cadence®

Power IR/EM User Guide

Product Version ICADVM20.1
October 2020

© 2020 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.

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

Preface 11
Licensing Requirements 12
Related Documentation 12
Voltus-Fi Custom Power Integrity Solution Documentation 12
What's New and KPNS 12
Other Documentation 12
Additional Learning Resources 13
Video Library 13
Virtuoso Videos Book 13
Rapid Adoption Kits 13
Help and Support Facilities 13
Customer Support 14
Feedback about Documentation 14
Typographic and Syntax Conventions 15

1
Product and Licensing Information 17
Overview 18
About Virtuoso Power System Products 18
About Virtuoso Power System Licenses 19
Checking Out Licenses for VPS Products 19
License and Feature Matrix 20

2
Getting Started 21
Product and Installation Information 22
Supported and Compatible Platforms 22
Specifying the 64-Bit or 32-Bit Version of Virtuoso Power System Applications 22

© 2020 All Rights Reserved.

Starting the Software . . . 24
Running Power IR/EM from the GUI . . . 24
Running Power IR/EM in Batch Mode . . . 34
Log Files for Power IR/EM . . . 34
Accessing Documentation and Help . . . 37
Launching Cadence Help from the Command Prompt . . . 37
Accessing Documentation and Help from the GUI . . . 37
Other Source of Information . . . 38

3
IR/EM Analysis Flow . . . 39
Overview . . . 40
Running Pegasus LVS or PVS LVS . . . 42
Running Pegasus or PVS Quantus QRC . . . 44
Running Extraction with Quantus (Calibre) Interface . . . 52
Running a Spectre Simulation . . . 53
Running an APS Simulation . . . 56
Running an UltraSim Simulation . . . 60

4
Setting Up the Power IR/EM Environment . . . 63
Setting Up the Environment Variables . . . 64
Manual Net Name Resolution . . . 69

5
IR Drop Analysis . . . 71
Overview . . . 72
Package Effect in IR Drop Analysis . . . 72
IR Drop Analysis . . . 73
Analyzing IR Drop Across Design Resistors . . . 79
Writing Out an IR Command File . . . 80
Opening Power IR/EM Directly from Extracted View . . . 82
IR Drop Results . . . 82
Importing Results of a Previous Analysis . . . 83

© 2020 All Rights Reserved.

Selecting Options in the IR (voltage) Drop Results Form to View Results 85
Customizing Error Layer Display for IR Drop Results 91

6
EM Analysis 105
Overview 106
EM Analysis of the Power Grid 106
Layermap File for EM Analysis 123
Analyzing EM Across Design Resistors 124
Writing Out an EM Command File 125
Opening Power IR/EM Directly from Extracted View 126
EM Analysis of Signal Nets 126
EM Analysis Results 128
Importing Results of a Previous EM Analysis 128
Viewing Electromigration Analysis Results for the Current Run 131
Customizing Error Layer Display for EM Analysis Results 140
Customizing the EM Analysis Results Display 142
Using EM Analysis Results for Cross-Probing in Power IR/EM 145

7
Viewing Results in vsaPlot 147
Overview 148
Launching vsaPlot 148
Launching vsaPlot from the Shell 148
Launching vsaPlot from Power IR/EM 149
Viewing Results in vsaPlot 152
Main Menus of vsaPlot 154
Toolbar Widgets 161
Main Components of vsaPlot 164
Querying IR Drop and EM Results 170
Zooming into the Worst Violation in the Selected Area 170
Retrieving the Values of All the Violations at a Specified Point on the Layout 171
Displaying the Colormap for Point-to-Point and Region-to-Region Selection 172
Synchronizing Multiple vsaPlot Windows 173
Opening a Sub-Window to Initiate Synchronism 174

© 2020 All Rights Reserved.

Synchronizing Options . . . 174
Synchronizing vsaPlot with the Layout or Extracted View . . . 176

8
Power-Grid View Creation . . . 179
Overview . . . 180
Data Requirements and Flow for PGV Creation . . . 180
Creating Power-Grid Views . . . 182
Writing Out a PGV Command File . . . 185
Outputs of PGV Generation . . . 187
File Formats . . . 187

9
EM Rules Specification . . . 191
Overview . . . 193
Reading EM Rules from qrcTechFile . . . 193
Supported Limit-Based Parameters . . . 193
Supported JMAX Parameters . . . 194
Rules for Defining Limit-Based Parameters . . . 195
EM Rule Selection . . . 200
Reading EM Rules from emDataFile . . . 202
EM Rules Specification File Syntax . . . 203
User Procedures for Electromigration JMAX Checking . . . 215
Custom Parameters in emDataFile . . . 218
SKILL Functions . . . 220
vsaDefVariable . . . 220
vsaGetVariable . . . 221
vsaWarn . . . 222
vsaError . . . 222
vsaDbType . . . 223
vsaRegisterEMDataFile . . . 224
vsaRegisterQrcTechFile . . . 224
vsaRegisterLayerMapFile . . . 225
vsaGetR . . . 226
vsaResGetIrms . . . 227

© 2020 All Rights Reserved.

vsaResGetIpeak . . . 227
vsaResGetIavg . . . 228
vsaResGetX . . . 229
vsaResGetY . . . 230
vsaResGetLayer . . . 230
vsaGetNode . . . 231
vsaNodeGetIR . . . 232
vsaEMGetQRCDirAndRunName . . . 233
vsaEMGetSimDirAndAnalysisType . . . 233
vsaIRGetSimDirAndAnalysisType . . . 234
Call-Back Functions . . . 236
Sample emDataFile . . . 239

10
Voltage Source Regions . . . 243

Overview . . . 244
Creating Voltage Source Regions . . . 244
SKILL Function . . . 250
vsaNetlistProc . . . 250

11
Batch Mode Execution . . . 251

Overview . . . 252
Batch Command Syntax . . . 252
Command File Syntax . . . 253
Common Commands . . . 253
_vsa_extracted_lib_cell_view “library” “cell” “view” . . . 254
_vsa_testbench_lib_cell_view “library” “cell” “view” . . . 255
_vsa_hierarchy “hierarchy_name” . . . 256
_vsa_pinmap "extrPinName" "testBenchPinName" . . . 257
_vsa_simulation_directory “sim_dir_path” . . . 258
_vsa_pres_cellname “cellname” . . . 259
_vsa_analysis_data “TRANSIENT” | “DCOP” . . . 260
_vsa_vavo_db_enable t | nil . . . 261
exit . . . 262

© 2020 All Rights Reserved.

IR Drop Analysis Commands 263
_vsa_use_explicit_resistor “cellname” 264
_vsa_drop_on_allseg “true” | “false” 265
_vsa_ir_pin_name “all” | “pin1 pin2 ...” 266
_vsa_ir_pin_name_file “pin_list.txt” 267
_vsa_ir_sig_name “all” | “sig1 sig2 ...” 268
_vsa_ir_sig_name_file “signal_list.txt” 269
_vsa_output_ir_file “../path/fileName” 270
_vsa_analyze_ir “start_time” “stop_time” 271
_vsa_irdrop_peak “true” | “false” 272
EM Analysis Commands 273
_vsa_em_pin_name “all” | “pin1 pin2 ...” 274
_vsa_em_pin_name_file “pin_list.txt” 275
_vsa_em_sig_name “all” | “sig1 sig2 ...” 276
_vsa_em_sig_name_file “signal_list.txt” 277
_vsa_qrc_run_directory “run_dir_name” 278
_vsa_rcx_run_name “run_name” 279
_vsa_em_data_file “../path/fileName” 280
_vsa_output_em_file “../path/fileName” 281
_vsa_em_job_name “jobName” 282
_vsa_em_peak “true” | “false” 283
_vsa_em_avg “true” | “false” 283
_vsa_em_avg_abs “true” | “false” 283
_vsa_em_rms “true” | “false” 284
_vsa_em_dc_rms “true” | “false” 284
_vsa_em_peak_ac “true” | “false” 284
_vsa_em_custom “true” | “false” 285
_vsa_em_save_only_failed_results “true” | “false” 286
_vsa_analyze_em “startTime” “stopTime” 287
_vsa_em_norundir "true" | “false” 288
Sample Command File 289
Sample EM Analysis Output 290

© 2020 All Rights Reserved.

Glossary 293

Index 295

© 2020 All Rights Reserved.

© 2020 All Rights Reserved.

Preface

This manual describes the data preparation and steps necessary for power-grid and *electromigration* (EM) analysis using the Virtuoso Power system (VPS) product, Power IR/EM. This product enables you to verify the integrity of your power and ground networks in analog blocks within the Cadence® Analog Design Environment (ADE). It also lets you find signal nets that can fail electromigration (EM) specifications in analog blocks.

The target designs include the analog, analog/mixed-signal (AMS) and custom digital designs of varying sizes that are created using Virtuoso.

This user guide is aimed at developers and designers of integrated circuits and assumes that you are familiar with:

- The Virtuoso design environment and application infrastructure mechanisms designed to support consistent operations between all Cadence® tools.
- The applications used to design and develop integrated circuits in the Virtuoso design environment, notably, the Virtuoso Layout Suite, and Virtuoso Schematic Editor.
- The Virtuoso design environment technology file.
- Component description format (CDF), which lets you create and describe your own components for use with Layout XL.

This preface contains the following topics:

- Licensing Requirements
- Related Documentation
- Additional Learning Resources
- Customer Support
- Feedback about Documentation
- Typographic and Syntax Conventions

© 2020 All Rights Reserved.

Licensing Requirements

For information about the licenses required for running the VPS product options, see the "Product and Licensing Information" chapter.

