `extview.tmp` contains length and width in meters. If this is not the case, which means that `extview.tmp` contains length and width in `um`, then specify the factor as `1` using the environment variable `VFI_EXTVIEW_UNIT_FACTOR` so that the software assumes that the length and width in the `extview.tmp` file is in `um`.
- With Quantus release version earlier than 15.1, the software assumes that `extview.tmp` contains length and width in `um`. If this is not the case, which means that `extview.tmp` contains length and width in meters with Quantus version earlier than 15.1, then specify the factor as `1e06` using the environment variable `VFI_EXTVIEW_UNIT_FACTOR` to convert the length and width in the `extview.tmp` file into `um`.

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-L license. By default, the software waits for 24 hours or 86400 seconds.

Example:

© 2023 All Rights Reserved.

```
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 5 degree Celsius. The variable, `deltaT`, can also be specified using either the `set_variable` command or in the Variables form that opens from the EM tab of the IR/EM Results form. For details, see the “Variables” chapter.

Example:

```
setenv VOLTUSFI_DELTA_T 15
```

VOLTUSFI_DYNAMIC_AC_PEAK

```
setenv VOLTUSFI_DYNAMIC_AC_PEAK true | false | multiplePeak
```

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 the “Variables” chapter.

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 the “Variables” chapter.

You can also see Rules for Specifying EM Analysis Type for Power and Ground Nets in the “EM Rules Specification” chapter.

Example:

```
setenv VOLTUSFI_SPLIT_ACDC_RULES true
```


© 2023 All Rights Reserved.

VFI_POINT_QUERY

```
setenv VFI_POINT_QUERY value
```

When set, runs the IR/EM analysis results query is performed within the defined square area around the specified point query. The value is specified in nanometer (nm). When this variable is not set, the software queries the nearest matching data point.

Set this variable before starting the GUI.

Example:

```
setenv VFI_POINT_QUERY 5
```

For a point selected on x- and y-coordinates, the software will query the following square area:

```
x-5,x+5,y-5,and y+5
```

ENABLE_VFI_L_RON_DATA_READING_FROM_PSF

```
setenv ENABLE_VFI_L_RON_DATA_READING_FROM_PSF t
```

When specified, enables the computation of on-resistance (RON) values.

Set this variable before running VFI-L in the batch mode.

Example

```
setenv ENABLE_VFI_L_RON_DATA_READING_FROM_PSF t
```

VSAPLOT_USE_DEFAULT_LAYER_ORDER

```
setenv VSAPLOT_USE_DEFAULT_LAYER_ORDER true | false
```

This is relevant only for viewing results in vsaplot or vfiplot. When specified, the software uses the original layer order created by Voltus-Fi-L and Voltus-Fi-XL in the VSA_RESULTS directory.

Example

```
setenv VSAPLOT_USE_DEFAULT_LAYER_ORDER true
```


© 2023 All Rights Reserved.

VSAPLOT_EXTERNAL_LAYER_ORDER_FILE_PATH

```
setenv VSAPLOT_EXTERNAL_LAYER_ORDER_FILE_PATH string
```

This is relevant only for viewing results in vsaplot or vfiplot. When specified, the software uses the control_layer file at the specified location.

Example

```
setenv VSAPLOT_USE_DEFAULT_LAYER_ORDER \amsPLL_oa_flow\VSA_Results
```

_vsaLSFMachineString

```
setenv _vsaLSFMachineString "bsub -q lnx64"
```

Specifies the name of the LSF command string that will launch Voltus-Fi-L in batch mode. This is only applicable for the power-grid view (PGV) flow.

_vsaUncheckViewFile

```
setenv _vsaUncheckViewFile "true"
```

Specifies that files generated during PGV creation should not open for viewing by default. This is only applicable for the PGV flow. In the GUI, the *View file* option of the Save PGV Commands To A File form to view the command files is checked by default. You can specify this variable if you do not want to view these files.

vfiEnableGUI

```
setenv voltus_fi.results vfiEnableGUI boolean { t | nil }
```

Specifies whether to hide or display the warning message about the deprecation of Voltus™-Fi Custom Power Integrity Solution in the next IC base release. The default value is `nil`.

Example

```
setenv voltus_fi.results vfiEnableGUI boolean t
```


© 2023 All Rights Reserved.

© 2023 All Rights Reserved.

6

IR Drop Analysis Results

© 2023 All Rights Reserved.

© 2023 All Rights Reserved.

Overview

Voltus-Fi-L uses the results of the simulation performed using the `vavo-db` or the parameter storage format (`PSF`) data to perform IR drop and EM analysis. The software provides both batch mode and GUI support for performing these analyses.

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`).

For details, see the Signal Net IR Drop Analysis section in the “IR Drop Analysis Results” chapter of the *Voltus-Fi Custom Power Integrity Solution XL User Guide.*

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 presistors, while in the stick diagram display, the presistors are connected through 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 Quantus QRC.

For details, see the Solid Shape Highlighting section in the “IR Drop Analysis Results” chapter of the *Voltus-Fi Custom Power Integrity Solution XL User Guide.*

© 2023 All Rights Reserved.

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-L 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. Voltus-Fi supports a finer display of the color gradient for violations in different segments of such layers. The shape polygons are split into subpolygons and then the subpolygons are colored with the worst violation value for all resistors connected to the nodes in the split shape. This feature is turned on by default for all layers. However, you can select specific layers to view their finer gradient instead of viewing for all layers.

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, click *Deselect All* and then select the PGDB layers for which you want to view finer gradient. For details of the steps involved, see Displaying Finer Color Gradient for Selected Layers in IR and EM Plots in the “IR Drop Analysis Results” chapter of the *Voltus-Fi Custom Power Integrity Solution XL User Guide.*

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'
  "amsPLL_oa_flow/df2layermap")
  ```

  Where,

  `voltus_fi.results` is the name of the tool

  `vfiDfiiLayerMapFile` is the name of the variable

  `amsPLL_oa_flow` is the path to the layer map file

© 2023 All Rights Reserved.

`df2layermap` is the name of the layer map file

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

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

Setting a Layer Index for Displaying Violations on the Layout

Voltus-Fi-L 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-L, 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
  ```


© 2023 All Rights Reserved.

The IR Drop Analysis Flow

In this section, the use model for the IR drop analysis flow is detailed for batch mode and GUI mode.

- The IR Drop Analysis Batch Mode Flow
- The IR Drop Analysis GUI Flow

The IR Drop Analysis Batch Mode Flow

The IR drop analysis batch mode flow can be run using either the EMIR configuration file used in Voltus-Fi-XL or the command file used by legacy (VPS-L). The complete details of the batch mode flow are covered in the “Batch Mode Execution” chapter.

For details of the Voltus-Fi-L batch mode flow, see the following sections in the “Batch Mode Execution” chapter:

- The Command File Support
  - Supported VPSL Batch File Commands
- The Configuration File Support
  - EMIR Control File Options Supported in Voltus-Fi-L

For details about migrating from legacy (VPS-L) to Voltus-Fi-L in batch mode, see Migrating from Legacy (VPS-L) to Voltus-Fi-L in Batch Mode in the “Batch Mode Execution” chapter.

The batch mode IR drop analysis flow involves the following key tasks, which are detailed in the sections below:

- Loading the IR Drop Analysis Results
- Printing the IR Drop Analysis Reports

Loading the IR Drop Analysis Results

When the legacy (VPS-L) command file is used for running Voltus-Fi-L in the batch mode, you can load the IR drop analysis results by writing the following command in the `vfibatch` command file:

load_ir_results_extview `-i cmd_file`

Where,

© 2023 All Rights Reserved.

`cmd_file` is the same batch file that is used in the legacy (VPS-L) batch mode execution.

Printing the IR Drop Analysis Reports

When the EMIR configuration file is used for running Voltus-Fi-L in the batch mode, the IR drop analysis report, `#.rpt_ir`, is generated automatically by the tool.

When the legacy (VPS-L) batch command file is used for running Voltus-Fi-L in the batch mode, the command, `print_ir_report`, is used to generate the IR drop analysis reports. This is the same command that is used to print IR drop analysis reports in Voltus-Fi-XL.

The syntax of the command is as follows:

```
print_ir_report
     -net {all_power | netname}
     -type {ir | iravg | rc | rcavg | rcrms}
     [-threshold threshold_value]
     -filename output_file_name]
```

Note: In the `-type` parameter, only the analysis types listed in the syntax above are supported in the Voltus-Fi-L flow.

For details of the above command and its parameters, see print_ir_report in the “Batch Mode Execution” chapter.

Sample Command Files for IR Drop Analysis

Sample command files used for running the IR drop analysis in batch mode in Voltus-Fi-L are provided below.

- A sample batch command file, same as the legacy (VPS-L) command file, used for IR drop analysis is provided below.

```
;vsa batch mode command file
;common commands for EM and IR
_vsa_extracted_lib_cell_view “amsPLL” “vco” “av_extracted_sol”
_vsa_testbench_lib_cell_view “amsPLL” “TB1_vco_single” “schematic”
_vsa_simulation_directory “/custom/TB1_vco_single/spectre/schematic/”
_vsa_pres_cellname “presistor”
_vsa_analysis_data “TRANSIENT”

;IR specific commands
;_vsa_analyze_IR “0” “170n”
_vsa_analyze_ir
```


© 2023 All Rights Reserved.

```
exit
```

- A sample batch command file, same as the EMIR configuration file, used for IR drop analysis is provided below.

```
DC Operating Point (DCOP) Analysis
==================================
net name=[*] analysis=[vavg vmax]
emirutil view=[lib=amsPLL cell=vco view=av_extracted_vco_oa]
emirutil view_tb=[lib=amsPLL cell=TB1_vco_single view=schematic]
emirutil analysisType="dc"
emirutil analysisName="dcOp-dc"
emirutil presCellName="presistor"
emirutil qrc_output=[runDir=vco_lvs_oa runName=vco]
emirutil hierarchy="/X1”
```

The IR Drop Analysis GUI Flow

To view the IR drop analysis results in Voltus-Fi-L, ensure the following are provided:

- The `emir` mode is enabled while performing simulation
- The name of the simulation result directory
- The DFII layermap file
- The Quantus QRC run directory
- The Quantus QRC run name
- The hierarchy name
- The presistor cell name

The following topics are covered in this section:

- Viewing the IR Drop Analysis Results
- Displaying and Querying EMIR Results

Viewing the IR Drop Analysis Results

Follow these steps to view the IR drop analysis results:

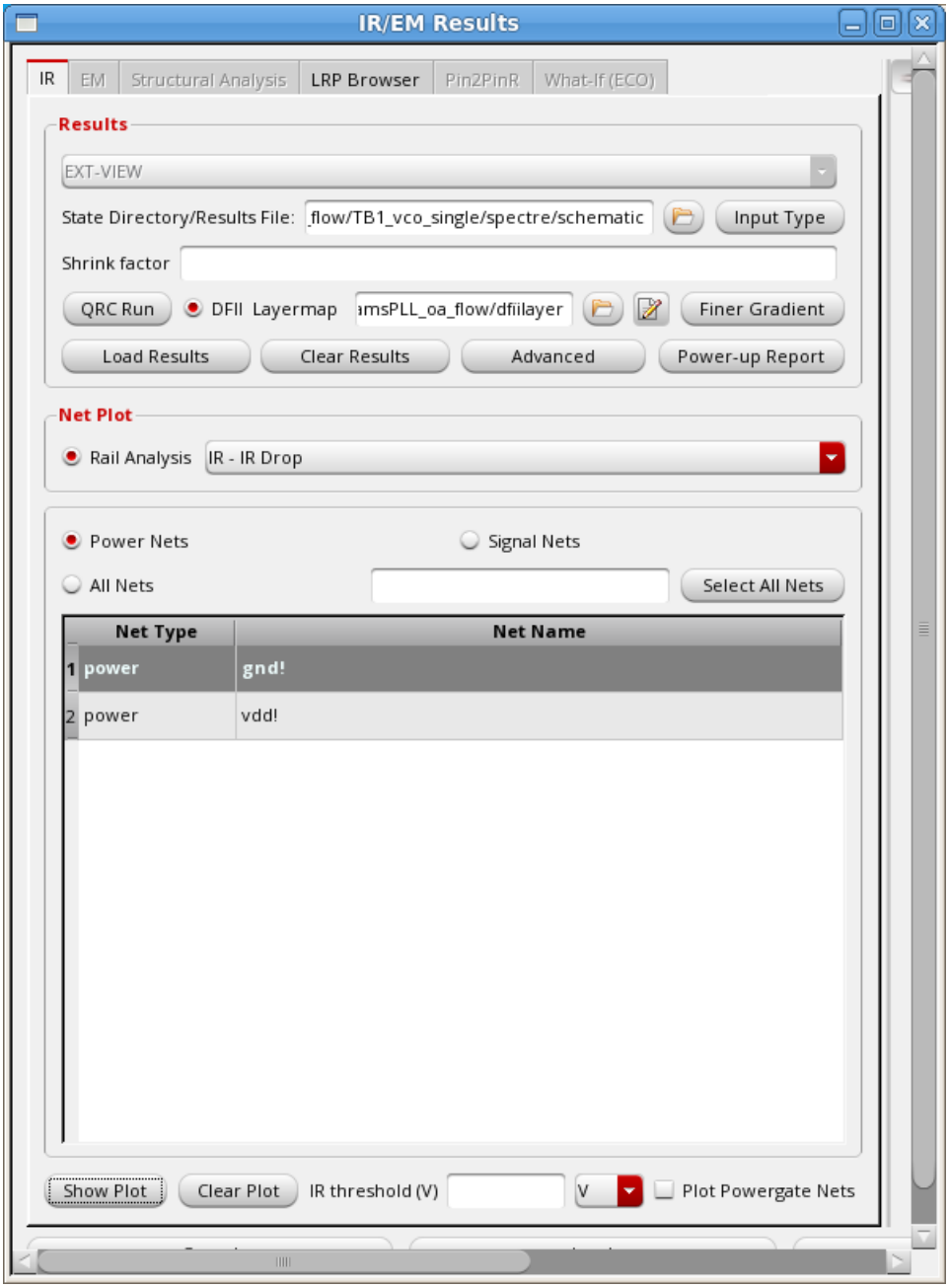
- In the Voltus-Fi-L console, choose *IR/EM Analysis* menu – *Rail Analysis Results*. The IR/EM Results form opens.

© 2023 All Rights Reserved.

Note: In this form, only the IR, EM, and LRP Browser tabs are available. All other tabs are currently disabled.

- Click the IR tab to plot the IR drop analysis results. This form is shown below.

Figure 6-1 IR/EM Results Form – The IR Tab



© 2023
All Rights Reserved.

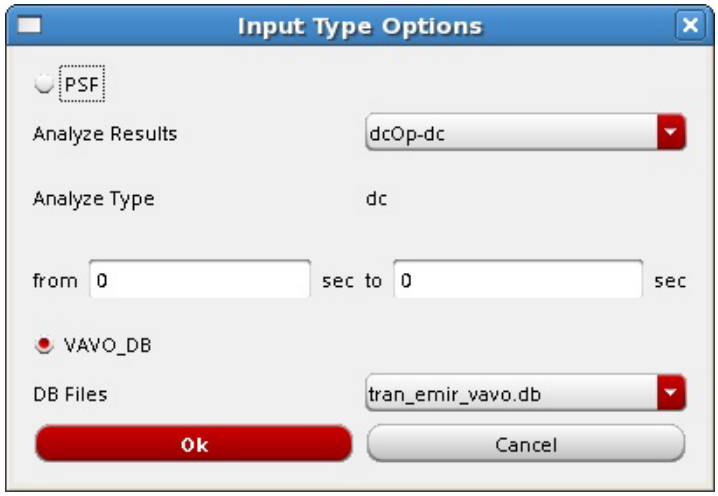
In the *Results* group box:

- The EXT_VIEW is already selected to enable the extracted view support.
- Specify the *State Directory/Results File*, which is the path to the simulation directory that stores the results of the simulation.
- Specify the *Shrink factor*, if any, by which the extracted view was shrunk. This is used for flows where the extracted view is generated with a shrunk technology but the layout remains on the original technology. For example, if the original extracted view was shrunk by 80%, the shrink factor should be specified as 0.8. The shrink factor value is less than 1.
- Specify the *Input Type Options* to specify the type of data, PSF or VAVO_DB, to be used for the IR drop analysis. When you click this button, the Input Type Options pop-up window opens. This is shown below.

  Note: The input type must be specified before running the IR drop or EM analysis.

  Note: You can also specify the input type options using the SKILL function, vsaSetInputType.

Figure 6-2 Input Type Options



In the above form, provide the following inputs:

- Select either *PSF* or *VAVO_DB* depending upon the input data being used for the analysis. If your simulation data is in the `vavo.db` format, the *VAVO_DB* option is selected by default. You can also specify this option in the batch mode by using the _vsa_vavo_db_file command.

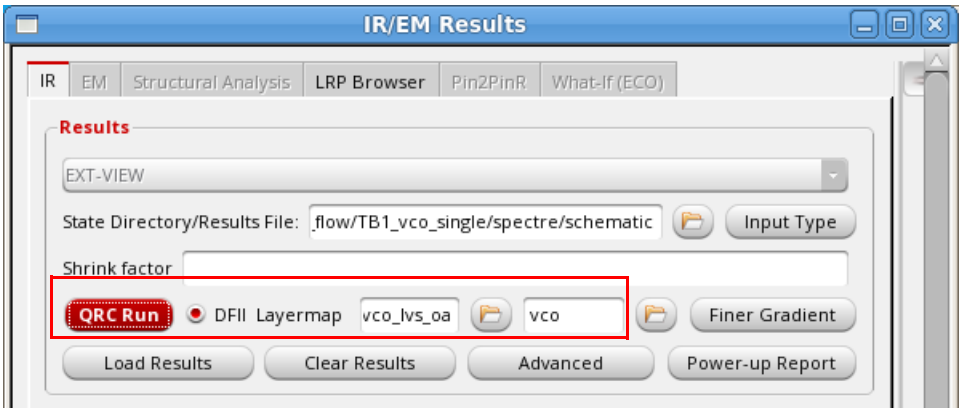
© 2023 All Rights Reserved.

- For the PSF data, specify the analysis type in the *Analyze Results* cyclic field. You can specify either DC operating point (DCOP) analysis or transient (`tran`) analysis.
  - The DCOP analysis is recommended for large analog designs that are difficult to simulate in Spectre using transient analysis, especially when all parasitics are included.
  - The transient analysis option sets the simulation time in seconds automatically when the from/to fields are selected. For transient analysis, separate start and stop times can be specified using the *from* and *to* fields.

  Note: You can also specify the start and stop time in the batch mode by using the vsa_analyze_ir command in the command file for IR drop analysis. If you are using the `emir.conf` file, specify the `emirutil` command options, `analysisStartTime` and `analysisStopTime`. If the start and stop times are not specified, the default is the full duration of the simulation.

  Note: For details of the `emirutil` commands supported in Voltus-Fi-L, see the EMIR Control File Options Supported in Voltus-Fi-L in the "Batch Mode Execution" chapter.

- When the analysis type is specified, the type of analysis is displayed in the *Analyze Type* field.
- For the VAVO_DB data, select the *DB Files* from the list. This is the `vavo_db` file to be used for the analysis.
- Click *OK*.
- Select *QRC Run.* Two text fields with browse buttons appear in the form. Click on the Browse buttons to open the pop-up windows, *Select QRC Run Directory* and *Select QRC Run Name* to specify the path to the QRC *Run Directory* and the QRC *Run Name*, respectively.



Note: When you switch between the IR and EM tabs, the selected *QRC Run*

© 2023
All Rights Reserved.

information remains intact. This means that the QRC run selections, run directory and run name fields, made on one tab remain populated for the other tab even after clearing the analysis results.

- Specify the *DFII Layermap*. This layermap file is the extracted view-to-DFII layermap file. When this file is specified, the plots displayed on the extracted view show solid shape highlighting. For details, see Solid Shape Highlighting. For a sample file, see DFII Layer Map File in the "File Formats" chapter.

  You can also specify this file by using the `vfiDfiiLayerMapFile` environment variable in either the `.cdsenv` or the `.cdsinit` file. For more information, see Loading the DFII Layer Map File by Default.

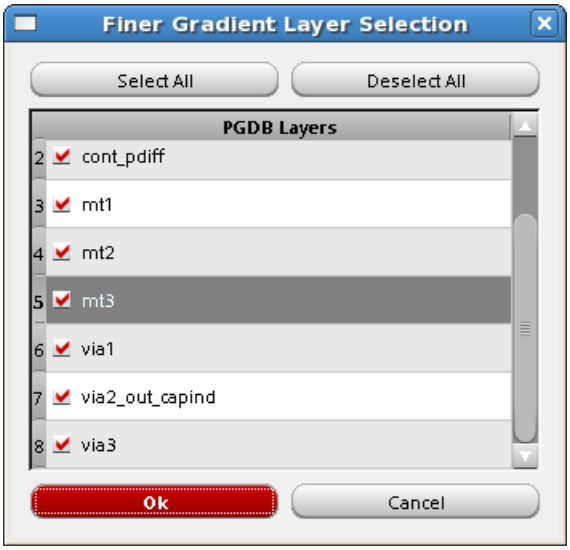
  The following considerations apply to the DFII layermap flow:

  - When the DFII layermap file is specified, the visibility of the DFII layers on the layout is synchronized 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.
  - 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 the figure below.

© 2023 All Rights Reserved.

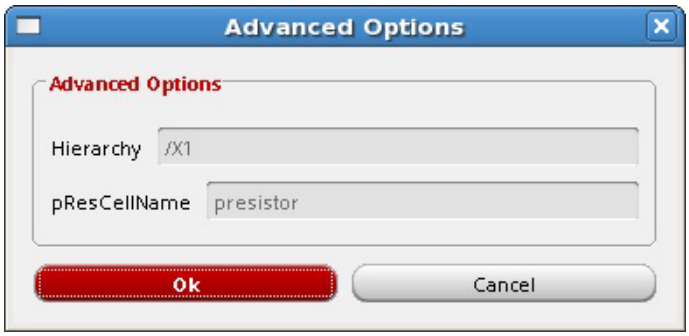
Figure 6-3 Selecting Layers for Viewing Finer Gradient



In this window, all the layers are selected by default. Click *Deselect All* and then select specific PGDB layers for which you want to view finer gradient and click *OK*. For details, see Displaying Finer Color Gradient for Selected Layers in IR and EM Plots.

- Select *Advanced*. The Advanced Options form opens.

Figure 6-4 Advanced Options



In this form, specify the following:

- *Hierarchy*—the hierarchy of the design
- *pResCellName*—the cell name of the parasitic resistor that is added to the extracted view. The default name is `presistor`. This is shown above.

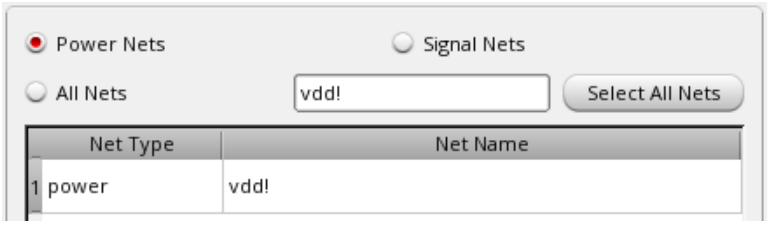
- Click *Load Results* to load the results of the IR drop Analysis. After the results are loaded, the list in the Net Plot group box is enabled.

© 2023 All Rights Reserved.

- Once the results are loaded, the *Rail Analysis* cyclic field in the *Net Plot* group box becomes enabled. From the list, select the type of plot you want to view. The details of the plot types available are provided in Types of IR Drop Analysis Plots.
- Click *Power Nets* to list all power nets in the list box.
- Click *Signal Net*s to list all signal nets in the list box.
- Click *All Nets* to list all power and signal nets in the list box.
- Click *Select All Nets* to select all the nets listed in the table. For example, if you select *Power Nets* and click *Select All Nets*, all power nets listed in the table are selected. 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 on page 91.

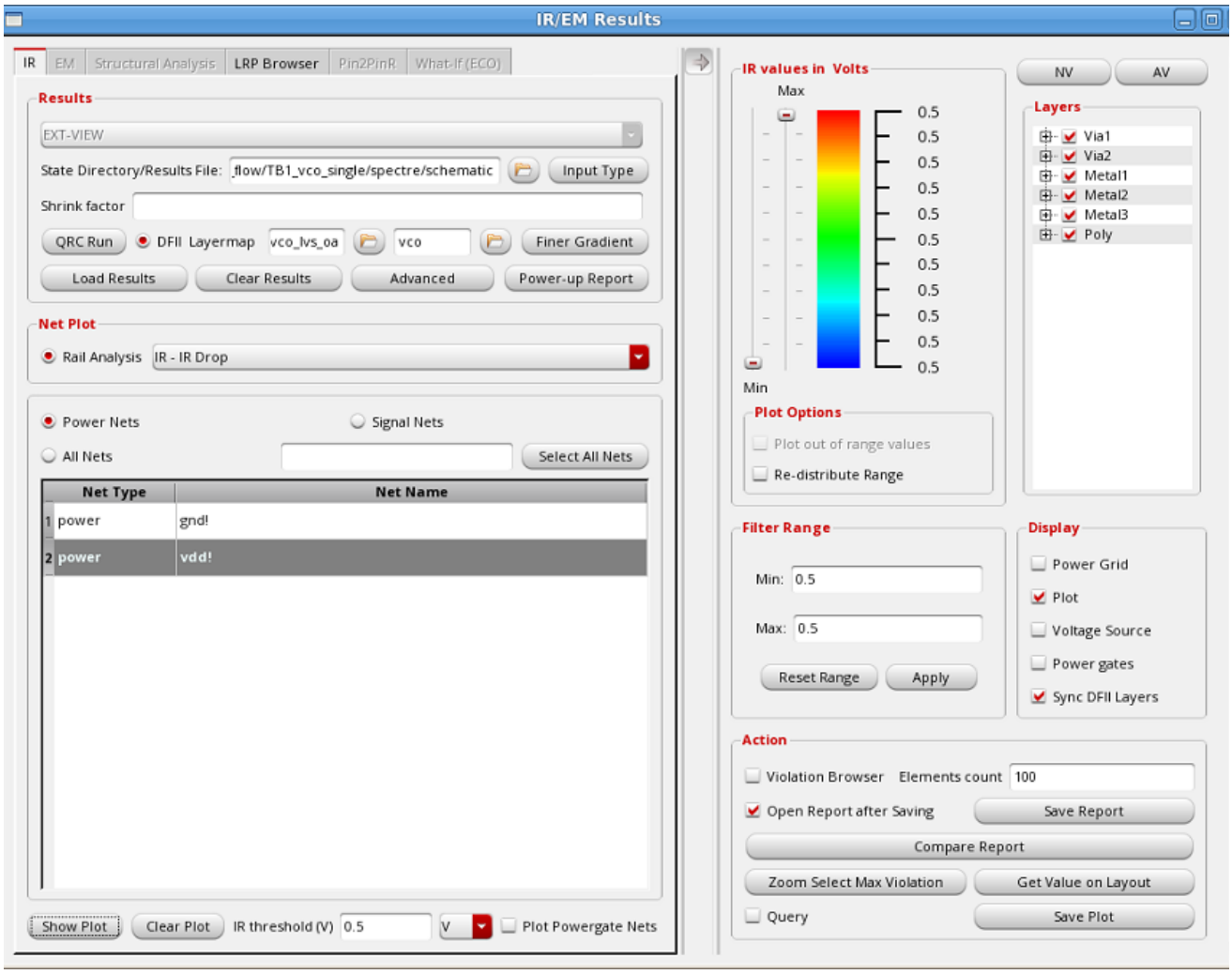
- Click *Show Plot* to view the plots for the selected nets on the extracted view. When you click *Show Plot*, the IR/EM Results form expands to show the available plot display options. In this form, you can customize the IR drop analysis plot displays.

  For details, see Displaying and Querying EMIR Results.

- Click *Clear Plot* to clear the plot for the selected net from the Virtuoso layout. For this, select the net for which you want to clear the plot in the list box and then click *Clear Plot*.
- Specify *IR threshold (V)* 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 0.5. So, the *Min* value of the slider range is updated to 0.5.

© 2023 All Rights Reserved.

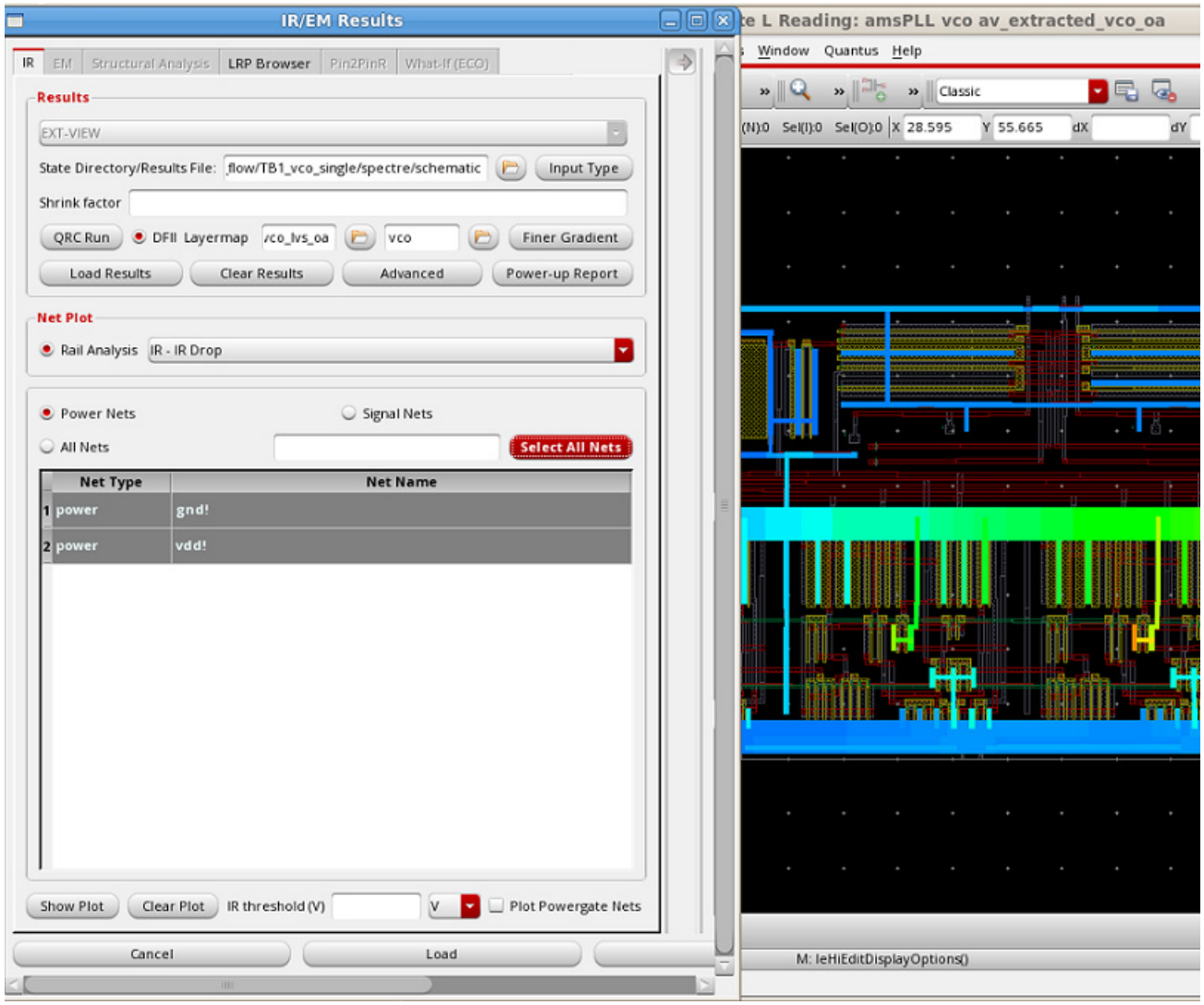
Figure 6-5 Specifying the Threshold Value for Viewing IR Drop Violations



- 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.

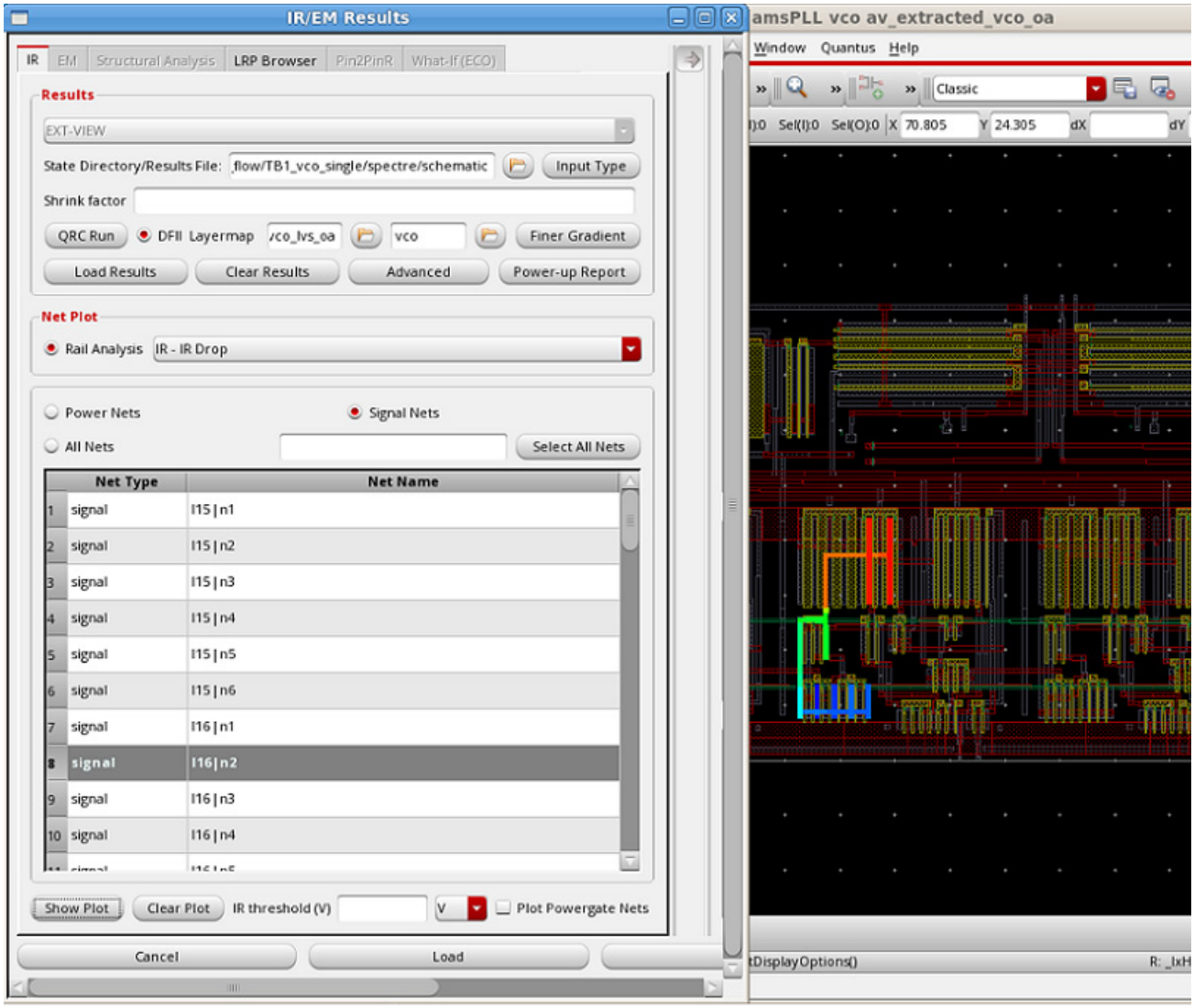
© 2023
All Rights Reserved.

Figure 6-6 Viewing the IR Plot for All Power Nets



© 2023
All Rights Reserved.

Figure 6-7 Viewing IR Plot for Selected Signal Net



© 2023
All Rights Reserved.

Displaying and Querying EMIR Results

Voltus-Fi-L 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.

In addition to specifying settings for the 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.

You can also compare reports that are generated for different analysis types either in the GUI or the batch mode. You can compare any two reports; one GUI and one batch report, two GUI reports, or even two batch reports. For this, *Compare Report* button is provided in the Display form.

For detailed descriptions of the display options and query feature in the GUI mode, see Displaying and Querying EMIR Results in the “IR Drop Analysis Plots” chapter of the *Voltus-Fi Custom Power Integrity Solution XL User Guide.*

The section, Displaying and Querying EMIR Results includes the following topics and subtopics:

- Viewing EMIR Violations in the Annotation Browser
  - Information Displayed in the EM/IR Tab of the Annotation Browser
  - Information Displayed in the Description Section of the Annotation Browser
- Querying EMIR Results
  - Viewing worst violations
  - Viewing all violations in selected area
  - Querying Specific Points and Areas on the Layout for Resistor Information
- Specifying the Display Options for IR/EM Plots
  - The following topics are covered in this section:
  - Selecting Layers for Viewing Violations
  - Customizing the Range for Viewing Violations

© 2023 All Rights Reserved.

- Customizing the Display Options for Plots
- Viewing and Managing Violations on the Layout
- Viewing Worst Violations
- Viewing all Violations in Selected Area
- Querying Specific Points and Areas on the Layout for Resistor Information
- Saving the IR Drop Analysis Result Plots and Reports
- Comparing Reports

For the batch command flow of the Query feature, see Batch Command for RON Reports in the “Batch Mode Execution” chapter.

Types of IR Drop Analysis Plots

Following types of IR drop analysis results plots are available in the list in the *Rail Analysis* field in the IR/EM Results form.

- IR – IR Drop: analyzes and reports the voltage drop
- IRAVG – IR Avg Drop: analyzes and reports the average voltage drop
- RC – Resistor Current: analyzes and reports peak resistor currents
- Power – Resistor Power: analyzes and reports resistor power, which is the value of current through a resistor multiplied by the voltage across the resistor
- RCAVG – Average Resistor Current: analyzes and reports average resistor currents
- RCRMS - RMS Resistor Current: analyzes and reports RMS resistor currents

Note: For details of the above plots, see Types of IR Drop Analysis Plots in the “IR Drop Analysis” chapter of the *Voltus-Fi Custom Power Integrity Solution XL User Guide*.

Note: The following IR drop analysis plots, currently supported in Voltus-Fi-XL, are not supported in Voltus-Fi-L:

- IV – Transistor Voltage Plot
- PI – Powergate Current Plot
- PV – Powergate Voltage Plot
- REffective – Effective Resistance Plot

© 2023 All Rights Reserved.

- TC – Tap Current Plots

© 2023 All Rights Reserved.

Plotting the Least-Resistive Path

The least-resistive path (LRP) plot lets you identify the 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-L lets you plot LRP on demand for any node. 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 Node* and *Get Marker Node*, 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 Node 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 analysis results:

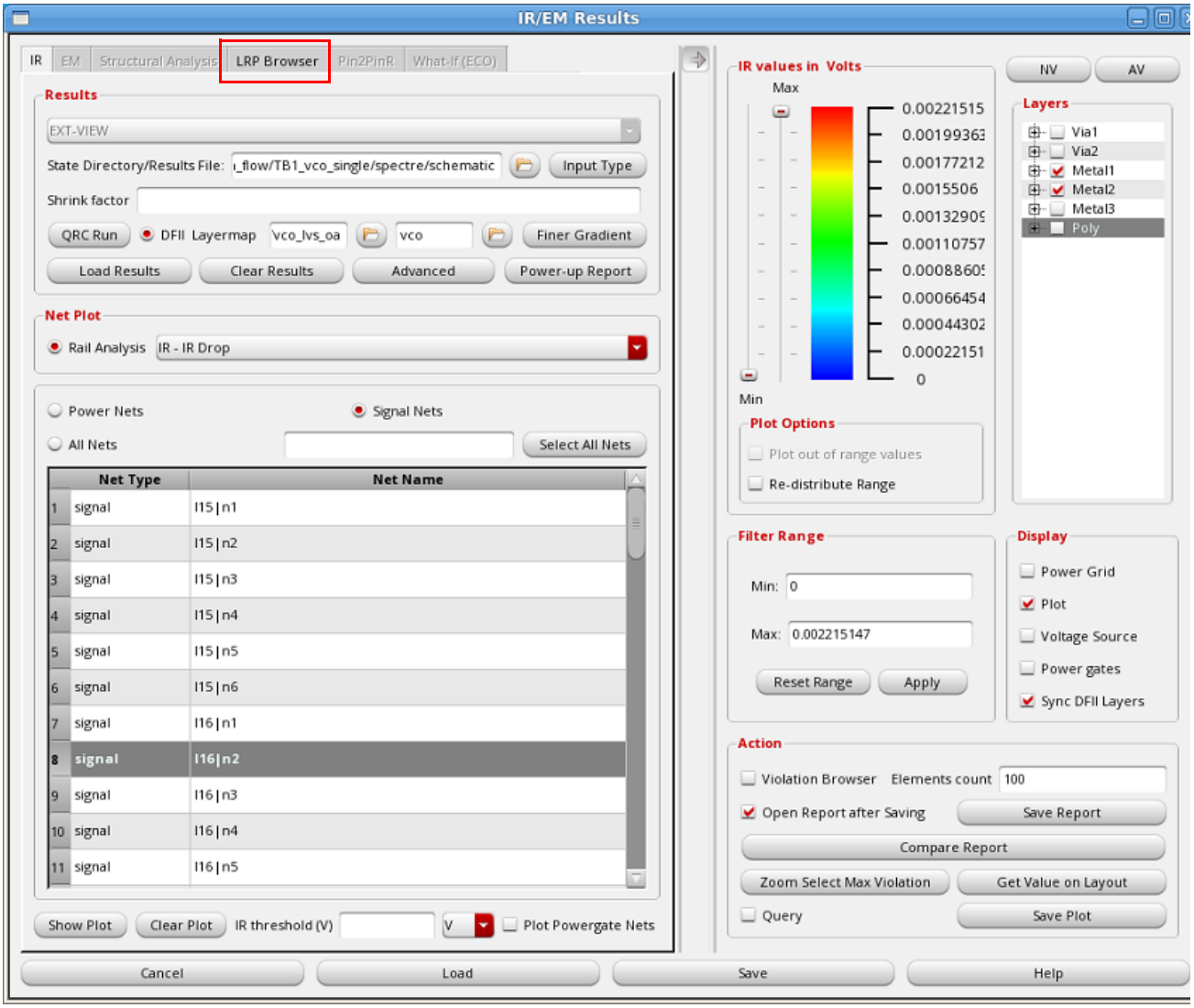
- Click *Select Nets* to select the net for which you want to view the LRP plot.

  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.

© 2023 All Rights Reserved.

Figure 6-8 IR/EM Results Form – Plotting the Least-Resistive Path



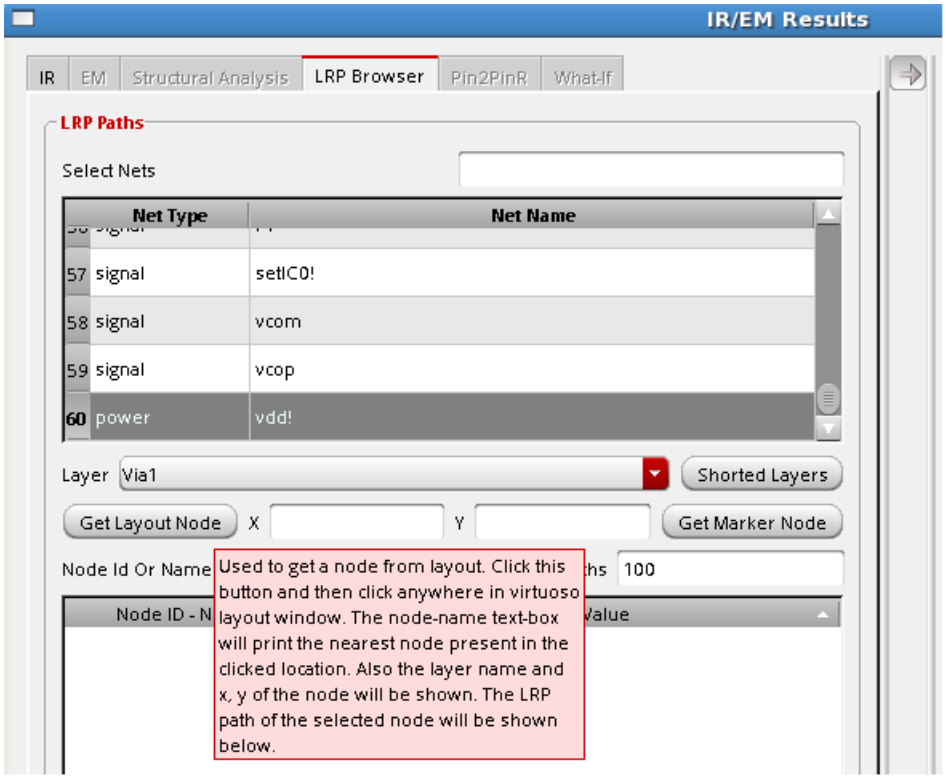
The LRP Browser

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

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

© 2023 All Rights Reserved.

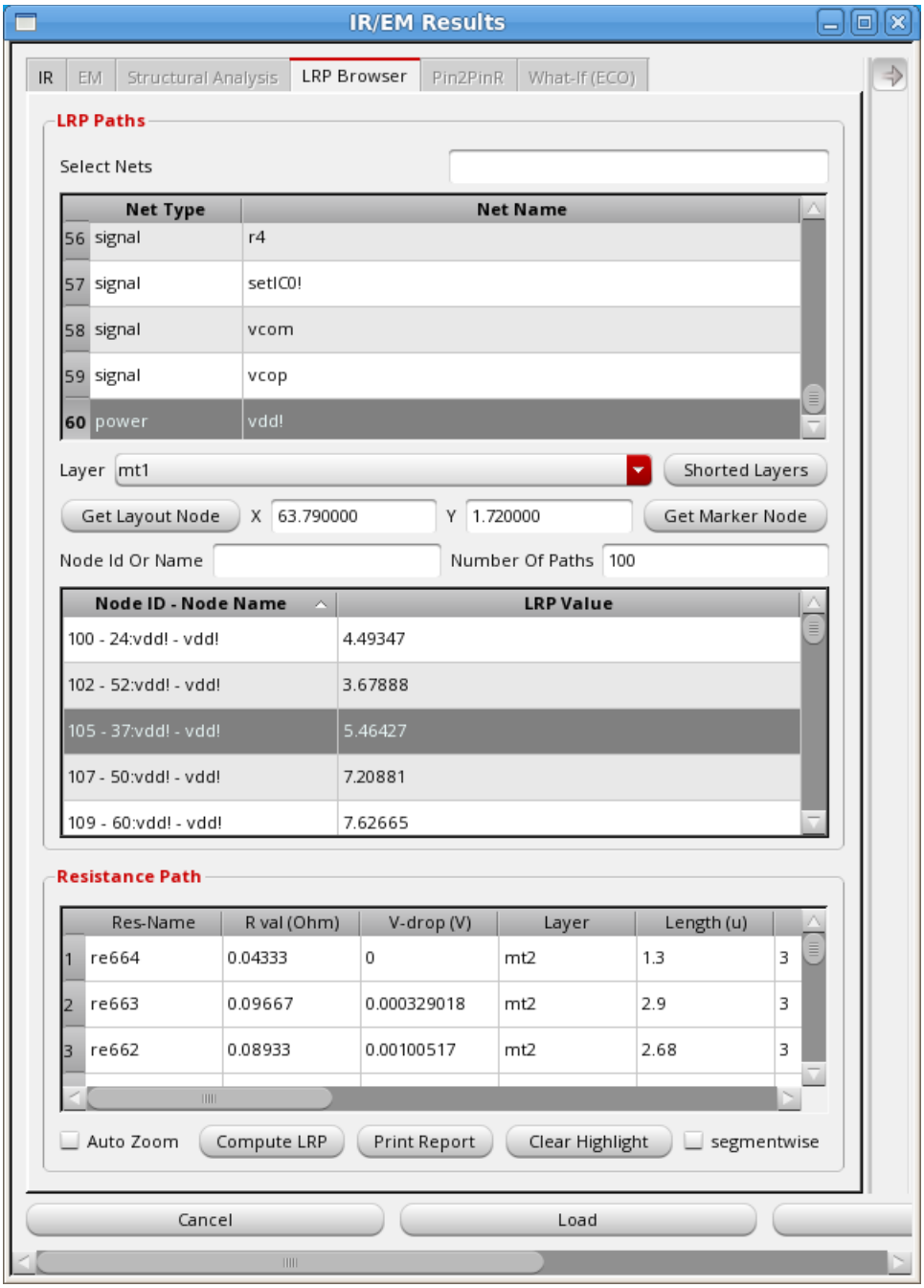
Figure 6-9 Tool Tips for LRP Browser Tab



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

© 2023 All Rights Reserved.

Figure 6-10 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.

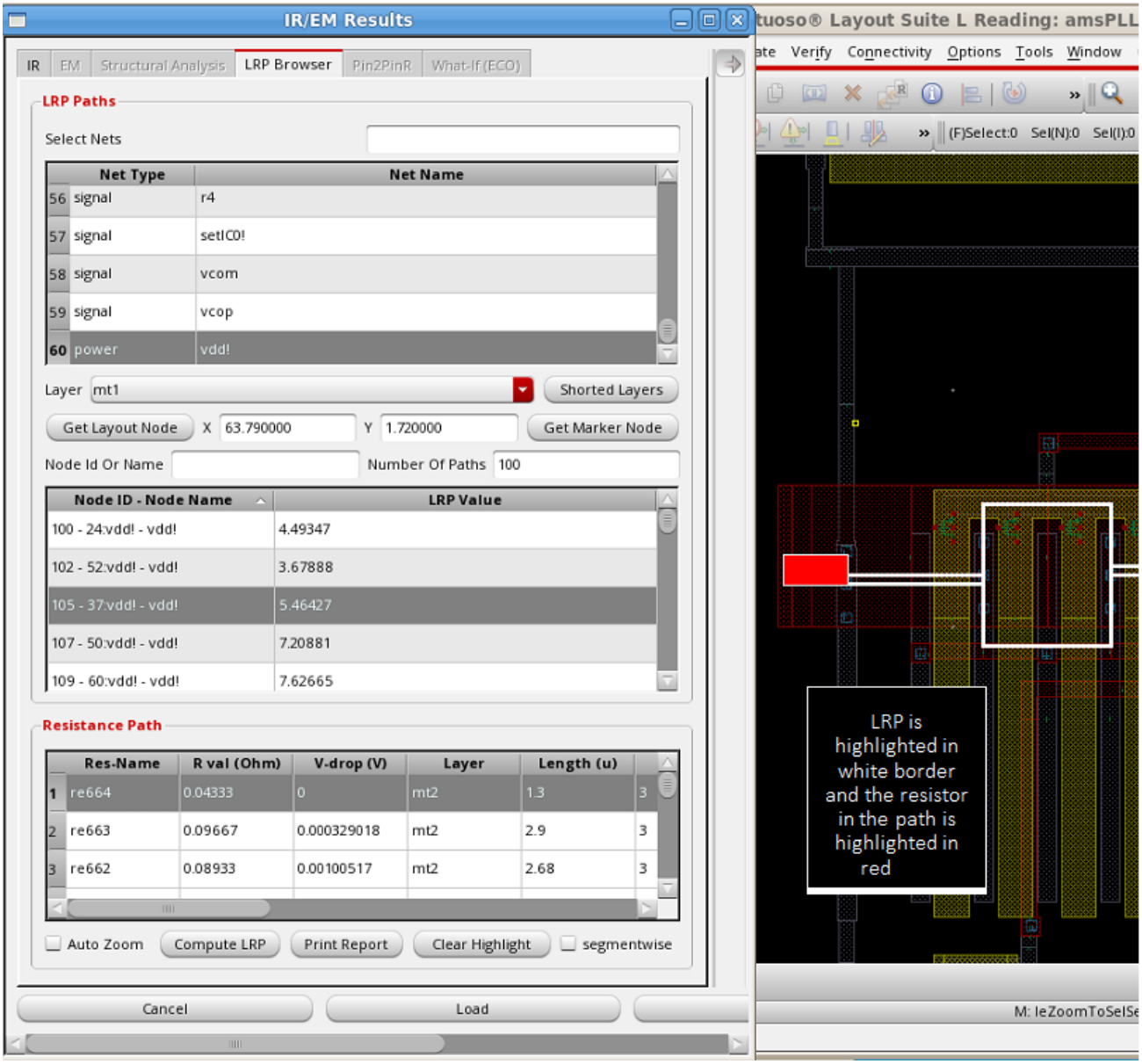
© 2023 All Rights Reserved.

- Click *Compute LRP* to compute the LRP values for all the nodes of the selected net. The information is populated in the *Node ID - Node Name* and the *LRP Value* columns of the table in the *LRP Paths* group box.
- Click *Shorted Layers* to short specific layers for the LRP analysis. For details, see Shorting Layers for LRP Analysis.
- Click *Get layout Node* to select a node on the layout. Click this button and then click anywhere in the Virtuoso layout window. The node-name text box will print information about the *X*- and *Y*- co-ordinates, and the *Layer* name for the node present nearest to the location of the click. The LRP path of the selected node will be shown in the table below.
- Click *Get Marker Node* to view the LRP values for a node, if present, on a selected marker. For this, enable the Annotation Browser, select a marker and click this button. If the marker is associated with a node name, the LRP for the node will be shown in the table below. For details, see Displaying LRP for the Node on the Selected Marker.
- Type the name of the node in the *Node Id or Name* to retrieve the LRP value for the specific node. Node names can also be regular expressions. The LRP table shows all nodes matching the regular expression. This saves time otherwise 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 nodes defined for the net in the *Node ID - Node 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 the image below.
- In the *Resistance Path* group box of the LRP Browser tab, you can view the list of resistors for a path. For details of the columns in the table, see Resistance Path Group Box Table Information. 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.

© 2023 All Rights Reserved.

Figure 6-11 Least-Resistive Path for a Resistor Highlighted on the Virtuoso Layout



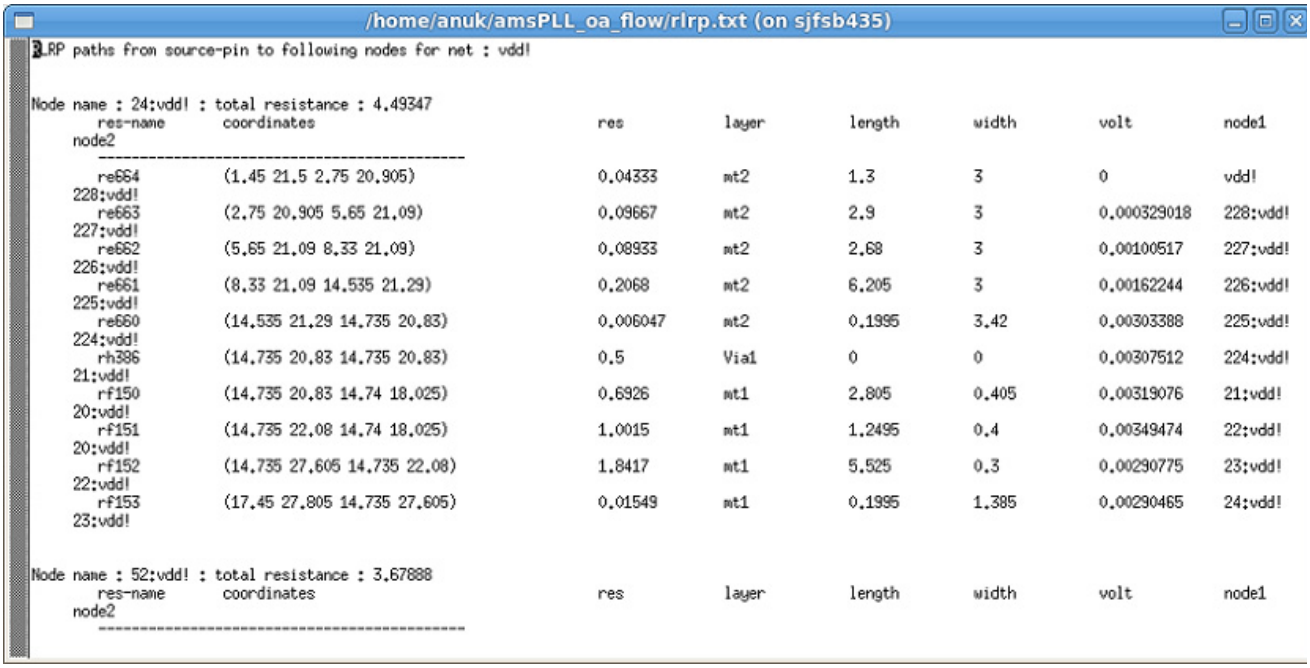
Note: If the tables in the LRP browser tab do not show any entries, ensure the following:

- The selected net should have at least one pin location or voltage-source location
- The selected net should have nodes connected to it

© 2023
All Rights Reserved.

- 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. This is shown below.

Figure 6-12 Sample RLRP Report



/home/anuk/amsPLL_oa_flow/rlrp.txt (on sjfsb435)