Related Documentation

Voltus-Fi Custom Power Integrity Solution Documentation

- *Voltus-Fi Custom Power Integrity Solution XL User Guide*
- *Voltus-Fi Custom Power Integrity Solution L User Guide*

What's New and KPNS

- The following chapters in the *Virtuoso What's New* manual:
  - *Voltus-Fi Custom Power Integrity Solution L*
  - *Voltus-Fi Custom Power Integrity Solution XL*
- *Power IR/EM Known Problems and Solutions*

Other Documentation

- *Virtuoso Design Environment User Guide*
- *Virtuoso Analog Design Environment L User Guide*
- *Virtuoso Layout Suite XL User Guide*
- *Virtuoso UltraSim Simulator User Guide*
- *Quantus QRC Extraction Users Manual*
- *Cadence Physical Verification User Guide*

© 2020 All Rights Reserved.

Additional Learning Resources

Video Library

The Video Library on the Cadence Online Support website provides a comprehensive list of videos on various Cadence products.

To view a list of videos related to a specific product, you can use the *Filter Results* feature available in the pane on the left. For example, click the *Virtuoso Layout Suite* product link to view a list of videos available for the product.

You can also save your product preferences in the Product Selection form, which opens when you click the *Edit* icon located next to *My Products*.

Virtuoso Videos Book

You can access certain videos directly from Cadence Help. To learn more about this feature and to access the list of available videos, see Virtuoso Videos.

Rapid Adoption Kits

Cadence provides a number of Rapid Adoption Kits that demonstrate how to use Virtuoso applications in your design flows. These kits contain design databases and instructions on how to run the design flow.

To explore the full range of training courses provided by Cadence in your region, visit Cadence Training or write to training_enroll@cadence.com.

Note: The links in this section open in a separate web browser window when clicked in Cadence Help.

Help and Support Facilities

Virtuoso offers several built-in features to let you access help and support directly from the software.

- The Virtuoso *Help* menu provides consistent help system access across Virtuoso tools and applications. The standard Virtuoso *Help* menu lets you access the most useful help and support resources from the Cadence support and corporate websites directly from the CIW or any Virtuoso application.

© 2020 All Rights Reserved.

- The Virtuoso Welcome Page is a self-help launch pad offering access to a host of useful knowledge resources, including quick links to content available within the Virtuoso installation as well as to other popular online content.

  The Welcome Page is displayed by default when you open Cadence Help in standalone mode from a Virtuoso installation. You can also access it at any time by selecting *Help – Virtuoso Documentation Library* from any application window, or by clicking the *Home* button on the Cadence Help toolbar (provided you have not set a custom home page).

For more information, see Getting Help in *Virtuoso Design Environment User Guide.*

Customer Support

For assistance with Cadence products:

- Contact Cadence Customer Support

  Cadence is committed to keeping your design teams productive by providing answers to technical questions and to any queries about the latest software updates and training needs. For more information, visit https://www.cadence.com/support.

- Log on to Cadence Online Support

  Customers with a maintenance contract with Cadence can obtain the latest information about various tools at https://support.cadence.com.

Feedback about Documentation

You can contact Cadence Customer Support to open a service request if you:

- Find erroneous information in a product manual
- Cannot find in a product manual the information you are looking for
- Face an issue while accessing documentation by using Cadence Help

You can also submit feedback by using the following methods:

- In the Cadence Help window, click the *Feedback* button and follow instructions.
- On the Cadence Online Support Product Manuals page, select the required product and submit your feedback by using the *Provide Feedback* box.

© 2020 All Rights Reserved.

Typographic and Syntax Conventions

The following typographic and syntax conventions are used in this manual.

text	Indicates names of manuals, menu commands, buttons, and fields.
`text`	Indicates text that you must type exactly as presented. Typically used to denote command, function, routine, or argument names that must be typed literally.
`z_argument`	Indicates text that you must replace with an appropriate argument value. The prefix (in this example, *`z_`*) indicates the data type the argument can accept and must not be typed.
`|`	Separates a choice of options.
`{ }`	Encloses a list of choices, separated by vertical bars, from which you **must** choose one.
`[ ]`	Encloses an optional argument or a list of choices separated by vertical bars, from which you **may** choose one.
`[ ?argName` *`t_arg`* `]`	Denotes a *key argument*. The question mark and argument name must be typed as they appear in the syntax and must be followed by the required value for that argument.
`...`	Indicates that you can repeat the previous argument. Used with brackets to indicate that you can specify zero or more arguments. Used without brackets to indicate that you must specify at least one argument.
`,...`	Indicates that multiple arguments must be separated by commas.
`=>`	Indicates the values returned by a Cadence® SKILL® language function.
`/`	Separates the values that can be returned by a Cadence SKILL language function.

If a command-line or SKILL expression is too long to fit within the paragraph margins of this document, the remainder of the expression is moved to the next line and indented. In code excerpts, a backslash (\ ) indicates that the current line continues on to the next line.

© 2020 All Rights Reserved.

© 2020 All Rights Reserved.

1

Product and Licensing Information

- Overview on page 18
- About Virtuoso Power System Products on page 18
- About Virtuoso Power System Licenses on page 19
  - Checking Out Licenses for VPS Products on page 19
  - License and Feature Matrix on page 20

© 2020 All Rights Reserved.

Overview

Cadence® Virtuoso® Power System (VPS) products are sold as part of a product package. Each product has a corresponding license.

The software uses licenses to determine the available features when the software runs.

About Virtuoso Power System Products

This release of the VPS products includes the following:

Table 1-1 Virtuoso Power System Products

Name	Abbreviation	Prod. Num.	Description
Power IR/EM	VPS-L	VPS100	Provides a view of IR drop, electromigration (EM), and signal net EM analysis. It delivers functionality equivalent to that of Virtuoso Analog VoltageStorm Option (VAVO) 34570 and Virtuoso Analog ElectronStorm Option (VAEO) 34580 products.
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 (Beta feature) ■ Hierarchical SHE analysis and modeling for Voltus, which is similar to the PGV Flow.

© 2020 All Rights Reserved.

About Virtuoso Power System Licenses

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

The following table lists the VPS product names and their corresponding license strings.

Table 1-2 Products and corresponding license strings

Products	License String
Power IR/EM (VPS-L)	`Virtuoso_Power_System_L`
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`

Note: If you have the license for the Voltus-Fi-L product, you can also run Power IR/EM or VPS-L. If you have the license for the Voltus-Fi-XL product, you can also run Voltus-Fi-L and Power IR/EM or VPS-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*.

Checking Out Licenses for VPS Products

The VPS products are integrated in the Virtuoso® Design Environment. These products use a form-based approach to drive the analysis based on user requirements. The results of the analysis are displayed in the Virtuoso GUI, overlaid on the design layout.

Running VPS products requires check-out availability of the Cadence® Design Framework II license (known as the "product 111 license"). Once a DFII license check is performed successfully, you will then be able to access VPS functionality.

For information on licenses for the Virtuoso® Design Environment, see Virtuoso Design Environment Licensing Setup in the *Virtuoso Software Licensing and Configuration Guide*.

© 2020 All Rights Reserved.

The VPS products leverage existing Cadence tools for performing extraction and simulation but do not include licenses for these tools. These licenses must be purchased separately.

For extraction, you require a license for the following:

- 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
- UltraSim version 10.1 and above
- APS version 10.1 and above
- ViVA version IC 6.1.5 and above
- ADE L version IC 6.1.5 and above

License and Feature Matrix

The following table lists the VPS product names and their key features:

Product Name	Simulator Support	Simulation-Based Dynamic Power Analysis (IR & EM)	Simulation-Based Dynamic Signal Analysis EM	Static Power Analysis (IR & EM)
Power IR/EM	Spectre, APS, and UltraSim	Yes	Yes	
Voltus-Fi Custom Power Integrity Solution-XL	Spectre® APS/ XPS, and UltraSim	Yes	Yes	Yes

© 2020 All Rights Reserved.

2

Getting Started

- Product and Installation Information on page 22
- Supported and Compatible Platforms on page 22
- Specifying the 64-Bit or 32-Bit Version of Virtuoso Power System Applications on page 22
- Starting the Software on page 24
  - Running Power IR/EM from the GUI on page 24
  - Running Power IR/EM in Batch Mode on page 34
- Log Files for Power IR/EM on page 34
- Accessing Documentation and Help on page 37

© 2020 All Rights Reserved.

Product and Installation Information

The Virtuoso Power System (VPS) products are launched within the Virtuoso Platform. To use these products, you must have the Virtuoso® IC 12 release version.

To check whether the VPS products are available in your Virtuoso® Design Environment, type the following in the unix, shell, or xterm window:

```
vsabatch -v
```

This prints the version of Power IR/EM available in your Virtuoso Design Environment.

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

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

Supported and Compatible Platforms

This product is supported on the following platforms:

- Linux (32 and 64 bit)
- IBM

Specifying the 64-Bit or 32-Bit Version of Virtuoso Power System Applications

You can run Power IR/EM in both 32-bit and 64-bit modes. The 32-bit version and 64-bit version of the software are installed in the same tools hierarchy.

By default, the software runs in the 32-bit mode, if it is available. To specify the 64-bit version, you need to set the `CDS_AUTO_64BIT` environment variable before starting the software. For more information, see the section below.

Using the CDS_AUTO_64BIT Environment Variable

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

© 2020 All Rights Reserved.

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, type the following command:

     ```
     setenv CDS_AUTO_64BIT ALL
     ```

   - To run specific applications in 64-bit mode, such as UltraSim and vpsbatch, and all other applications in 32-bit mode, type the following command:

     ```
     setenv CDS_AUTO_64BIT ultrasim:vpsbatch
     ```


© 2020 All Rights Reserved.