```
LRP paths from source-pin to following nodes for net : vdd!

Node name : 24:vdd! : total resistance : 4.49347
      res-name      coordinates                          res        layer    length   width    volt         node1
    node2
      ---------------------------------------------
      re664         (1.45 21.5 2.75 20.905)              0.04333    mt2      1.3      3        0            vdd!
    228:vdd!
      re663         (2.75 20.905 5.65 21.09)             0.09667    mt2      2.9      3        0.000329018  228:vdd!
    227:vdd!
      re662         (5.65 21.09 8.33 21.09)              0.08933    mt2      2.68     3        0.00100517   227:vdd!
    226:vdd!
      re661         (8.33 21.09 14.535 21.29)            0.2068     mt2      6.205    3        0.00162244   226:vdd!
    225:vdd!
      re660         (14.535 21.29 14.735 20.83)          0.006047   mt2      0.1995   3.42     0.00303388   225:vdd!
    224:vdd!
      rh386         (14.735 20.83 14.735 20.83)          0.5        Via1     0        0        0.00307512   224:vdd!
    21:vdd!
      rf150         (14.735 20.83 14.74 18.025)          0.6926     mt1      2.805    0.405    0.00319076   21:vdd!
    20:vdd!
      rf151         (14.735 22.08 14.74 18.025)          1.0015     mt1      1.2495   0.4      0.00349474   22:vdd!
    20:vdd!
      rf152         (14.735 27.605 14.735 22.08)         1.8417     mt1      5.525    0.3      0.00290775   23:vdd!
    22:vdd!
      rf153         (17.45 27.805 14.735 27.605)         0.01549    mt1      0.1995   1.385    0.00290465   24:vdd!
    23:vdd!


Node name : 52:vdd! : total resistance : 3.67888
      res-name      coordinates                          res        layer    length   width    volt         node1
    node2
      ---------------------------------------------
```

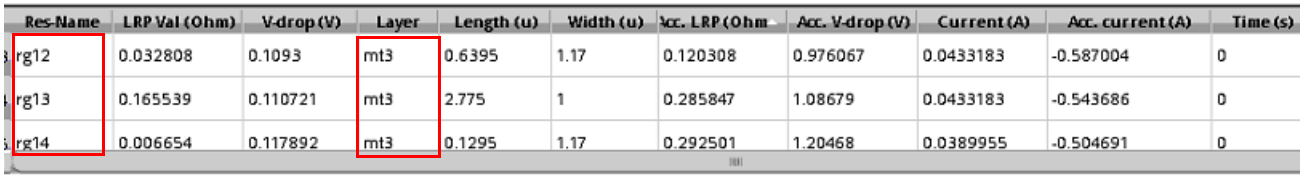
You can also generate RLRP reports in the batch mode by using the print_rlrp_report command.

A sample report is shown in RLRP Report in the “IR Drop Analysis” chapter of the *Voltus-Fi Custom Power Integrity Solution XL User Guide.*

- Click *Clear Highlight* to clear the LRP highlighted on the layout.
- Check *segmentwise* to view the combined resistance values for a layer in the Resistance Path table. When this option is enabled, all the resistors on a layer, metal or via, will be merged into one segment for the purpose of LRP reporting. In the image below, the resistors, `rg12`, `rg13`, and `rg14` on layer `mt3` are shown in the Resistor Path table. This is the display when the *segmentwise* option is not selected.

© 2023 All Rights Reserved.

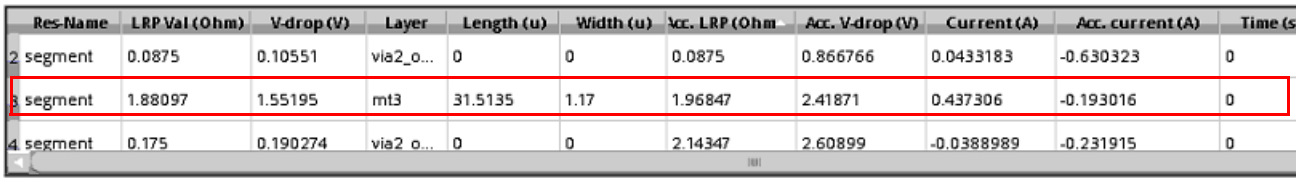
Figure 6-13 Display of Resistor Paths



Res-Name	LRP Val (Ohm)	V-drop (V)	Layer	Length (u)	Width (u)	Acc. LRP (Ohm	Acc. V-drop (V)	Current (A)	Acc. current (A)	Time (s)
rg12	0.032808	0.1093	mt3	0.6395	1.17	0.120308	0.976067	0.0433183	-0.587004	0
rg13	0.165539	0.110721	mt3	2.775	1	0.285847	1.08679	0.0433183	-0.543686	0
rg14	0.006654	0.117892	mt3	0.1295	1.17	0.292501	1.20468	0.0389955	-0.504691	0

In the image below, the resistors for the layer, `mt3` are combined to display the resistor path for the segment. This is highlighted in red in the image below.

Figure 6-14 Segment-wise Display of Resistor Paths



Res-Name	LRP Val (Ohm)	V-drop (V)	Layer	Length (u)	Width (u)	Acc. LRP (Ohm	Acc. V-drop (V)	Current (A)	Acc. current (A)	Time (s
2 segment	0.0875	0.10551	via2_o...	0	0	0.0875	0.866766	0.0433183	-0.630323	0
3 segment	1.88097	1.55195	mt3	31.5135	1.17	1.96847	2.41871	0.437306	-0.193016	0
4 segment	0.175	0.190274	via2_o...	0	0	2.14347	2.60899	-0.0388989	-0.231915	0

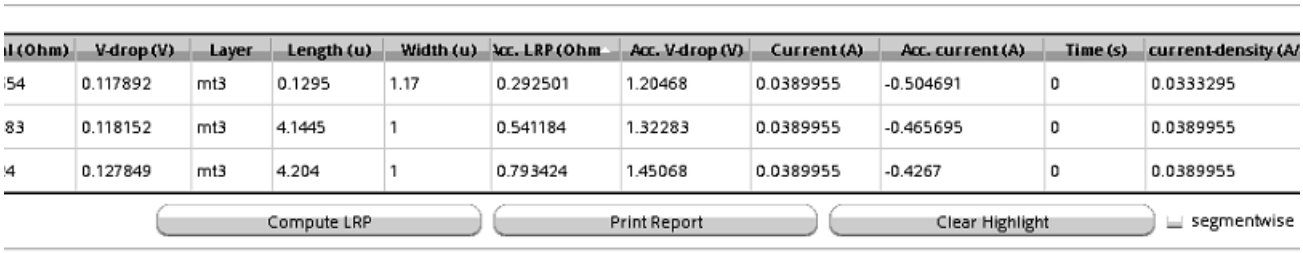
In the above image, the resistors for the layer, `mt3` are combined to display the resistor path for the segment. This is highlighted in red in the above image.

Resistance Path Group Box Table Information

The Resistance Path table lists all the resistors in the LRP path for the node selected in the *Node ID - Node Name* table. You can click any row to highlight the resistor in the layout window.

A section of the *Resistance Path* table is shown in the image below. The details of the column information is provided subsequently.

Figure 6-15 Information in the Resistance Path Table



l (Ohm)	V-drop (V)	Layer	Length (u)	Width (u)	Acc. LRP (Ohm	Acc. V-drop (V)	Current (A)	Acc. current (A)	Time (s)	current-density (A/
54	0.117892	mt3	0.1295	1.17	0.292501	1.20468	0.0389955	-0.504691	0	0.0333295
83	0.118152	mt3	4.1445	1	0.541184	1.32283	0.0389955	-0.465695	0	0.0389955
4	0.127849	mt3	4.204	1	0.793424	1.45068	0.0389955	-0.4267	0	0.0389955

Compute LRP | Print Report | Clear Highlight | segmentwise

The following information is provided in the table:

© 2023 All Rights Reserved.

- *Res-Name*: Name of the resistor on the layer
- *R Val (Ohm)*: The resistance value of the resistor
- *V-Drop (V)*: The voltage drop across the resistor
- *Layer*: Name of the layer on which the resistor is located
- *Length (u)*: The length of the resistor in micron
- *Width (u)*: The width of the resistor in micron
- *Acc.LRP (ohm)*: The accumulative resistance along the LRP
- *Acc.V-Drop (ohm)*: The accumulative voltage drop along the LRP
- *Current (A)*: The current across the resistor
- *Acc.Current (A)*: The accumulative current along the LRP
- *Time*: The simulation time captured from the simulation result file
- *current-density (A/u)*: The Current Density across the resistor
- *Location (u)*: The location of the resistor

Shorting Layers for LRP Analysis

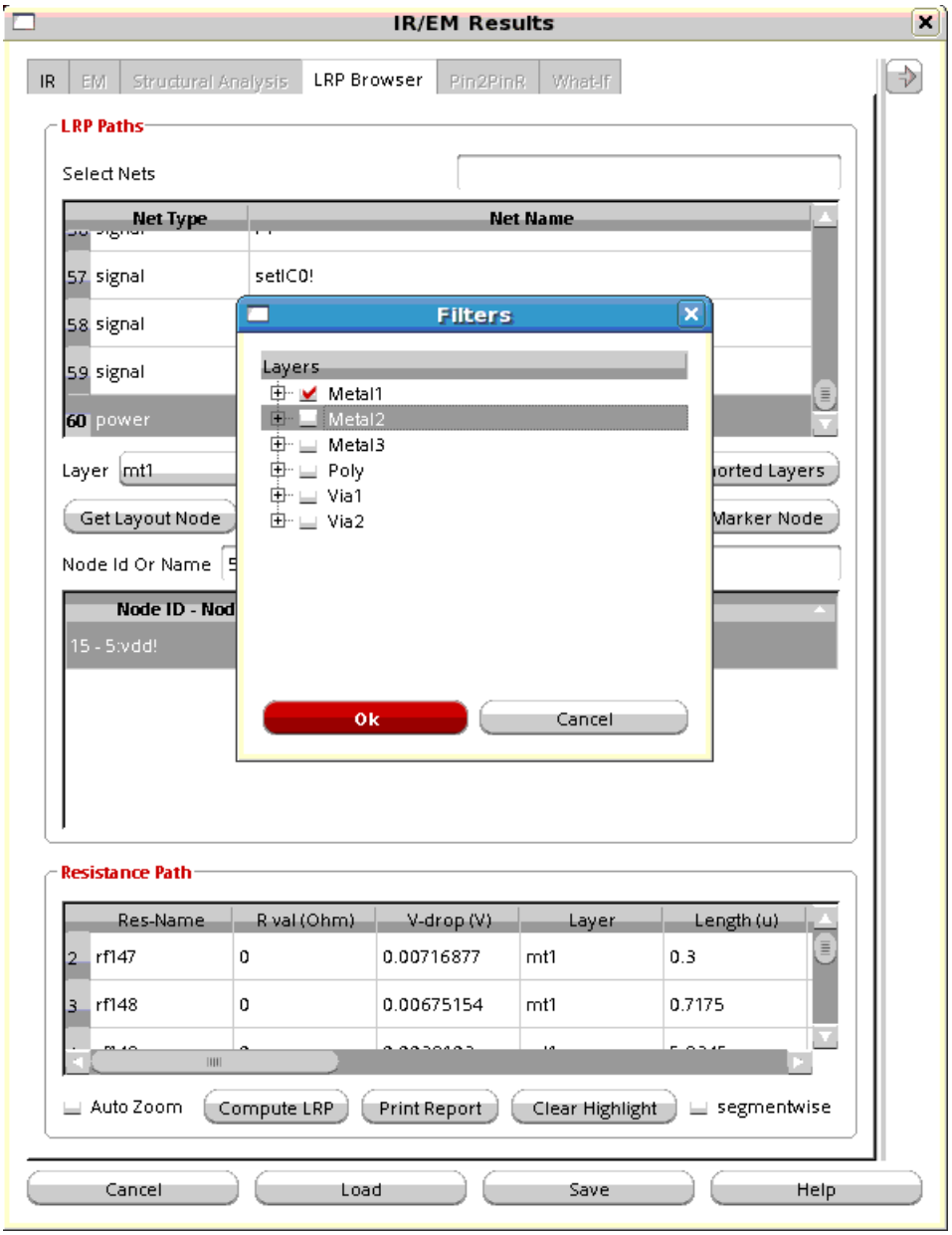
While computing LRP for a node or instance, you can short specific layers. When you do this, all the resistance on the specified layer is shorted. That means, the LRP display shows a value of 0 for all resistors on the shorted layer. This is shown in the images below.

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` or `mt1` is shorted.

© 2023 All Rights Reserved.

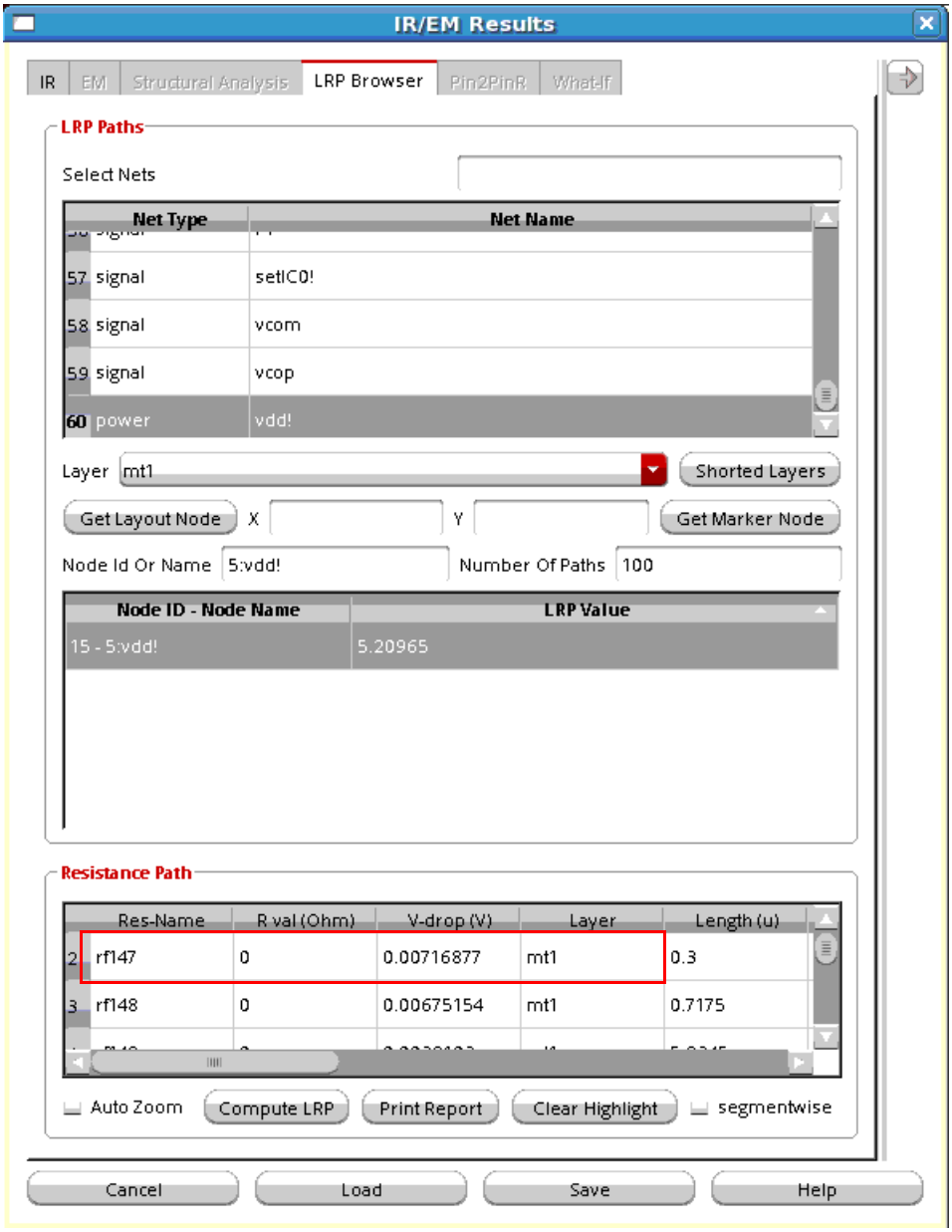
Figure 6-16 Specifying the layers to be shorted for the LRP Analysis



When a layer is shorted, the resistance value information in the Resistance Path group box shows a value of `0` in the *R val (Ohm)* column for all the resistors on that layer. For example, in the image below, the LRP value for resistor `rf147` on layer `mt1` is 0.

© 2023
All Rights Reserved.

Figure 6-17 LRP Value for Resistors on the Shorted Layer



Displaying LRP for the Node on the Selected Marker

You can display the LRP for a node 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 node name, you can view the LRP for the node.

© 2023
All Rights Reserved.

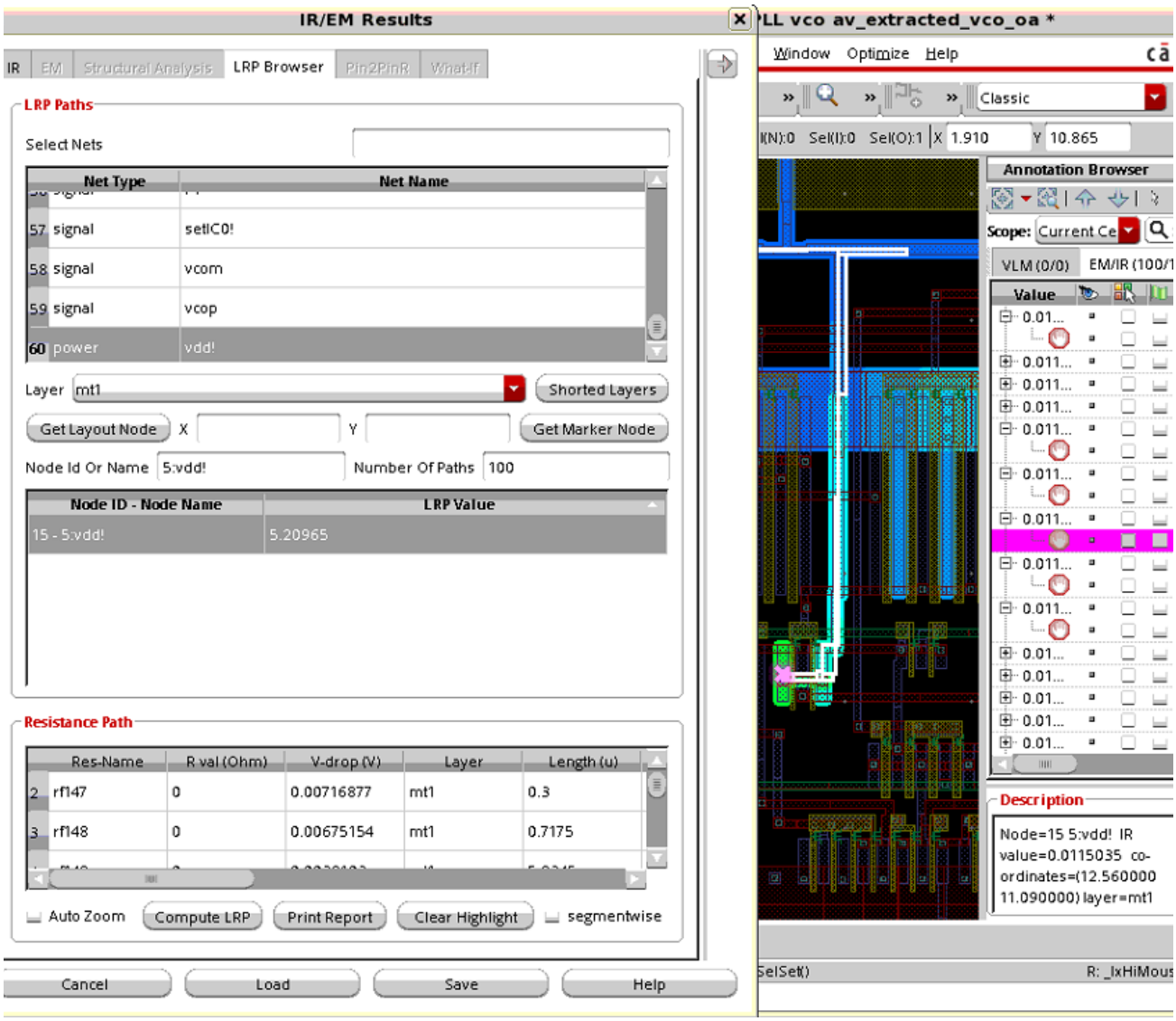
The steps are detailed below.

- In the IR tab, select 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.
- In the LRP Browser tab, click *Get Marker Node*. The information for the node – the x- and y- co-ordinates and the layer name – on the marker is populated in the LRP browser tab.
- Click the node name in the LRP table and then click *Compute LRP* to compute the LRP value for the node and to view the least-resistive path for the node on the layout.

  An example is shown below.

© 2023 All Rights Reserved.

Figure 6-18 Displaying LRP for the Node on the Selected Marker



© 2023
All Rights Reserved.

Generating EMIR Analysis Reports in Voltus-Fi-L

The following topics are covered in this section:

- Reports Generated Using the EMIR Control File
- Default Reports Generated by Voltus-Fi-L for EMIR Analysis
- RLRP Report

Reports Generated Using the EMIR Control File

You can generate both textual and html reports for EMIR results in Voltus-Fi-L 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 details, see Reports Generated using the EMIR Control File in the "IR Drop Analysis" chapter of the *Voltus-Fi Custom Power Integrity Solution XL User Guide.*

Important

For details of the supported EMIR control file options that can be specified, see EMIR Control File Options Supported in Voltus-Fi-L in the "Batch Mode Execution" chapter.

You can also see the EMIR Control FIle Options Supported in Voltus-Fi-XL in the "Data Preparation" chapter of the *Voltus-Fi Custom Power Integrity Solution XL User Guide.*

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 by Voltus-Fi-L for EMIR Analysis

By default, Voltus-Fi-L generates the following reports for EMIR analysis:

- IR drop analysis reports with the extension, `#.rpt_ir`
- EM analysis reports with the extension, `#.rpt_em`
- Summary report (`summary.rpt`), which includes reports for the IR drop and EM analyses performed in the current run

© 2023 All Rights Reserved.

Note: All default reports print the software version number in the header.

For details of the report formats and sample reports, see Default Reports Generated for EMIR Analysis in the “IR Drop Analysis” chapter of the *Voltus-Fi Custom Power Integrity Solution XL User Guide.*

RLRP Report

RLRP analysis report (`RLRP.rpt`) includes the LRP values of the instances or tap-nodes of the specified net.

For details of the report format and sample report, see RLRP Report in the “IR Drop Analysis” chapter of the *Voltus-Fi Custom Power Integrity Solution XL User Guide.*

© 2023 All Rights Reserved.

Reporting On-Resistance Values

Overview

Voltus-Fi-L reports the on-resistance (RON) values for all layers and devices between specified pairs of pins in a design. The software reports the combined resistance contribution of both pins in the pair and the contribution from each pin. The resistance values are reported in ohms.

For this feature, `report_vfi_L_ron` command has been provided. For details, see report_vfi_L_ron.

Note: The flow is currently enabled only after setting the variable shown below:

```
setenv ENABLE_VFI_L_RON_DATA_READING_FROM_PSF t
```

Generating the RON Report

This flow is supported only in the batch mode. To enable this flow, perform the following steps:

- Create a command file including the information below. This command file is used to load results in Voltus-Fi and report RON.
  - `set_variable extViewExtraCmdFile "`*`file_name`*`"`
  - `load_em_results_extview -i cmd_file`
  - ```
    report_vfi_L_ron
    -pin_pair_file file_name
    [-device_ron true | false]
    -param_list_file file_name
    -output_file file_name
    ```
- Use the above command file in Voltus-Fi as follows:
  `vfibatch -cmd cmd_file`

Example

These set of commands are included in the main command file, `main.cmd`.