Starting the Software

You can run Power IR/EM interactively from the GUI or you can execute it as a batch command. But before you run Power IR/EM, you need to extract parasitic resistance and capacitance from the power and ground nets, create extracted views of the design, and simulate these views. For more information, see the "IR/EM Analysis Flow" chapter.

- Running Power IR/EM from the GUI on page 24
- Running Power IR/EM in Batch Mode on page 34

Running Power IR/EM from the GUI

To run Power IR/EM in the Virtuoso Design Environment, you will be performing the following tasks:

- Starting the Virtuoso Design Environment on page 24
- Using the CIW to Open the Virtuoso® Schematic Editor L on page 25
- Descending into Design Hierarchy on page 29
- Running Power IR/EM on page 32

Starting the Virtuoso Design Environment

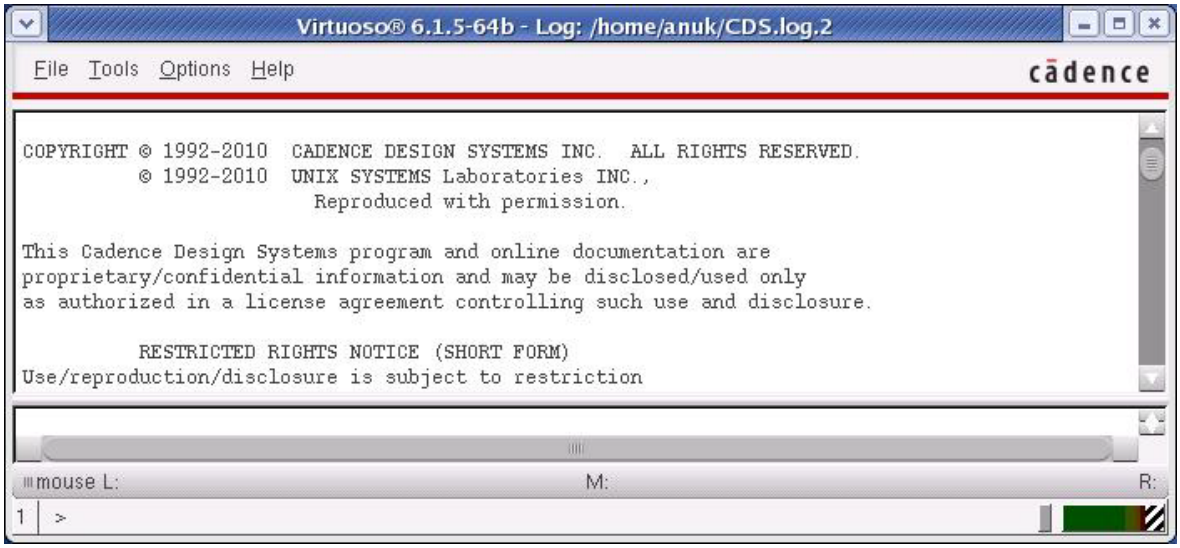
Once you have set up the Virtuoso Design Environment, you can start Virtuoso by typing the following in the unix, shell, or xterm window:

```
virtuoso &
```

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

© 2020 All Rights Reserved.

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.

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

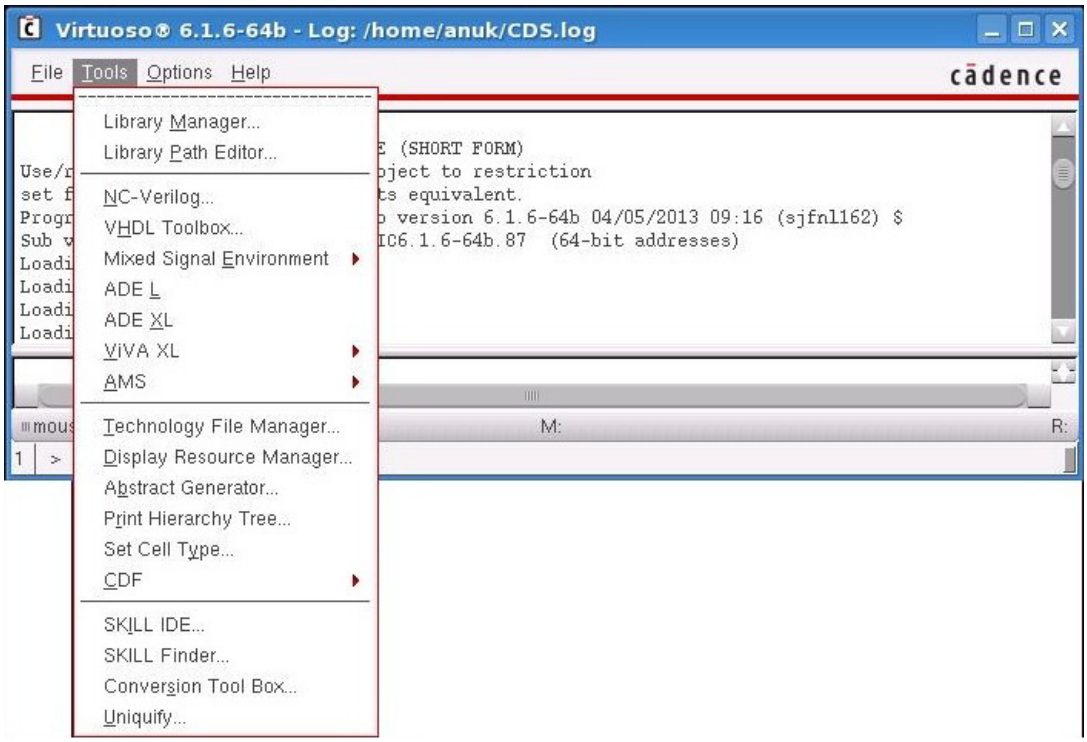
For information about how to open the schematic view of Virtuoso from the *File* menu in the CIW, see, "Working with Cellviews", in the *Virtuoso Design Environment User Guide*.

© 2020 All Rights Reserved.

Using the Tools Menu

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

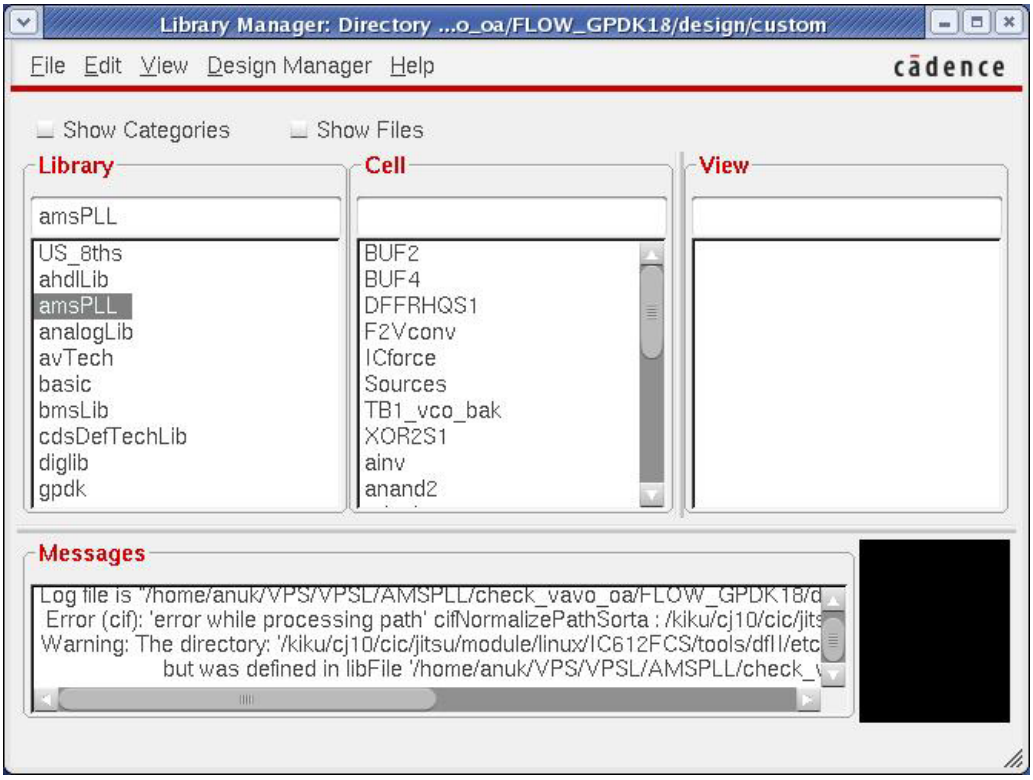
Figure 2-2 Command Interpreter Window – Opening the Library Manager Form



The Library Manager form opens. This is shown below.

© 2020 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

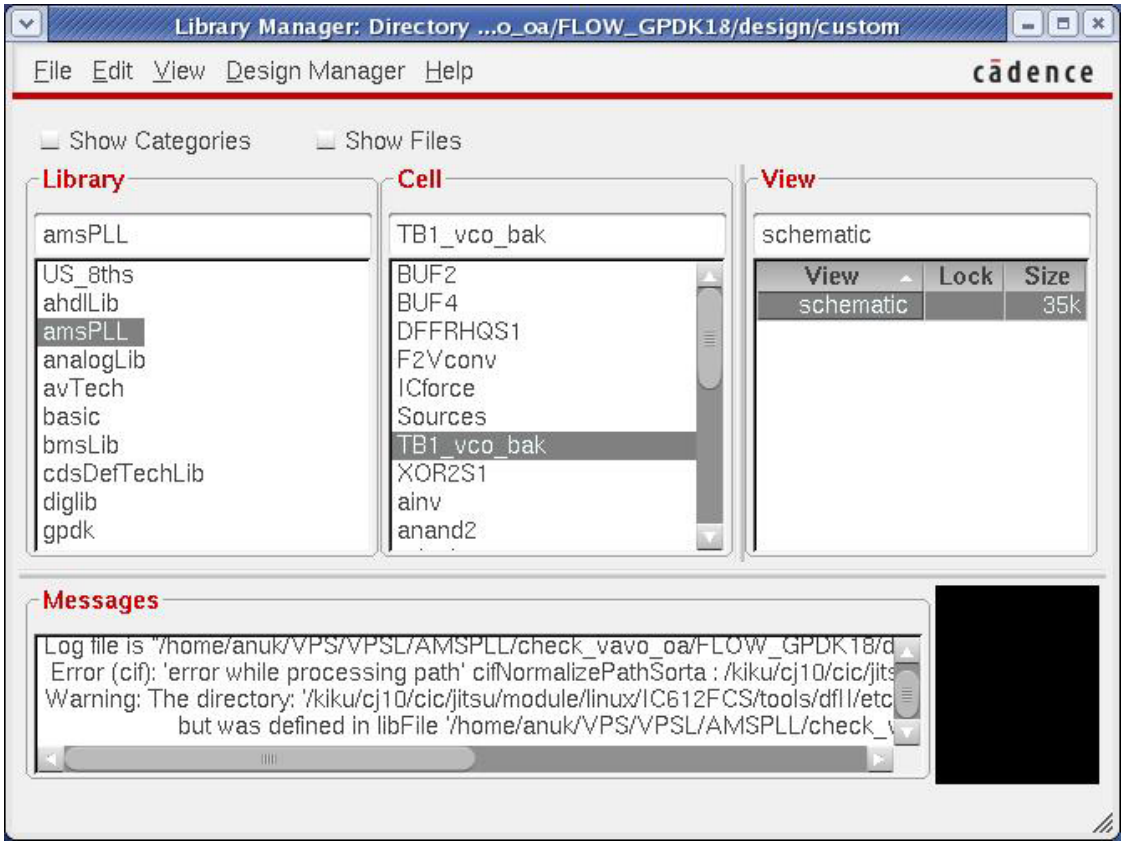
In the Library Manager form, you can choose to display library information (library, cell, view, file, category) using either list boxes (in View – Lists mode) or a hierarchical tree structure (in View - Tree mode).

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. This is shown below.

© 2020 All Rights Reserved.

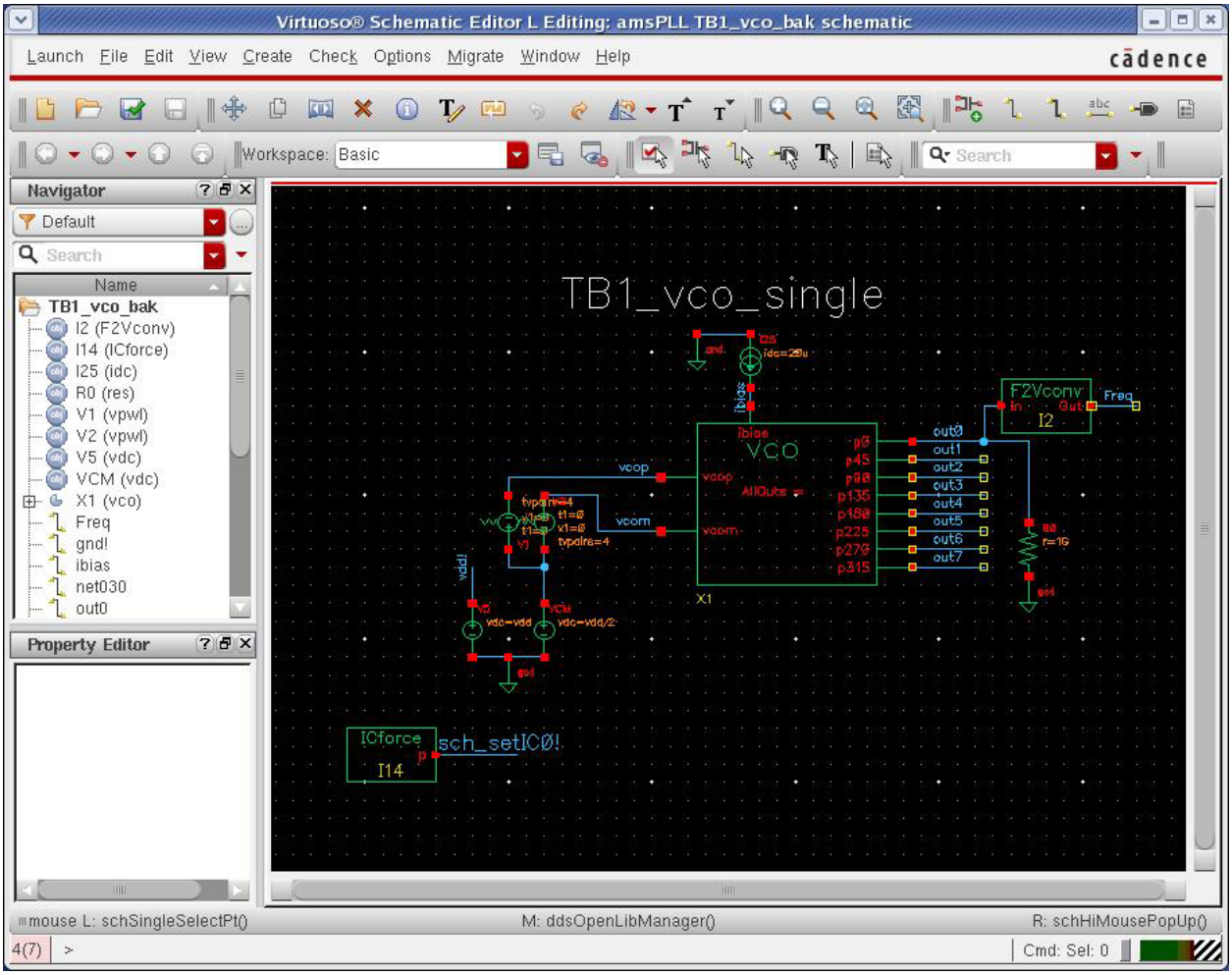
Figure 2-4 Library Manager – Opening the Schematic View of a Design



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

© 2020 All Rights Reserved.

Figure 2-5 Virtuoso Schematic Editor – Displaying the Design



Descending into Design Hierarchy

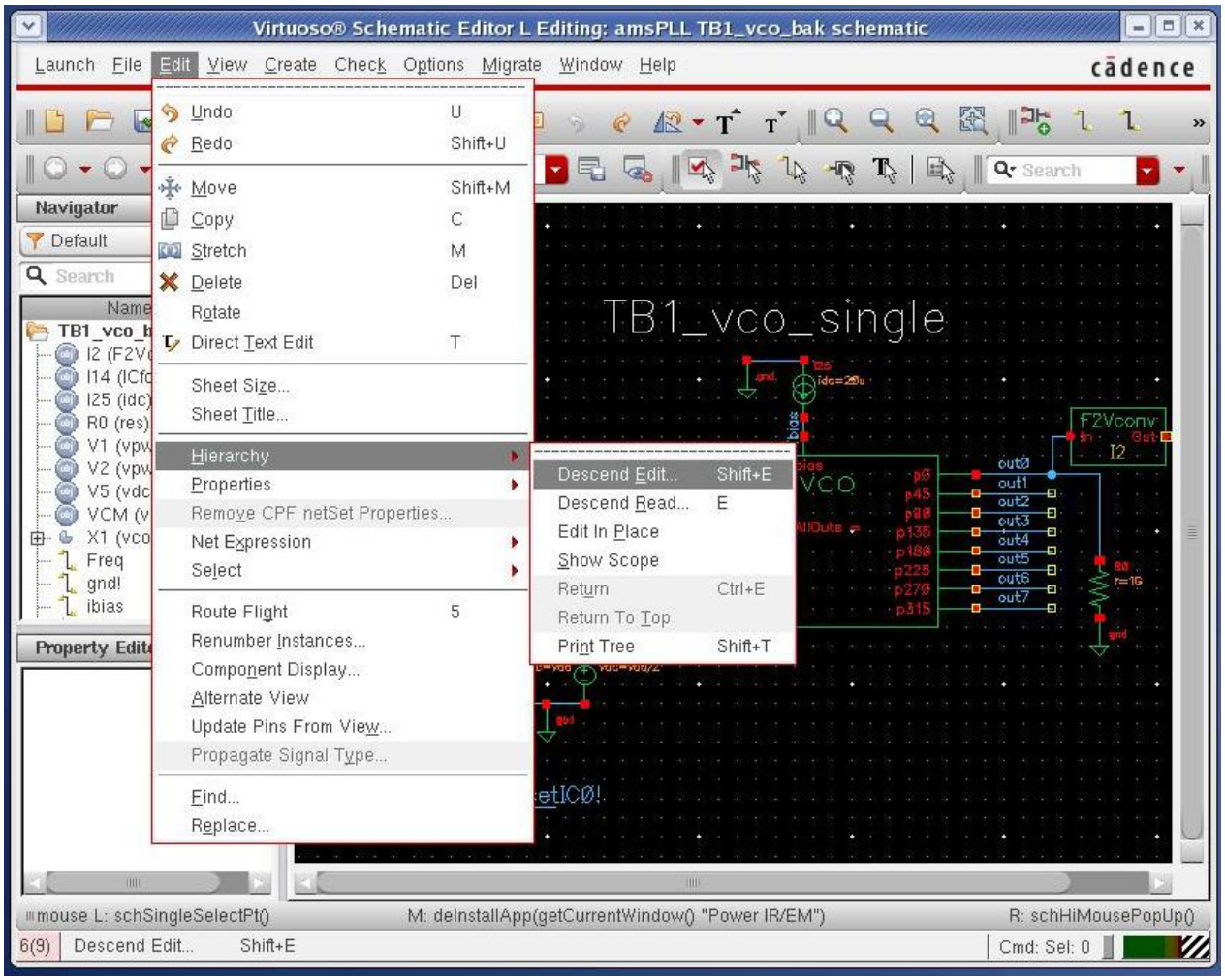
In the Schematic Editor, before you run Power IR/EM, you need to descend into the design hierarchy. A design hierarchy consists of many levels of a single design. When you descend into the design hierarchy, you view the extracted layout of the design you want to analyze.