- `set_variable extViewExtraCmdFile "extViewExtraCmdFile"`

  Specifies the command file, `extViewExtraCmdFile,` for running EM analysis in batch mode in Voltus-Fi-L. For more information on the contents of this file, see Contents of the extViewExtraCmdFile.

© 2023 All Rights Reserved.
    ```

- `load_em_results_extview -i vsaEM_allpin_vavodb_voltus_fi.cmd`

  Loads the EM results stored in the, `vsaEM_allpin_vavodb_voltus_fi.cmd` file.

- `report_vfi_L_ron -pin_pair_file pin_pair.txt -output_file output.rpt`

  Creates a RON report in the `output.rpt` file for the pin-pairs specified in the `pin_pair.txt` file.

- `vfibatch -cmd main.cmd`

  The main.cmd file is then used in Voltus-Fi, as shown above.

Contents of the extViewExtraCmdFile

The extracted-view data is loaded through the cmd_file, `extViewExtraCmdFile`. A sample command file for EM analysis is shown below.

```
;vsa batch mode command file
;common commands
_vsa_extracted_lib_cell_view "amsPLL" "vco" "av_extracted_sol"
_vsa_testbench_lib_cell_view "amsPLL" "TB1_vco_single" "schematic"
_vsa_simulation_directory "/custom/TB1_vco_single/spectre/schematic/"
_vsa_pres_cellname "presistor"
_vsa_analysis_data "TRANSIENT"

;commands for EM analysis
_vsa_qrc_run_directory "/custom/WORK/AV/LVS_emir"
_vsa_em_data_file "emDataFile.txt"
;_vsa_analyze_em "0" "1n"
_vsa_analyze_em

exit
```

Output

The RON report includes the following information:

- Voltage source names and their corresponding terminal/pin names, voltage values, and current values
- Total RON values for devices and wires for the specified pin pairs
- Parameters for the devices being used for calculating the device RON

© 2023 All Rights Reserved.

- Layer-based wire RON for both pins in the pin-pair combinations
- Layer-based wire RON for each pin in the pin-pair combinations separately

Sample Pin-Pair File

```
D S
```

Where `D` and `S` are names of pins separated by a space.

Sample RON Report

A sample RON report is shown below.

© 2023 All Rights Reserved.

Figure 6-19 Sample RON Report

```
REPORT FILE CREATED = 2018-Mar-24 04:08:45 (2018-Mar-24 11:08:45 GMT)
Simulation psf file used: psf
QRC run directory: qrc_run_dir

Pins and voltage-source used:

/* Vsource     | Terminal Name| Voltage[V]  | Current[A]
/* ------------------------------------------------------------ */
   vsrc_D      | D            | 0.100000    | -0.252190
   vsrc_G      | G            | 5.000000    | -0.000000
   vsrc_S      | S            | 0.000000    | 0.000000
/* ------------------------------------------------------------ */
   dummy       | D2           | 0.000000    | 0.000000
/* ------------------------------------------------------------ */

Device Resistance & Wire Resistance for pair (Pin1 & Pin2):

 < Resistance Between Pins >
/* ------------------------------------------------------------------------------ */
/* (No)| Pin1    | Pin2        |         Ron Information     R[Ohm]
/* ------------------------------------------------------------------------------ */
/* ( 1)| D       | S           |
/*     |         |             | [ Parallel ]
/*     |         |             | Device   (I1\|device_1\@13) :   0.028803
/* ------------------------------------------------------------------------------*/
/*     |         |             |  Total Device : 0.037551      (9.470% )
/*     |         |             |  Total Wire : 0.358975        (90.530% )
/*     |         |             |  Total : 0.396526             (100.000% )
/* ------------------------------------------------------------------------------*/
Device Parameter:
I1\|device_1\@13|(" D" " G" " S")(I1\|device_1\@13) nmos40v ("w=0.00032"
"l=2.2e-06" "m=0.000416667") Con:Parallel
 [2400]

Pins Ron:
[Result No.(1) : D / S ]
/* ------------------------------------------------------ */
/* Wire & Via Name     | Resistance[Ohm]
/* ------------------------------------------------------ */
  META      (ra )       | 3.475778e-01 (87.656%)
  M2        (rb )       | 5.635546e-03 (1.421%)
  M1        (rc )       | 4.152733e-03 (1.047%)
  VIAB      (rob)       | 5.952597e-04 (0.150%)
  CS        (rl )       | 4.538119e-04 (0.114%)
  Total                 | 3.589751e-01 (90.530%)
/* ------------------------------------------------------ */
[Result No.(1) : EM_D ]
/* ------------------------------------------------------ */
/* Wire & Via Name     | Resistance[Ohm]
/* ------------------------------------------------------ */
  METALB      (ra )     | 1.616493e-01 (40.766%)
  M2          (rb )     | 2.748089e-03 (0.693%)
  M1          (rc )     | 2.025045e-03 (0.511%)
  VIAB        (rob)     | 3.011315e-04 (0.076%)
  CS          (rl )     | 4.538119e-04 (0.114%)
  Total                 | 1.672359e-01 (42.175%)
/* ------------------------------------------------------ */
[Result No.(1) : S ]
/* ------------------------------------------------------ */
/* Wire & Via Name     | Resistance[Ohm]
/* ------------------------------------------------------ */
  METALB     (ra )      | 1.859286e-01 (46.889%)
  M2         (rb )      | 2.887456e-03 (0.728%)
  M1         (rc )      | 2.127687e-03 (0.537%)
  VIAB       (rob)      | 2.941282e-04 (0.074%)
  CS         (rm )      | 4.454470e-04 (0.112%)
  Total                 | 1.917391e-01 (48.355%)
/* ------------------------------------------------------ */

Device Resistance & Wire Resistance for pair (D & G):
Device Resistance & Wire Resistance for pair (D & S):
```


© 2023 All Rights Reserved.

© 2023 All Rights Reserved.

7

EM Analysis Results

© 2023 All Rights Reserved.

Overview

Voltus-Fi-L uses the results of the simulation performed using the vavo-db or the PSF data to perform EM analysis and displays the results of the 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.

EM Rules Specification

Before running EM Analysis, the EM reliability rules need to be specified. These rules are read either from the QRC technology file (`qrcTechFile`), or from the `emDataFile`, which includes user-specified rules.

For more information, see the "EM Rules Specification" chapter.

Types of EM Analysis

This section details the types of analyses that are available in Voltus-Fi-L.

- **Peak (max)**—Calculates the Current Density violations based on the maximum DC current for metal lines, vias, and contacts.

  $\mathbf{I}_{peak} = \mathbf{I}_{peak\_dc}$ = maximum value of **I**

- **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(t)`" is value of current.

© 2023 All Rights Reserved.

- **AC-Peak (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.

  $\mathbf{I}_{peak\_ac}=\mathbf{I}_{peak}$

  Note: The above values are from simulation.

  $\mathbf{I}_{peak\_ac(limit)}=\mathbf{I}_{peak\_dc(limit)}/(r)^{1/2}$

  where the $I_{peak\_dc(limit)}$ is specified in the ICT file

  and duty ratio "r" is:

  $r=\mathbf{I}_{abs\text{-}avg}/\mathbf{I}_{peak}$, by default ("`false`")

  Note: The $I_{abs\text{-}avg}$ and I_{peak} values are from simulation.

  And

  $r=\mathbf{T}_d/\mathbf{T}_{total}$, when the dynamicACPeak variable is set as either "`true`" or "`multiplePeak`".

  $\mathbf{T}_{total}$=total transient time

  $\mathbf{T}_d$=the time duration in micro second or the total "On Time" period, where `abs(I)>(max(abs(I)/2.0)` during transient analysis

  Note: Voltus-Fi-L does not exactly calculate $\mathbf{T}_d$ as the `max(abs(I)/2.0)` period. Instead, it calculates $\mathbf{T}_d$ by measuring the pulse at half energy level. This is because of performance reasons. The parameter `pwc_threshold` defines the current threshold for finding the start of the pulse. The `pwc_threshold` value is specified using the `emirutil` command in the EMIR configuration file. The default value is `1e-06(1uA)`

  When the parameter `halfPeakAnalysis` is set to `true`, Voltus-Fi-L exactly calculates the $\mathbf{T}_d$ as the `max(abs(I)/2.0)` period.

  $\mathbf{T}_d$ has the following different values depending upon the value of `dynamicACPeak`:

  - $\mathbf{T}_d$=the time duration of maximum peak, when `dynamicACPeak` is set to `true`
  - $\mathbf{T}_d$=sum of time durations of different peaks, $\mathbf{T}_{d1}+\mathbf{T}_{d2}+\mathbf{T}_{dN}$.... (when `dynamicACPeak` is set to `multiplePeak`)

  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 $\mathbf{I}_{peak\_ac}$ 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.

© 2023 All Rights Reserved.

- **Avg-Abs (avgabs)**—Calculates the Current Density violations based on the average of the absolute current for metal lines, vias, and contacts.

$$I_{abs\text{-}avg} = \left[\left(\int_0^{\tau} |I(t)| dt \right) / \tau \right]$$

where "**T**" is time and "**I**`(t)`" is value of current.

- **RMS (rms)**—Calculates the Current Density violations based on the root mean square (RMS) of the AC 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(t)`" is value of current.

Design Resistor EM Current Analysis

By default, the EM current analysis is only performed on parasitic resistors. If required, EM analysis for design resistors can be enabled by specifying design resistor cell names by using commands listed below in a file.

```
net design_res_models=[name=nplusres l=l w=w layer=rhim_n unit=1e06]
net design_res_models=[name=pplusres l=l w=w layer=rhim_p unit=1e06]
```

Where,

`l` is the length property name of the cell

`w` is the width property name of the cell

`layer` is the technology layer to be used for EM analysis.

`unit` is the factor to convert `l/w` in `um`

This file is then specified in the Voltus-Fi-L command file using the variable listed below, before running the `load_em_results_extview` or the `load_ir_results_extview` commands.

© 2023 All Rights Reserved.

```
set_variable extViewExtraCmdFile <filename>
```

The design resistor related EM results are reported in the EM analysis report file.

For more details, see Design Resistor EM Current Analysis in the “EM Analysis Results” chapter of the *Voltus-Fi Custom Power Integrity Solution XL User Guide*.

© 2023 All Rights Reserved.

The EM Analysis Flow

In this section, the use model for the EM analysis flow is detailed below for the batch mode and GUI mode.

- The EM Analysis Batch Mode Flow
- The EM Analysis GUI Flow

The EM Analysis Batch Mode Flow

The EM analysis batch mode flow can be run using either the EMIR configuration file used in Voltus-Fi-XL or the command file used by legacy (VPS-L). The complete details of the batch mode flow are covered in the “Batch Mode Execution” chapter.

For details of the Voltus-Fi-L batch mode flow, see the following sections in the “Batch Mode Execution” chapter:

- The Command File Support
  - Supported VPSL Batch File Commands
- The Configuration File Support
  - EMIR Control File Options Supported in Voltus-Fi-L

For details about migrating from legacy (VPS-L) to Voltus-Fi-L in batch mode, see Migrating from Legacy (VPS-L) to Voltus-Fi-L in Batch Mode in the “Batch Mode Execution” chapter.

The batch mode EM analysis flow involves the following key tasks, which are detailed in the sections below:

- Loading the EM Analysis Results
- Printing the EM Analysis Reports

Loading the EM Analysis Results

When the legacy (VPS-L) command file is used for running Voltus-Fi-L in the batch mode, you can load the EM analysis results by writing the following command in the `vfibatch` command file:

```
load_em_results_extview -i cmd_file
```

Where,

© 2023 All Rights Reserved.

`cmd_file` is the batch file used in legacy (VPS-L) batch mode execution.

Printing the EM Analysis Reports

When the EMIR configuration file is used for running Voltus-Fi-L in the batch mode, the EM analysis report, `#.rpt_em`, is generated automatically by the tool.

When the legacy (VPS-L) batch command file is used for running Voltus-Fi in the batch mode, the command, print_em_report is used to generate the EM analysis reports. This is the same command that is used to print EM analysis reports in Voltus-Fi-XL.The syntax of the command is as follows:

```
print_em_report
     -net {all_power | all_signal | all_nets | netname}
     -type {javg | jmax | jabsavg | jacpeak | jacrms | rc | rcavg | rcrms}
     [-threshold threshold_value]
     -filename output_file_name
     [-rule custom_em_rule_name]
```

Note: In the `-type` parameter, only the analysis types listed in the syntax above are supported in the Voltus-Fi-L flow.

A Sample Command File for EM Analysis

Sample command files used for running the EM analysis in batch mode in Voltus-Fi-L are provided below.

- A sample batch command file, same as the legacy (VPS-L) command file, used for EM analysis is provided below.

```
;vsa batch mode command file
_vsa_extracted_lib_cell_view "amsPLL" "vco" "av_extracted_sol"
_vsa_testbench_lib_cell_view "amsPLL" "TB1_vco_single" "schematic"
_vsa_simulation_directory "/custom/TB1_vco_single/spectre/schematic/"
_vsa_pres_cellname "presistor"
_vsa_analysis_data "TRANSIENT"

;following command are for EM analysis
_vsa_qrc_run_directory  "/custom/WORK/AV/LVS_emir"
_vsa_em_data_file "emDataFile.txt"
;_vsa_analyze_em "0" "1n"
_vsa_analyze_em

exit
```


© 2023 All Rights Reserved.

- A sample batch command file, same as the EMIR configuration file, used for EM analysis is provided below.

```
For Transient Analysis
===========
net name=[*] analysis=[iavg irms imax vavg vmax]
emirutil view=[lib=amsPLL cell=vco view=av_extracted_vco_oa]
emirutil view_tb=[lib=amsPLL cell=TB1_vco_single view=schematic]
emirutil analysisType="tran"
emirutil analysisName="tran-tran"
emirutil presCellName="presistor"
emirutil qrc_output=[runDir=vco_lvs_oa runName=vco]
emirutil analysisStartTime="0"
emirutil analysisStopTime="170n"
emirutil hierarchy="/X1"
emirutil techfile="qrcTechFile"
```

The EM Analysis GUI Flow

To view the EM analysis results in Voltus-Fi-L, ensure the following are provided:

- The `emir` mode is enabled while performing simulation
- The name of the simulation result directory
- The DFII layermap file for mapping the extracted view names and the DFII layer names
- The location of the qrcTechFile/ICT file or emDataFile
- The Quantus QRC run directory
- The Quantus QRC run name
- The layermap file for mapping layer names in the simulation database to the layer names in the technology file
- The hierarchy name
- The presistor cell name

The following topics are covered in this section:

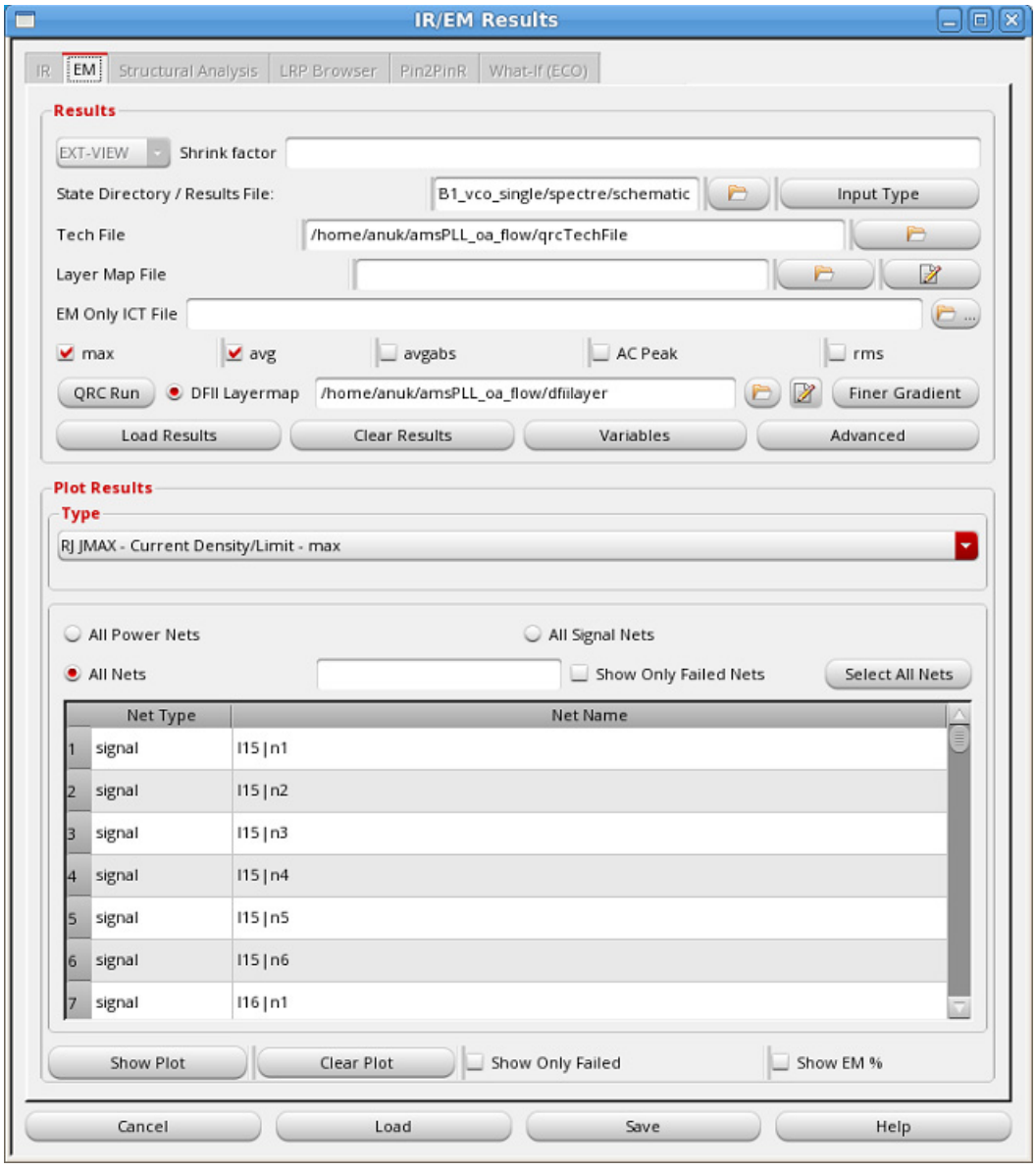
- Viewing the EM Analysis Results
- Displaying and Querying EMIR Results

© 2023 All Rights Reserved.

Viewing the EM Analysis Results

- 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.

Figure 7-1 IR/EM Results Form – The EM Tab



In the Results group box:

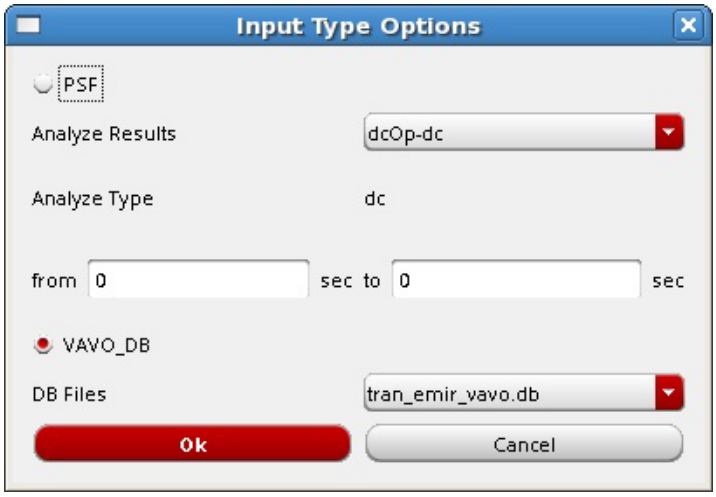
- The EXT_VIEW option is selected to enable the extracted view support.

© 2023 All Rights Reserved.

- Specify the *State Directory/Results file*, which is the path to the Spectre Simulation directory that stores the results of the simulation.
- Specify the *Shrink factor*, if any, by which the extracted view was shrunk. This is used for flows where the extracted view is generated with a shrunk technology but the layout remains on the original technology. For example, if the original extracted view was shrunk by 80%, the shrink factor should be specified as 0.8. The shrink factor value is less than 1.
- Specify the *Input Type Options* to specify the type of data, PSF or VAVO_DB, to be used for the EM analysis. When you click this button, the Input Type Options pop-up window opens. This is shown below.

  Note: The input type must be specified before running the IR drop or EM analysis.

Figure 7-2 Input Type Options



In the above form, provide the following inputs:

- Select either *PSF* or *VAVO_DB* depending upon the input data being used for the analysis. If your simulation data is in the `vavo.db` format, the *VAVO_DB* option is selected by default. You can also specify this option in the batch mode by using the _vsa_vavo_db_file command.
- For the PSF data, specify the analysis type in the *Analyze Results* cyclic field. You can specify either DC operating point (DCOP) analysis or transient (`tran`) analysis.
  - The DCOP analysis is recommended for large analog designs that are difficult to simulate in Spectre using transient analysis, especially when all parasitics are included.

© 2023 All Rights Reserved.

- The transient analysis option sets the simulation time in seconds automatically when the from/to fields are selected. For transient analysis, separate start and stop times can be specified using the *from* and *to* fields.

Note: You can also specify the start and stop time in the batch mode by using the _vsa_analyze_em command in the command file for EM analysis, respectively. If you are using the `emir conf` file, specify the `emirutil` command options, `analysisStartTime` and `analysisStopTime`. If the start and stop times are not specified, the full duration of the simulation is the default.

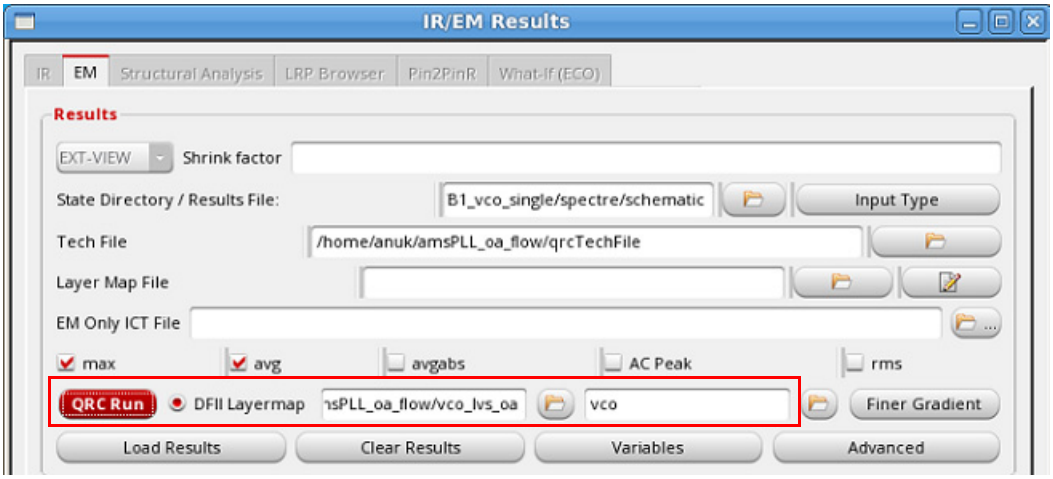
Note: For details of the `emirutil` commands supported in Voltus-Fi-L, see the EMIR Control File Options Supported in Voltus-Fi-L in the "Batch Mode Execution" chapter.

- When the analysis type is specified, the type of analysis is displayed in the *Analyze Type* field.
- For the VAVO_DB data, select the *DB Files* from the list. This is the `vavo_db` file to be used for the analysis.
- Click *OK*.

- Specify the technology file in the *Tech File* field. The following technology files are supported in this flow:
  - The interconnect technology (`ICT`) file
  - The QRC technology file (`qrcTechFile`)
  - The `emDataFile`, which is converted into the ICT file. For details, see Converting emDataFile to ICT File for EM Analysis.
- 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 the Layer Map File in the "File Formats" chapter.
- Specify the *EM Only ICT File*, which includes information about the 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`.
  - Select *QRC Run.* Two text fields with browse buttons appear in the form. Click on the Browse buttons to open the pop-up windows, *Select QRC Run Directory* and *Select QRC Run Name* to specify the path to the QRC *Run Directory* and the QRC *Run Name*, respectively.

© 2023 All Rights Reserved.

Figure 7-3 QRC Run Location



Note: When you switch between the IR and EM tabs, the selected *QRC Run* remains intact. This means that the QRC run selections, run directory and run name fields, made on one tab remain populated for the other tab even after clearing the analysis results.

- Specify the *DFII Layermap*. This layermap file is the extracted view-to-DFII layermap file. When this file is specified, the plots displayed on the extracted view show solid shape highlighting. For details, see Solid Shape Highlighting. For a sample file, see DFII Layer Map File in the “File Formats” chapter.

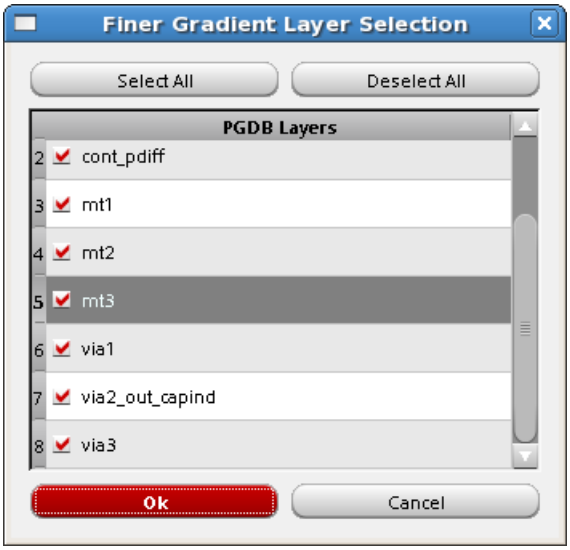
  The following considerations apply to the DFII layermap flow:

  - When the DFII layermap file is specified, the visibility of the DFII layers on the layout is synchronized 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.
  - When this file is not specified, the pop-up-window shown below opens. If you click *Yes*, the plots are displayed using stick diagrams.

© 2023
All Rights Reserved.

- 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 figure below.

Figure 7-4 Selecting Layers for Viewing Finer Gradient



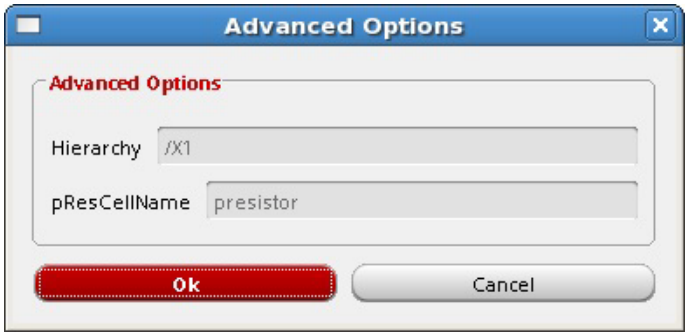
In this window, all the layers are selected by default. Click *Deselect All* and select the PGDB layers for which you want to view the finer gradient and click *OK*. For details, see Displaying Finer Color Gradient for Selected Layers in IR and EM Plots in the “IR Drop Analysis Results” chapter of the *Voltus-Fi Custom Power Integrity Solution XL User Guide.*

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.

- Select *Advanced*. The Advanced Options form opens.

© 2023 All Rights Reserved.

Figure 7-5 Advanced Options



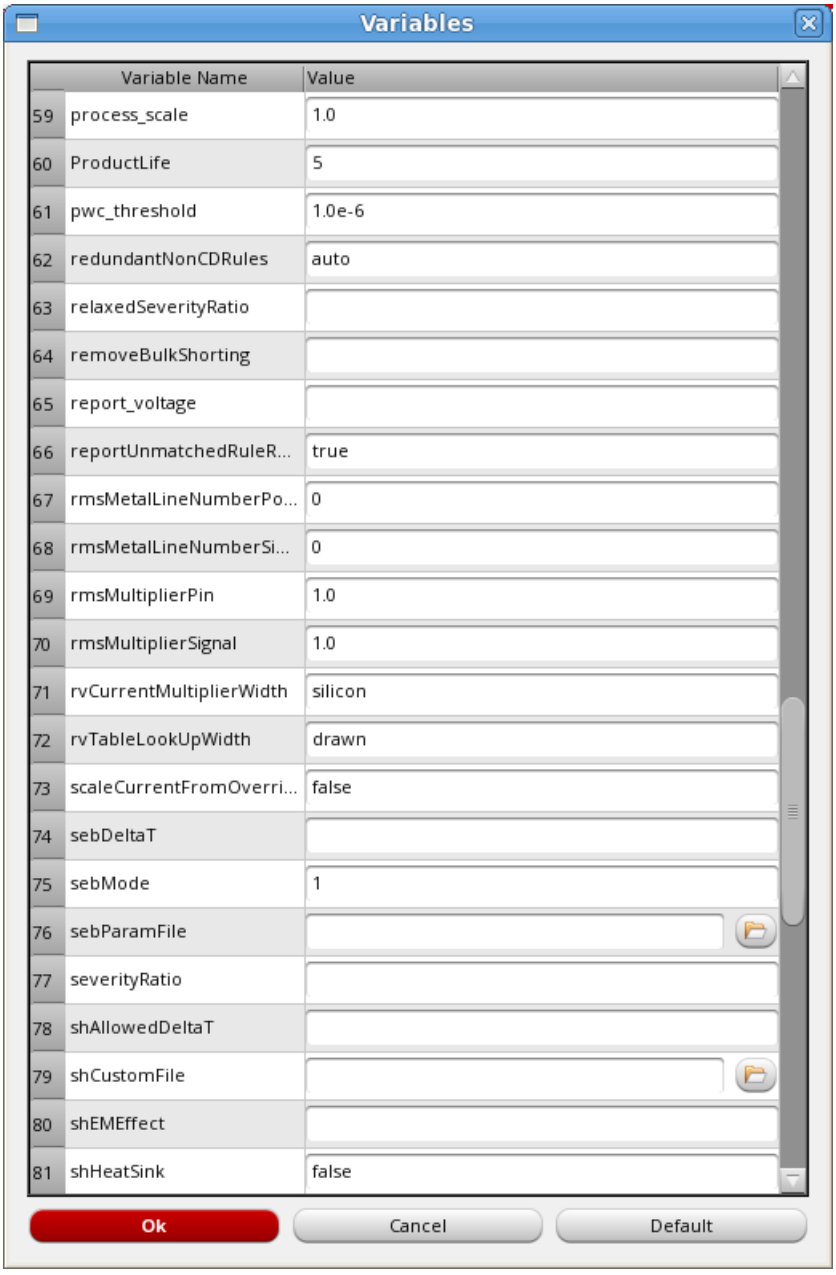
In this form, specify the following:

- *Hierarchy*—the hierarchy of the design
- *pResCellName*—the cell name of the parasitic resistor that is added to the extracted view. The default name is `presistor`. This is shown above.

- Click *Variables* to open the Variables form. In this form, you can view and/or edit the variables that are used for EM analysis. Double-click the value of any variable to edit it. For variables that require a path to a file or a directory to be specified, click the *Browse* button next to the variable field and specify the path. This is shown in the figure below.

© 2023 All Rights Reserved.

Figure 7-6 The Variables Form



You can also set the above variables in the batch mode by using the `set_variable` command.

Note: For descriptions of all the variables supported in Voltus-Fi-L, see the "Variables" chapter.

© 2023 All Rights Reserved.

- 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
  - **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. It is only supported in the `qrcTechFile` flow. Even if this analysis type is selected in the `emDataFile` flow, the corresponding plot types, JACPEAK and J_ACPeak, will not be available in the plot types list.
  - **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 EM Analysis. After the results are loaded, the 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, RC plot type is available for all EM analyses.

  - **RC - Resistor Current**: analyzes and reports peak resistor currents

  For details of the RC plots, see RC, RCAVG, and RCRMS – Resistor Current Plots, in the “IR Drop Analysis Results” chapter of the *Voltus-Fi Custom Power Integrity Solution XL User Guide*.

  Note: Once the results are loaded, the plot types are enabled. You can select from the available plot types. For more information about the different EM plots, see Types of EM Analysis Plots in the “EM Analysis Results” chapter of the *Voltus™-Fi Custom Power Integrity Solution XL User Guide*.

- Click *Clear Results* to clear the results.
- Click *All Power Nets* to view EM plots for all the power nets.
- Click *All Signal Nets* to view EM plots for all the signal nets.

© 2023 All Rights Reserved.

- 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.

  Note: The net name is case-sensitive.

- 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.

- 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.

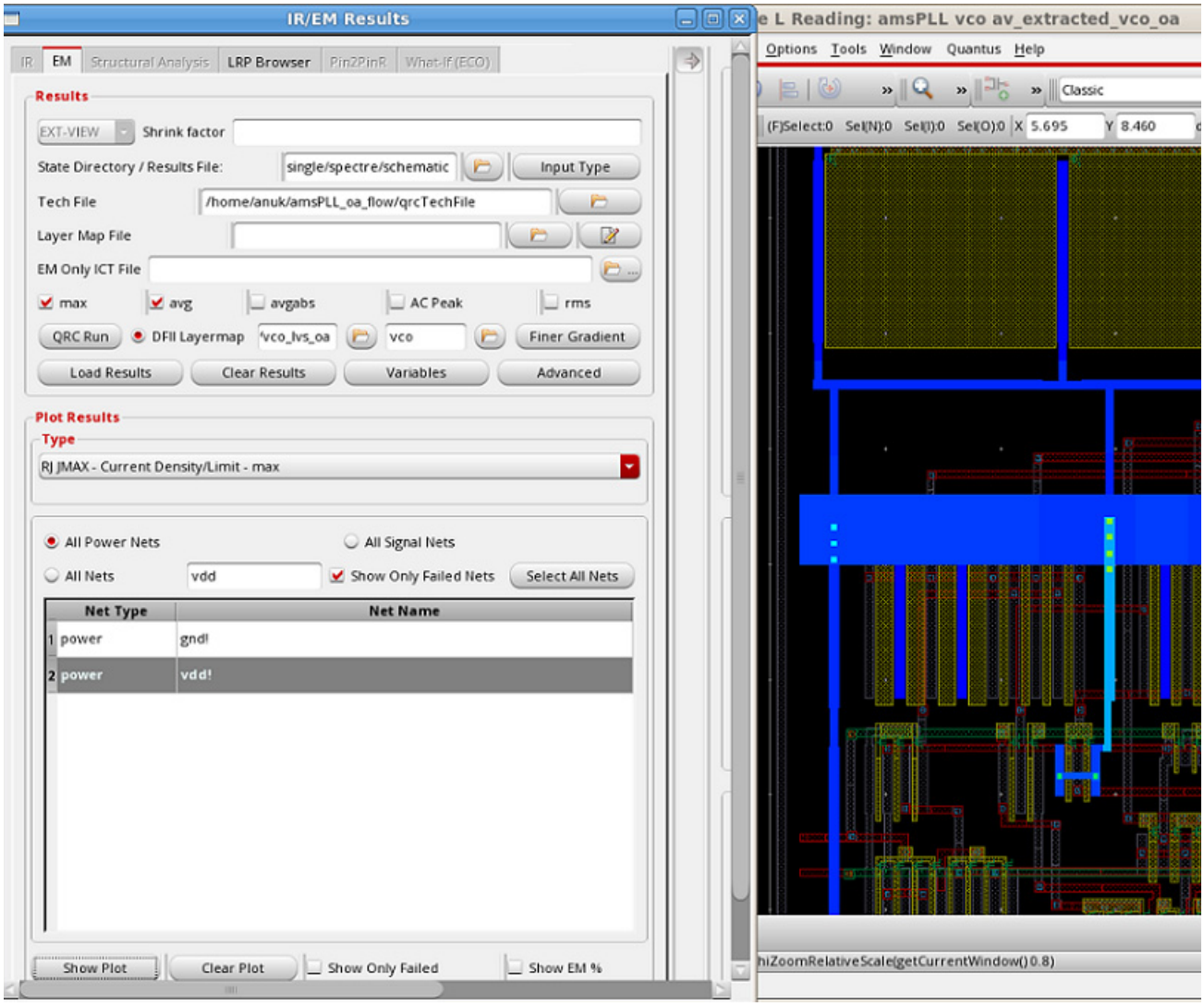
  For details, see Displaying and Querying EMIR Results in the “IR Drop Analysis Plots” chapter.

  Note: Tap Current plots are not currently supported in Voltus-Fi-L. A sample EM analysis plot is shown below.

- Click *Clear Plot* to clear the plot for the selected net from the Virtuoso layout. For this, select the net for which you want to clear the plot in the list box and then click *Clear Plot*.

© 2023 All Rights Reserved.

Figure 7-7 Current Density Plot in Voltus-Fi-L



Displaying and Querying EMIR Results

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

In addition, 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.