To descend into the design hierarchy, follow these steps:

1. In the Virtuoso Schematic Editor L, choose *Edit – Hierarchy –* and *Descend Read*. You can choose the *Descend Edit* option if you need to make any change in the extracted layout.

© 2020 All Rights Reserved.

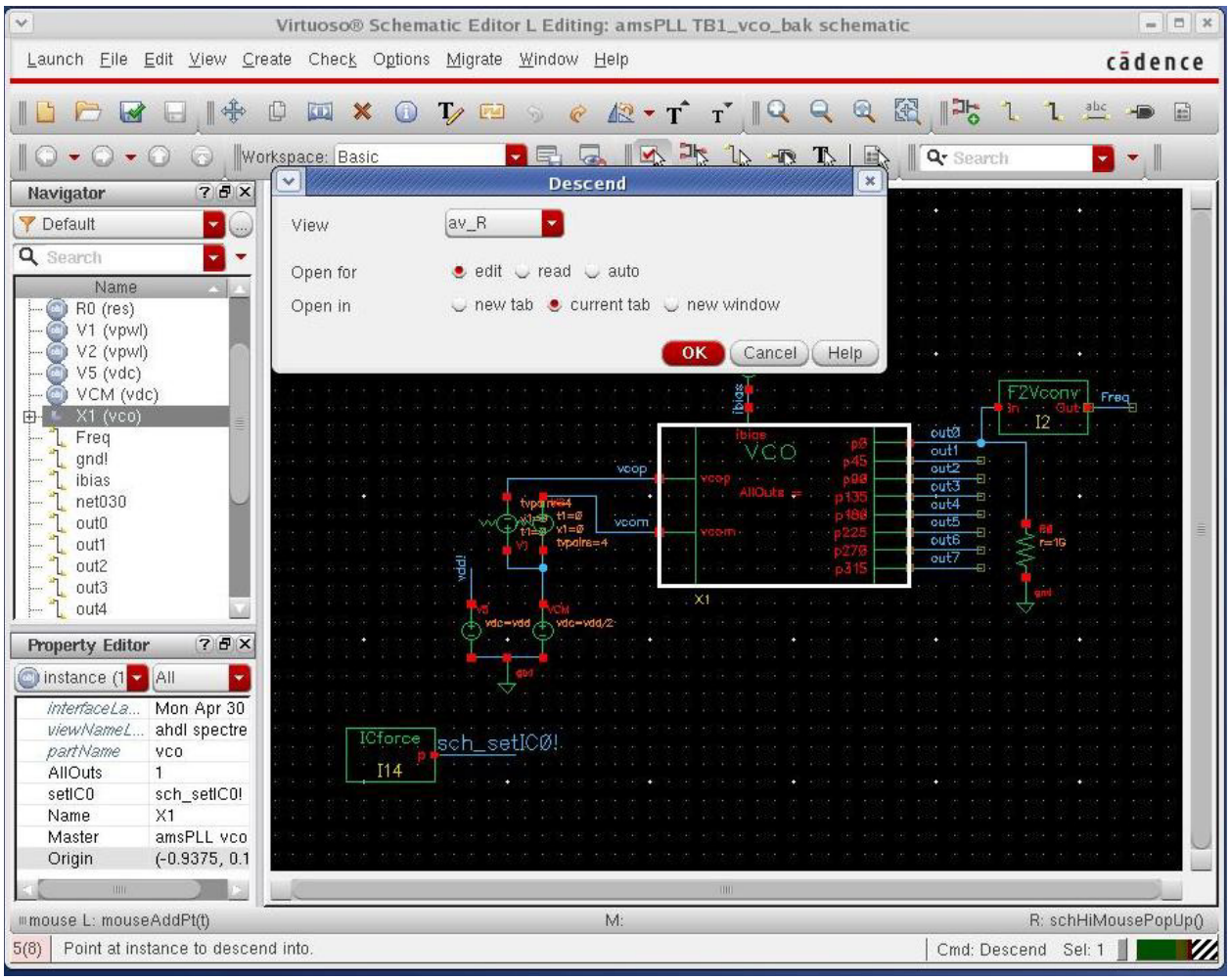
Figure 2-6 Virtuoso Schematic Editor L – Descending into Design Hierarchy



2. Use the pointer to select the instance to which you want to descend. The Descend form opens.

© 2020 All Rights Reserved.

Figure 2-7 Descend Form – Selecting the Design View



3. 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_r`.

4. Click *read* to open the extracted view for reading. You can select *edit* if you want to modify something in the extracted view.

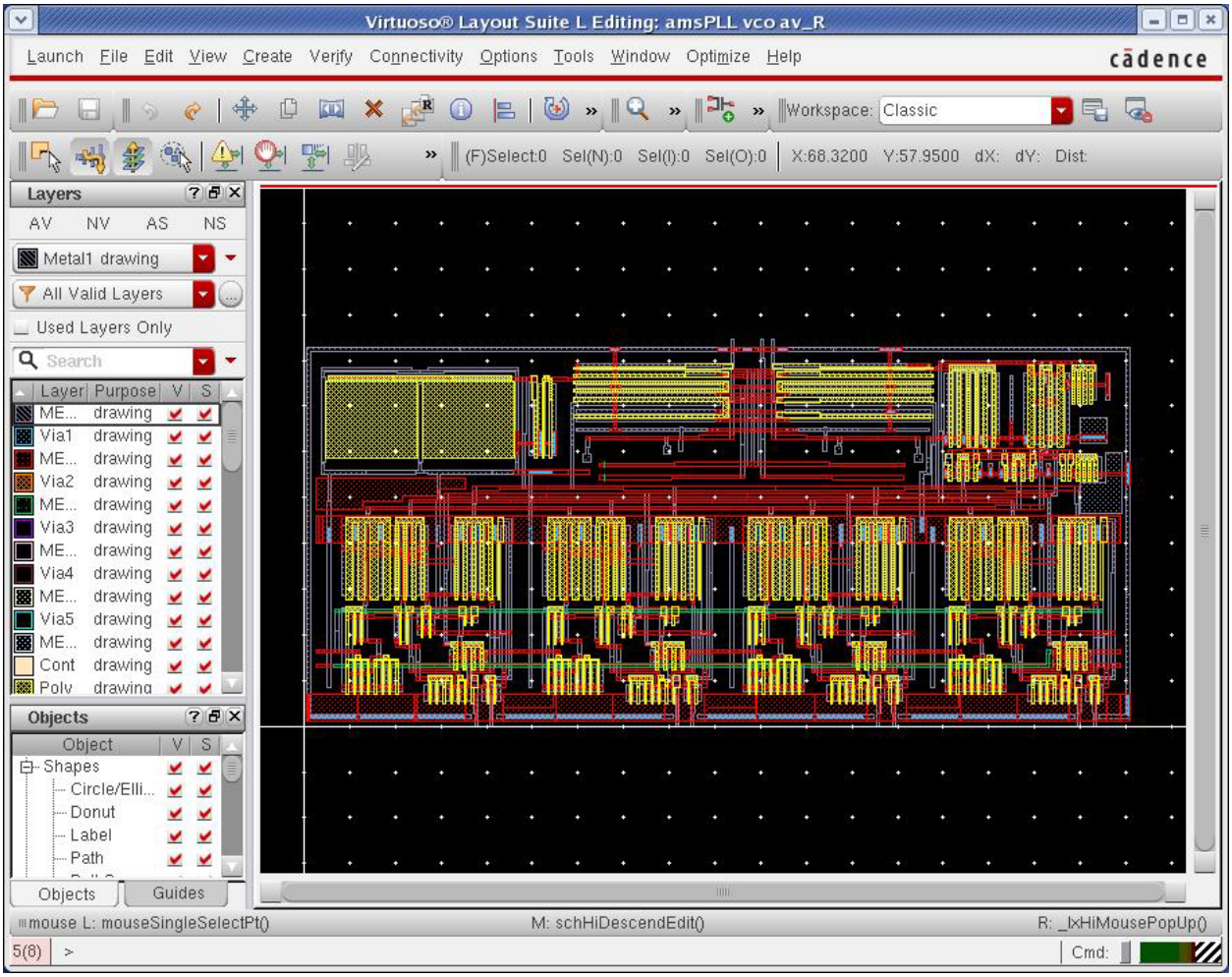
5. Choose where you want to open the extracted view by selecting *Open in – new tab, current tab,* or *new window.*

6. Click *OK*.

7. The selected view of the instance opens in Virtuoso® Layout Suite L. This is shown in the figure below.

© 2020 All Rights Reserved.