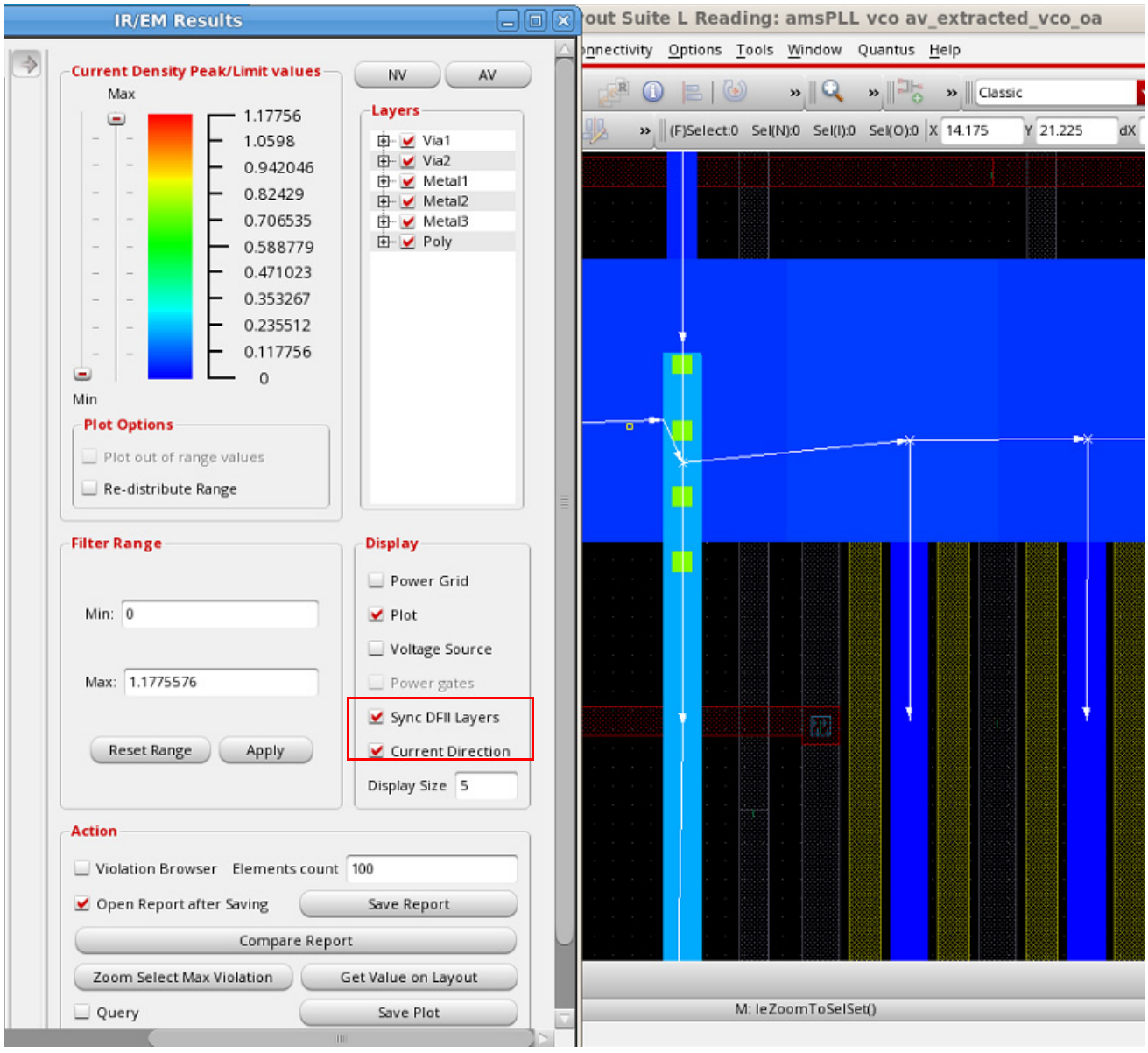
For detailed descriptions of the display options and the query feature, see Displaying and Querying EMIR Results in the “IR Drop Analysis Results” chapter of the *Voltus-Fi Custom Power Integrity Solution XL User Guide.*

© 2023 All Rights Reserved.

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.

Figure 7-8 Viewing the Current Direction



The following options are provided in the *Display* group box:

© 2023 All Rights Reserved.

- *Current Direction*: Check 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 plot honors the following user settings:

- Layer selection specified in the *Layers* group box
- The *Select Nets* option in the *Type* group box of the IR/EM Results form, and
- *Show Only Failed* check in the IR/EM Results form

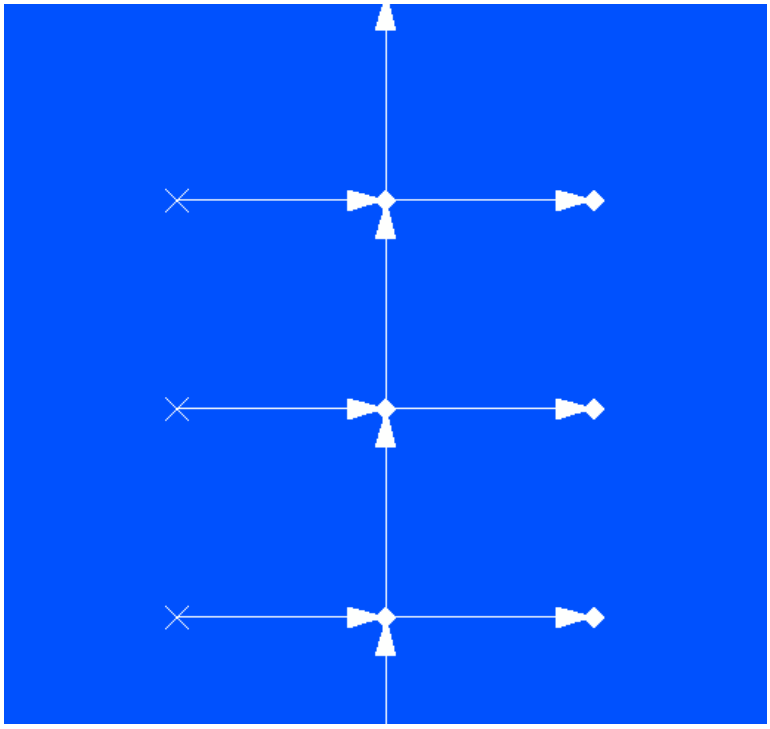
Note: The *Min/Max* slider range setting is not honored by the current direction plot. The plot displays current directions for all nodes.

- 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.

© 2023 All Rights Reserved.

Figure 7-9 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
  ```

For details, see Reporting the Direction of Currents in the EM Result File in the “EM Analysis Results” chapter of the *Voltus-Fi Custom Power Integrity Solution XL User Guide.*

© 2023 All Rights Reserved.

Converting emDataFile to ICT File for EM Analysis

Voltus-Fi supports the specification of an emDataFile as a technology file for performing EM analysis. The file can be specified in the following ways:

- In the batch mode by using the variable, `_vsa_em_tech_file`
- In the GUI mode, by specifying the path to the `emDataFile` in the *Tech File* field.

When the emDataFile is specified, the following files are generated in the current working directory:

- `toolgenerated.ict`, or the converted ICT file
- `additional.cmd` file
- `emdatafiletoict.log`, which is generated only in batch mode

The software uses the `toolgenerated.ict` for EM rules and the `additional.cmd` file for EM variable values for running EM analysis.

© 2023 All Rights Reserved.

Types of EM Analysis Plots

The different plot types available and the corresponding analysis types are listed below:

- RC plots – max, avg, avgabs, AC Peak, rms analyses
- RCAVG, RCRMS plots – max, avg, avgabs, AC Peak, rms analyses (only available in the `qrcTechFile` flow)
- RJ JMAX and J_MAX plots – max analysis
- JAVG and J_AVG plots – avg analysis
- JABSAVG and J_ABSAVG plots – avgabs analysis
- JACRMS and J_ACRMS plots – rms analysis

Important

TC or tap current plots are not currently supported in Voltus-Fi-L.

Note: JACPEAK and J_ACPEAK plot types are only available in the `qrcTechFile` flow. In addition to the above, the RC plots are common to all analysis types. However, the RCAVG, and RCRMS plots are only available in the `qrcTechFile` flow and the EM Only ICT File flow. They are not supported in the `emDataFile` flow.

For details of the above plots, see Types of EM Analysis Plots in the "EM Analysis Results" chapter of the *Voltus™-Fi Custom Power Integrity Solution XL User Guide*.

Generating EM Analysis Reports in Voltus-Fi-L

For details about the EMIR reports generated in Voltus-Fi-L, see the following sections in the "IR Drop Analysis" chapter of the *Voltus-Fi Custom Power Integrity Solution XL User Guide*:

- Reports Generated using the EMIR Control File
- Default Reports Generated for EMIR Analysis

Important

For details of the supported EMIR control file options that can be specified, see the EMIR Control File Options Supported in Voltus-Fi-L in the "Batch Mode Execution" chapter.

© 2023 All Rights Reserved.

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.

© 2023 All Rights Reserved.

8

EM Rules Specification

© 2023 All Rights Reserved.

EM Rule File Requirements

Before running EMIR analysis in Voltus-Fi-L, 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 of the *Voltus-Fi Custom Power Integrity Solution XL User Guide*.

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

- EM Rules in qrcTechFile
- EM Rules in EM Data File

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-L only supports parameters for limit-based EM analysis.

Parameters for Limit-Based Analysis

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

© 2023 All Rights Reserved.

Table 8-1 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. This parameter is similar to the viaWidthList parameter in emDataFile.
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 **Note:** 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.
em_jmax_dc_rms_metal_line_factor	Defines the relationship between the value of power on metal lines and the I_{rms} rating values for DC RMS analysis.
em_jmax_ac_rms_metal_line_factor	Defines the relationship between the value of power on metal lines and the I_{rms} rating values for AC RMS analysis.

Current Density (JMAX) Keywords

The table below lists the Current Density keywords that are supported in Voltus-Fi-L.

Table 8-2 Current Density Keywords

Keyword	Description
em_jmax_dc_avg*	Optional keyword that specifies the DC AVG analysis in Voltus-Fi-L
em_jmax_ac_avg*	Optional keyword that specifies the AC AVG analysis in Voltus-Fi-L
em_jmax_dc_peak*	Optional keyword that specifies DC PEAK analysis in Voltus-Fi-L

© 2023 All Rights Reserved.

Keyword	Description
em_jmax_ac_peak*	Optional keyword that specifies AC_PEAK analysis in Voltus-Fi-L
em_jmax_dc_rms*	Optional keyword that specifies DC RMS analysis in Voltus-Fi-L
em_jmax_ac_rms*	Optional keyword that specifies AC RMS analysis in Voltus-Fi-L

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

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_life <life1> <scale1> [<life2> <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-L. 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.

© 2023 All Rights Reserved.

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`^{2} or `mA` for metal layers (width based)
  - `A/via` or `mA` for via layers (per via)
- For a PWL table:
  - `A/cm`^{2} or `mA`, with width specified in microns for metal layers

© 2023 All Rights Reserved.

- `A/cm`^{2} or `mA`, with area specified in square microns for via layers

■ 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`.

If the specified temperature, `T`, falls between the defined minimum and maximum temperatures, the software will calculate a new scale factor (RT) using the formula, `TTF=A*J^(-n)*exp(Ea/kT)`.

For example:

```
RT = RT1*exp((Ea/kT)*(1/T-1/T1))
```

where

```
Ea/kT=(T1*T2/(T1-T2))*ln(RT2/RT1)
```

Note: If the specified temperature, `T`, is either below or above the defined minimum and maximum temperatures, the software will use the minimum and maximum temperature for calculating the scale factor, respectively.

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,

© 2023 All Rights Reserved.

`L` = length
`W` = Width
`Lu` = Upper metal length
`Lb` = Bottom metal length
`Wu` = Upper metal width
`Wb` = Lower metal width
`Lv2v` = Distance between terminal vias
`Itolerance` = Difference in the current values of terminal vias. For example, if a difference of up to 5 percent is acceptable, then specify '`Itolerance <= 0.05`'.
T_d = Time duration in micro second or total 'On Time" period
`r` = Duty ratio
`a` = Area of the single cut in the via or in the via array
`N` = Number of vias in the via array

Voltus-Fi-L 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, `Lu/Wu` refers to dimensions of the metal above the via and `Lb/Wb` refers to dimensions of the metal below the via.

- **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 Life**

  The keyword, `jmax_life` provides the ability to set the scaling factor that applies to the Current Density limits for different lifetimes. The syntax is as follows:

```
jmax_life <life1> <scale1> [<life2> <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 `Lb/Wb` 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 Lu/Wu for VIA Jmax factor

© 2023 All Rights Reserved.

- `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:

© 2023 All Rights Reserved.

```
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=="ppoly 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
  ```


© 2023 All Rights Reserved.

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 for both scenarios, when the variable is set and when it is not set.

Table 8-3 Rules for Power and Signal Nets

splitACDCRules false	splitACDCRules true	
Power and Signal Nets	**Power Nets**	**Signal Nets**
em_jmax_dc_peak	em_jmax_dc_peak	em_jmax_ac_peak
em_jmax_dc_avg	em_jmax_dc_avg	em_jmax_ac_avg
em_jmax_ac_rms	em_jmax_dc_rms	em_jmax_ac_rms

EM Rule Selection Priority

This section details the order of priority in which the EM rules are applied by Voltus-Fi-L 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

© 2023 All Rights Reserved.

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:

  - Match rule 1 because it has a matching `sub_conductor` keyword

© 2023 All Rights Reserved.

- 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

- `color`

© 2023 All Rights Reserved.

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* …
  ```

  Then the software follows the order of priority provided below:

© 2023 All Rights Reserved.

- 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
- When the variable, optimisticEMRuleSelection, is set to `true`, then rule 2 is selected because it is more optimistic as compared to the other rules

© 2023 All Rights Reserved.

- 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 Rules in 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.

Single line comments can be added by beginning the line with a semicolon (';'). For example:

```
; This is a comment line
```


© 2023 All Rights Reserved.

For a sample EM Data file, see EM Data File in the “File Formats” chapter of the *Voltus-Fi Custom Power Integrity Solution XL User Guide*.

EM Rules Specifications Supported in EmData File

The EM Rule specifications supported in Voltus-Fi-L are listed below:

- `currentDensityMPV`: Specifies the Current Density of via in milliamps per via instead of the standard milliamps per micron.
- `deltaT`: Specifies the 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, Voltus-Fi-L considers a default temperature of 5 degrees Celsius. This is an ideal condition where the design must be EM free.

  `deltaT` can be used in defining the Current Density limit: `Irms`

  For example,

  ```
  (nil layer "M1" minW 0.09 maxW 12.0 currentDensity
  (((sqrt(18.33*deltaT*(w-.016)*(w-.016)*(w-.016+.352)/(w-.016+.0443) )/w),
  110)))
  ```

- `recoveryFactor`: Calculates the effective DC electromigration in bipolar current wave forms. This accounts for some 'sweep back recovery' of EM during the negative portion of the current wave form. The recovery factor is specified in the EM Rules specification file as:

  `recoveryFactor=<`*`value`*`>`

  where the user-specified value ranges from 0 to 1.0.

  The following equation shows how the recovery factor is used by the software while calculating the total average EM current:

  ```
  I (avg) = max( I+, I- ) - (recovery factor)* min( I+, I-)
  ```

  where `I+` is the average of `I` in the positive direction and `I-` is the average of `I` in the negative direction

- `recoveryFactorList`: Specifies the list of layers and the corresponding recovery factors to be used for these layers during EM analysis. The following example shows how different recovery factors are specified for layers, `Via1` and `mt1`.

  ```
  recoveryFactorList=(("Via1" 0.7) ("mt1" 0.3))
  ```

  Note: If both `recoveryFactor` and `recoveryFactorList` are specified, the list of layers and their corresponding values specified using the `recoveryFactorList` keyword will get preference.

© 2023 All Rights Reserved.

- `cadGrid`: Specifies the minimum width of parasitic resistor. The unit of measurement is micrometer (`um`) or micron.
- `routingLayers`: The routing layers are the names of the DFII layers that are used for routing. Include only the poly and metals. An example follows:

  ```
  routingLayers = ("poly1" "m1" "m2" "m3" "m4" "m5" "m6" "m7" "m8" "m9" "MD")
  ```

- `viaLayers`: Specifies the DFII layer name of via layers.

  ```
  ( Metal1      Cont            Poly       )  // the via layer is Cont
  ```

  An example of the viaLayers syntax follows:

  ```
  viaLayers=("cw" "v1" "v2" "v3" "v4" "v5" "v6" "v7" "v8")
  ```

- `xrefLayers`: These are the cross-reference layers that provide mapping between the Quantus QRC layer name, Current Density name, and the DFII layer name. If a layer is not present in `xreflayers` mapping, its DFII name is used as the Current Density name and Quantus QRC name.

  ```
  (("QRCName" ("currentDensityName" "DFIIName"))
  ```

- `CurrentDensSpecList`: Refers to the Current Density specification declared by the foundry. The syntax allows one layer to support two different Current Density specifications according to the width of the material. The following Current Density specifications are provided in the emDataFile: `avgCurrentDensSpecList, avgAbsCurrentDensSpecList, rmsCurrentDensSpecList, DCrmsCurrentDensSpecList, peakCurrentDensSpecList, and ACpeakCurrentDensSpecList`

  The following optional parameters are supported:

  ```
  minW, maxW, minL, maxL, minLu, maxLu, minWu, maxWu, minWb, maxWb, minLb, maxLb
  ```

- Layer Stack: The layerstack contains a list of layer names and corresponding upper metal and bottom metal layer names.

  ```
  (("LayerName" ("upperMetalLayerName" "bottomMetalLayerName"))
  ```

  - *`LayerName`* is the layer name specified in the extracted view
  - *`upperMetalLayerName`* is the upper metal layer name in the extracted view
  - *`bottomMetalLayerName`* is the bottom metal layer name in the extracted view

- `viaWidthList`: Specifies the minimum via width. For example:

  ```
  viaWidthList=(("Cont" 0.2)("Via1" 0.2))
  ```

  Where,

  `Cont` is the DFII name of via layer.

  `0.2` is the minimum width of the default via for this layer.

© 2023 All Rights Reserved.

- `extViaWidthList`: Specifies the minimum via width. For example:

  ```
  extViaWidthList=(("extCont" 0.2)("extVia1" 0.2))
  ```

  Where

  `extCont` is the Quantus QRC (Extraction) name of via layer.

  `0.2` is the minimum width of the default via for this layer.

  Note: `viaWidthList` will be ignored if `extViaWidthList` is specified.

- `DesignResistors`: Specifies the list of design resistor cell names. For each cell name, you can specify the resistor property name, width property name, length property name, and units for width and length property names for which the design resistors are to be analyzed. Specifying the resistor property name, width property name, length property name, and unit name is optional. If any of these property names is not specified, the software uses “r”, “w”, “l”, and “meter”, respectively, as the default values for these properties.

  The width and length property names are specified when the names of these properties are different from the “w” and “l” in the extracted view.

  Syntax:

  ```
  DesignResistors = ("CellName1" "CellName2" "CellName3")
  DesignResistors = ("CellName.*")
  ```

- `junctionTemp`: Specifies the temperature to be used for EM analysis. Used as `tj` in ICT.

- `applyRThreshold`: Specifies the threshold ratio `"r"` for AC-Peak analysis. The default value of threshold ratio is `0.05`.

- `rmsMultiplierPin`: Specifies the RMS relaxation factor for EM power-grid analysis. The default value is `1.0`.

- `rmsMultiplierSignal`: Specifies the RMS relaxation factor for signal EM analysis. The default value is `1.0`.

EM Rules Specifications not Supported in Voltus-Fi-L

- `Jmaxlife`: Specifies the scaling factor that applies to the Current Density limits for different lifetimes.

- `BulkTerminal`: Specifies the bulk terminals of the design resistors that are to be ignored while performing EM analysis.

- `minI`: Used to limit the reporting of small currents which are of little or no interest.

© 2023 All Rights Reserved.

- `durationTime`: Specifies the maximum time period for which violations are allowed.
- `technologyNode`: Specifies the technology-dependent calculation.
- `viaLengthList`: Specifies the via length.
- `extViaLengthList`: Specifies the via length. It get preference over viaLengthList.
- `selectResistors`: Selects resistors by their names. All other resistors will be ignored for the analysis.
- `selectLayers`: Selects resistors based on the extracted layer name. All resistors on other layers will be ignored for the analysis.
- `ignoreLayers`: Ignores resistors based on the extracted layer name. All resistors on other layers will be selected for the analysis.
- `ignoreResistors`: Ignores resistors by their name. All other resistors will be selected for the analysis.
- `meshR`: Specifies regions for mesh resistance extraction.
- `fracture_vias`: Specifies whether or not there are dummy vias present.
- `Thickness`: Specifies the thickness of the layers.

© 2023 All Rights Reserved.

© 2023 All Rights Reserved.

9

SKILL Functions

© 2023 All Rights Reserved.

Overview

This section provides syntax, description, and examples for the Cadence® SKILL functions associated with the Voltus™-Fi Custom Power Integrity Solution (Voltus-Fi) 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.

The following functions are common to both VFI-L and VFI-XL. They are documented in the VFI-XL manual. The names of the functions and links to the document are provided below.

- vsaCreateMarker
- vsaLoadNets
- vsaSetLayersToMergeDuringResultsLoading
- vsaSetInputType
- vsaSetEMIRConfig
- vsaSetEMResultsFile
- vsaSetEMLayerMapFileName
- vsaSetEMTechFileName
- vsaSetEMTypes
- vsaSetDfiiLayerMapFileName
- vsaSetQRCData
- vsaSelectFailedNets
- vsaSetEMOnlyICTFile
- vsaSetEMPlot
- vsaLoadEM
- vsaSetIRResultsFile
- vsaSetIRThreshold
- vsaSetIRPlot
- vsaSetLRPShortedLayers

© 2023 All Rights Reserved.

- vsaLoadIR
- vsaSetFinerDisplayOff

© 2023 All Rights Reserved.

Voltus-Fi SKILL Functions

The following types of functions are provided only for Voltus-Fi L:

- vsaOpenSchematic
- vsaDescendView

© 2023 All Rights Reserved.

vsaOpenSchematic

```
vsaOpenSchematic(
     libname
     cellname
     viewname
     )
```

Description

Opens the design on the Virtuoso layout for running Voltus-Fi-L.

Arguments

libname	Specifies the library name for the design to be opened.
cellname	Specifies the cell name for the design to be opened.
viewname	Specifies the view name for the design to be opened.

Values Returned

None.

Example

```
vsaOpenSchematic("testlib" "testcell" "schematic")
```

The above example code shows how to specify the library name, cell name, and schematic view name of the design you want to open.

© 2023 All Rights Reserved.

vsaDescendView

```
vsaDescendView(
     libname
     cellname
     viewname
     )
```

Description

Descends in the hierarchy and opens the child view of the design in Virtuoso for running Voltus-Fi-L.

Arguments

libname	Specifies the library name for the design in which you want to descend.
cellname	Specifies the child cell name for the design in which you want to descend.
viewname	Specifies the child view name for the design in which you want to descend.

Values Returned

None.

Example

```
vsaDescendView("testlib" "testcell" "av_extracted")
```

The above example code shows how to specify the library name, child cell name, and child view name of the design in which you want to descend.

© 2023 All Rights Reserved.

10

Power-Grid View Creation

© 2023 All Rights Reserved.

The Power-Grid View Flow

Voltus-Fi-L 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 the power rail and power distribution information for each instance of the cell in the design for dynamic or static full-chip power-grid analysis.

For more information about PGVs, refer to the “Power-Grid Library Generation” chapter in the *Voltus IC Power Integrity Solution User Guide*.

© 2023 All Rights Reserved.

Data Requirements and Flow for PGV Creation

The following table shows the required inputs for generating power-grid views in Voltus-Fi-L. The inputs are categorized into the following: 1) from the design; and 2) from the user.

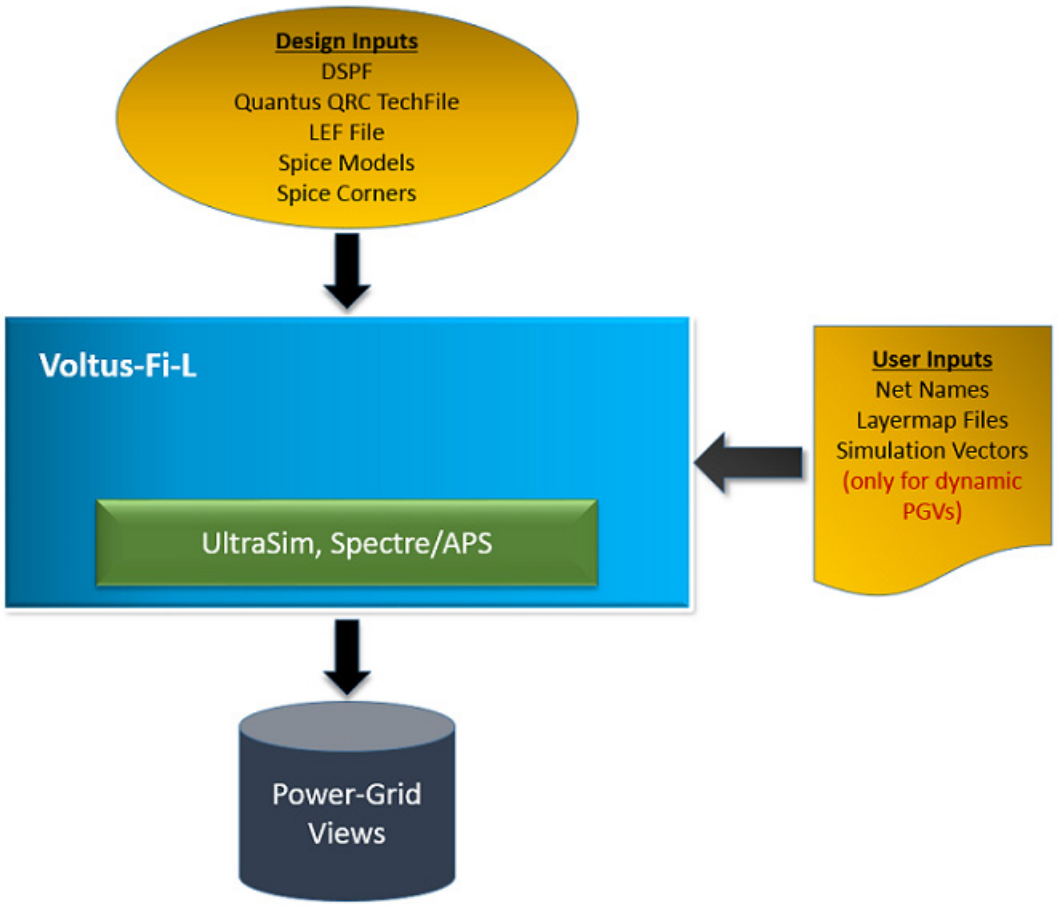
Type of Inputs	Required Inputs	Details
Design Inputs	`Detailed Standard Parasitic Format (DSPF)`	Required for providing the following information: ■ All power ground nets extracted ■ Layer information with each resistor ■ x, y coordinates of the parasitic resistors ■ A valid layermap external file that lists all layers ■ Lumped C of the signal nets
	`Spice models and corresponding spice corners`	Required for SPICE simulation
	`QRC Techfile`	Required for power-grid extraction
	`LEF file`	Required for port connectivities
User Inputs	`Net names`	Required for extraction and pin information
	`DSPF layermap file`	Required to provide information about the order and type of layers in the DSPF file.
	`LEF layermap file`	Required for providing mapping between the layer names in the LEF file and the layer names in the technology file.

© 2023 All Rights Reserved.

Type of Inputs	Required Inputs	Details
	`Stimulus file`	Required for providing information about simulation vectors, start time/stop time, and the time interval for running the simulation. **Note:** This is required only for dynamic PGV generation.

The following diagram illustrates the flow of PGV creation in Voltus-Fi-L.

Figure 10-1 Power-Grid View Generation Flow in Voltus-Fi-L



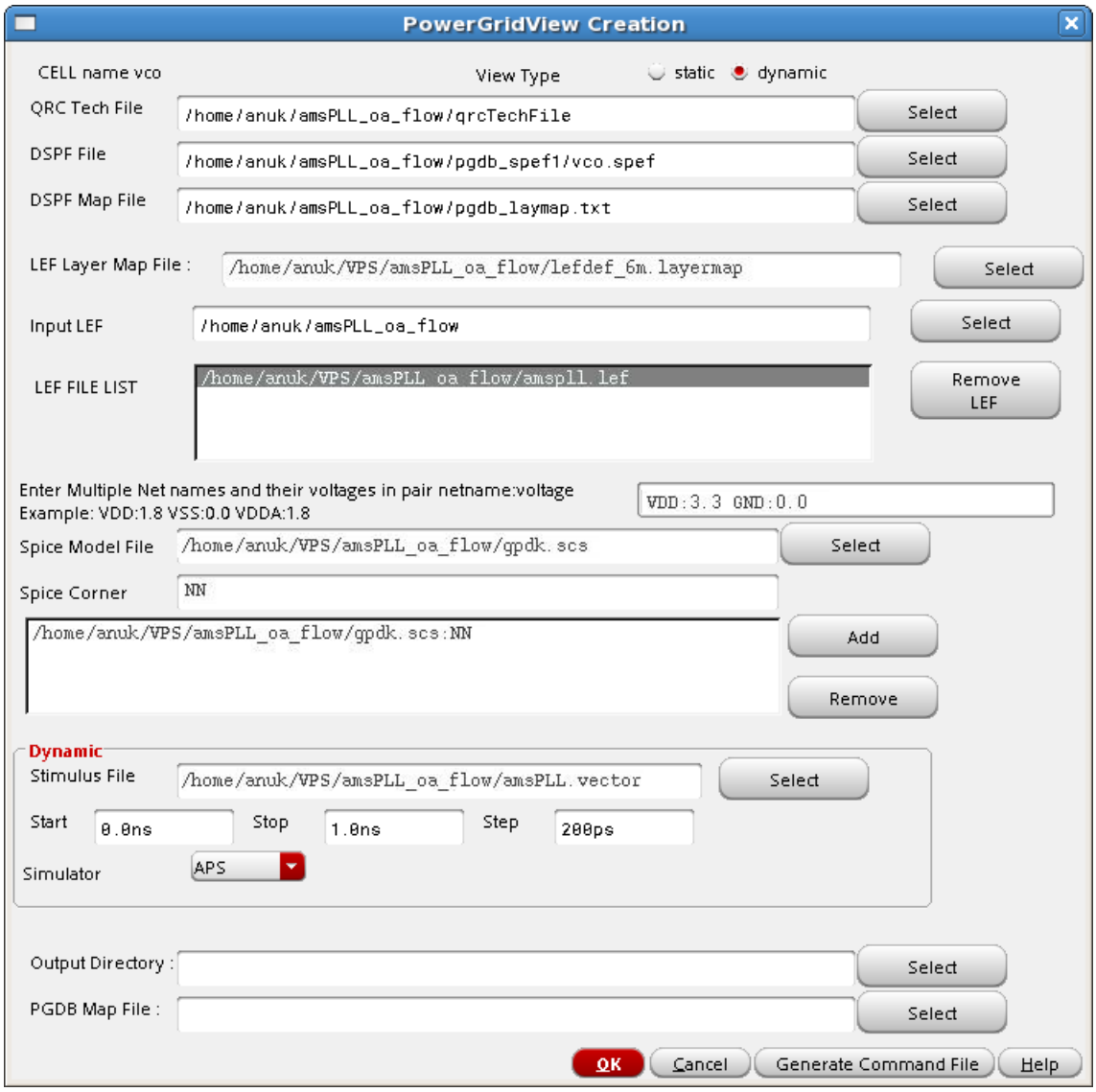
© 2023
All Rights Reserved.

Creating Power-Grid Views

To create PGVs in Voltus-Fi-L, perform the following steps:

- In the Voltus-Fi-L console, select *IR/EM Analysis – Create Power Grid View.*
- The PowerGridView Creation form opens. This form is shown below.

Figure 10-2 The PowerGridView Creation Form



- The *CELL name* field is already populated with the name of the cell for which the power-grid view is being created. This is chosen from the open extracted view.

© 2023
All Rights Reserved.

- In *View Type*, choose either *static* or *dynamic*, to specify the type of analysis for which you want to generate the view.
- Select the *QRC Tech File*, which is the name of the Quantus QRC technology file that will be used for PGV generation.
- Select the *DSPF File*, which is the third party or the xDSPF netlist file that is to be imported.
- Select the *DSPF Map File*, which is the layermap file that has the layer type and layer stack information of the DPSF file, from the top-most layer to the bottom-most layer. The Layer type column specifies the type of layer: metal, via, or local; while the Process layername column specifies the name of the layer. To view an example, see DSPF Layermap File in the "File Formats" chapter.

  Note: In the DSPF map file, ensure that the layer information is provided in the top-to-bottom order. If this order is not followed, the power-grid database (PGDB) will not be created.
- Specify the *LEF Layer Map File*, which contains information for mapping the layer names in the LEF file to the layer names in the technology file. To view an example, see LEF Layermap File in the "File Formats" chapter.
- Select the technology LEF files and the Macro LEF files in the *Input LEF* field. The technology LEF files contain the LEF information for the technology data, and the Macro LEF files contain the LEF data for the design. The selected filenames will appear in the *LEF FILE LIST*. Click *Remove LEF* to remove any file from the *LEF FILE LIST*.
- In the *Enter Multiple Net names and their voltages in pair netname:voltage* field, type the names of the nets and their corresponding voltages in the specified format. For example, `VDD:1.8 VSS:0.0`. You can specify multiple net names separated by a space.
- To specify the Spice Models information, perform the following steps:
  - Select the *Spice Model File* used by the SPICE netlist.
  - Type the corresponding *Spice Corner* to be used from the model file.
  - Click *Add* to add the spice model – spice corner pair to the table.
  - Continue till all models and their corresponding corners are specified.
  - Click *Remove* to remove a pair from the table.
- The Dynamic group box is activated only when you choose the *View type* as *dynamic*. In this group box, provide the specifications for dynamic PGV generation.

© 2023 All Rights Reserved.

- Select the *Stimulus File*, which is the input file for the simulator. Choose this option to specify the vector information to be used for the simulation.
- Specify the *Start, Stop* time in nanoseconds (`ns`), and *Step Size* in picoseconds (ps).This specifies the simulation start time, stop time, and step size. The default value of step size is 200 `ps`.
- Choose APS in the *Simulator* cyclic field.

- Select the *Output Directory* to write the output data. 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.

- Select the *PGDB Map File*, which is the layermap file that 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. To view a sample file, see PGDB Layermap File in the “File Formats” chapter.

- Click *OK*.

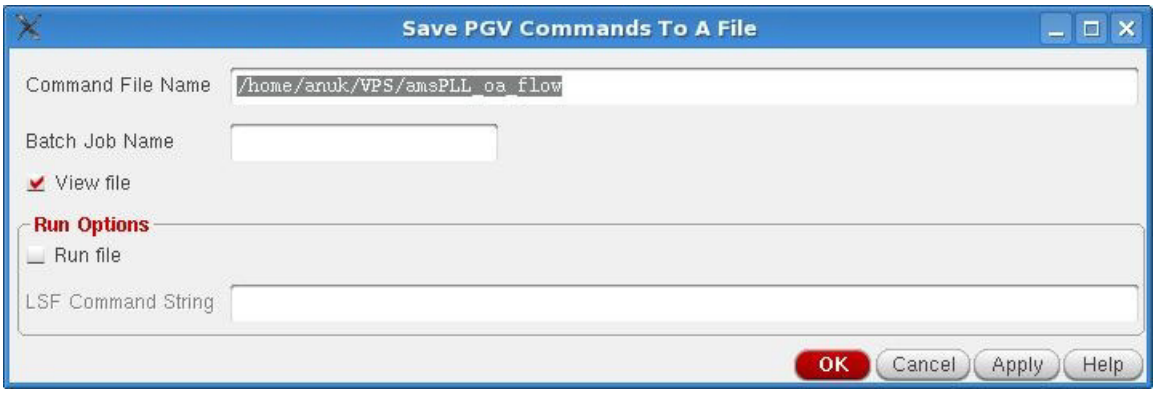
© 2023 All Rights Reserved.

Writing Out a PGV Command File

To generate a batch mode command file for PGV generation, follow these steps:

- In the PowerGridView Creation form, click *Generate Command File*. The Save PGV Commands To A File form opens. This form is shown below.

Figure 10-3 PGV Command File Generation



- In this form, specify the *Command File Name*. If the command filename is not specified, the software either uses `<Batch job Name>.cmd` as the default command file name, or if the batch job name is not provided, then it uses `vsaPGVBatch.cmd` as the default command filename.
- Specify the *Batch Job Name*, which is a unique job name to distinguish between different jobs. This allows you to run multiple command files – generated by different job names – in the same working directory.
- Click *View file* to view the command file. This option is selected by default. If you do not want to view the command file, you can change the default selection of this option by setting the `_vsaUncheckViewFile` environment variable to `true`. For details, see _vsaUncheckViewFile in the “Environment Variables” chapter.
- In the *Run Options* group box, click *Run file* to run the generated command file in the background.
- Specify the Local Server Farm or the *LSF Command String*, which is the name of the wrapper string command that will launch the tool in the batch mode. For example, you can specify bsub -q lnx64 as the LSF command string. You can also specify the LSF string by using the environment variable, `_vsaLSFMachineString`. When this environment variable is set, the *LSF Command String* field is pre-filled with the specified command. For details, see _vsaLSFMachineString in the “Environment Variables” chapter.

© 2023 All Rights Reserved.

- Click *OK*.

Outputs of PGV Generation

Following are the outputs of PGV generation in Voltus-Fi-L:

- 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.

Sample Batch Command file

A sample PGV batch command file is shown below:

```
;vsa batch mode command file
;commands for PGV generation
_vsa_testbench_lib_cell_view "amsPLL" "TB1_vco_single" "schematic"
_vsa_extracted_lib_cell_view "amsPLL" "vco" "av_R"
_vsa_pgv_output_directory "dyn_pgv"
_vsa_pgv_tech_file "qrcTechFile"
_vsa_pgv_lef_layer_map_file "leflayermap"
_vsa_pgv_pgdb_map_file "pgdb.map"
_vsa_pgv_net "vdd!" " 1.2"
_vsa_pgv_net "gnd!" "0.0"
_vsa_pgv_lef_files "amspll.lef tech.lef"
_vsa_pgv_dynamic_options "amspll.vector" "0.0ns" "120.0ns" "250ps"
_vsa_pgv_spice_model "cln90g_lk.l" "TT tt_hvt"
_vsa_pgv_spice_model "cln90g_sr.l" "TT_sr"
_vsa_pgv_dspf_file "vco.xdspf"
_vsa_pgv_dspf_layermap "layermap_dspf"
_vsa_create_pgv
exit
```


© 2023 All Rights Reserved.

© 2023 All Rights Reserved.

11

Viewing Results in vsaplot or vfiplot

© 2023 All Rights Reserved.

© 2023 All Rights Reserved.

Overview

The vsaplot/vfiplot utility is used to view the results of the IR drop and EM analysis that is performed in Voltus-Fi, Voltus-Fi-L (VFI-L) and Voltus-Fi-XL (VFI-XL), respectively.

The functionality offered by vsaplot and vfiplot is essentially the same, with the following key differences:

- The vfiplot utility requires a VFI_XL license checkout, whereas vsaplot can be run using either the VFI_L or the VFI_XL license.
- The vsaplot utility provides analysis of only those results that are created using the VFI-L flow whereas vfiplot provides analysis of results created using both, VFI-L and VFI-XL.
- The Relative-R feature, which is used to calculate the relative resistance between nodes, is supported in vfiplot but not supported in vsaplot.

  Note: This document refers to the utility as vsaplot for all common features and a distinction is made for the Relative-R feature that is applicable only to vfiplot.

The vsaplot utility has its own display area, menus, and windows that provide options for customizing the display of IR drop and EM violations in the main display area of the tool.

The key benefits of viewing the results of IR drop and EM analysis in vsaplot are as follows:

1. **No dependency on the Virtuoso layout**: vsaplot lets you view the results of the analyses in its own display area, which means, you do not need the Virtuoso layout for displaying the plots.
2. **Faster display**: In vsaplot, the results are loaded and displayed quickly and when you change your selections, the plots are updated quickly. This performance enhancement is seen because vsaplot is independent of Virtuoso. Therefore, the time that would otherwise be spent in calling the Virtuoso APIs to display the results on the Virtuoso layout is saved.

© 2023 All Rights Reserved.

Launching vsaplot

Launching the vsaplot involves the following steps:

- Generating the VSA_RESULTS Database in Voltus-Fi
  - Generating the VSA_RESULTS Database in the GUI Mode
  - Generating the VSA_RESULTS Database in the Batch Mode
- Launching vsaplot from the Shell

Generating the VSA_RESULTS Database in Voltus-Fi

Before launching the vsaplot, you need to generate the vsaplot database or the `VSA_RESULTS` directory. This can be done both from the Voltus-Fi GUI and batch mode. Both methods are detailed below.

Generating the VSA_RESULTS Database in the GUI Mode

- Set the following environment variable before launching Voltus-Fi:

  ```
  setenv VFI_VSAPLOT_DB 1
  ```

- Launch Voltus-Fi and open the IR/EM Results form. Provide the inputs listed below for generating the `VSA_RESULTS` database for the IR drop analysis or EM analysis results on the IR and EM tabs, respectively.
  - The *State Directory/Results File,* which is the path to the simulation directory that stores the results of the simulation in the Voltus-Fi-L flow and a file containing the EMIR simulation result data in the Voltus-Fi-XL flow.
  - The *DFII Layermap* file, which contains the mapping between the extracted view and the DFII layer names in the Voltus-Fi-L flow and the xDSPF and the DFII layer names in the Voltus-Fi-XL flow.
  - The *QRC Run,* which includes the Quantus (QRC) *Run Directory* and the *Run Name.*
  - For generating the `VSA_RESULTS` for EM analysis:
    - *The Tech File*, or the technology file, which can either be the `emDataFile` or the `qrcTechFile`.

  Note: Both, the DFII layermap file and QRC Run information, is required in Voltus-Fi-L but only one of them needs to be specified for Voltus-Fi-XL.

© 2023 All Rights Reserved.

- Click *Load Results*. In the GUI mode, the plots for all the analysis types that are selected on the EM tab of the IR/EM Results form while loading the results will be available in vsaplot. The following analysis types are available on the EM tab; `max`, `avg`, `avgabs`, `AC Peak`, and `rms`.
- The `VSA_RESULTS` directory is created in the current directory containing the IR drop and EM analysis results.

Generating the VSA_RESULTS Database in the Batch Mode

This section provides details of the commands used to generate the vsaplot db directory or the `VSA_RESULTS` directory. This directory is then used to run the vsaplot.

To generate the `VSA_RESULTS` directory, run the following command:

```
vfibatch -cmd <cmd_file_name>
```

where the *<cmd_file_name>* contains the name of the file with various commands.

Note: No environment variable is required for generating the `VSA_RESULTS` directory when running in the batch mode.

The following commands are used to generate the `VSA_RESULTS` directory for IR drop and EM analysis in vsaplot for both Voltus-Fi-L and Voltus-Fi-XL. However, there are some parameters that are specific to either Voltus-Fi-L or Voltus-Fi-XL. The distinction is provided in the parameter descriptions.

- write_ir_vsaplot_db
- write_em_vsaplot_db

Specifying the Order of Layers in vsaplot

By default, the layer order used for viewing results in vsaplot is the same as that in the PGDB Layermap file. However, you can specify the order of layers for viewing in vsaplot by specifying the layer order in a file, `control_layer` file. In this file, the layer name and order is specified in the following format:

```
layerName <Layerorder>
```

For layers that are not to be displayed, use the following format in the file:

```
layerName ~
```


© 2023 All Rights Reserved.

After generating the `control_layer` file, specify the following environment variables before launching vsaplot.

- VSAPLOT_USE_DEFAULT_LAYER_ORDER – Uses the original layer order created by Voltus-Fi-XL and Voltus-Fi-L in VSA_RESULTS directory.

  OR

- VSAPLOT_EXTERNAL_LAYER_ORDER_FILE_PATH – Uses the control_layer file at the specified location.

For details, see VSAPLOT_USE_DEFAULT_LAYER_ORDER and VSAPLOT_EXTERNAL_LAYER_ORDER_FILE_PATH in the “Environment Variables” chapter.

The software will follow the following order of priority while searching for the control_layer file:

- If VSAPLOT_USE_DEFAULT_LAYER_ORDER is specified, use the default order.
- If VSAPLOT_EXTERNAL_LAYER_ORDER_FILE_PATH is specified, use the file at the specified location.
- If neither of the above variables are specified, search for the control_layer file in the default location, VSA_RESULTS directory
- If no control_layer file is available, use the layer order from previous session, if available
- If the above is not available, use the default layer order

Note: If the control_layer file is provided by the user in one run, the tool saves the layer order and will use the file for the subsequent vsaplot/vfiplot runs.

© 2023 All Rights Reserved.

Launching vsaplot

You can launch vsaplot in two ways, from the *Voltus-Fi* console menu, or directly from the UNIX, shell, or xterm window.

Note: While vsaplot can be enabled by checking out either the VFI-L or the VFI-XL license, vfiplot requires a VFI-XL License checkout.

Launching vsaplot from the Shell

You can open vsaplot directly from the UNIX, shell, or xterm window as follows:

`vsaplot <VSA_Results_DB>`

or

`vfiplot <VSA_Results_DB>`

Where:

`<VSA_Results_DB>` is the path of the result directory in which the results of the IR drop or EM analysis are saved.

Note: You can also run vsaplot/vfiplot by typing just `vsaplot` or `vfiplot` in the shell without specifying the results directory. You can then click the *File* menu and *Open* submenu in the vsaplot window to specify the results directory. This is detailed in the File Menu section.

Launching vsaplot from the GUI

You can launch the vsaplot from the IR/EM Analysis menu in the Voltus-Fi console in the following ways:

1. Click the *Connect To vsaplot* submenu of the IR/EM analysis menu. When you click this option, the Connect pop-up window opens. Provide the name of the *MW Server Port* and click *Connect*. This is shown in the images below.

   Note: For vfiplot, the submenu name is *Connect to vfiplot.*

© 2023 All Rights Reserved.

Figure 11-1 Launching vsaplot from IR/EM Analysis – Connect To vsaplot/vfiplot Menu

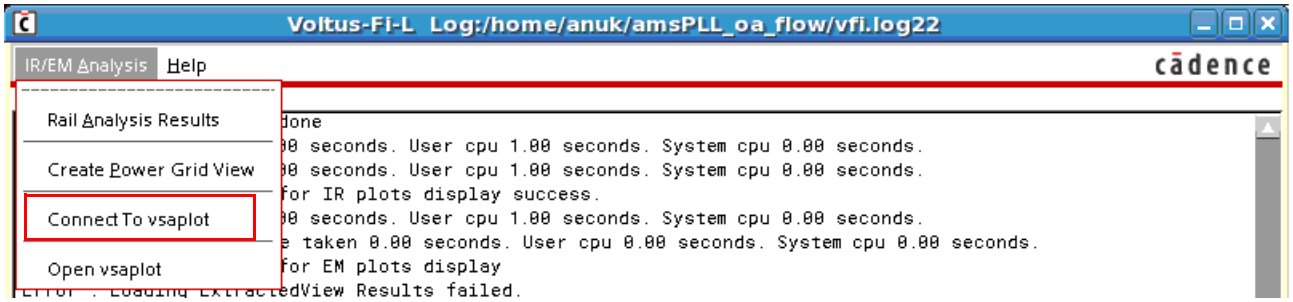
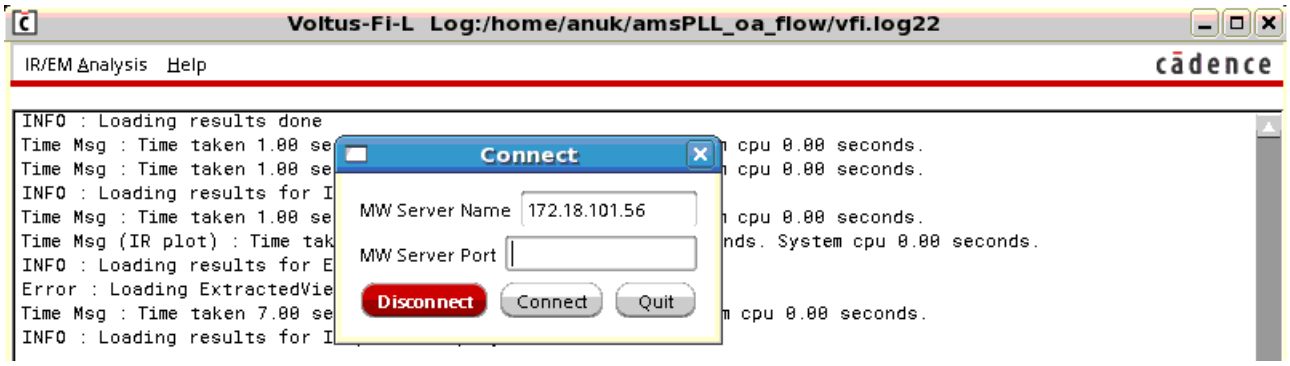


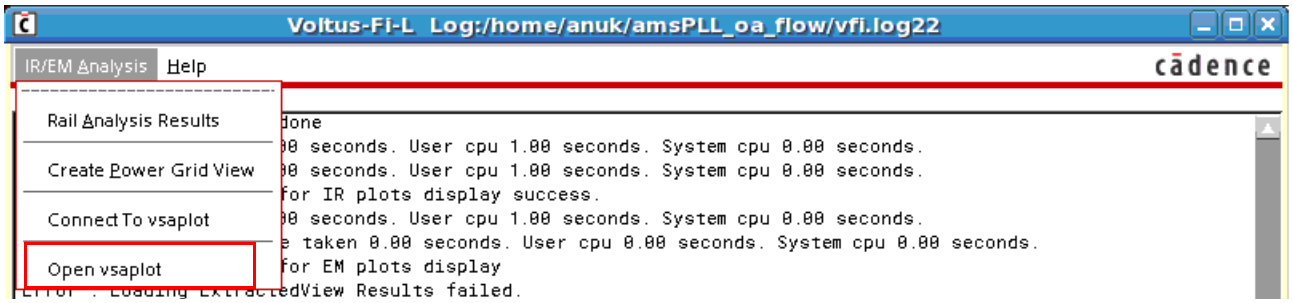
Figure 11-2 Opening the Connect Pop-Up Window



2. Click *Open vsaplot* submenu of the *IR/EM Analysis* menu. When you click this option, the vsaplot window opens.

   Note: For vfiplot, the submenu name is *Open vfiplot*.

Figure 11-3 Launching vsaplot from IR/EM Analysis – Open vsaplot/vfiplot Menu



© 2023 All Rights Reserved.

Viewing Results in vsaplot

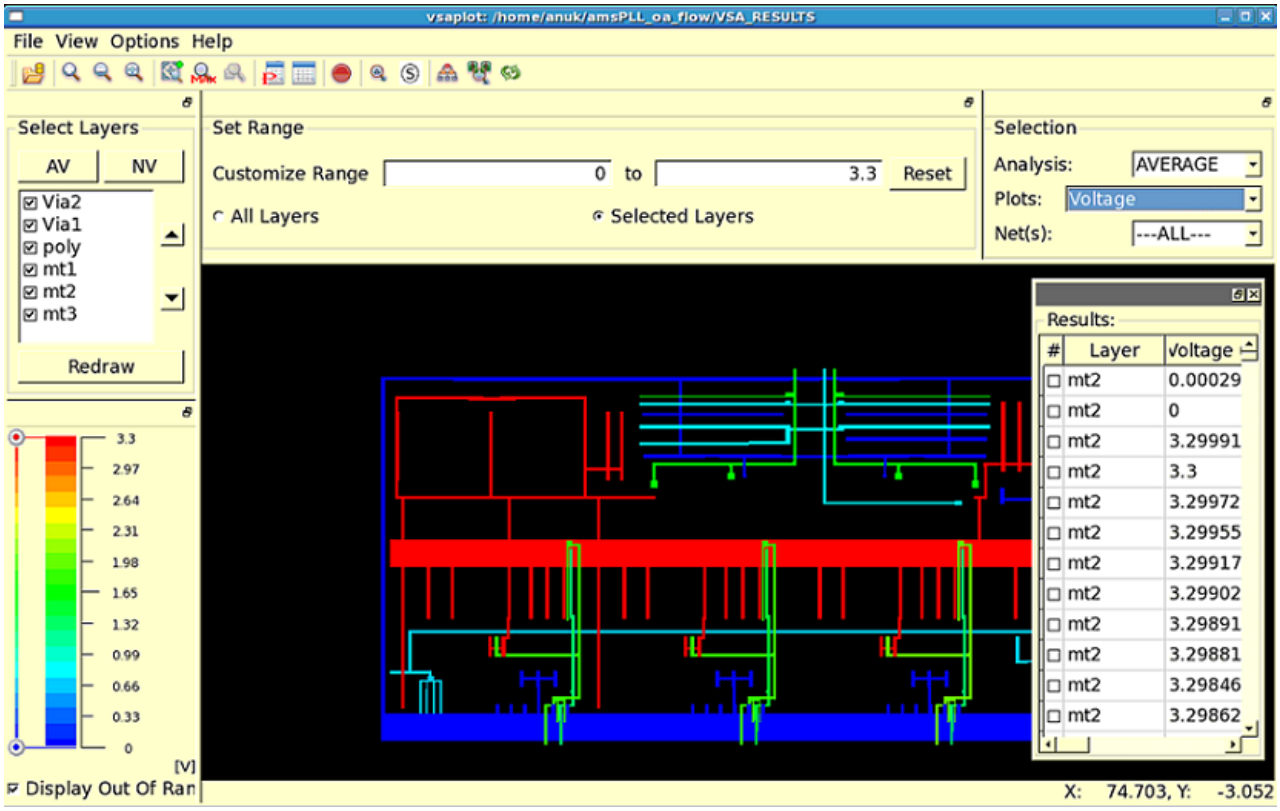
When you launch vsaplot either from the GUI or in the batch mode, the vsaplot window opens. Before detailing the options available for viewing and customizing results in vsaplot, let us first look at the main menus, submenus, and components of vsaplot.

The following topics are covered in this section:

- Main Menus of vsaplot
- Toolbar Widgets
- Main Components of vsaplot

The main window of the vsaplot is shown in the figure below.

Figure 11-4 The vsaplot Window



© 2023
All Rights Reserved.

Main Menus of vsaplot

The following main menus are available in vsaplot.

- File
- View
- Options
- Help

File Menu

The File menu has the following submenus:

- Open
- Sync
- Save
- Quit

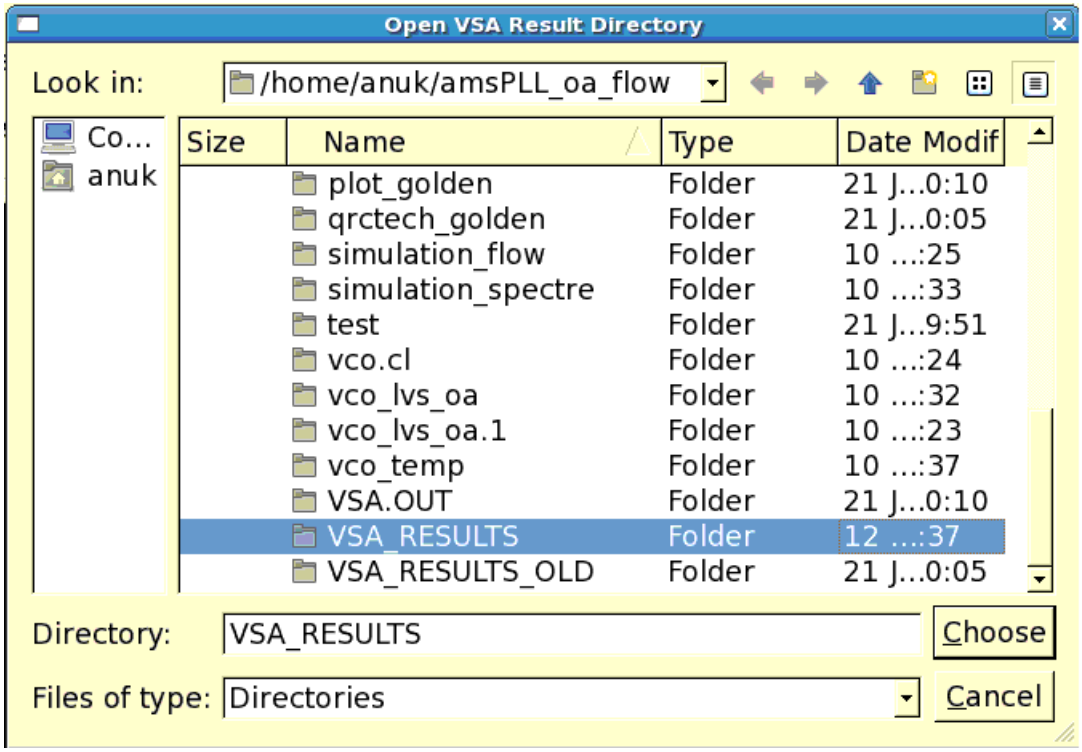
These submenus are detailed below.

- The *File – Open* submenu provides the following options:
  - *Open Results*: Opens the VSA Result Directory pop-up window. Select the results directory and click *Choose*.

    Note: You do not need to specify the result directory here if you have already specified it while launching the tool using the `vsaplot <Result_DB>` or the `vfiplot <Result_DB>` command in the shell.

© 2023 All Rights Reserved.

Figure 11-5 The Open VSA Result Directory Form

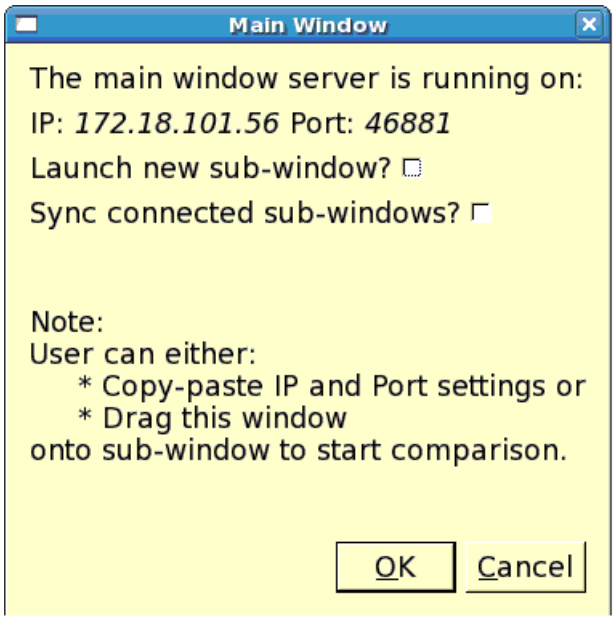


The extracted view or the layout view of the design will open in the display area or the layout area of vsaplot.

- *Open New Window*: Opens the VSA Result Directory pop-up window. Select the results directory and click *Choose.* This opens a new vsaplot window for the specified results directory.
- *Open New Window and Sync*: Opens the Main Window pop-up window that lets you launch a sub-window and synchronize connected windows as shown below.

© 2023 All Rights Reserved.

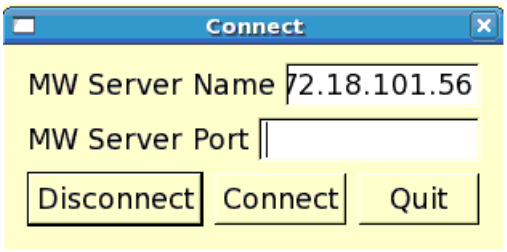
Figure 11-6 The Main Window



- *Open New Window and Close*: Opens the VSA Result Directory pop-up window. Select the results directory and click *Choose.* This opens a new vsaplot window for the specified results directory and closes the already open window.

- The *File – Sync* submenu provides the following options:
  - *Connect*: Opens the Connect pop-up window that lets you provide the IP address and the port number of the window you want to connect to the current window.

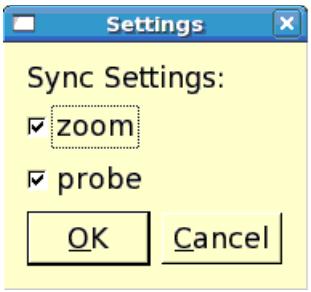
Figure 11-7 The Connect Window



  - Settings: Opens the Settings pop-up window that lets you specify whether you want to synchronize zoom, probe, or both for the connected windows.

© 2023 All Rights Reserved.

Figure 11-8 The Settings Window



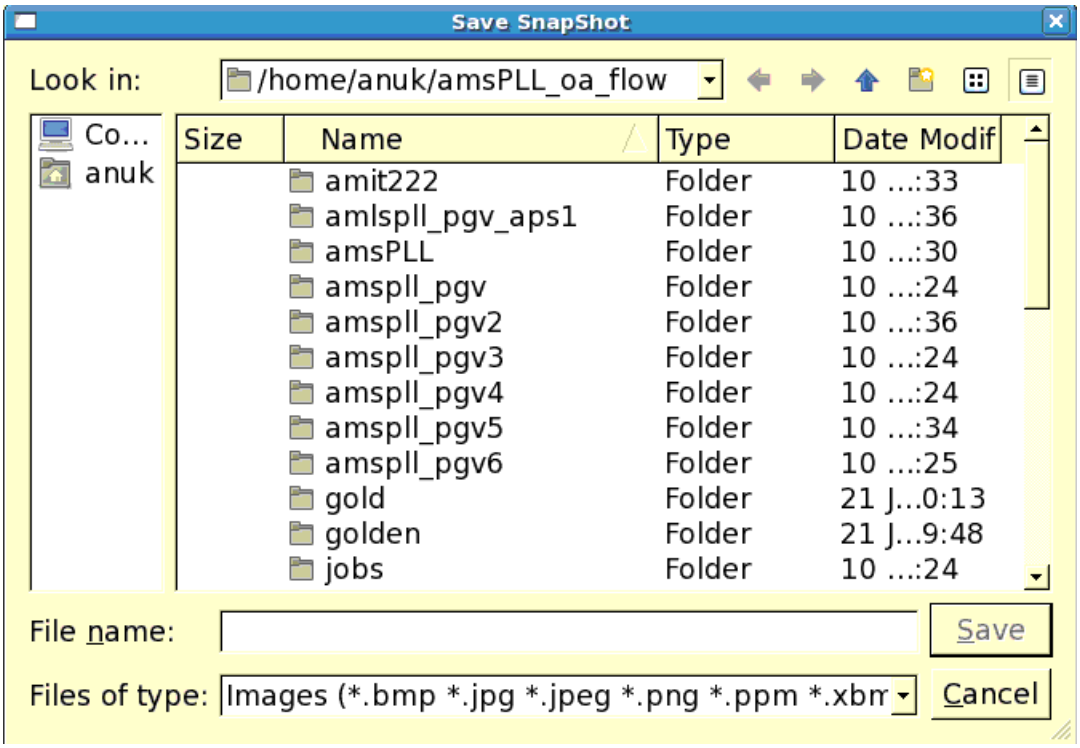
For details of the synchronism function in vsaplot, see Synchronizing Multiple vsaplot Windows.

- The *File – Save* submenu provides options to save a snapshot of the vsaplot window. The following options are available:
  - *Window Region*: Saves a snapshot of the entire window.
  - *Image Region*: Saves a snapshot of the display area of vsaplot.

    Note: When you select either of the two options, the Save Snapshot form opens. Specify the name and the format in which you want to save the image and click *Save*. This form is shown below.

© 2023 All Rights Reserved.

Figure 11-9 The Save Snapshot Form



- The File – *Quit* submenu closes the vsaplot window.

View Menu

The View menu has the following submenus:

- zoom
- Results
- Region2Region
- RelativeR

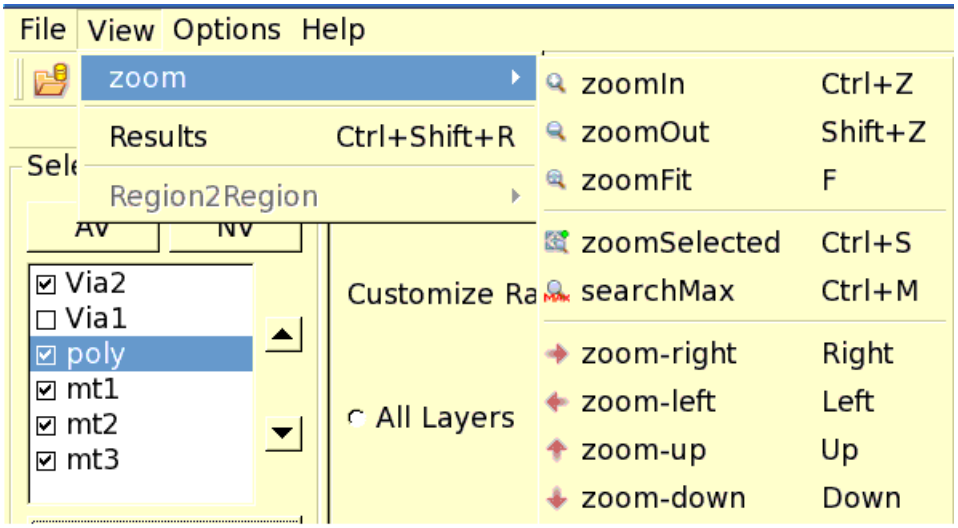
Note: The *RelativeR* menu is only available for `vfiplot`. It is not available in `vsaplot`.

These options are detailed below.

- The *View – zoom* submenu provides options to zoom into or out of specific areas of the design in the display area. The available options and their corresponding keyboard shortcuts are shown in the figure below.

© 2023 All Rights Reserved.

Figure 11-10 The View Menu of vsaplot



- *zoomIn*: Use this option to zoom into a portion of the design and view it in greater detail.

  You can also use the scroll wheel to zoom into a portion of the design. Each forward scroll of the scroll wheel zooms in one level.

- *zoomOut*: Use this option to display a larger area of the design in less detail. Each click zooms out two levels.

  You can also use the scroll wheel to zoom out of a detailed view of the design. Each backward scroll of the scroll wheel zooms out one level.

- *zoomFit*: Use this option to fit the entire design within the design display area.

- *zoomSelected*: Use this option to zoom into a specific selected area of the design.

- *searchMax*: Use this option to zoom into the worst violation in the selected area of the design.

  Note: The options, *zoomSelected* and *searchMax* are mutually exclusive.

- *zoom-right*: Use this option to pan the viewable window to the right. The equivalent bindkey is the Right arrow key.

- *zoom-left*: Use this option to pan the viewable window to the left. The equivalent bindkey is the Left arrow key.

- *zoom-up*: Use this option to pan up the viewable window. The equivalent bindkey is the Up arrow key.

© 2023 All Rights Reserved.

- *zoom-down*: Use this option to pan down the viewable window. The equivalent bindkey is the Down arrow key.

- The *View – Results* submenu provides the option to hide or unhide the Results window in vsaplot. By default, this option is checked and the Results window is displayed above the display area.
- The *View – Region2Region* submenu provides the option to view and compare results between specified regions in the display area of the vsaplot. Select *Point* to specify two points on the layout or specify *Rectangle* or *Ellipse* to specify regions as rectangular shapes or ellipses, respectively. This option is enabled only if a particular net is specified in the *Net(s):* field. It is disabled if ALL nets are selected.

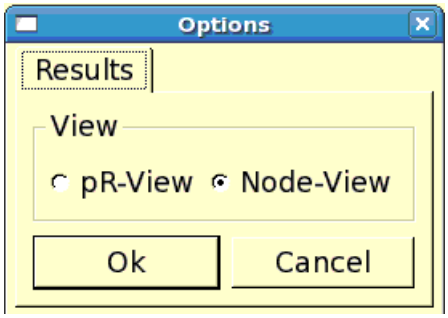
  For details, see Displaying the Colormap for Point-to-Point and Region-to-Region Selection.

Options Menu

When you click the *Options* menu, the Options form shown below opens. You can use this form to specify whether you want to view the analysis results in the pR-View (presistor view) or the Node-View (node view).

Note: The *Options* menu is available only if your results directory has both IR drop and EM analysis results. If the results directory has only IR drop analysis results, the *Options* menu is not available in the main menu of vsaplot because the IR drop violations only have a node view.

Figure 11-11 The Options Form

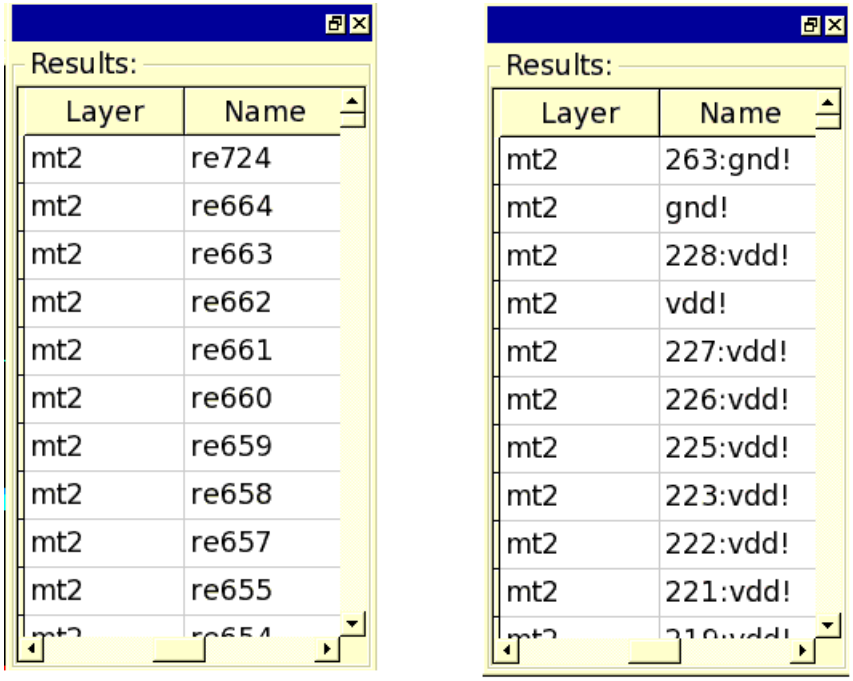


By default, the Results window displays results in the node view.

When you change your view selection, the Results window is updated accordingly. The updated “Name” column in the Results window for both views is shown below.

© 2023 All Rights Reserved.

Figure 11-12 Viewing Results in Presistor View and Node View



Results in Presistor view **Results in Node view**

Toolbar Widgets

The following row of widgets, located below the menus and above the design display area, includes shortcuts for the menu options detailed in the above section.

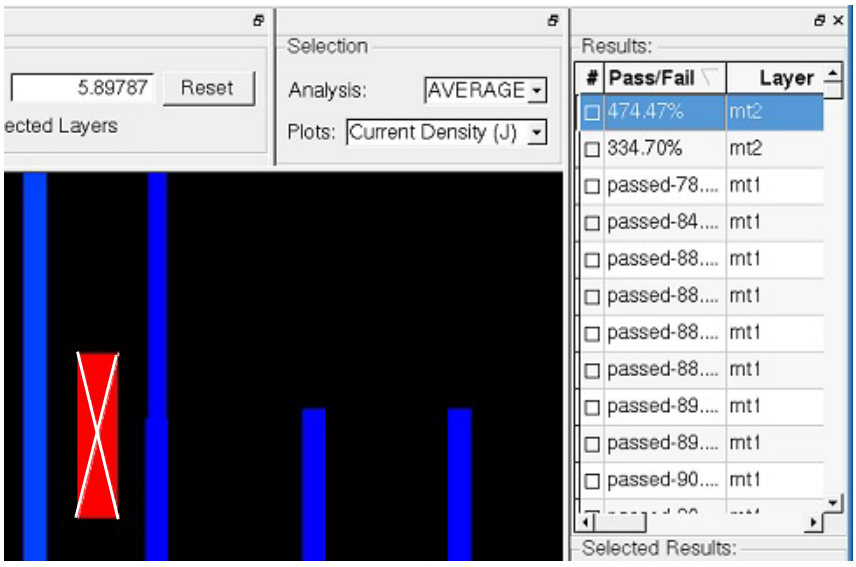
The descriptions of the widgets are provided in the table below.

© 2023 All Rights Reserved.

Table 11-1 Toolbar Widgets of vsaplot

Widget	Description
	Open – opens the result directory.
	zoomIn – zooms into the design in the display area.
	zoomOut – zooms out of the design in the display area.
	zoomFit – fits the entire design within the design display area.
	Search Maximum Value – zooms into and highlights the worst violation in the selected area of the design in the display area.
	Enables region-to-region mapping. **Note:** This widget is enabled only if a particular net is specified in the *Net(s):* field. It is disabled if ALL nets are selected.
	Partial/Full Results – updates the Results window to show partial or full results. In partial view, only the Pass/Fail, Layer, and J/Jmax columns are displayed. In full view, all the columns are displayed. This is shown in Figure 11-20 on page 201. By default, the Results window opens in partial view.
	Open/Close Results – toggles the Results window.

© 2023 All Rights Reserved.

Widget	Description
	Click to reset results markings – clears the markers in the display area.
	Auto Zoom – click to select a node or a presistor in the Results window. The display area will zoom into the selected presistor or node as shown below. 
	Shows markers on screen – shows the markers for the selected layers in the display area.
	Opens the Main Window pop-up window that lets you launch a sub-window and synchronize connected windows.
	Opens the Connect pop-up window that lets you provide the IP address and the port number for the window you want to connect to the current window.
	Opens the Settings pop-up window that lets you specify whether you want to synchronize zoom, or probe, or both for connected windows.

© 2023
All Rights Reserved.

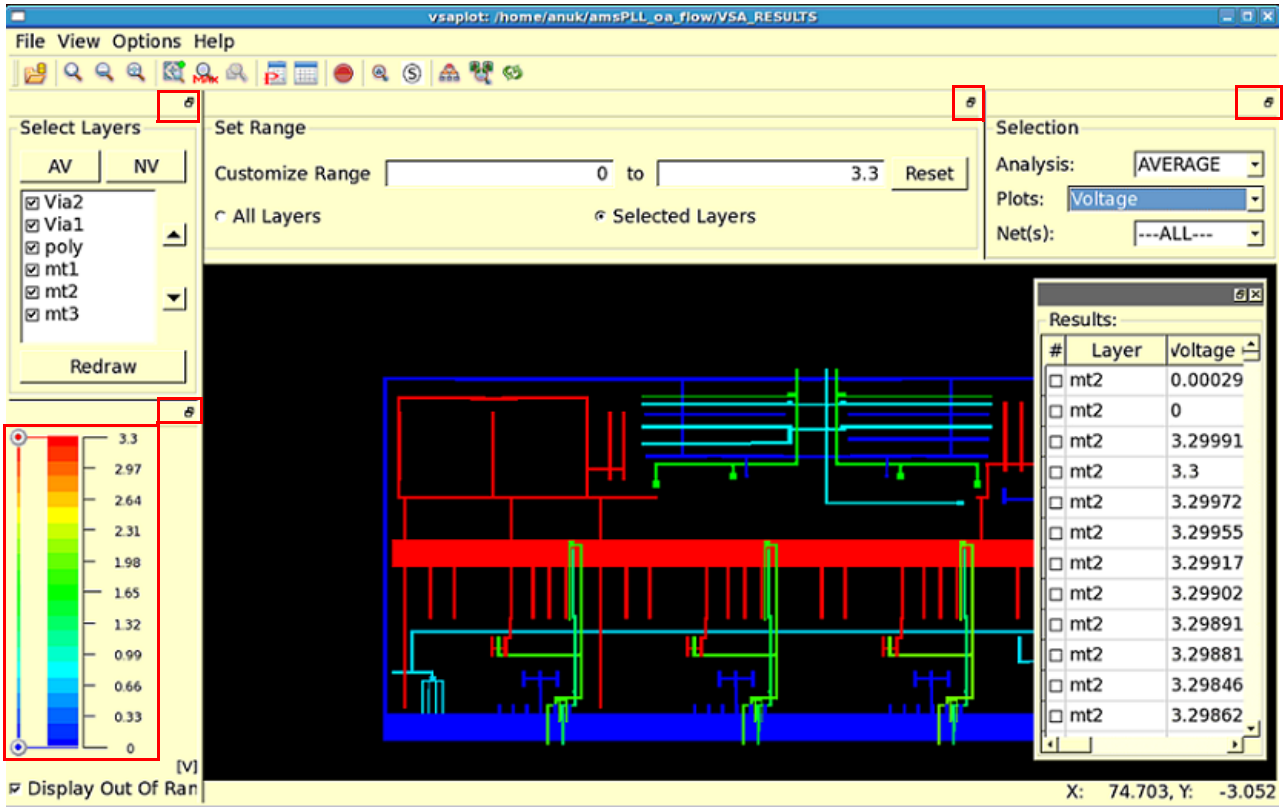
Main Components of vsaplot

In addition to the menus and widgets provided in the vsaplot, the main window of vsaplot includes a display area and the following sub-windows that are used to customize the display of violations.

- Select Layers
- Slider Range
- Set Range
- Selection
- Results

The main window of the tool highlighting the sub-windows is shown below.

Figure 11-13 Main Components of vsaplot



Note: You can dock any sub-window to the side, top, or bottom of the display area. Click the dock/undock handle (highlighted in red in image above) and drag it to the desired location.

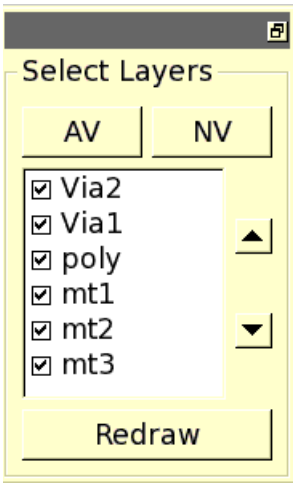
© 2023 All Rights Reserved.

The sub-windows are described in detail below.

Selecting the Layers

The *Select Layers* window is shown below.

Figure 11-14 Select Layers



In the Select Layers window:

- Select the layers for which you want to view the violations. Click *AV* or *NV* to select or deselect all layers in the list box, respectively. Click the layer names to select specific layers. By default, the first layer is selected.

  When you change your selection, click *Redraw* to apply your selection and update the view in the display area.

  Note: The view is not updated automatically when you change your selection.

- Change the order of layers by using the two buttons, with up and down arrows, provided next to the layers listed in the list box. Select any layer and then click the up or down arrow button to move the layer up or down in the list. The updated layer order is reflected in the extracted view of the design in the display area.

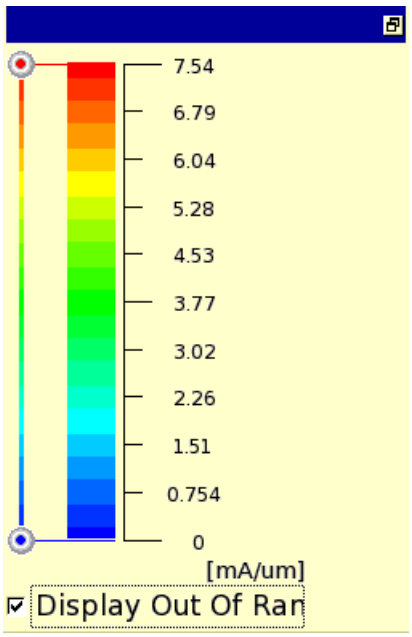
  Note: You can also specify the order of layers by setting environment variables before launching vsaplot. For details, see Specifying the Order of Layers in vsaplot.

© 2023 All Rights Reserved.

Specifying the Slider Range

In this window, a slider is provided to let you customize the violation range you want to view using a continuous RGB gradient. The vast range of color options provided by the RGB gradient makes it easier to view a specific range of violations. The ruler range of violation values for all or selected layers is provided next to the color spectrum as shown below.

Figure 11-15 The Slider Range



Min and Max slider buttons (the blue and red button, respectively) are provided for customizing the ruler range for viewing the violations that fall within the specified range. You can move the Min Max slider buttons up and down to update the ruler ranges according to the new slider button positions.

When the slider buttons are used to specify the ruler range for violations, the selection is updated in the *Customize Range* fields in the *Set Range* sub-window. The violations above and below the slider marks are displayed in “deep red” and “deep blue”, respectively.

Select *Display Out of Range* to view the violations with values outside the specified range.

Specifying the Violation Range

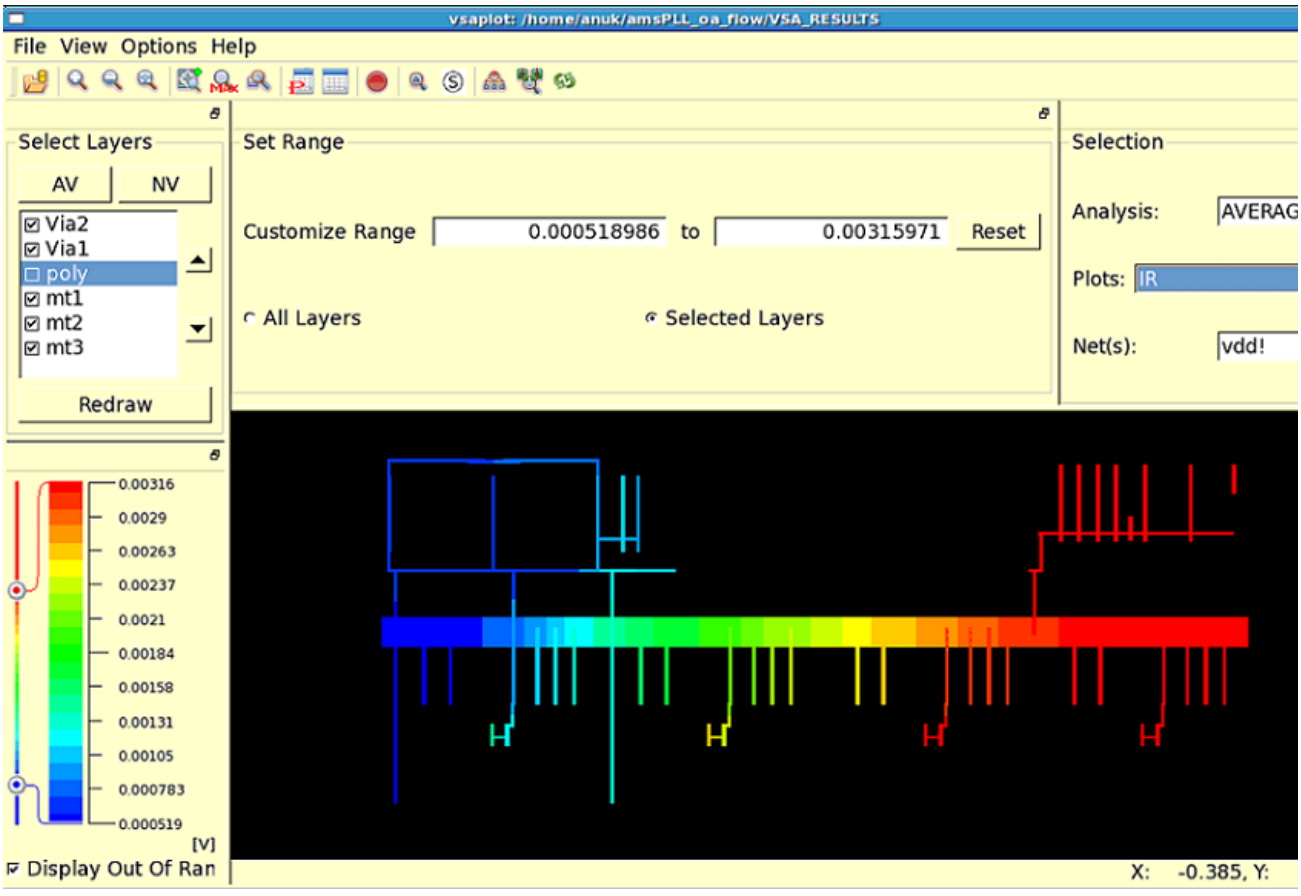
In the *Set Range* sub-window, you can specify the violation range you want to view.

© 2023 All Rights Reserved.

- Click *All Layers* to specify the total violation range. When *All Layers* is selected, the total violation range remains constant, irrespective of whether the layers are selected or deselected in the *Select Layers* window described in the previous section.
- Click *Selected Layers* to view the violation range for the layers selected in the *Select Layers* window. The specified range is updated in the slider range as well as in the *Customize Range* fields for the violations of the selected layers. The layout is highlighted accordingly.
- Type the range for which you want to view the violations in the text fields provided next to *Customize Range*. You can click *Reset* to reset the range to the previous setting.

The figure below shows the IR drop violations for the selected layers within the specified slider range.

Figure 11-16 Customizing the Slider Range for Viewing IR Drop Violations for Selected Layers



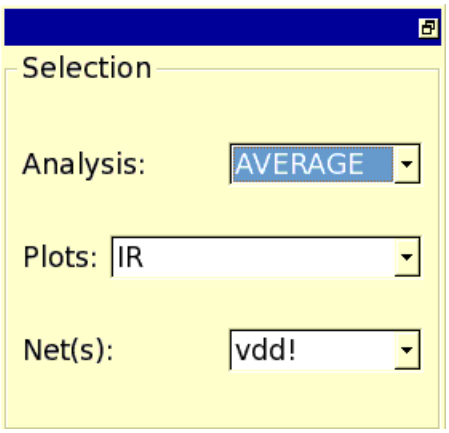
© 2023
All Rights Reserved.

Selecting the Type of Analysis and Plots

In this window, you can select the type of analysis and the type of plot for which you want to view violations. The following three cyclic fields are available in this window, as shown below:

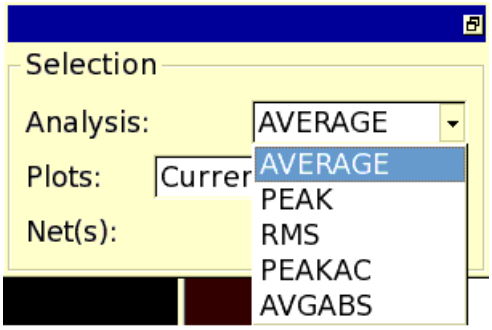
- *Analysis:* Types of analyses that can be performed
- *Plots:* Types of plots available for the different analysis types
- *Net(s):* Nets on which the analysis is to be performed

Figure 11-17 The Selection Window



Note: The analysis types available in the *Analysis:* cyclic field depends upon the selection of analysis types on the EM tab of the IR/EM form in the GUI mode while loading results or those specified using the `-type` parameter of the `write_em_vsaplot_db` command in the batch mode. When all analysis types are selected on the EM tab, the following types are displayed in the *Analysis:* field of the Selection window.

Figure 11-18 The Analysis Types in the Selection Window



© 2023
All Rights Reserved.

- *AVERAGE*: Displays the Current Density violations based on average value of the DC currents
- *PEAK*: Displays the Current Density violations based on the maximum or peak value of the DC currents
- *RMS*: Displays the Current Density violations based on the root mean square (RMS) values of the AC currents
- *PEAKAC*: Displays the Current Density violations based on the peak AC current
- *AVGABS*: Displays the Current Density violations based on the average of the absolute current

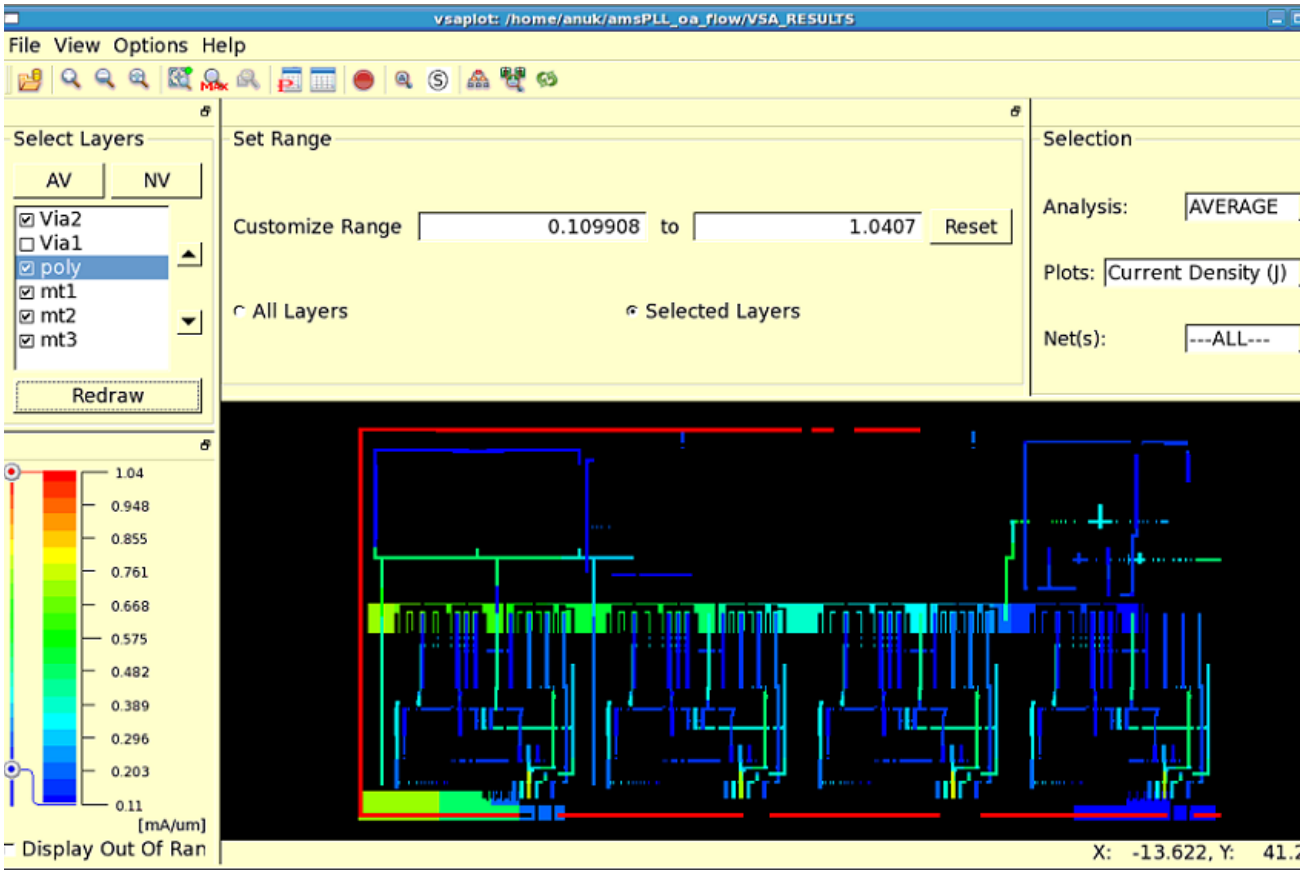
The following types of plots are available for the above analysis in the Plots cyclic field:

- Current (I): Displays the current violations for all or selected layers
- Current Density (J): Displays the Current Density violations for all or selected layers
- J/Jmax: Displays the Current Density ratio violations for all or selected layers
- IR: Displays the voltage violations for all or selected layers
- Voltage: Displays the voltage for all or selected layers
- R: Displays the ratio of voltage drop and current (`IR/I`) for all or selected layers

The results for Current Density ratio violations based on the average current for selected layers is shown in the figure below. The range of violations is customized using the slider.

© 2023 All Rights Reserved.

Figure 11-19 Viewing Current Density Violations for the Average Current for Selected Layers

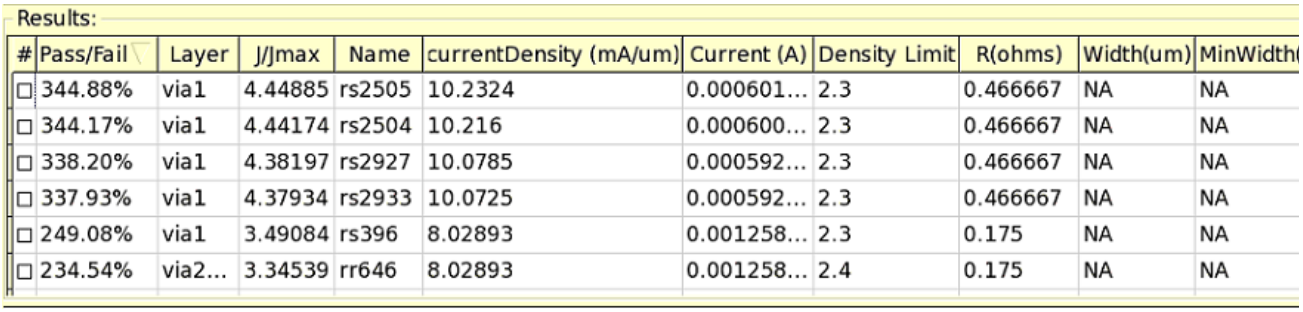


Viewing the Results

The Results window displays the results of the IR drop or EM analysis. By default, the Results window opens in partial view, in which only the layer name, net name, and plot type columns are visible. You can click the Partial/Full Results widget to display the full results, which includes all the columns shown in the figure below.

© 2023 All Rights Reserved.

Figure 11-20 The Results Window Displaying Full Results in pR-View


Results:

#	Pass/Fail	Layer	J/Jmax	Name	currentDensity (mA/um)	Current (A)	Density Limit	R(ohms)	Width(um)	MinWidth(
☐	344.88%	via1	4.44885	rs2505	10.2324	0.000601...	2.3	0.466667	NA	NA
☐	344.17%	via1	4.44174	rs2504	10.216	0.000600...	2.3	0.466667	NA	NA
☐	338.20%	via1	4.38197	rs2927	10.0785	0.000592...	2.3	0.466667	NA	NA
☐	337.93%	via1	4.37934	rs2933	10.0725	0.000592...	2.3	0.466667	NA	NA
☐	249.08%	via1	3.49084	rs396	8.02893	0.001258...	2.3	0.175	NA	NA
☐	234.54%	via2...	3.34539	rr646	8.02893	0.001258...	2.4	0.175	NA	NA

Tip

You can save your markings in the Results window. Select the check box against the node or resistor for which you have viewed the violation. When you open the vsaplot the next time, your markings will be visible in the Results window. This feature helps you save time by identifying violations that have already been viewed.

Querying IR Drop and EM Results

The following layout query features are supported:

- Zooming into the Worst Violation in the Selected Area
- Retrieving the Values of All the Violations at a Specified Point on the Layout

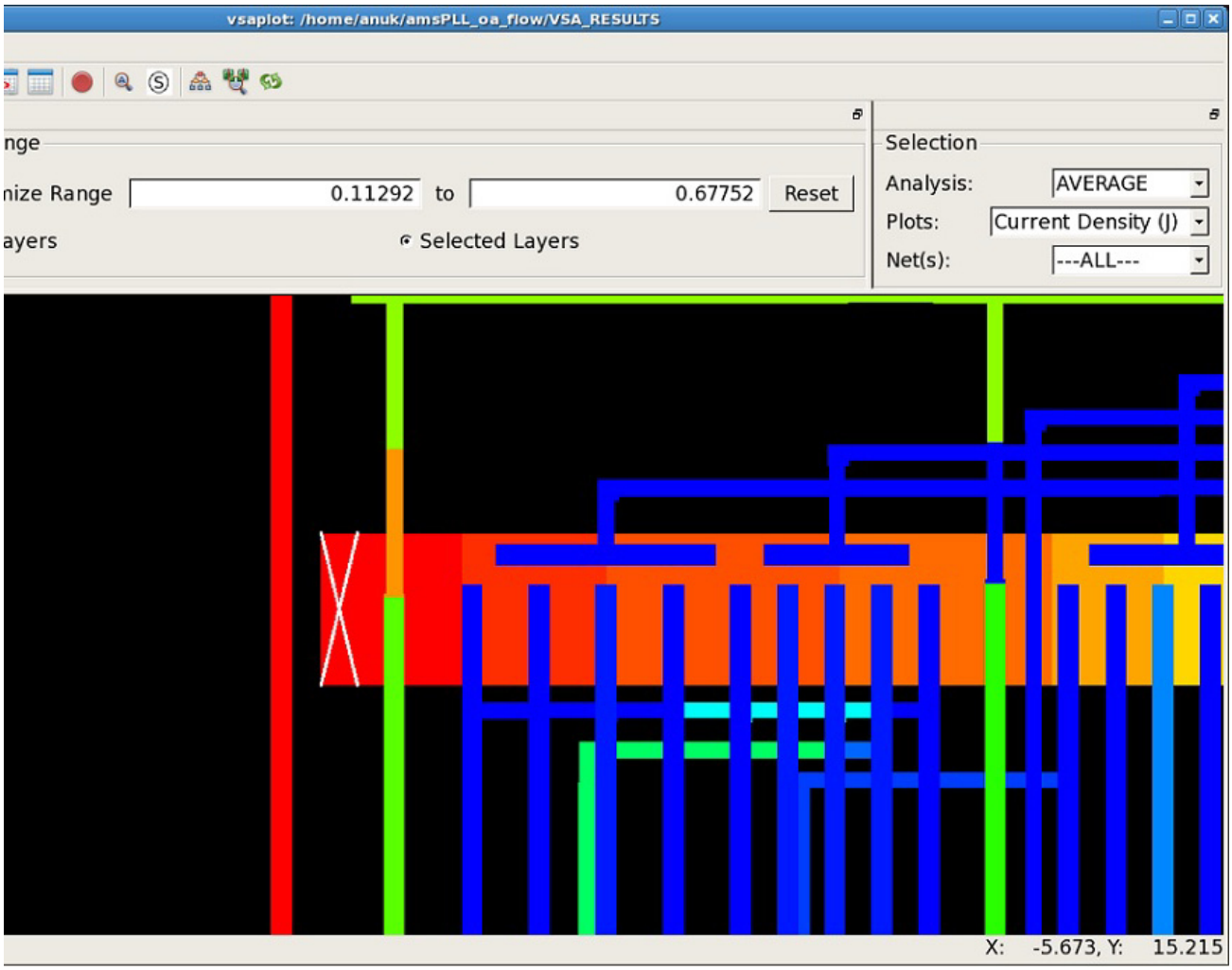
Zooming into the Worst Violation in the Selected Area

You can select an area on the layout and zoom into the worst violation in the selected area by using the *searchMax* submenu in the *View* menu, or by clicking the corresponding toolbar widget.

The display area zooms into the worst violation in the selected area. The location of the maximum violation (x and y coordinates) is displayed in the bar at the bottom of the vsaplot window as shown below.

© 2023 All Rights Reserved.

Figure 11-21 Zooming into the Worst Violation in the Selected Area

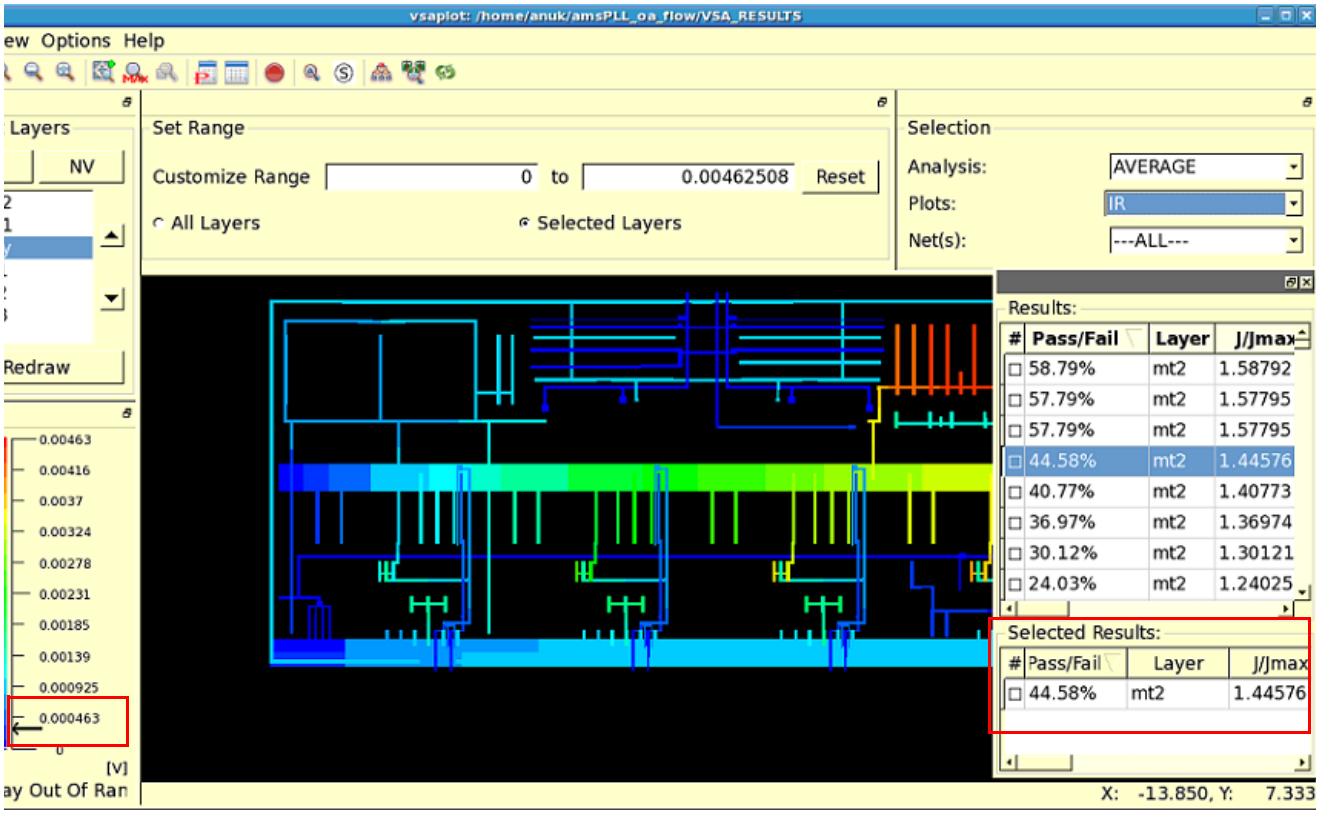


Retrieving the Values of All the Violations at a Specified Point on the Layout

You can click anywhere on the design in the display area and retrieve the values of all the violations at that point. The results are displayed in the *Selected Results* section of the Results window. This section is only visible when you click anywhere on the design in the display area. The vsaplot window displays the violations at that point and the worst violation value is marked by an arrow on the ruler range as shown below.

© 2023 All Rights Reserved.

Figure 11-22 Retrieving the Values of the Violations at a Specified Point



Displaying the Colormap for Point-to-Point and Region-to-Region Selection

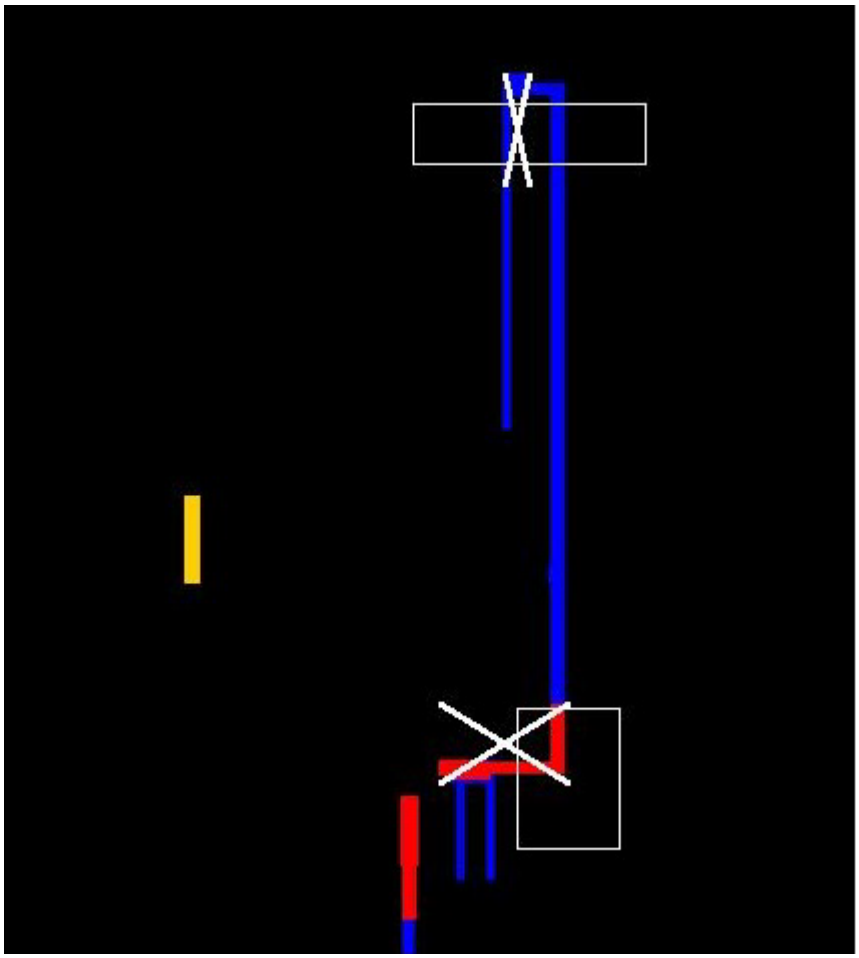
The vsaplot or vfiplot utility supports the display of the colormap between two points or the regions selected on the layout. For this, the *Region2Region* submenu of the *View* menu is provided with four options; *Point*, *Rectangle*, *Ellipse*, and *Clear*. The *Clear* option is used to clear the selection.

Select *Point* to specify two points on the layout, or specify *Rectangle* or *Ellipse* to specify regions as rectangular shapes or ellipses in the display area, respectively. The tool resets the violation range, displaying colors from blue to red, between the two specified points or regions. Rectangular regions are specified in the figure below.

Note: You can also enable the region-to-region mapping using the toolbar widget.

© 2023 All Rights Reserved.

Figure 11-23 Displaying Region-to-Region Selection – Rectangular Shapes



To clear the point or region selection, click the *Clear* option in the *View - Region2Region* submenu or click the *Esc* button. The rectangles or ellipses will be cleared.

Click *Reset* in the *Set Range* group box to clear the colormap display between the selected points or regions.

Calculating Relative Resistance Values between the Nodes on a Net in vfiplot

This feature is only available for vfiplot. The vfiplot utility supports the calculation of relative resistance between two specified points or nodes on the specified net and plots a colormap for these in the display area. This feature uses the colormap feature of vfiplot for point-to-point

© 2023 All Rights Reserved.

and region-to-region selection to specify the points for which relative resistance is to be calculated and plotted.

For pairs of selected nodes, the software calculates the relative resistance value between the selected nodes and displays the colormap with colors ranging from blue to red, between the two nodes.

For pairs of selected regions, the software uses the lowest voltage value from the high voltage region and the highest voltage value for the low voltage region to calculate the relative resistance between the specified regions.

Note: This feature is only enabled when the following are specified:

- vfiplot is run
- The R plot type is specified in the *Plots:* field of the *Selection* group box
- A particular net is selected in the *Net(s):* field of the *Selection* group box

The flow, including the requirements and the reports generated for relative resistance calculations, is documented in detail in subsequent sections.

Requirements for Running the Relative-R Flow

To run the relative resistance (Relative-R) flow, ensure the settings for Voltus-Fi-L that are listed below.

- Set the following variables:
  `setenv VFI_VSAPLOT_DB 1` (for the GUI flow)
  `setenv VFI_VSAPLOT_DUMP_DSPF 1`
- Run Voltus-Fi-L and create the `VSA_RESULTS` directory. A `basic.dspf` file will be created in the `VSA_RESULTS` directory.
- Run `vfiplot`.

Note: The `vfiplot` option will check out the VFI-XL license and allow you to use the Relative-R or the R Map feature. If the license is not available, it will error out.

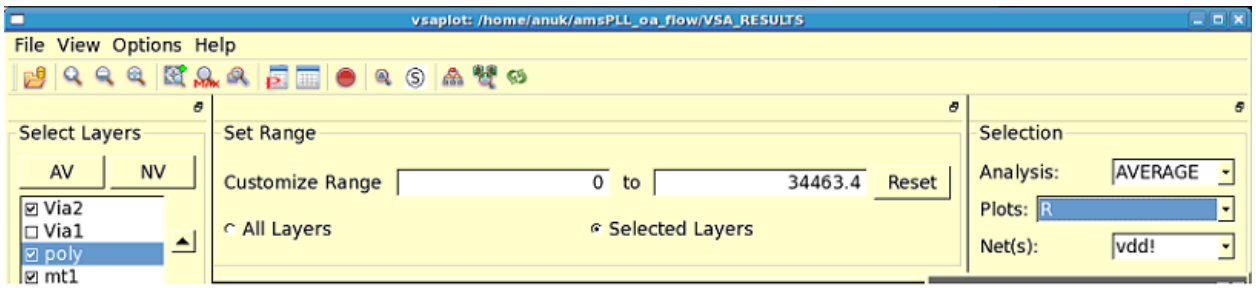
© 2023 All Rights Reserved.

Running the Relative-R Flow

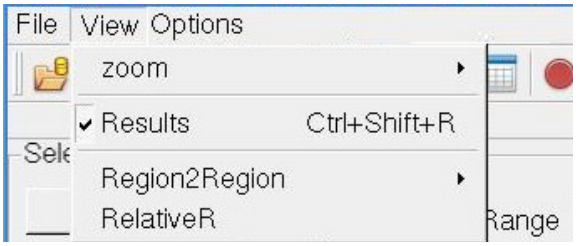
Follow these steps to run the relative resistance flow in vfiplot.

1. In the *Selection* group box, as shown below:
   a. Select the net on which you want to perform the relative resistance analysis.
   b. Select R as the plot type.

Figure 11-24 Selecting the Net and Plot Type for the Relative-R Flow



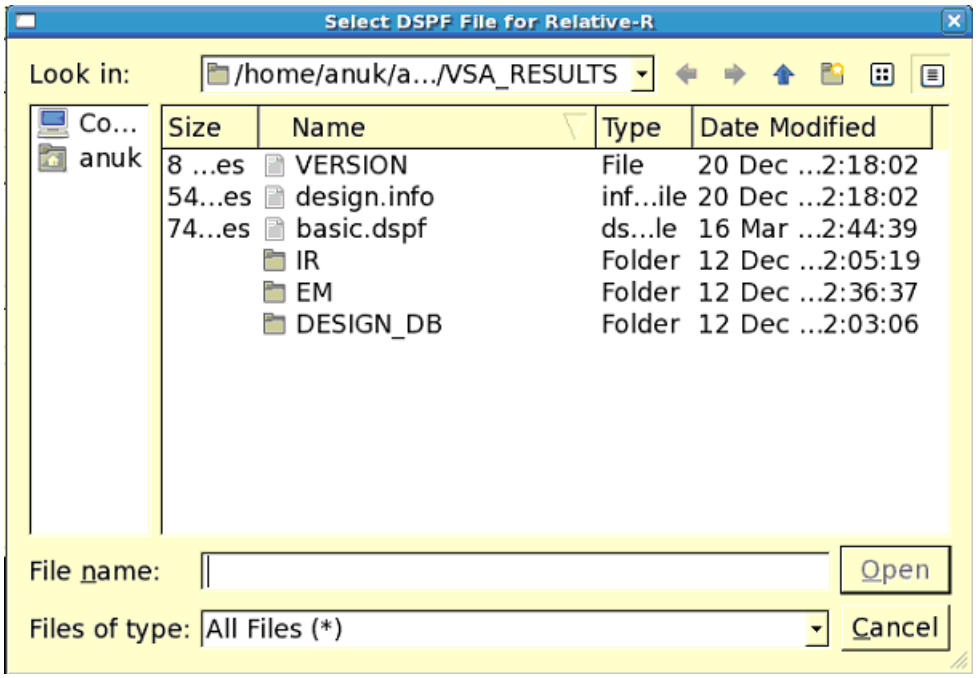
2. The *RelativeR* submenu is enabled as shown below.



3. In the *View* menu, click *RelativeR*. The *Select DSPF File for Relative-R* pop-up window opens. The VSA_RESULTS directory opens by default.
4. Select the DSPF file that you want to use. This file will get copied to the `basic.dspf` file, which is present in the VSA_RESULTS directory.

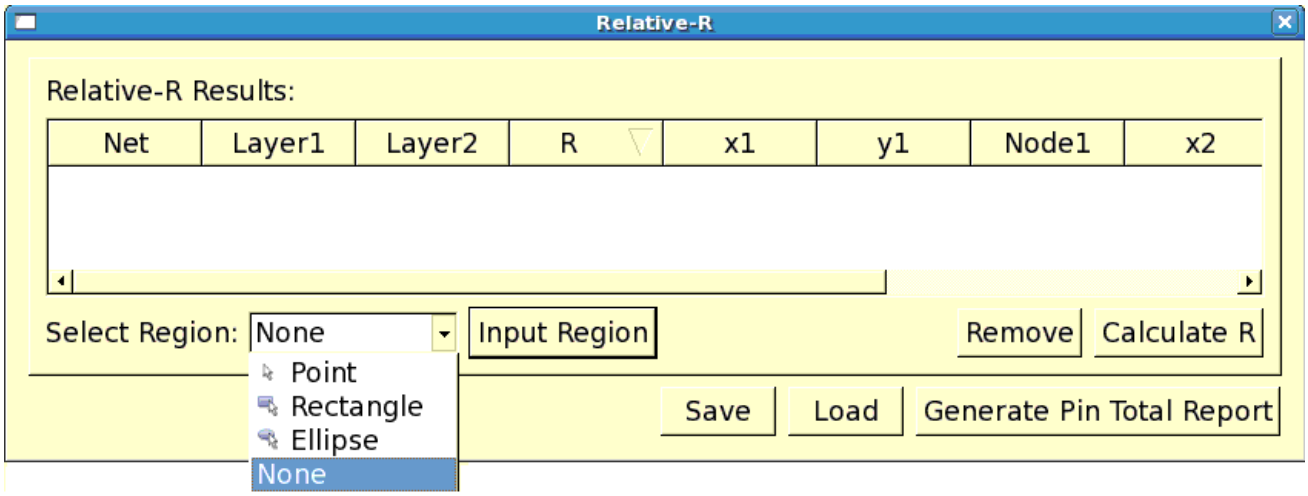
© 2023 All Rights Reserved.

Figure 11-25 Selecting the DSPF File



5. Click Open. The Relative-R form opens.

Figure 11-26 The Relative-R Form



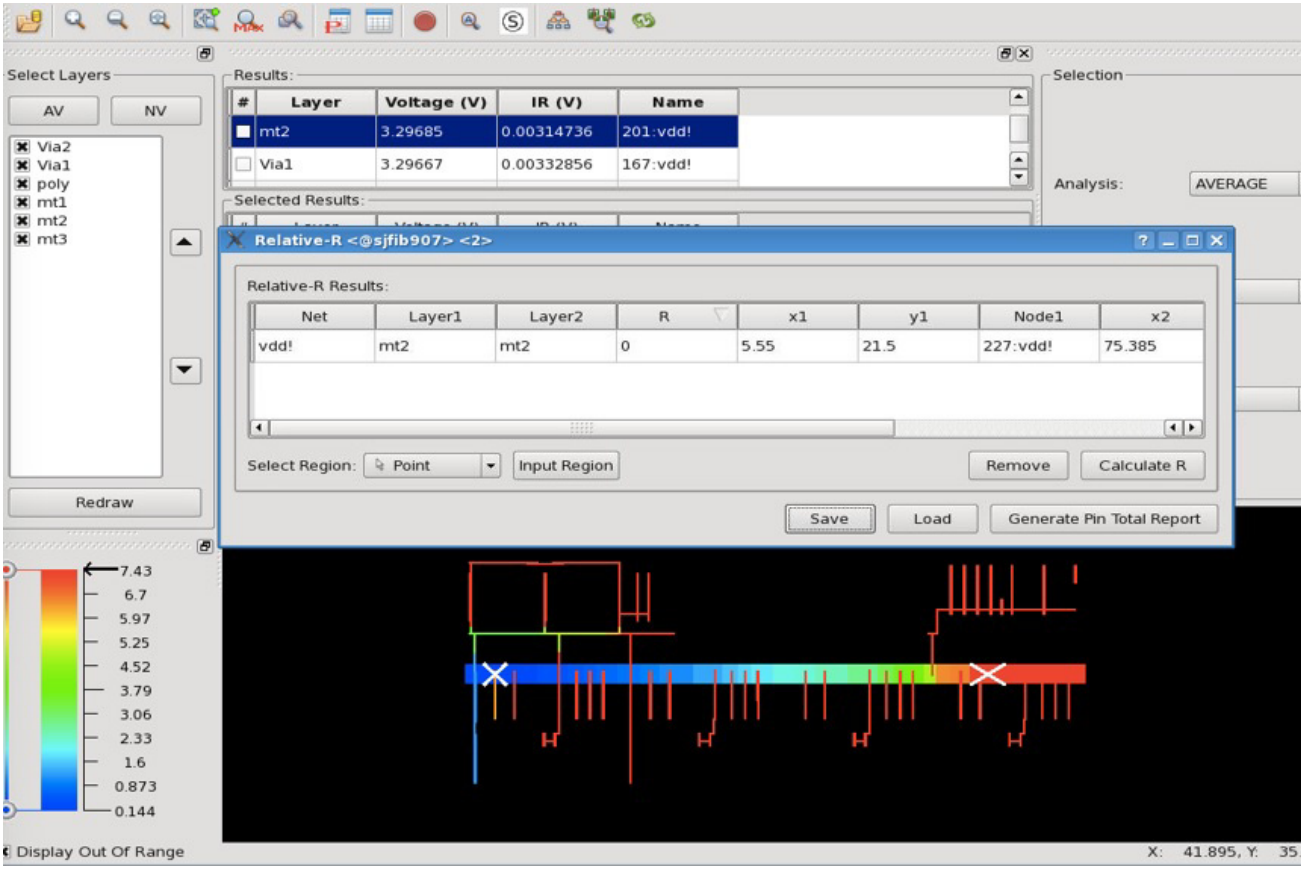
The above form displays the relative resistance results for the pairs of selected nodes. the steps are detailed below.

6. Click Select Region to specify how you want to select the region. You can choose from the *Point*, *Rectangle*, and *Ellipse* options.

© 2023
All Rights Reserved.

7. Click *Input Region* to select the region in the display area. When two points, rectangles, or ellipses are selected in the Display region, the table in the Relative-R form is populated with the information for the two points or nodes, including their x- and -y coordinates, node names, and layer names as shown below.

Figure 11-27 The Relative-R form – Displaying Information for Selected Pairs of Nodes

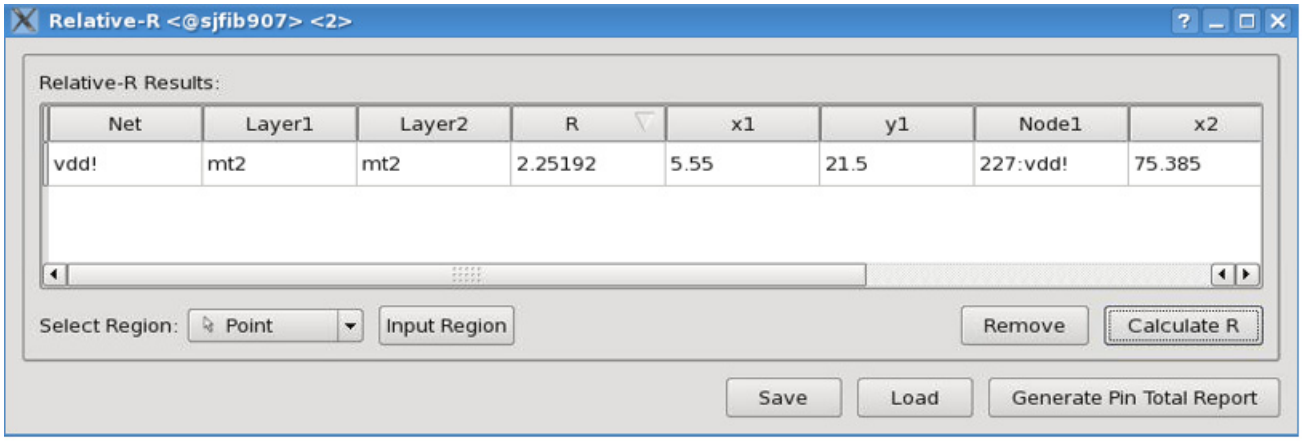


You can select multiple pairs of nodes in the display area by using the *Input Region* button. These pairs will be listed in the Relative-R form.

8. In the Relative-R form, click any row and then click *Calculate R* to calculate the relative resistance between the pair of specified nodes in the form. The software uses the static power grid solver (SPGS) feature of Spectre to calculate the value of the resistance and populates the “R” column in the table of the Relative-R form as shown below.

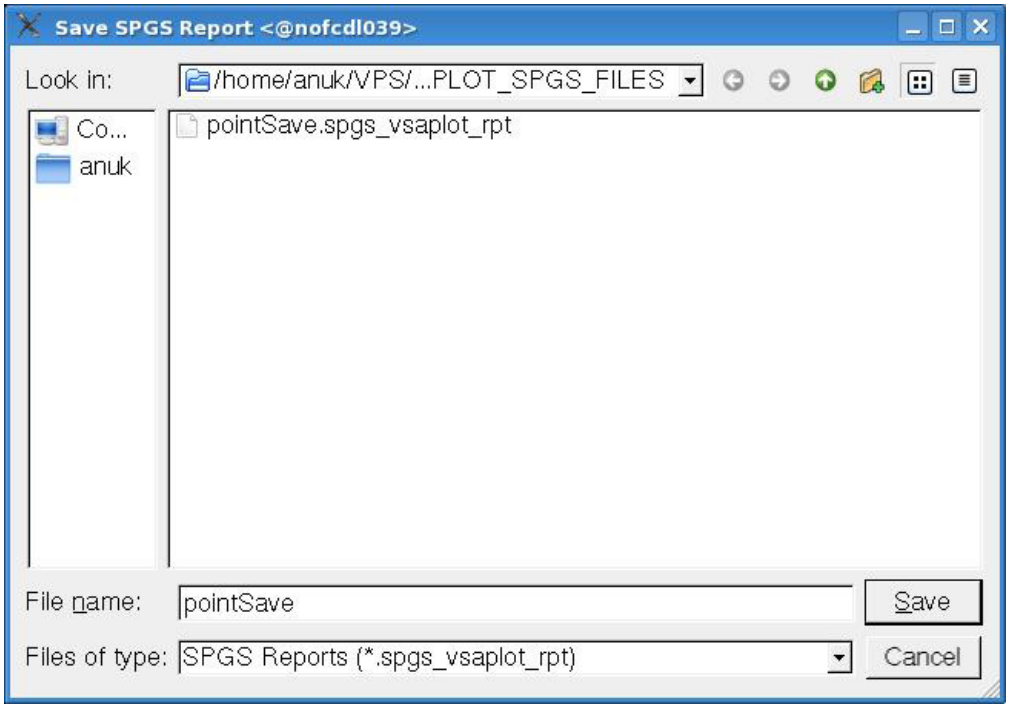
© 2023 All Rights Reserved.

Figure 11-28 The Relative-R form – Calculating Relative-R between Specified Pairs of Nodes



9. Click *Save* to save the information filled out in the Relative-R form in a file. You can load this file in subsequent runs to populate the Relative-R form. The *Save SPGS Report* window opens as shown below.

Figure 11-29 The Save SPGS Report Window



Specify a name for the file. By default, the file is saved in your working directory (`VSAPLOT_SPGS_FILES`). A sample file is shown below.

© 2023 All Rights Reserved.

Figure 11-30 A Sample File with the Relative-R Information