Figure 2-8 Virtuoso Layout Suite L – Displaying the Selected Design View



Running Power IR/EM

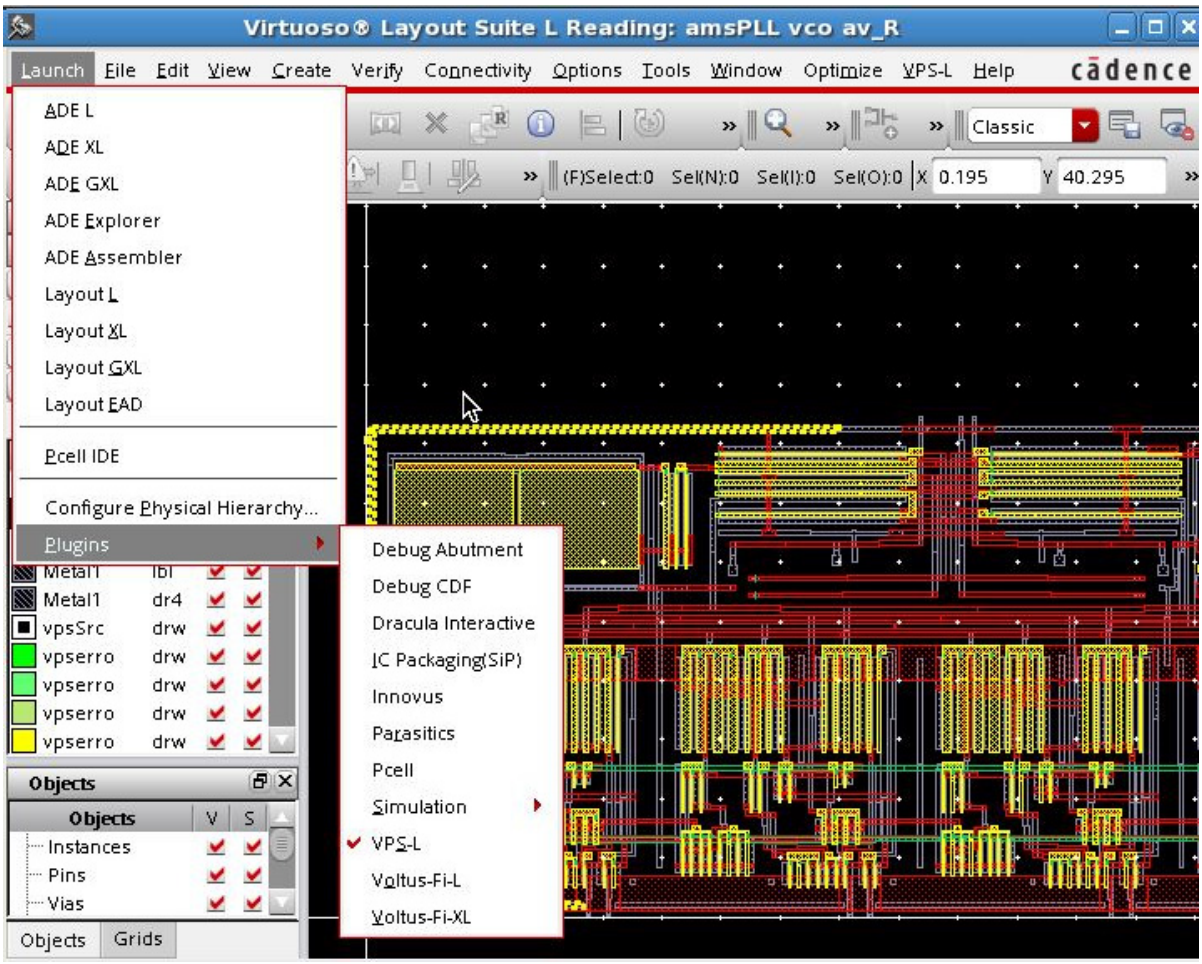
Once you have opened the detailed view of the selected instance of the design, you can run Power IR/EM by activating the *VPS-L* menu option in the Main menu of the Virtuoso® Layout Suite L.

To activate the *VPS-L* menu, perform the following steps:

1. In the Virtuoso Layout Suite, choose *Launch – Plugins – VPS-L*.

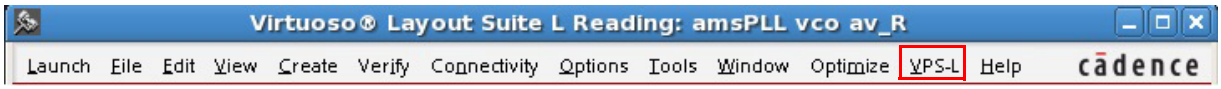
© 2020 All Rights Reserved.

Figure 2-9 Virtuoso Layout Suite - Running Power IR/EM



2. The *VPS-L* menu option appears in the Main menu of the Virtuoso Layout Suite L. This is shown below.

Figure 2-10 Virtuoso Layout Suite L - VPS-L Menu



© 2020 All Rights Reserved.

Running Power IR/EM in Batch Mode

You can run Power IR/EM as a batch command, running from Virtuoso in no-graph mode. Batch mode is typically used to perform the initial pass/fail analysis of the design's power-grid networks.

Once you have run the analysis in batch mode, you can use the interactive GUI to view and analyze the results of the analysis.

Batch Command Syntax

The batch mode is executed from the unix, shell, or xterm window as follows:

```
vsabatch -cmd <cmd_file>
```

Where:

<cmd_file> is a list of VPS commands that define the input data, control the execution of analysis, and select the output location.

For more information on Batch commands and the sample command file, see the "Batch Mode Execution" chapter.

Log Files for Power IR/EM

When you run Power IR/EM either in the GUI or in the batch mode, a standalone log file is created to record the complete Power IR/EM session. This file is created in addition to the `CDS.log` file that records the ongoing events of the design session in Virtuoso. This Power IR/EM-specific log file is saved in the following location by default.

working_dir/VSA.OUT/VSA.LOG.n

Example:

working_dir/VSA.OUT/VSA.LOG.1

For each session, a new log file is created, such as `VSA.LOG.1, VSA.LOG.2,` and so on. You can save up to twenty log files, after which the last log file, `VSA.LOG.20`, 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.

© 2020 All Rights Reserved.

```
#This is a tool generated log file and contains log for the entire VPS session
LOG FILE CREATED: "Apr 30 11:44:16 2014"
LOG FILE VERSION: 1.0
STARTED IR Drop Analysis
VSA CONTEXT VERSION       = v10.10-dev.rosingh
Command file parsed Successfully
License "Virtuoso_Power_System_L" exists
Checkout of license "Virtuoso_Power_System_L" was successful

Running IR Drop Analysis
  STARTING TO GET NETS AND INSTANCES ...
     FINISHED in 0 hours 0 minutes 0 seconds
     MEMORY USAGE 0.00 MB

  STARTING TO FIND THE DEVICES CONNECTED TO THE NETS ...
     FINISHED in 0 hours 0 minutes 0 seconds
     MEMORY USAGE 0.30 MB

  STARTING TO CALCULATE THE SIMULATION RESULTS ...
INFO: ** loadNetlist command done ( time 0 sec )
STATUS: instances (analyzed: 312     failed: 12      total: 312   )
WARNING: 12 instances failed, please see the log file.
INFO: ** run command done ( time 0 sec )

     FINISHED in 0 hours 0 minutes 1 seconds
     MEMORY USAGE 27.37 MB

  STARTING TO WRITE THE IRDROP RESULTS ...
You specified an existing file "/home/rosingh/work/IRDrop.results".
Overwriting?
Successfully wrote IR drop results to
      "/home/rosingh/work/IRDrop.results"
```


© 2020 All Rights Reserved.

```
     FINISHED in 0 hours 0 minutes 1 seconds
     MEMORY USAGE 0.00 MB
TOTAL TIME WAS 0 hours 0 minutes 2 seconds
TOTAL MEMORY USAGE WAS 38.43 MB
```


© 2020 All Rights Reserved.

Accessing Documentation and Help

You can access the Power IR/EM documentation and help system using the following methods:

- Launching Cadence Help from the Command Prompt on page 37
- Accessing Documentation and Help from the GUI on page 37
- Other Source of Information on page 38

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

1. Select *Help* on the Main Menu of the Virtuoso® Layout Suite L.
2. Open the *Documentation Browser*. A list of all Virtuoso documents appears.
3. Choose *Virtuoso Power System – Power IR/EM User Guide*. This opens the Table of Contents page of the user guide.

Select Help on a Power IR/EM Form

Click the *Help* button on the bottom right corner of a form.

- Clicking the *Help* button opens the Power IR/EM user guide entry for the form in the Cadence Help window.

© 2020 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.

© 2020 All Rights Reserved.

3

IR/EM Analysis Flow

- Overview on page 40
- Running Pegasus LVS or PVS LVS on page 42
- Running Pegasus or PVS Quantus QRC on page 44
- Running Extraction with Quantus (Calibre) Interface on page 52
- Running a Spectre Simulation on page 53
- Running an APS Simulation on page 56
- Running an UltraSim Simulation on page 60

© 2020 All Rights Reserved.

Overview

A typical IR/EM analysis flow is as follows:

- Compare the schematic design with the layout design to verify the logical connectivity of the design.This is done by running *Pegasus* LVS or PVS LVS (Layout as compared to Schematic) or Calibre® LVS. For more information, see Running Pegasus LVS or PVS LVS.

  Note: Pegasus, the new physical verification system, is available in the 18.2 physical verification system release.

- Extract parasitic resistance and capacitance from the power and ground nets and create an extracted view of the design using Quantus QRC. For more information, see Running Pegasus or PVS Quantus QRC and Running Extraction with Quantus (Calibre) Interface.