```
Relative-R Results:

Net     Layer1   Layer2    R        x1      y1       Node1     x2      y2          Node2
vdd!     mt2      mt2    2.25192   5.550   21.500   227:vdd!  75.385  21.500       201:vdd!
```

10. Click *Load* to load an already saved configuration file in the form. All pairs of nodes listed in the file are added to the table in the Relative-R form.

11. Click *Generate Pin total Report* to generate the total resistance report for all pins. This feature is described in detail in the section below.

Calculating the Total Resistance for All Pins

The vfiplot utility lets you calculate the total resistance for all the pins in your design. For every pin, resistance is calculated for all the wires on every layer of the pin, and the percentage of the resistance on each layer of the pin as compared to the total resistance on the pin.

This feature is useful because it lets you view the resistance for every layer of the pin and the percentage resistance for each layer so that you can identify the layers contributing the highest resistance percentage in the pin.

The total pin report includes the following information:

```
Pin: <name of pin>
/* ------------------------------------------------------ */
/*   Wire & Via Name  |    Resistance    |   Percentage
/* ------------------------------------------------------ */
```

where,

The *Wire & Via Name* column lists the names of all layers of the pin

The *Resistance* column lists the resistance values of all wires of each layer, and

The *Percentage* column lists the percentage resistance – the resistance on each layer as a percentage of the total resistance on the pin – on each layer

The following set of equations show how the resistances are calculated:

© 2023 All Rights Reserved.

Step 1: Calculating the total resistance on the pin

$$P_{total} = \Sigma I^2 R$$

$$R_{total} = P_{total} / I_{total}$$

where,

P_{total} is the total wire power – the summation of power dissipated on all parasitic resistors belonging to the pin – drawn by the pin

I_{total} is the total current drawn from the pin, and

R_{total} is the total wire resistance on the pin

Step 2: Calculating the resistance on a particular layer, say `M1`:

$$P_{M1} = \Sigma I^2 R$$

$$R_{M1} = (P_{M1} / P_{total}) \times R_{total}$$

where,

P_{M1} is the summation of power dissipated on all parasitic resistors belonging to the pin lying on layer M1

R_{M1} is the total wire resistance on layer M1

A sample report is shown below.

Example 11-1 A Sample Pin Report

```
Pin Total Report:

Pin: gnd!
/* ------------------------------------------------------- */
/*   Wire & Via Name  |    Resistance    |   Percentage
/* ------------------------------------------------------- */
                  mt1        7.66013e-05         6.02065
                  mt3                  0               0
```


© 2023 All Rights Reserved.

```
                  poly                  0               0
                   mt2        0.000676434         53.1658
                  Via1        0.000519275         40.8135
                  Via2                  0               0
/* ------------------------------------------------------- */
                 Total         0.00127231             100
/* ------------------------------------------------------- */


Pin: ibias
/* ------------------------------------------------------- */
/*   Wire & Via Name  |    Resistance    |   Percentage
/* ------------------------------------------------------- */
                   mt1        9.94715e-05         35.4389
                   mt3                  0               0
                  poly        4.07399e-10     0.000145145
                   mt2        0.000101207         36.0572
                  Via1        8.00056e-05         28.5037
                  Via2                  0               0
/* ------------------------------------------------------- */
                 Total        0.000280685             100
/* ------------------------------------------------------- */
```

Synchronizing Multiple vsaplot Windows

The vsaplot/vfiplot utility supports the synchronism function, which involves opening multiple windows and synchronizing them using the menu options provided in the GUI. Three main synchronism functions are provided through the *Open* submenu of the *File* menu. These are the following:

- For a design open in a vsaplot window, you can open a sub-window of another design to compare the two designs by using the *Open New Window* option.
- For two designs open in separate vsaplot windows, you can specify the IP and port number of one window in the other, to compare the two designs by using the *Open New Window and Sync* option.
- For two connected windows, you can specify the information you want to synchronize for comparing the two designs.

The above functions can also be performed using the toolbar widgets. For details, see Toolbar Widgets.

© 2023 All Rights Reserved.

The synchronism function is detailed in the following sections:

- Opening a Sub-Window to Initiate Synchronism
- Synchronizing Options

Opening a Sub-Window to Initiate Synchronism

To initiate synchronism and start the comparison, follow any of these methods:

- Open a sub-window. For this, select *File* and then *Open*. Click *Open New Window and Sync*. The Main Window pop-up opens. Select *Launch new sub-window?* to open a sub-window of another design and start the comparison between the two designs.
- Open two designs in separate windows. Then open the Main Window pop-up window in one window and drag it into the sub-window to connect the two windows and start the comparison.
- Open two designs in separate windows. Then open the Connect pop-up window and manually provide the IP and port settings of the sub-window to connect the two windows and start the comparison. This method can be used to connect two designs on different machines.

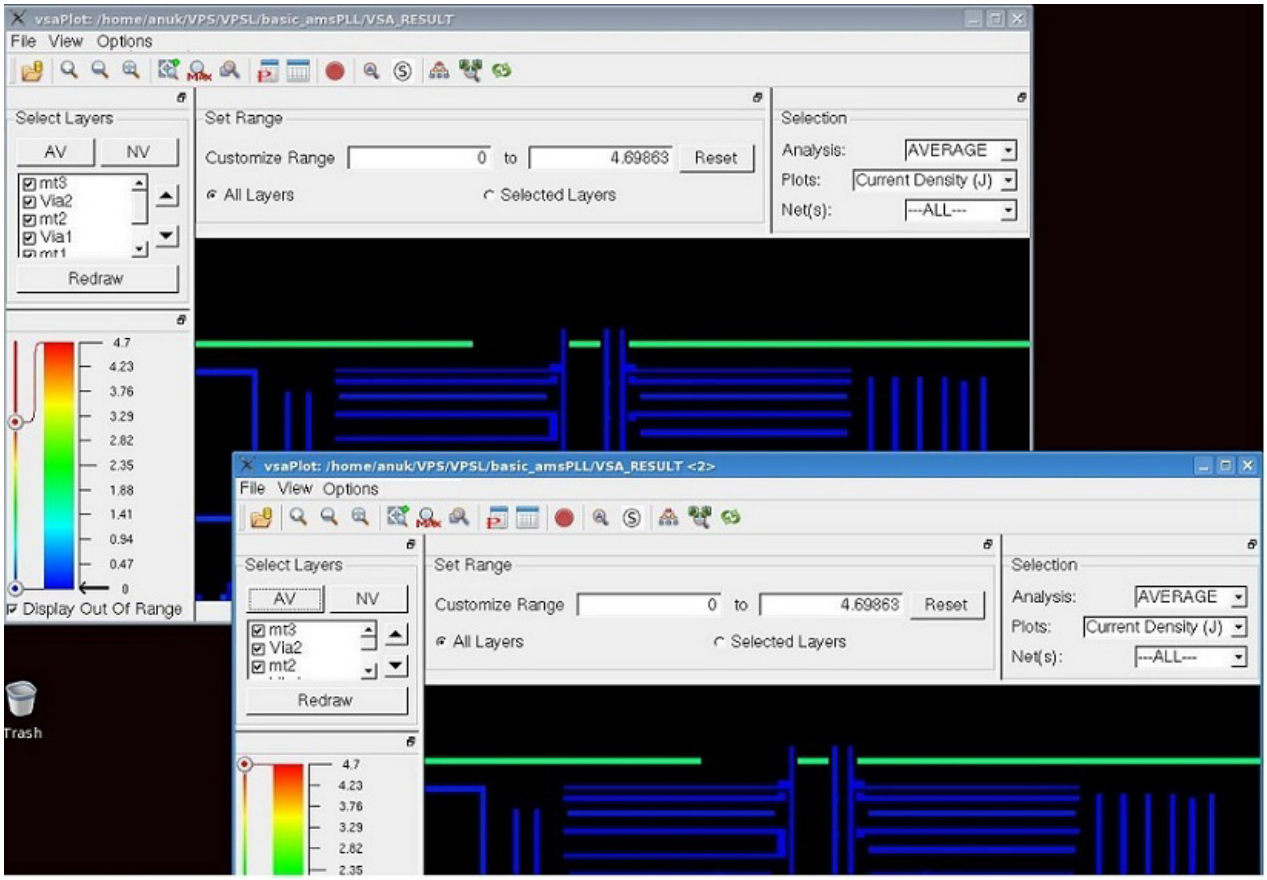
Synchronizing Options

Once the windows are open for comparison, you can select the options for synchronism depending upon your requirements. The following options are available:

- **Static synchronism**: Once the windows are connected, the sub-window gets the state of the main window. This means that the main window's properties, such as the type of analysis selected in the *Selection* group box, the layers selected in the *Select Layers* group box, the display area settings, and the slider range, are copied to the sub-window. However, these settings are copied only once. If these settings are changed in the main window, you have to manually synchronize the sub-window from the main window again to bring the two windows back into the same state as shown below.

© 2023 All Rights Reserved.

Figure 11-31 Opening a Sub-Window

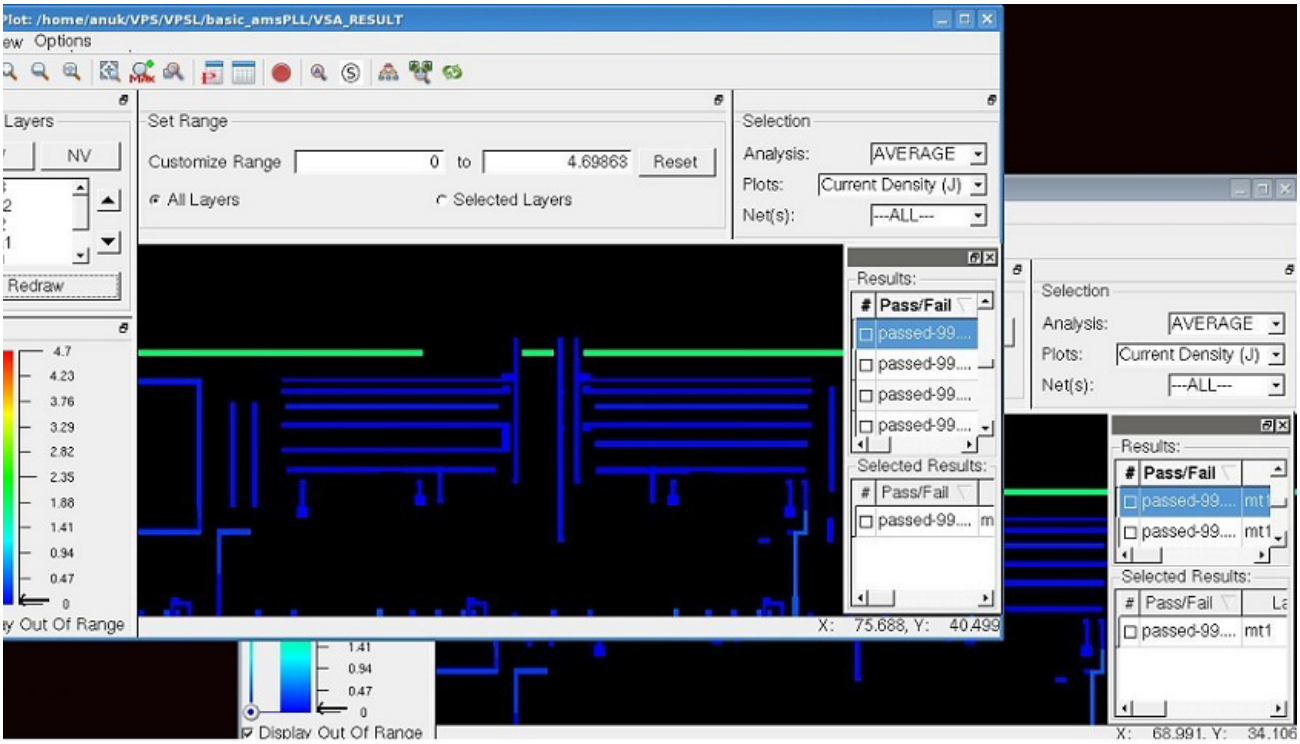


- **Dynamic synchronism**: Properties such as zooming and probing are supported dynamically. You can filter the instruction received by the sub-window from the main window by using the synchronization settings. Open the *Synch* submenu in the *File* menu. Click Settings. The Settings pop-up window opens. Specify whether you want to synchronize zoom or probe, or both for the connected windows.

  Selecting zoom means any zoom in or zoom out action performed in the main window will be replicated in the sub-window. Selecting probe means that if you probe any part of the design in the main window, the same action will be performed in the sub-window as shown below. By default, both zoom and probe actions are synchronized.

© 2023 All Rights Reserved.

Figure 11-32 Dynamic Synchronism Probing between Connected Windows



Synchronizing vsaplot with the Layout or Extracted View

In addition to synchronizing multiple vsaplot windows, you can synchronize a vsaplot window with the Virtuoso layout/extracted cellview window. Once they are synchronized, the actions of zooming or scrolling in the vsaplot window are replicated in the Virtuoso layout/extracted cellview window. This helps you map problematic areas seen in the vsaplot to those in the Virtuoso layout/extracted cellview window correctly.

To synchronize the vsaplot window with the layout/extracted view, follow these steps:

- Load the results directory, `VSA_RESULT`, using the following command:

  ```
  vsaplot VSA_RESULT
  ```

  or

  ```
  vfiplot VSA_RESULT
  ```

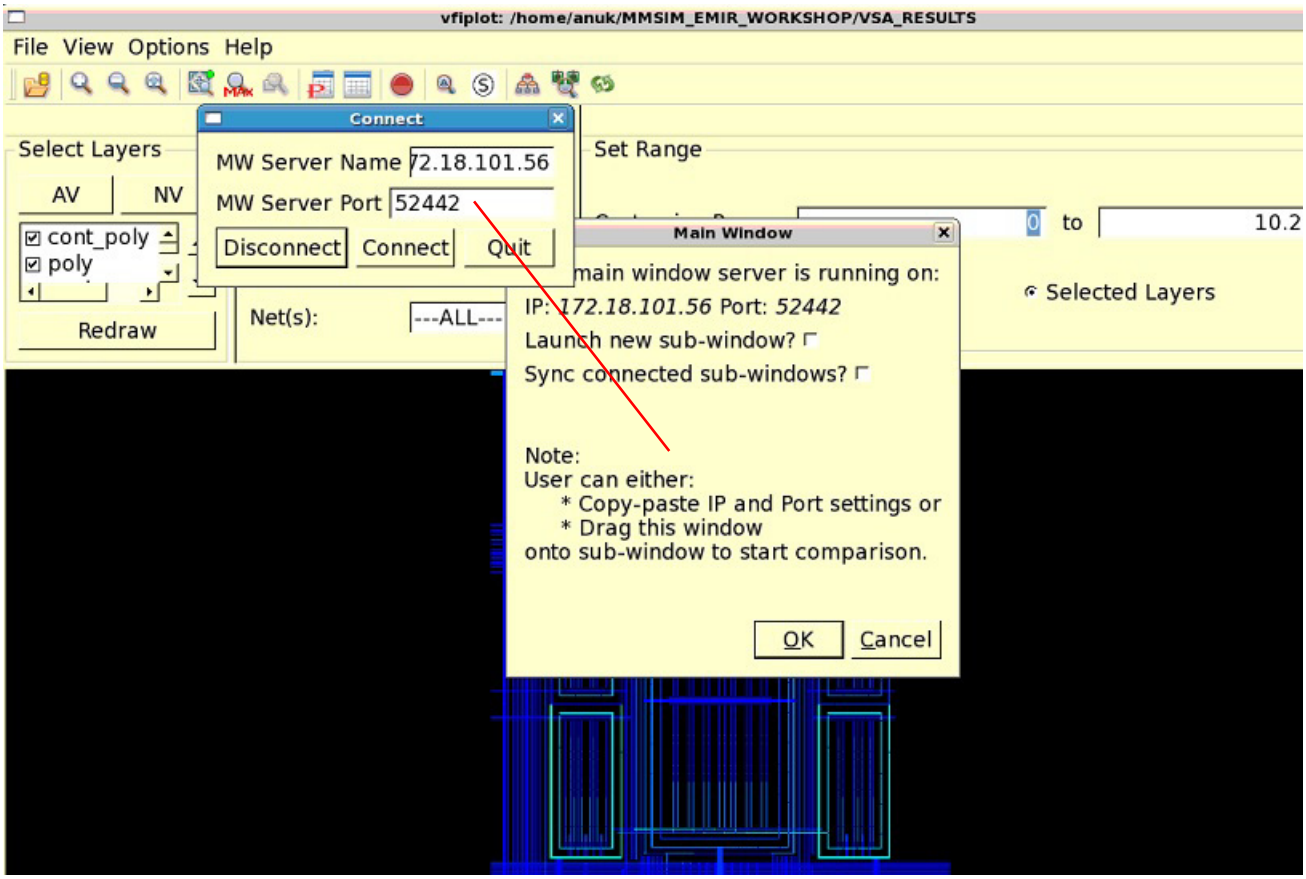
- In the *Open* submenu of the *File* menu, click *New Window and Sync*. The *Main Window* pop-up window opens. Drag and drop the *Main Window* to the *Connect* pop-up window. The library/cell/view settings of the *Main Window* are populated in the *MW*

© 2023 All Rights Reserved.

Server Name field of the *Connect* window. The port information is populated in the *MW Server Port* field as shown below.

Note: You can also open the Connect pop-up window from the *Connect To vsaplot* submenu of the IR/EM analysis menu. When you click this option, the *Connect* pop-up window opens. Provide the name of the *MW Server Port* and click *Connect*. For details, see Launching vsaplot from the GUI.

Figure 11-33 Transferring the Lib/Cell/View Information



- Click Connect in the Connect pop-up window. The specified extracted or layout view opens. This is synchronized with the vsaplot window.

© 2023 All Rights Reserved.

12

Batch Mode Execution

© 2023 All Rights Reserved.

Overview

This chapter covers details of how to run Voltus-Fi-L in the batch mode including all the data requirements.

Migrating from Legacy (VPS-L) to Voltus-Fi-L in Batch Mode

The batch mode flow of Voltus-Fi-L supports both the configuration file, or the `conf` file used in Voltus-Fi-XL for IR/EM analysis, and the command file, `cmd_file`, used in the legacy (VPS-L) batch mode.

Support is provided to convert an existing legacy (VPS-L) command file to the Voltus-Fi-L configuration file using the command below:

```
vfibatch -convert <VPSL command file name> <new emir conf file name>
```

If the new `emir` configuration file name is not specified, then by default, the software will use the name, "`ext_emir.conf`".

Note: It is recommended that after converting the command file to the configuration file, check the new `emir.conf` file and add any missing or additional information before running Voltus-Fi in the batch mode.

The details of the batch mode flow using the `conf` file and the `cmd_file` are provided in the following sections:

- The Command File Support
- The Configuration File Support

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 EMIR Analysis Reports in Voltus-Fi-L” in the IR Drop Analysis Results chapter.

© 2023 All Rights Reserved.

The Command File Support

The command file is used to load the IR drop and EM analysis results for the Voltus-Fi-L extracted view flow. The command file consists of the legacy (VPS-L) command file, Voltus-Fi commands, and values of the environment variables that are specified as commands. The commands that should be available in the command file are listed in the section below.

The following topics are covered in this section:

- Supported Voltus-Fi Commands
- Supported VPSL Batch File Commands
- Sample Batch Mode Command File

© 2023 All Rights Reserved.

Supported Voltus-Fi Commands

The Voltus-Fi batch commands are specified in the command file for loading and viewing the results of different analyses types. These are specified in addition to the legacy (VPS-L) command file name. The following batch commands are available:

- Batch Commands for IR Reports
- Batch Commands for EM Reports
- Batch Command for RON Reports
- Batch Commands for Querying
- Batch Commands for Viewing Results in vsaplot and vfiplot

© 2023 All Rights Reserved.

Batch Commands for IR Reports

You can load and print the results for IR drop analysis in the Voltus-Fi-L flow using the following commands:

- load_ir_results_extview
- print_ir_report
- print_rlrp_report

© 2023 All Rights Reserved.

load_ir_results_extview

`load_ir_results_extview -i` *`cmd_file`*

Loads the results of IR drop analysis in the Voltus-Fi-L flow.

Parameter

`-i` *`cmd_file`*	Specifies the name of the batch file used in the Voltus-Fi-L batch mode execution.

Example

- The following command loads the results for IR drop analysis that are stored in the `vsaIR.cmd` file:

  ```
  load_ir_results_extview -i vsaIR.cmd
  ```


© 2023 All Rights Reserved.

print_ir_report

```
print_ir_report
     -net {all_power | netname}
     -type {ir |iv |rc |rcavg |rcrms}
     [-threshold threshold_value]
     -filename output_file_name
```

Prints the results of IR drop analysis in the Voltus-Fi-L flow. You can generate IR reports for all the power nets or for specific nets.

Note: Before running this command, ensure that the IR drop results are loaded using the `load_ir_results_extview` 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 | rc |rcavg |rcrms}`	

© 2023 All Rights Reserved.

	Specifies the analysis type for which the report is being generated. `ir` – specifies that the report is generated for IR drop analysis. `rc` – specifies that the report is generated for the peak resistor Current Density analysis. `rcavg` – specifies that the report is generated for the average resistor Current Density analysis. `rcrms` – specifies that the report is generated for the RMS resistor Current Density 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 `VSS` net for `rc` analysis in the `vss_rc.txt` file:

```
print_ir_report \
-net VSS \
-type rc \
-filename vss_rc.txt
```

- The following command prints the IR drop analysis report for all the 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
```


© 2023 All Rights Reserved.

print_rlrp_report

```
print_rlrp_report
     -net netname
     -filename output_file_name
     -tap tapname
```

Prints the RLRP analysis report in the Voltus-Fi-L flow.

Note: Before running this command, ensure that the IR drop results are loaded using the `load_ir_results_extview` 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 `VDD` net in the `RLRP.rpt` file:

```
load_ir_results_extview -i vsaIR
print_rlrp_report -net VDD -filename RLRP.rpt -tap MavD7_1_unmatched#d
print_rlrp_report -net VDD -filename RLRP.rpt -tap MavD7_2_unmatched#d
```


© 2023 All Rights Reserved.

Batch Commands for EM Reports

You can load and print the results for EM analysis in Voltus-Fi-L using the following commands:

- load_em_results_extview
- print_em_report

© 2023 All Rights Reserved.

load_em_results_extview

```
load_em_results_extview -i cmd_file
```

Loads the results of EM analysis in the Voltus-Fi-L flow.

Parameters

-cmd_file	Specifies the name of the batch file used in the Voltus-Fi-L batch mode execution.

Example

- The following command loads the results of EM analysis stored in the `vsaEM` file:

  ```
  load_em_results_extview -i vsaEM
  ```


© 2023 All Rights Reserved.

print_em_report

```
print_em_report
     -net {all_power | all_signal | all_nets | netname}
     -type {javg | jmax | jabsavg | jacpeak | jacrms | rc | rcavg | rcrms}
     [-threshold threshold_value]
     -filename output_file_name
     [-rule custom_em_rule_name]
```

Prints the EM analysis reports in the Voltus-Fi-L flow.

Note: Before running this command, ensure that the EM results are loaded using the `load_em_results_extview` 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`*`}`	
	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}`	

© 2023 All Rights Reserved.

	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. 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
```


© 2023 All Rights Reserved.

- 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
```


© 2023 All Rights Reserved.

Batch Command for RON Reports

You can report the on-resistance (RON) values of power devices in Voltus-Fi-L by using the following command:

- report_vfi_L_ron

© 2023 All Rights Reserved.

report_vfi_L_ron

```
report_vfi_L_ron
     -pin_pair_file file_name
     [-device_ron {true | false}
     -param_list_file file_name
     -output_file file_name
```

Generates the on-resistance (RON) report for IR drop and EM analysis for the specified pin-pairs. Layer-wise wire RON is reported by default. For reporting the device RON, specify the `-device_ron` parameter.

For details of the flow, see Reporting On-Resistance Values.

Parameters

`-device_ron {true | false}`	When set to `true`, the device RON is reported. This parameter is optional. *Default*: `false`, which means the device RON is not reported.
`-output_file` *`file_name`*	Specifies the name of the output file to be created with the RON report. This is a required parameter.
`-param_list_file` *`file_name`*	Specifies the parameters of the devices to be evaluated. This is a required parameter.
`-pin_pair_file` *`file_name`*	Specifies the name of the pin-pair file that contains information about all the pin pairs for which the RON report is to be generated. The pins in each pair are separated by a space. Multiple pin-pairs can be specified in different lines. The software reports the RON between each pin-pair listed in the file. This is a required parameter.

Example

- The following command creates a RON report in the `output.rpt` file for the pin-pairs specified in the `pin_pair.txt` file:

© 2023 All Rights Reserved.

```
report_vfi_L_ron -pin_pair_file pin_pair.txt -output_file output.rpt
```


© 2023 All Rights Reserved.

Batch Commands for Querying

You can query resistors and nodes in Voltus-Fi-L 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:

```
emirutil postTclCmdFile=[conf.tcl]
```

For details of the above variable, see EMIR Control File Options Supported in Voltus-Fi-L.

Listed below are the batch commands that can be specified for querying specific data. Links to the command descriptions in the “Batch Mode Execution” chapter of the *Voltus-Fi Custom Power Integrity Solution XL User Guide* are provided below.

- search_res_id
- res_id_info
- search_node_id
- node_id_info

© 2023 All Rights Reserved.

Batch Commands for Viewing Results in vsaplot and vfiplot

The following commands are used to generate the VSA_RESULTS directory for IR drop and EM analysis in vsaplot for both Voltus-Fi-L and Voltus-Fi-XL. However, there are some parameters that are specific to either Voltus-Fi-L or Voltus-Fi-XL. The distinction is provided in the parameter descriptions.

- write_ir_vsaplot_db
- write_em_vsaplot_db

The details of the commands are provided below.

write_ir_vsaplot_db

```
write_ir_vsaplot_db
     -libname library_name
     -cellname cell_name
     -viewname view_name
     -extractedlibname library_name
     -extractedcellname cell_name
     -extractedviewname view_name
     -dfII_layermap_file dfII_layer_file
     -qrc_run_directory run_dir
     -qrc_run_name run_name
     -input_type {psf | vavodb}
     [-analysis_name analysis_name]
     [-anaysis_type analysis_type]
     [-vavodb_file file_name]
     [-start_time start_time]
     [-stop_time stop_time]
     results_file_name
```

Generates the vsaplot database for the IR drop analysis results.

Note: The parameters in “blue” in the above syntax are only applicable for Voltus-Fi-L.

Parameters

-analysis_name *analysis_name*

© 2023 All Rights Reserved.

	Specifies the analysis name, which could be either DC operating-point (DCOP) analysis or transient analysis. **Note:** This parameter is required to be specified when the `-input_type` is `psf`. **Note:** This parameter is only applicable for Voltus-Fi-L.
`-analysis_type` *`analysis_type`*	
	Specifies the type of analysis, `dc` for DCOP and `tran` for transient analysis. For transient analysis, you can specify separate start and stop times by using the `-start_time` and `-stop_time` parameters of this command. **Note:** This parameter is mandatory when the `-input_type` is `psf`. **Note:** This parameter is only applicable for Voltus-Fi-L.
`-cellname` *`cell_name`*	Specifies the schematic testbench cell name for the design in Voltus-Fi-L. Specifies the cell name for the design in Voltus-Fi-XL. This is a required parameter.
`-dfII_layermap_file` *`dfII_layer_file`*	
	Specifies the name of the DFII layermap file that provides the mapping between the extracted view names and the DFII layer names in Voltus-Fi-L. Specifies the name of the DFII layermap file that provides the mapping between the xDSPF file layer names and the DFII layer names in Voltus-Fi-XL. This is a required parameter. The parameters, `-dfII_layermap_file` and `-qrc_run_dir` are mutually exclusive in Voltus-Fi-XL but they are both required in Voltus-Fi-L.
`-extractedcellname` *`cell_name`*	Specifies the extracted view cell name for the design. This is a required parameter. **Note:** This parameter is only applicable for Voltus-Fi-L.
`-extractedlibname` *`lib_name`*	

© 2023 All Rights Reserved.

	Specifies the extracted view library name for the design. This is a required parameter. **Note:** This parameter is only applicable for Voltus-Fi-L.
`-extractedviewname` *`view_name`*	Specifies the extracted view name for the design. This is a required parameter. **Note:** This parameter is only applicable for Voltus-Fi-L.
`-input_type {psf | vavodb}`	Specifies the type of data input. You can specify either the vavo database (`vavodb`) or the `psf` database. This is a required parameter. **Note:** When `psf` data type is specified, the `-analysis_name` and `-analysis_type` parameters must be specified. **Note:** This parameter is only applicable for Voltus-Fi-L.
`-libname` *`lib_name`*	Specifies the schematic testbench library name for the design in Voltus-Fi-L. Specifies the library name for the design in Voltus-Fi-XL. This is a required parameter.
`-qrc_run_directory` *`run_dir`*	
	Specifies the name of the Quantus (QRC) run directory. This is a required parameter. **Note:** The parameters, `-dfII_layermap_file` and `-qrc_run_dir` are mutually exclusive in Voltus-Fi-XL but they are both required in Voltus-Fi-L.
`-qrc_run_name` *`run_name`*	
	Specifies the name of the Quantus (QRC) run name. This is a required parameter. **Note:** This parameter is always specified with the `-qrc_run_directory` parameter.
`results_file_name`	Specifies the name of the simulation directory in Voltus-Fi-L. Specifies the name of the file that contains the EMIR simulation results data in Voltus-Fi-XL. This is a required parameter.
`-start_time` *`start_time`*	

© 2023 All Rights Reserved.

	Specifies the start time for the transient analysis. If not specified, the software analyzes the full duration of the simulation. **Note:** This parameter is only specified when the `-input_type` selected is `psf` and the `-analysis_type` is `tran`. This parameter must be specified if the `-stop_time` parameter is specified. **Note:** This parameter is only applicable for Voltus-Fi-L.
`-stop_time` *`stop_time`*	Specifies the stop time for the transient analysis. If not specified, the software analyzes the full duration of the simulation. **Note:** This parameter is only specified when the `-input_type` selected is `psf` and the `-analysis_type` is `tran`. This parameter must be specified if the `-start_time` parameter is specified. **Note:** This parameter is only applicable for Voltus-Fi-L.
`-vavodb_file` *`file_name`*	
	Specifies the `vavo.db` database file required for the vavo_db flow. **Note:** This parameter is mandatory if `vavodb` is specified as the `-input_type`. **Note:** This parameter is only applicable for Voltus-Fi-L.
`-viewname` *`view_name`*	Specifies the schematic testbench view name for the design in voltus-Fi-L. Specifies the view name for the design in Voltus-Fi-XL. This is a required parameter.

Example:

- The following example shows how to generates the vsaplot database for the IR drop analysis results for Voltus-Fi-L.

```
write_ir_vsaplot_db \
```


© 2023 All Rights Reserved.

```
-libname amsPLL \
-cellname TB1_vco_single \
-viewname schematic \
-extractedlibname amsPLL \
-extractedcellname vco \
-extractedviewname av_extracted_vco_oa \
-dfII_layermap_file dfiilayer \
-qrc_run_directory vco_lvs_oa \
-qrc_run_name vco \
-input_type vavodb \
-vavodb_file tran_emir_vavo.db \
./simulation_flow/TB1_vco_single/spectre/schematic
```

- The following example shows how to generates the vfiplot database for the IR drop analysis results for Voltus-Fi-XL.

```
write_ir_vsaplot_db \
-libname testlib \
-cellname testcell \
-viewname testview \
-dfII_layermap_file dfiilayer \
xps.emirtap.emir0_bin
```


© 2023 All Rights Reserved.

write_em_vsaplot_db

```
write_em_vsaplot_db
-libname library_name
-cellname cell_name
-viewname view_name
-extractedlibname library_name
-extractedcellname cell_name
-extractedviewname view_name
-tech_file tech_file_name
[-layer_mapfile layer_map_filename]
[-em_only_ict_file em_only_ict_file_name]
-dfII_layermap_file dfII_layer_file
-qrc_run_directory run_dir
-qrc_run_name run_name
-input_type {psf | vavodb}
[-analysis_name analysis_name]
[-analysis_type analysis_type]
[-vavodb_file vavodb_file_name]
[-start_time start_time]
[-stop_time stop_time]
-type {javg jmax jabsavg jacpeak jacrms}
results_file_name
```

Generates the vsaplot database for the EM analysis results.

Note: The parameters in “blue” in the above syntax are only applicable for Voltus-Fi-L.

Parameters

-analysis_name *analysis_name*	
	Specifies the analysis name, which could be either DC operating-point (DCOP) analysis or transient analysis. **Note:** This parameter is required to be specified when the `-input_type` is `psf`. **Note:** This parameter is only applicable for Voltus-Fi-L.
-analysis_type *analysis_type*	
	Specifies the type of analysis, `dc` for DCOP and `tran` for transient analysis. For transient analysis, you can specify separate start and stop times by using the `-start_time` and `-stop_time` parameters of this command. **Note:** This parameter is mandatory when the `-input_type` is `psf`. **Note:** This parameter is only applicable for Voltus-Fi-L.

© 2023 All Rights Reserved.

`-cellname` *`cell_name`*	Specifies the schematic testbench cell name for the design in Voltus-Fi-L. Specifies the cell name for the design in Voltus-Fi-XL. This is a required parameter.
`-dfII_layermap_file` *`dfII_layer_file`*	
	Specifies the name of the DFII layermap file that provides the mapping between the extracted view names and the DFII layer names in Voltus-Fi-L. Specifies the name of the DFII layermap file that provides the mapping between the xDSPF file layer names and the DFII layer names in Voltus-Fi-XL. This is a required parameter. **Note:** The parameters, `-dfII_layermap_file` and `-qrc_run_dir` are mutually exclusive in Voltus-Fi-XL but they are both required in Voltus-Fi-L.
`-em_only_ict_file` *`em_only_ict_file_name`*	
	Specifies the name of the ICT file along with the process and EM model information that will be used for EM analysis. This is an optional parameter.
`-extractedcellname` *`cell_name`*	
	Specifies the extracted view cell name for the design. This is a required parameter. **Note:** This parameter is only applicable for Voltus-Fi-L.
`-extractedlibname` *`lib_name`*	
	Specifies the extracted view library name for the design. This is a required parameter. **Note:** This parameter is only applicable for Voltus-Fi-L.
`-extractedviewname` *`view_name`*	
	Specifies the extracted view name for the design. This is a required parameter. **Note:** This parameter is only applicable for Voltus-Fi-L.

© 2023 All Rights Reserved.

`-input_type {psf | vavodb}`	Specifies the type of data input. You can specify either the vavo database (`vavodb`) or the `psf` database. This is a required parameter. **Note:** When `psf` data type is specified, the `-analysis_name` and `-analysis_type` parameters must be specified. **Note:** This parameter is only applicable for Voltus-Fi-L.
`-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. **Note:** This parameter is only required if the layer names in the simulation database are different from the ones in the technology file.
`-libname` *`lib_name`*	Specifies the schematic testbench library name for the design in Voltus-Fi-L. Specifies the library name for the design in Voltus-Fi-XL. This is a required parameter.
`-qrc_run_directory` *`run_dir`*	
	Specifies the name of the Quantus (QRC) run directory. **Note:** The parameters, `-dfII_layermap_file` and `-qrc_run_dir` are mutually exclusive in Voltus-Fi-XL but they are both required in Voltus-Fi-L.
`-qrc_run_name` *`run_name`*	Specifies the name of the Quantus (QRC) run name. **Note:** This parameter is always specified with the `-qrc_run_directory` parameter. This is a required parameter.
`results_file_name`	Specifies the name of the simulation directory in Voltus-Fi-L. Specifies the name of the file that contains the EMIR simulation results data in Voltus-Fi-XL. This is a required parameter.

© 2023 All Rights Reserved.

`-start_time` *`start_time`*	Specifies the start time for the transient analysis. If not specified, the software analyzes the full duration of the simulation. **Note:** This parameter is only specified when the `-input_type` selected is `psf` and the `-analysis_type` is `tran`. This parameter must be specified if the `-stop_time` parameter is specified. **Note:** This parameter is only applicable for Voltus-Fi-L.
`-stop_time` *`stop_time`*	Specifies the stop time for the transient analysis. If not specified, the software analyzes the full duration of the simulation. **Note:** This parameter is only specified when the `-input_type` selected is `psf` and the `-analysis_type` is `tran`. This parameter must be specified if the `-start_time` parameter is specified. **Note:** This parameter is only applicable for Voltus-Fi-L.
`tech_file` *`tech_file_name`*	Specifies the name of the `qrcTechFile` or the `emDataFile` used for EM analysis. This is a required parameter.
`-type {javg jmax jabsavg jacpeak jacrms}`	

© 2023 All Rights Reserved.

	Specifies the type of analysis for which you want to view results in the vfiplot display window. `javg`: For viewing the results of average Current Density analysis data `jmax`: For viewing the results of peak Current Density data analysis `jabsavg`: For viewing the results of average absolute Current Density data analysis `jacpeak`: For viewing the results of AC peak Current Density data analysis `jacrms`: For viewing the results of RMS Current Density analysis data analysis This is a required parameter.
`-vavodb_file` *`file_name`*	Specifies the `vavo.db` database file required for the vavo_db flow. **Note:** This parameter is mandatory if `vavodb` is specified as the `-input_type`. **Note:** This parameter is only applicable for Voltus-Fi-L.
`-viewname` *`view_name`*	Specifies the schematic testbench view name for the design in Voltus-Fi-L. Specifies the view name for the design in Voltus-Fi-XL. This is a required parameter.

Example:

- The following example shows how to generates the vsaplot database for the EM analysis results for Voltus-Fi-L.

```
write_em_vsaplot_db \
-libname amsPLL \
-cellname TB1_vco_single \
-viewname schematic \
-extractedlibname amsPLL \
-extractedcellname vco \
-extractedviewname av_extracted_vco_oa \
-tech_file qrcTechFile \
```


© 2023 All Rights Reserved.

```
-layer_mapfile contactmapfile \
-dfII_layermap_file dfiilayer \
-qrc_run_directory vco_lvs_oa \
-qrc_run_name vco \
-input_type psf \
-analysis_name dcOp-dc \
-analysis_type dc \
-start_time 0 \
-stop_time 170 \
-type javg
./simulation_flow/TB1_vco_single/spectre/schematic
```

- The following example shows how to generates the vfiplot database for the EM analysis results for Voltus-Fi-XL.

```
write_em_vsaplot_db \
-libname testlib \
-cellname testcell \
-viewname testview \
-tech_file qrcTechFile \
-layer_mapfile contactmapfile \
-dfII_layermap_file dfiilayer \
-type jacrms
xps.emirtap.emir0_bin
```


© 2023 All Rights Reserved.

Supported VPSL Batch File Commands

Currently, only the commands listed below are supported in the Voltus-Fi-L extracted flow. If there are any additional commands in the specified legacy (VPS-L) `cmd_file`, they will be ignored.

The following commands are required to be in the specified batch file:

- _vsa_extracted_lib_cell_view
- _vsa_qrc_run_directory
- _vsa_qrc_run_name
- _vsa_simulation_directory
- _vsa_em_tech_file
- _vsa_em_data_file
- _vsa_analysis_data
- _vsa_vavo_db_file
- _vsa_testbench_lib_cell_view

The following commands are optional:

- _vsa_hierarchy
- _vsa_analyze_em
- _vsa_analyze_ir
- _vsa_pres_cellname
- _vsa_em_map_file

Note: The following hold true when using the above commands:

- If the required commands are not present in the specified batch file, the job is aborted with the associated error message.
- If the extracted view data cannot be loaded, or the appropriate `dbFile/extview.tmp` file is not located, the tool will throw an error before performing the analysis.
- If there are any errors while reading the Voltus-Fi data, the tool will flag these errors.

The descriptions of the commands are provided below.

© 2023 All Rights Reserved.

_vsa_extracted_lib_cell_view

```
_vsa_extracted_lib_cell_view "library" "cell" "view"
```

Specifies the RCX extracted view used as input for the analysis. This is a required command.

Parameters

`library`	Specifies the name of the library.
`cell`	Specifies the name of the cell within the library.
`view`	Specifies the name of the view within the cell.

Example

The following command specifies the RCX extracted view used as input for the analysis:

```
_vsa_extracted_lib_cell_view "amsPLL" "vco" "av_R"
```

_vsa_qrc_run_directory

```
_vsa_qrc_run_directory "run_dir_name"
```

Specifies the directory name of the Quantus QRC run. This is a required command.

Parameter

`run_dir_name`	Specifies the directory name of the Quantus QRC run.

Example

The following command specifies the directory name of the Quantus QRC run:

```
_vsa_qrc_run_directory "../custom/LVS"
```

_vsa_qrc_run_name

```
_vsa_qrc_run_name "run_name"
```


© 2023 All Rights Reserved.

Specifies the Quantus QRC run name. This is a required command.

Example

```
_vsa_qrc_run_name "vco"
```

_vsa_simulation_directory

```
_vsa_simulation_directory "sim_dir_path"
```

Specifies the input simulation run. This is a required command.

Parameter

`sim_dir_path`	Specifies the path of the simulation directory.

Example

The following command specifies the path of the simulation directory:

```
_vsa_simulation_directory "/custom/simulation/TB1_vco_bak/spectre/schematic"
```

_vsa_em_tech_file

```
_vsa_em_tech_file "../path/fileName"
```

Specifies the path and filename of the technology file. This is required for EM analysis. If the technology file is not specified, the software gives an error. The following files are supported: The ICT file, the QRC tech file, and the emDataFile.

Note: The `_vsa_em_tech_file` and the `_vsa_em_data_file` commands are only required in the EM analysis flow. One of these two commands must be specified.

Example

```
_vsa_em_tech_file "../amsPLL_oa_flow/qrcTechFile"
```

_vsa_em_data_file

```
_vsa_em_data_file "../path/fileName"
```


© 2023 All Rights Reserved.

Specifies the path and file name of the `emDataFile` required for EM analysis. This is an optional command. If no other location is specified, the current working directory is used. If the `emDataFile` is not found, an error is returned.

Note: The `_vsa_em_tech_file` and the `_vsa_em_data_file` commands are only required in the EM analysis flow. One of these two commands must be specified.

Parameter

`../path/fileName`	Specifies the path and name of the EM rule specification file.

Example

The following command specifies the path and filename of the `emDataFile` required for EM analysis:

```
_vsa_em_data_file "../custom/oa/gpdk/emDataFile.txt"
```

_vsa_analysis_data

```
_vsa_analysis_data "TRANSIENT" | "DCOP"
```

Specifies the type of analysis to perform.

For transient analysis, separate start and stop times can be specified for both IR Drop and EM analysis using the `_vsa_analyze_ir` and `_vsa_analyze_em` commands. If start and stop times are not specified, the default is the full duration of the simulation.

Default: `"TRANSIENT"`

Parameters

`TRANSIENT`	Specifies that transient analysis should be performed.
`DCOP`	Specifies that DC operating point analysis should be performed.

© 2023 All Rights Reserved.

Examples

- The following command specifies that transient analysis be performed for the full duration:

  ```
  _vsa_analysis_data "TRANSIENT"
  ```

- The following command specifies that DCOP analysis be performed:

  ```
  _vsa_analysis_data "DCOP"
  ```

_vsa_vavo_db_file

```
_vsa_vavo_db_file "../path/FileName"
```

Specifies the `vavo.db` database file required for the `vavo_db` flow.

Example

```
_vsa_vavo_db_file "../amsPLL_oa_flow/vavo.db"
```

_vsa_testbench_lib_cell_view

```
_vsa_testbench_lib_cell_view "library" "cell" "view"
```

Specifies the schematic testbench. This is a required command.

Parameters

`library`	Specifies the name of the library.
`cell`	Specifies the name of the cell within the library.
`view`	Specifies the name of the view within the cell.

Example

The following command specifies the schematic testbench used as input for the analysis:

```
_vsa_testbench_lib_cell_view "amsPLL" "tb_vco_bak" "schematic"
```

_vsa_hierarchy

```
_vsa_hierarchy "hierarchy_name"
```

Specifies the hierarchy of the design. This is an optional command.

© 2023 All Rights Reserved.

Parameter

`hierarchy_name`	Specifies the hierarchy name for the design.

Example

The following command specifies the hierarchy of the design:

```
_vsa_hierarchy "/X1"
```

_vsa_analyze_em

```
_vsa_analyze_em
```

Specifies the start and stop times for transient EM analysis. If the start and stop times are not specified, the full duration of the simulation is analyzed.

To specify DCOP analysis or to analyze the full duration of a transient analysis, use `_vsa_analyze_em`.

Note: This command is only required in the EM analysis flow.

Parameters

`startTime`	Specifies the start time for the DCOP or transient analysis.
`stopTime`	Specifies the stop time for the DCOP or transient analysis.

Example

The following command is used to specify the start and stop times for transient EM analysis:

```
_vsa_analyze_em _vsa_analyze_em "0" "1n"
```

_vsa_analyze_ir

Specifies the start and stop times for transient IR drop analysis. If the start and stop times are not specified, the full duration of the simulation is analyzed.

To specify DCOP analysis or to analyze the full duration of a transient analysis, use `_vsa_analyze_ir`.

© 2023 All Rights Reserved.

Parameters

`start_time`	Specifies the start time for the transient analysis.
`stop_time`	Specifies the stop time for the transient analysis.

Examples

- The following command specifies the start and stop times for transient IR drop analysis:

  ```
  _vsa_analyze_ir "0" "150n"
  ```

- The following command is used for running DCOP analysis or full duration of transient IR analysis:

  ```
  _vsa_analyze_ir
  ```

_vsa_pres_cellname

```
_vsa_pres_cellname "cellname"
```

Specifies the cell name of the parasitic resistor used by RCX that is added to the extracted view. This command is optional.

Default: `"presistor"`

Parameter

`cellname`	Specifies the cell name of the parasitic resistor within the cell.

Example

The following command specifies the name of the parasitic resistor used by RCX:

```
_vsa_pres_cellname "presistor"
```

_vsa_em_map_file

```
_vsa_em_map_file "../path/mapFileName"
```

Specifies the layermap file, which is the name of the mapping file to be used for EM analysis. It provides a mapping between the layer names in the simulation database and the layer names in the technology file specified using the `_vsa_em_tech_file` command.

© 2023 All Rights Reserved.

Note: This command is only required in the EM analysis flow.

Example

```
_vsa_em_map_file "../amsPLL_oa_flow/DFIIlayermap"
```


© 2023 All Rights Reserved.

Sample Batch Mode Command File

A sample legacy (VPS-L) batch mode command file is provided below.

```
;vsa batch mode command file
;common commands for EM and IR
_vsa_extracted_lib_cell_view “amsPLL” “vco” “av_extracted_sol”
_vsa_testbench_lib_cell_view “amsPLL” “TB1_vco_single” “schematic”
_vsa_simulation_directory “/custom/TB1_vco_single/spectre/schematic/”
_vsa_pres_cellname “presistor”
_vsa_analysis_data “TRANSIENT”

; IR specific commands
;_vsa_analyze_IR “0” “170n”
_vsa_analyze_ir

;following command are for EM analysis
_vsa_qrc_run_directory  “/custom/WORK/AV/LVS_emir”
_vsa_em_data_file “emDataFile.txt”
;_vsa_analyze_em "0" "1n"
_vsa_analyze_em

exit
```


© 2023 All Rights Reserved.

The Configuration File Support

Yo can use the configuration file to run Voltus-Fi-L in batch mode. The following command is used:

```
vfibatch -c <emir conf> -db <simulation directory path> -outdir
<output directory>
```

Note: The simulation directory path specified using the above command is the same as that specified in the VPS-L `cmd_file`.

Detailed information about the control file options that can be specified in the configuration file are provided in the subsequent section.

EMIR Control File Options Supported in Voltus-Fi-L

The following table summarizes the EMIR control file options supported in Voltus-Fi-L. These are specified in the configuration file using the `emirutil` command.

Table 12-1 Supported EMIR Control File Options

Keyword	Option Set	Explanation	Default Value
`emirutil`	`analysisName="analysis_name"`	Specifies the analysis name for DCOP analysis and transient analysis.	`none`
	`analysisType="dc | tran"`	Specifies the type of analysis, `dc` for DCOP analysis and `tran` for transient analysis.	`none`
	`analysisStartTime="value"`	Specifies the start time for transient analysis. If not specified, the software analyzes the full duration of the simulation. For example, `emirutil analysisStartTime="0"`	`none`

© 2023 All Rights Reserved.

	`analysisStopTime="value"`	Specifies the stop time for transient analysis. If not specified, the software analyzes the full duration of the simulation. For example, `emirutil analysisStopTime="170n"`	`none`
	`techfile="emDataFile"`	Specifies the EM rule file in the emdatafile, qrctechfile, or ictfile format. Case-insensitive layer names are supported allowing easier match between the extracted view and the technology files. Alternatively, you can use the `emdatafile="emDataFile"` option to specify the techfile in the emdata format, `qrctechfile="qrcTechFile"` to specify the Quantus QRC tech files, and `ictfile="ictfile"` to specify the file in ICT format. The `layermap` file provides the mapping between the extracted view layers and the layout database. It is applicable only for `qrcTechfile` and `ictfile`.	`none`
	`EMOnlyICTFile="emOnlyICTFile"`	Specifies the process and EM models to be used for EM analysis. **Note**: This file can only be provided with the `qrcTechFile` flow.	`none`

© 2023 All Rights Reserved.

	`postTclCmdFile=[conf.tcl]`	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. For details about the commands that can be written in the file (`conf.tcl`), see Batch Command for RON Reports.	`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`
	`report=[text | html]`	Defines the type of report created by `emirutil`. `text` - produces only text report. `html` - creates the report in html format. **Note**: Multiple entries are supported.	`text`
	`notation=[s | e]`	Notation for the text and html reports. `e` - engineering scale number (for example, 5.02m) `s` - scientific notation (for example, 5.02e-3)	`e`
	`sort_by_net=yes|no`	Report IR and EM results per net, or all nets. `yes` - report EMIR results per net `no` - combine EMIR results of all nets into one report.	
	`filter_ir_threshold=`*`value`*	Defines the IR voltage drop threshold for the node being reported. `filter_ir_threshold=0.01` reports all the nodes with IR drop above 0.01V.	

© 2023 All Rights Reserved.

	`filter_em_threshold=`*`value`*	Defines the EM analysis threshold value for the resistors being reported. The value for `filter_em_threshold` is the J/Jlimit value. `filter_em_threshold=1` reports only those resistors that have a J/Jlimit value above 1. **Note**: Specifying this option is equivalent to filtering all passed resistors.	
	`enableDesignResistorPeakFactor=[true | false]`	Specifies the addition of the factor, $1/(r)^{1/2}$, in the current limit calculation for the AC-Peak analysis of the design resistor. The software, by default, calculates the current limit for the AC-Peak analysis of the design resistor using the following equation: $\mathbf{I}_{peak\_ac(limit)}=\mathbf{I}_{peak\_dc(limit)}$ To apply the factor, $1/(r)^{1/2}$, in your calculation, set this variable to `true`. The current limit will then be calculated in the following manner: $\mathbf{I}_{peak\_ac(limit)}=\mathbf{I}_{peak\_dc(limit)}/(r)^{1/2}$	`false`
	`enableDesignResistorScaling=[true | false]`	Enables the scaling of the width and length values based on the `layout_scale` factor specified in the technology file before the EM rules are applied for EM analysis of design resistors. By default, the scaling of design resistors is enabled.	`true`

© 2023 All Rights Reserved.

	`extendPWLMatch=[ true | false]`	When the EM rules are specified in terms of PWL and the width of metal or the area of via does not exactly match any width/area of the PWL, the EM limit will not be calculated for this metal resistor/via. When set to `true`, the software interpolates the limit value for the `resistor/via` from the selected rule PWL as follows: minimum of PWL i^{th} value * (metal's width or via's area) / (PWL i^{th} width or PWL `area`).	`false`
	`process_scale=`*`value`*	Specifies the value by which the width and length values will be multiplied before the EM rules are applied.	`1.0`

© 2023 All Rights Reserved.

	`prioritizedPwlRulesLayers=[all | none | list_layers]`	Specifies the list of layers for which area-based PWL search is performed first. When no match is found, then non-PWL rules are used. The default value of this variable is `"all"`, which means that this behavior is enabled for all layers. To enable the behavior for specific layers, you can specify a list of layers. To disable this behavior for all layers, set the value of this variable to "`none`". In the GUI, you can set this variable in the Variables form. Example rules: `em_jmax_dc_avg 10 jmax_factor 90 2.384 95 2.078` - non-PWL baseline rule `em_jmax_dc_avg PWL 0.3582 0.001444 0.5694 0.003364 0.7164 0.0038 jmax_factor 90 2.384` - PWL rule When both the above rules are specified, the second rule is given priority over the first.	`all`
	`halfPeakAnalysis=[true|false]`	Defines the time duration, $\mathbf{T}_d$ as the duration for which the pulse current is greater than half the peak current (I_{peak}) value for dynamic AC peak analysis. By default, this parameter is set to `false`. For example, `emirutil halfPeakAnalysis=true`	`false`

© 2023 All Rights Reserved.

	`pwc_threshold=1e-06`	Defines the current (`I`) threshold value for locating the start of the pulse to calculate the time duration, $\mathbf{T_d}$ for dynamic AC peak analysis. The default value is `1e-06(1uA)`. For example, `emirutil pwc_threshold=1e-06`	`1e-06`
	`idirn=[true|false]`	Prints the current direction in the EM report (*Current Direction* column). For metal resistors, the current direction is printed. That is, *W* means that the current is flowing from east to west. For via resistors, the direction from/to layer is printed.	`false`
	`Tlife=hours | years`	Specifies the lifetime for which the EM analysis will be performed. You can specify the value in either years or `hours`. For example: `emirutil Tlife=”20000hours”`	`years`

© 2023 All Rights Reserved.

	`emLimitScale=`*`scale_factor_value`*
	|
	`"scale_factor_netfile"`
		Scales the EM limits of the parasitic resistors in the design before generating the EM reports. You can either apply the same scale factor to all the nets or specify different scale factors for different nets.
		
		To apply the same scale factor to the parasitic resistors of all the nets in the design, provide the value of the scale factor.
		
		`emirutil emLimitScale=`*`scale_factor_value`*
		
		For example,
		
		`emirutil emLimitScale=0.04`
		
		Note: This is applicable to `avg`, `avg abs`, and `rms` analyses. For the `avg` and `avg abs` analyses, the limit is directly multiplied by this factor. For rms analysis, the limit is multiplied by the root of this factor.
		
		To apply different scale factors to different nets in the design, provide the name of the file that contains information about the net names and the corresponding scale factors to be applied to these nets.
		
		`emirutil emLimitScale=`*`"scale_factor_netfile"`*
		
		For example,
		
		`emirutil emLimitScale="scaleLimitFile"`
			`none`

© 2023 All Rights Reserved.

	`rmsMetalLineNumberSignal=`*`number`*	Specifies the value of the power on the metal lines for RMS EM analysis of signal nets. If this option is not specified, the default value of `0` is considered, which means the software uses the `em_ac_rms_metal_line_num` value specified in the ICT file. However, if this option is specified, it overrides the `em_ac_rms_metal_line_num` value specified in the ICT file.	`0`
	`recovery_factor`	Defines the recovery factor for iavg calculation with recovery factor. If specified, then `iavg=max(|iavgpos|,|iavgneg|) - recovery_factor*min(|iavgpos|,|iavgneg|).` If not specified, then `iavg=iavgpos+iavgneg`	

© 2023 All Rights Reserved.

	`print_em_report`	Defines the EM analysis report file to be generated in the Voltus-Fi-L flow. `filename`: create a report with the specified filename `net`: create a report for the specified nets - all power nets (`all_power`), all signal nets (`all_signal`), all nets (`all nets`), or net (`netname`). `threshold` - create a report for resistors or nodes that have a threshold value of EM ratio above the specified value `type` - create a report for specified analysis type; `javg, jmax, jabsavg, jacpeak, jacrms, rc, rcavg, rcrms.` `rule` - specifies the custom EM rule for which the report should be generated	
	`extendedreport=true`	Specifies that the following information should be included in the EM analysis report file: ■ `Td`: time duration in micro second or total 'On Time" period ■ `dutyR`: duty ratio ■ I_{limit}: current limit	`false`
	`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`

© 2023 All Rights Reserved.

	`reportAllResistor=true | false`	When set to `true`, reports all the resistors of the specified nets in the `"#.rpt_all"` file. **Note:** If both, `reportUnmatchedRuleResistor` and `reportAllResistor`, are set to `true`, the `reportAllResistor` setting gets priority.	`false`
	`layer_filter=[layer1 layer2...]`	Specifies the layers that are to be filtered out from the EMIR analysis reports. For example, `emirutil layer_filter=[contodcont]` **Note:** If you have specified this option and also set the `reportAllResistor` option to `true`, the resistor information for the specified layers will be filtered out from the reports.	`none`
	`disableLayoutScale=true`	When set to `true`, disables the layout_scale factor defined in the ICT file for Voltus-Fi-L EM analysis. The layout_scale factor is the ratio of the scaled dimensions to 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.	`false`

© 2023 All Rights Reserved.

	`simpleSingleViaDefinition=true | false`	The keyword `single` is used to differentiate between a single square cut via and a square via array. It is applicable to a square via, with width equal to only '`vc_width/min_width`'. To support the `single` keyword for all via sizes, set the `simpleSingleViaDefinition` parameter to `true`.	`false`
	`hierarchy="`*`hierarchy_name`*`"`	Specifies the hierarchy of the design. For example, `emirutil hierarchy="/X1"`	`none`
	`presCellName="`*`presistor_name`*`"`	Specifies the name of the parasitic resistor that is added to the extracted view.	`presistor`
	`view=[lib=<`*`lib`*`> cell=<`*`cell`*`> view=<`*`layout`*`>]`	Specifies the name of the extracted view in the library, cell, view format.	`none`
	`view_tb=[lib=<`*`lib`*`> cell=<`*`cell`*`> view=<`*`schematic`*`>]`	Specifies the name of the schematic testbench in the library, cell, view format.	`none`

For details of the `emirutil` commands that are the same as those used in Voltus-Fi-XL, see EMIR Control File Options Supported in Voltus-Fi-XL in the “Data Preparation” chapter of the *Voltus-Fi Custom Power Integrity Solution XL User Guide*.

Sample Configuration Files

The sample `conf` files for different analyses types are provided below.

DC Operating Point (DCOP) Analysis

```
================================================
net name=[*] analysis=[iavg irms imax vavg vmax]
emirutil view=[lib=amsPLL cell=vco view=av_extracted_vco_oa]
emirutil view_tb=[lib=amsPLL cell=TB1_vco_single view=schematic]
```


© 2023 All Rights Reserved.

```
emirutil analysisType="dc"
emirutil analysisName="dcOp-dc"
emirutil presCellName="presistor"
emirutil qrc_output=[runDir=vco_lvs_oa runName=vco]
emirutil hierarchy="/X1"
emirutil techfile="qrcTecFile"
```

Transient Analysis

```
================================================
net name=[*] analysis=[iavg irms imax vavg vmax]
emirutil view=[lib=amsPLL cell=vco view=av_extracted_vco_oa]
emirutil view_tb=[lib=amsPLL cell=TB1_vco_single view=schematic]
emirutil analysisType="tran"
emirutil analysisName="tran-tran"
emirutil presCellName="presistor"
emirutil qrc_output=[runDir=vco_lvs_oa runName=vco]
emirutil analysisStartTime="0"
emirutil analysisStopTime="170n"
emirutil hierarchy="/X1"
emirutil techfile="qrcTecFile"
```

Analysis using the vavo-db File

```
================================================
net name=[*] analysis=[iavg irms imax vavg vmax]
emirutil view=[lib=amsPLL cell=vco view=av_extracted_vco_oa]
emirutil view_tb=[lib=amsPLL cell=TB1_vco_single view=schematic]
emirutil presCellName="presistor"
emirutil qrc_output=[runDir=vco_lvs_oa runName=vco]
emirutil hierarchy="/X1"
emirutil techfile="qrcTecFile"
emirutil vavoDBFile="tran_emir_vavo.db"
```


© 2023 All Rights Reserved.

© 2023 All Rights Reserved.

13

Variables

© 2023 All Rights Reserved.

Overview

This chapter provides a list of variables that you can specify in Voltus-Fi-L. You can set these variables 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 "The EM Analysis GUI Flow" 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 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-L GUI

Using the set_variable Command in the Voltus-Fi Command File

When using the `vfibatch` command to run Voltus-Fi-L 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.

© 2023 All Rights Reserved.

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 the batch mode, you can set the variables using the emirutil command in the control file. The command syntax and example are 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-L in the “Batch Mode Execution” 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 the 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.

© 2023 All Rights Reserved.

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 because 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-L 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
- deltaT
- disableLayoutScale
- dynamicACPeak
- emLimitScale
- enableDesignResistorPeakFactor
- enableDesignResistorScaling
- extendPWLMatch
- halfPeakAnalysis
- optimisticEMRuleSelection
- powerRailSupplyNets
- powerRailRules
- powerRailGroundNets
- process_scale
- prioritizedPwlRulesLayers
- pwc_threshold
- reportUnmatchedRuleResistor
- rmsMetalLineNumberSignal

© 2023 All Rights Reserved.

- rmsMultiplierPin
- rmsMultiplierSignal
- simpleSingleViaDefinition
- splitACDCRules
- Tj
- Tlife
- useRecoveryFactor

The above variables are described in detail in the subsequent sections.

applyRThreshold

Specifies the threshold ratio “r” for AC-Peak analysis. The default value of the threshold ratio is `0.05`.

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-L considers a default temperature of 5 degrees Celsius. This is an ideal condition in which 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 to the drawn dimensions. The ICT file is defined based on the scaled dimensions. Quantus QRC 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-L EM analysis flow. By default, this variable is set to `false`.

© 2023 All Rights Reserved.

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=$\mathbf{I}_{abs\text{-}avg}$ / $\mathbf{I}_{peak\_dc}$, by default (“`false`”)

And

`r`=$\mathbf{T}_d / \mathbf{T}_{total}$, when the `dynamicACPeak` variable is set.

$\mathbf{T}_{total}$=total transient time

$\mathbf{T}_d$=the time duration in micro second or the total “On Time” period, where `abs(I)>(max(abs(I)/2.0)` during transient analysis

Note: Voltus-Fi-L does not exactly calculate $\mathbf{T}_d$ as the `max(abs(I)/2.0)` period. Instead, it calculates $\mathbf{T}_d$ by measuring the pulse at half energy level. This is because of performance reasons. The parameter `pwc_threshold` defines the current threshold for finding the start of the pulse. The `pwc_threshold` value is specified using the `emirutil` command in the EMIR configuration file. The default value is `1e-06(1uA)`

When the parameter `halfPeakAnalysis` is set to `true`, Voltus-Fi-L exactly calculates the $\mathbf{T}_d$ as the `max(abs(I)/2.0)` period.

$\mathbf{T}_d$ has the following different values depending upon the value of `dynamicACPeak`:

- $\mathbf{T}_d$=the time duration of maximum peak, when `dynamicACPeak` is set to `true`
- $\mathbf{T}_d$=sum of time durations of different peaks, $\mathbf{T}_{d1}+\mathbf{T}_{d2}+\mathbf{T}_{dN}$.... (when `dynamicACPeak` is set to `multiplePeak`)

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 $\mathbf{I}_{peak\_ac}$ 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.

© 2023 All Rights Reserved.

emLimitScale

Scales the EM limits of the parasitic resistors in the design before generating the EM reports. You can either apply the same scale factor to all the nets, or specify different scale factors for different nets. Both use models are detailed below.

- To apply the same scale factor to the parasitic resistors of all the nets in the design, provide the value of the scale factor using this variable. For example,

  `emLimitScale` *`scale_factor`*

  ```
  emLimitScale 0.04
  ```

  Note: This is applicable to `avg`, `avg abs`, and `rms` analyses. For the `avg` and `avg abs` analyses, the limit is directly multiplied by this factor. For the `rms` analysis, the limit is multiplied by the root of this factor.

- To apply different scale factors to different nets in the design, provide the name of the file that contains information about the net names and the corresponding scale factors to be applied to these nets. For example,

  `emLimitScale` *`scale_factor_netfile`*

  An example of the scale factor file is as follows:

  ```
  net_name1 scale_factor1
  net_name2 scale_factor2
  ```

enableDesignResistorPeakFactor

Specifies the addition of the factor, $1/(r)^{1/2}$, in the current limit calculation for the AC-Peak analysis of the design resistor.

By default, the software calculates the current limit for the AC-Peak analysis of the design resistor using the following equation:

$$\mathbf{I}_{peak\_ac(limit)} = \mathbf{I}_{peak\_dc(limit)}$$

Apply the factor, $1/(r)^{1/2}$,in your calculation, then set the `enableDesignResistorPeakFactor` variable to `true`. The current limit will then be calculated in the following manner:

$$\mathbf{I}_{peak\_ac(limit)} = \mathbf{I}_{peak\_dc(limit)}/(r)^{1/2}$$

By default, this variable is set to `false`.

© 2023 All Rights Reserved.

enableDesignResistorScaling

Enables the scaling of the width and length values based on the `layout_scale` factor specified in the technology file before the EM rules are applied for EM analysis of design resistors. By default, this variable is set to `true`. In the GUI, you can set this variable in the Variables form.

In the batch mode, you can set this variable in the `vfibatch` command file by using the following command:

```
set_variable enableDesignResistorScaling false
```

You can also set this variable in the EMIR control file by using the following command:

```
emirutil enableDesignResistorScaling=false
```

extendPWLMatch

When the EM rules are specified in terms of piece-wise linear (PWL) and the width of metal or the area of via does not match exactly with any width/area of the PWL, the EM limit will not be calculated for this metal resistor/via.

When this variable is set to `true`, the software interpolates the limit value for the resistor/via from the selected rule PWL as follows:

The minimum of PWL i^{th} value *(metal's width or via's area)/(PWL i^{th} width or PWL area)

halfPeakAnalysis

Defines the time duration, $\mathbf{T}_d$ as the duration for which the pulse current is greater than half the peak current (I_{peak}) value for dynamic AC peak analysis. By default, this variable is set to `false`.

In the batch mode, you can set this variable in the `vfibatch` command file by using the following command:

```
set_variable halfPeakAnalysis true
```

You can also set this variable in the EMIR control file by using the following command:

```
emirutil halfPeakAnalysis=true
```


© 2023 All Rights Reserved.

optimisticEMRuleSelection

Specifies whether optimistic or pessimistic values are selected for the specified rules. This is applicable when multiple rules are selected after all the 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 the 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 layers
  - 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)`.

© 2023 All Rights Reserved.

- `""(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 Variables form that opens from the EM tab of the IR/EM Results form. 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
```

prioritizedPwlRulesLayers

Specifies the list of layers for which area-based PWL search is performed first. When no match is found, non-PWL rules are used. The default value of this variable is `"all"`, which means that this behavior is enabled for all layers.

To enable the behavior for specific layers, you can specify a list of layers.

To disable this behavior for all layers, set the value of this variable to "`none`". In the GUI, you can set this variable in the Variables form.

In the batch mode, you can set this variable in the `vfibatch` command file by using the following command:

```
set_variable prioritizedPwlRulesLayers none
```

You can also set this variable in the EMIR control file by using the following command:

```
emirutil prioritizedPwlRulesLayers=none
```


© 2023 All Rights Reserved.

pwc_threshold

Defines the current (`I`) threshold value for locating the start of the pulse to calculate the time duration, $\mathbf{T_d}$ for dynamic AC peak analysis. The default value is `1e-06(1uA)`.

In the batch mode, you can set this variable in the `vfibatch` command file by using the following command:

```
set_variable pwc_threshold 1e-06
```

You can also set this variable in the EMIR control file by using the following command:

```
emirutil pwc_threshold =1e-06
```

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-L.

rmsMetalLineNumberSignal

Specifies the value of power on the metal lines for RMS EM analysis of signal nets.

If this option is not specified, the default value of `0` is considered, which means the software uses the `em_ac_rms_metal_line_num` value specified in the ICT file.

However, if this option is specified, it overrides the `em_ac_rms_metal_line_num` value specified in the ICT file.

You can also set this variable in the config file in the following manner:

```
emirutil rmsMetalLineNumberSignal=number
```

The default value is `0`.

© 2023 All Rights Reserved.

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`.

simpleSingleViaDefinition

The keyword, `single` is used to differentiate between a single square cut via and square via array. It is applicable to a square via, with width only equal to '`vc_width/min_width`'. To support the `single` keyword for all the via sizes, set the `simpleSingleViaDefinition` variable to `true`.

By default, this variable is set to `false`.

You can also set this variable in the config file in the following manner:

```
emirutil simpleSingleViaDefinition=true
```

For details, see EMIR Control File Options Supported in Voltus-Fi-L.

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 “EM Rules Specification” 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:

© 2023 All Rights Reserved.

```
Tlife="20000hours"
```

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

© 2023 All Rights Reserved.

© 2023 All Rights Reserved.

14

File Formats

© 2023 All Rights Reserved.

Rail Analysis Files

This section lists the rail analysis file formats.

- DFII Layer Map File
- Layer Map File

© 2023 All Rights Reserved.

DFII Layer Map File

The APS/XPS-to-DFII layer map file provides the mapping between the layer names in the extracted view to the DFII layer names.

Multiple extraction layers can be mapped to the same DFII layer.

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 extracted view that is to be mapped to the DFII layer.

`dfII_layer_name` specifies the name of the DFII layer to which the extracted view 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
```

Important

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.

© 2023 All Rights Reserved.

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. It maps the layer names in the xDSPF file to the ICT layer names. It is needed under either of the following circumstances:

- When Quantus package uses the `layerSetupFile`, which includes the `"model"` keywords to define parasitic resistor model names. For example,

  ```
  pro_layer=M1 ext_layer=MET1 model=metal1
  ```

  In this case, the `"metal1"` to `"M1"` mapping needs to be specified in the layer map file.

  The sample command for the above in the `.cdsinit` file will be specified as follows:

  ```
  envSetVal("spectre.envOpts" "emLayerMap" 'string "/your-path/layermapfile")
  ```

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
```


© 2023 All Rights Reserved.

Power-Grid View Creation Files

This section lists the power-grid view creation file formats.

- DSPF Layermap File
- LEF Layermap File
- PGDB Layermap File

DSPF Layermap File

This layermap file contains the layer type and layer stack information for the DSPF from the top-most layer to the bottom-most layer. If this order is not followed, the PGDB will not be created.

Sample file name: `layermap_dspf.txt`

Format:

```
Layer type      Process layername
```

Format Description:

`Layer type` specifies the type of layer: metal, via, or local

`Process layername` specifies the name of the layer

Example:

```
#   Layer       Process
#   type        layername
#  ---------   ------------
    metal       m4
    via         via3
    metal       m3
    via         via2
    metal       m2
    via         via1
    metal       m1
    metal       po
    metal       mdtap
```


© 2023 All Rights Reserved.

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.

Sample file name: `lefdef.layermap`

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    M1

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.

© 2023 All Rights Reserved.

PGDB Layermap File

A layer mapping file is required for the 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.

Sample file name: `pgdb.map`

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    metal1
  via    VIA_1                 pgdb    VIA1
  metal  METAL_2               pgdb    metal2
  via    VIA_2                 pgdb    VIA2
  diff   OD                    pgdb    nwires
  diff   OD                    pgdb    mwires
```

Note: When generating PGVs from Voltus-Fi, some of the layers are not required for analysis, but must be preserved to maintain connectivity. Voltus-Fi supports mapping of these layers to a special layer "`connectivity`" in the layermap file. An example is shown below.

Example:

```
type    technology_layer_name  pgdb    xDSPF_layer_name
diff    connectivity           pgdb    nwires
diff    connectivity           pgdb    mwires
```


© 2023 All Rights Reserved.

© 2023 All Rights Reserved.