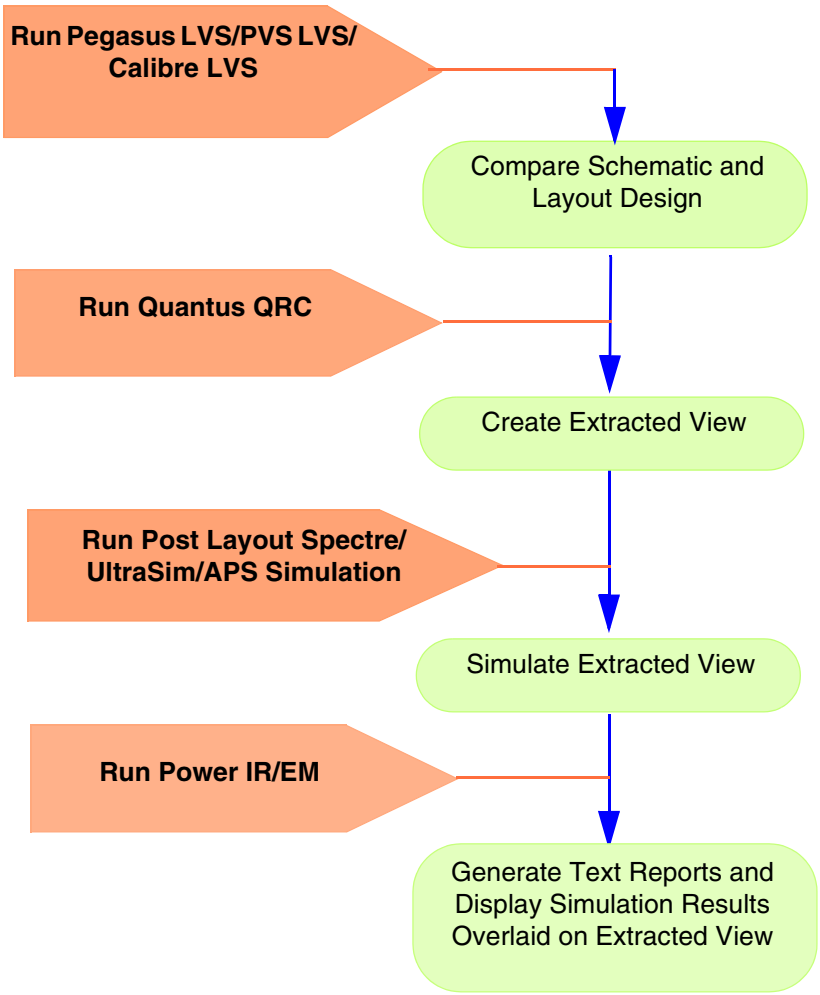
- Simulate the extracted view of the design using Spectre, UltraSim, or APS simulation. For more information, see Running a Spectre Simulation on page 53, Running an UltraSim Simulation on page 60, and Running an APS Simulation on page 56.

- Run Power IR/EM to begin power-grid and *electromigration* analysis on your design and view the results.

This flow is shown below.

© 2020 All Rights Reserved.

Figure 3-1 Power IR/EM Design Flow



Note: Quantus QRC creates an extracted view that can be simulated in Cadence® Analog Design Environment (ADE). You can then use Power IR/EM to superimpose the simulation data generated through circuit simulation using Spectre, UltraSim, or APS onto the Virtuoso extracted view. You can also get a pass/fail report for the electromigration (EM) analysis.

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 Power IR/EM.

Note: The data preparation can be limited to include only the nets of interest.

© 2020 All Rights Reserved.

Running Pegasus LVS 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. You do not need to make any special modifications to the Pegasus or PVS LVS decks.

Note: Pegasus, the new physical verification system, is available in the 18.2 physical verification system release.

1. In the *Virtuoso Layout Suite*, select the Run LVS sub menu from the Pegasus or PVS menu. This is shown below:

Figure 3-2 Run LVS Sub Menu from PVS Menu

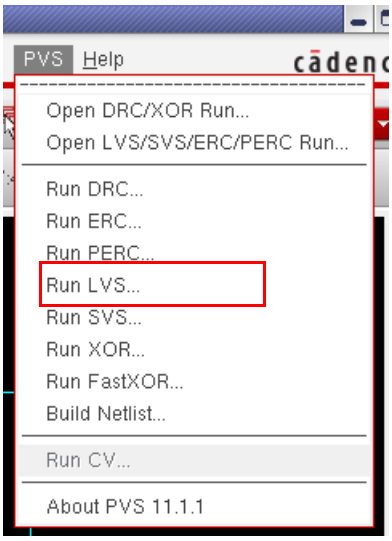
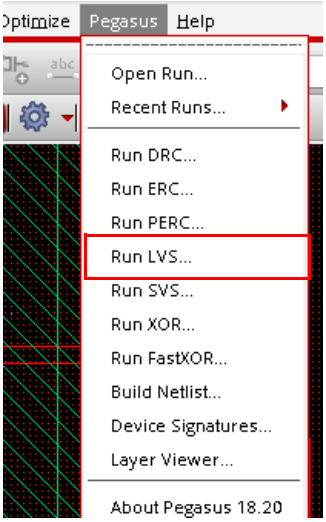


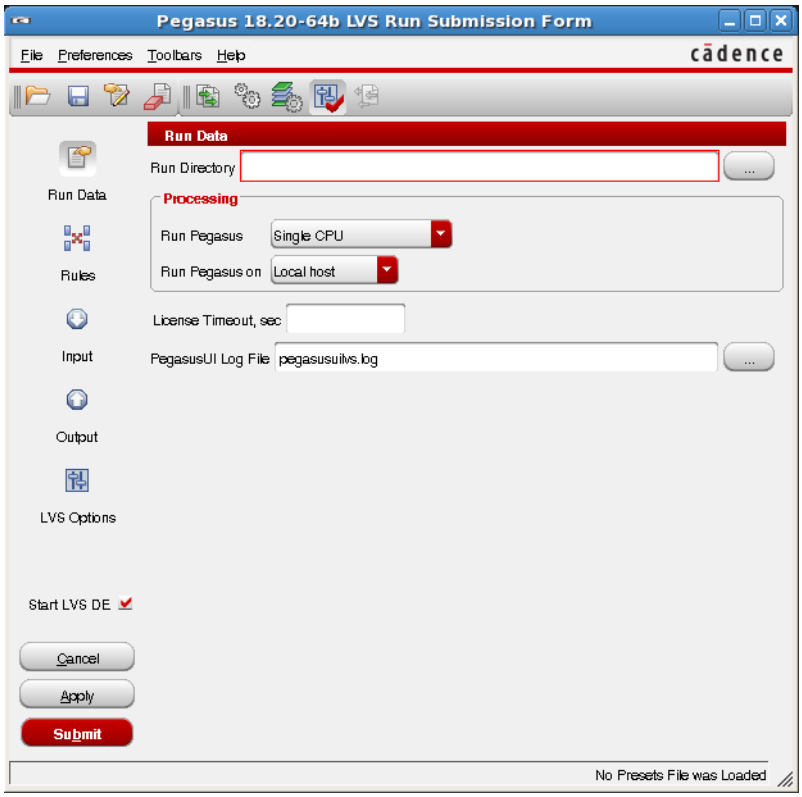
Figure 3-3 Run LVS Sub Menu from Pegasus Menu

© 2020 All Rights Reserved.



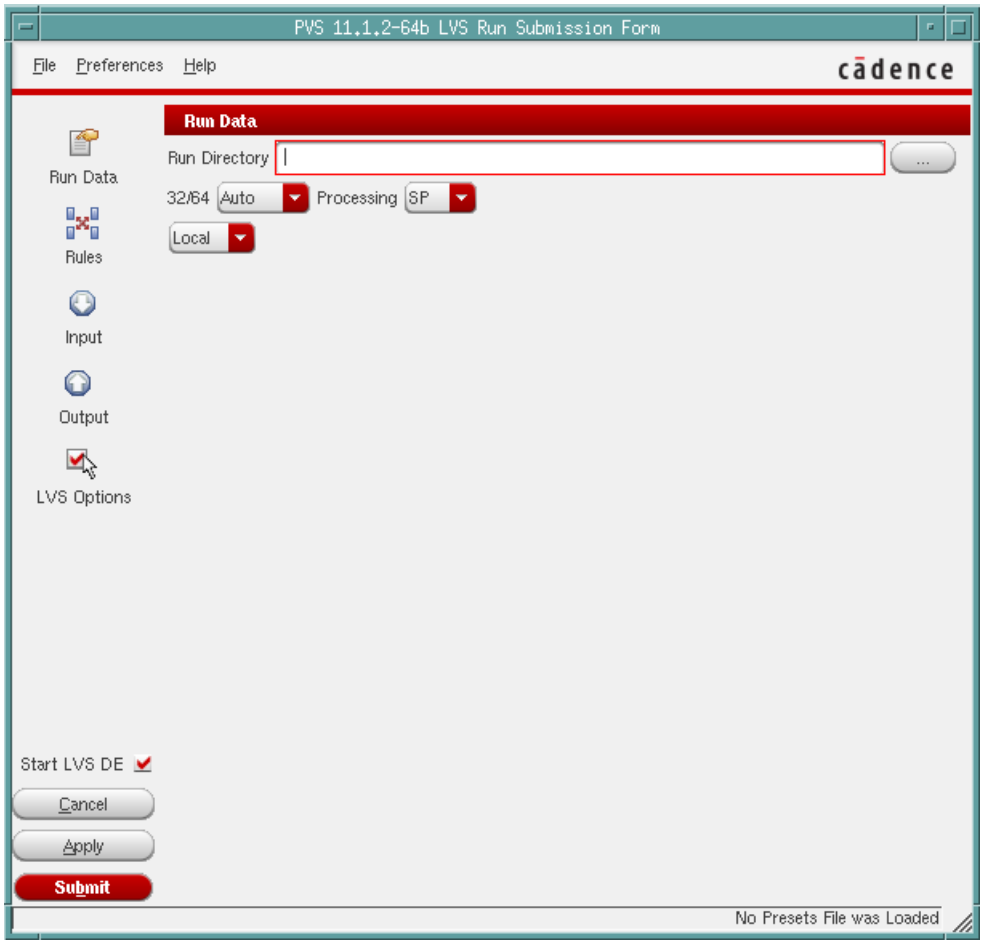
1. The Pegasus or PVS LVS Run Submission Form opens.

Figure 3-4 Pegasus LVS Run Submission Form



© 2020
All Rights Reserved.

Figure 3-5 PVS LVS Run Submission Form



For detailed information on running Pegasus or PVS LVS, see the *Pegasus™ Interactive User Guide* or *Virtuoso® IPVS User Guide.*

Running Pegasus or PVS Quantus QRC

After the LVS run is complete, you must extract parasitic resistance and capacitance from the power and ground nets for power-grid analysis, and from the signal nets for detecting electromigration failures in Power IR/EM. Although, no rule changes are required to run Quantus QRC, you need to enable some options to generate the extracted view correctly. For more information on running Quantus QRC, see the Quantus *QRC Extraction Users Manual*.

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

© 2020 All Rights Reserved.

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

Figure 3-6 Quantus (Pegasus) Interface Form

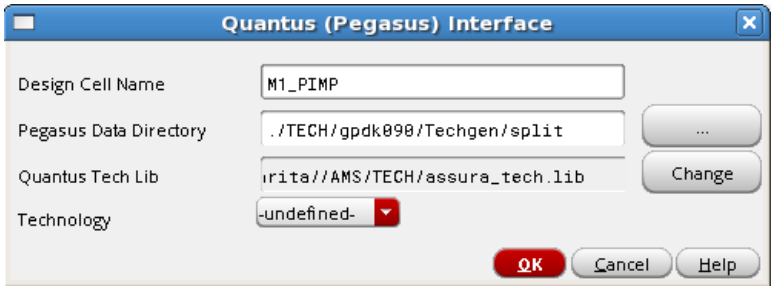
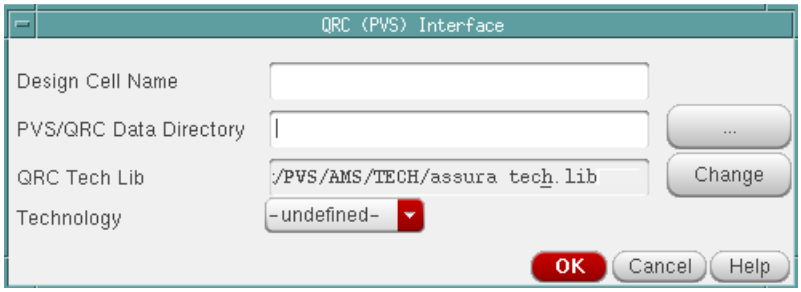


Figure 3-7 QRC (PVS) Interface Form

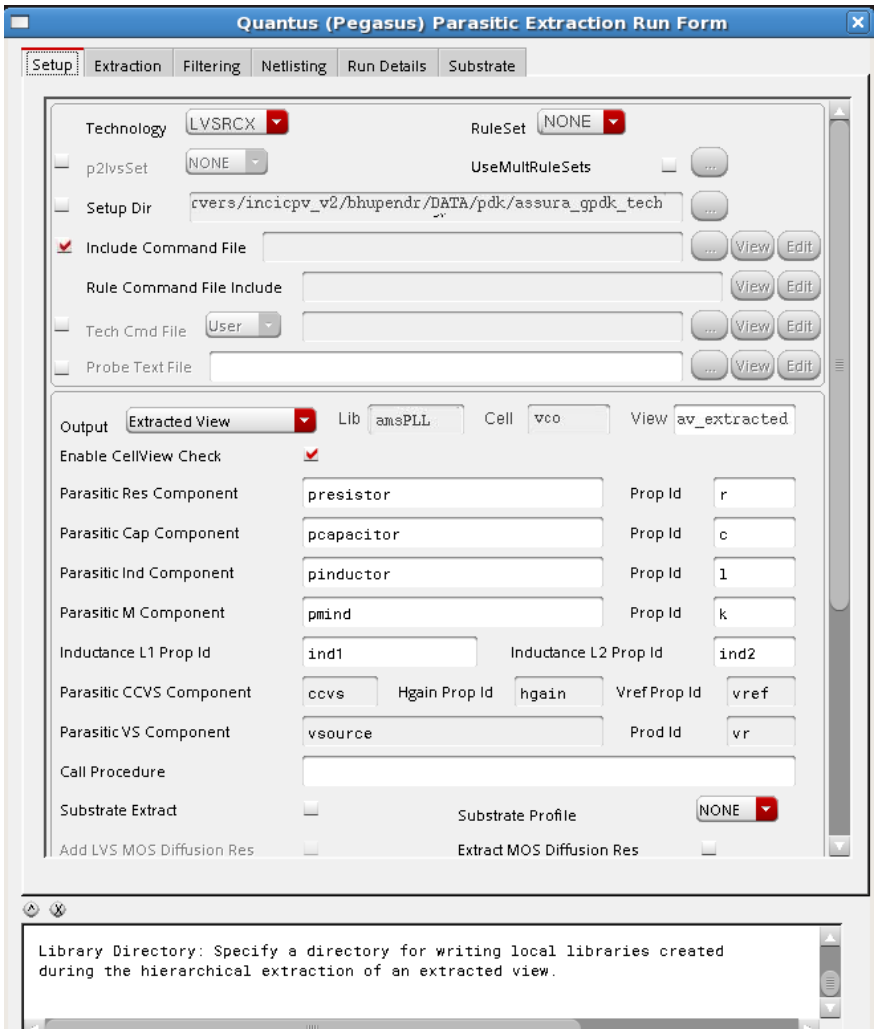


2. Specify the *Pegasus/PVS Data Directory* and the *Quantus/QRC Tech Lib* and click *OK*.

3. The Quantus (Pegasus)/QRC (PVS) Parasitic Extraction Run Form opens. In this form, select the *Setup* tab.This form is shown below.

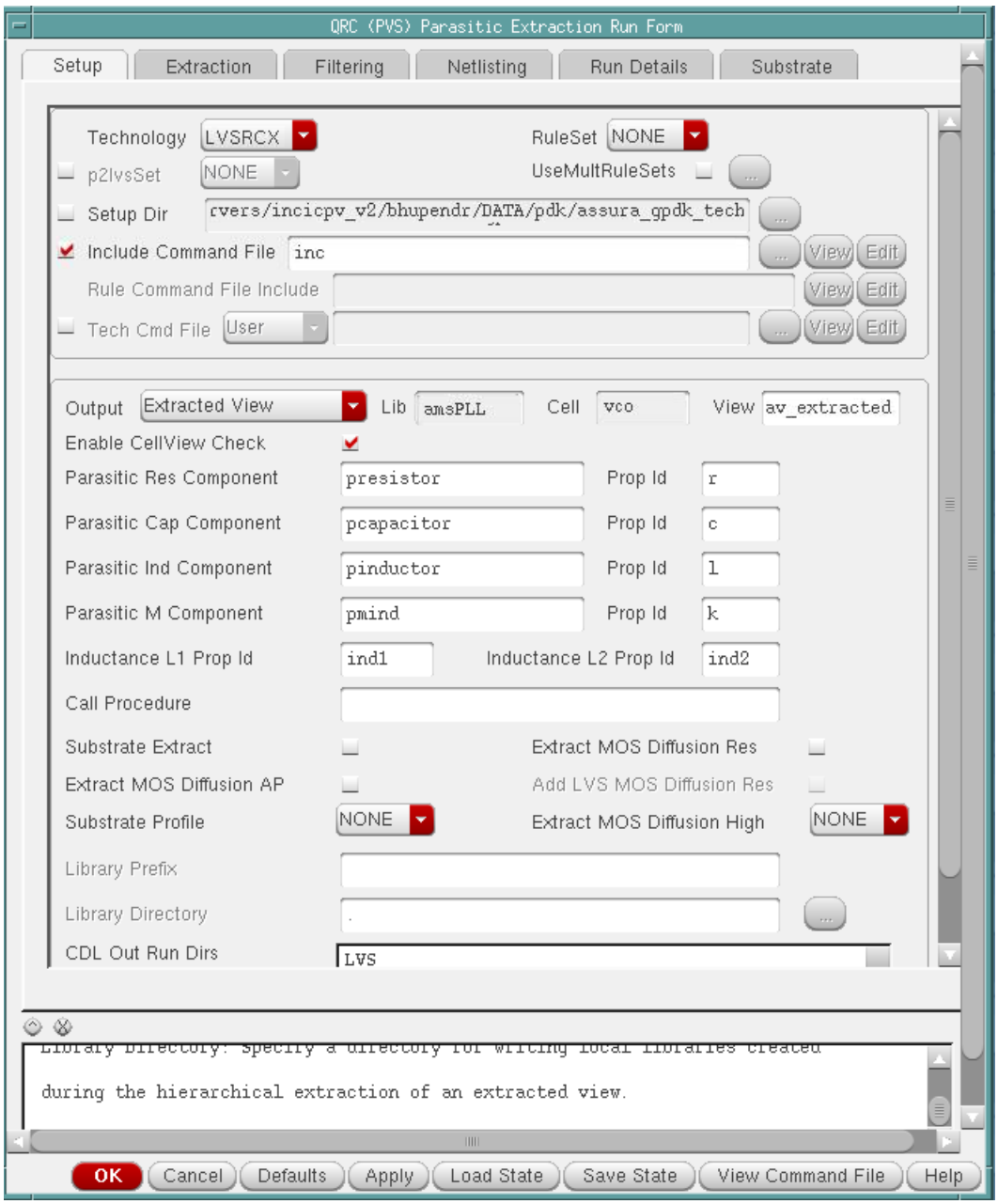
© 2020 All Rights Reserved.

Figure 3-8 Quantus (Pegasus) Parasitic Extraction Run Form – Setup Tab



© 2020 All Rights Reserved.

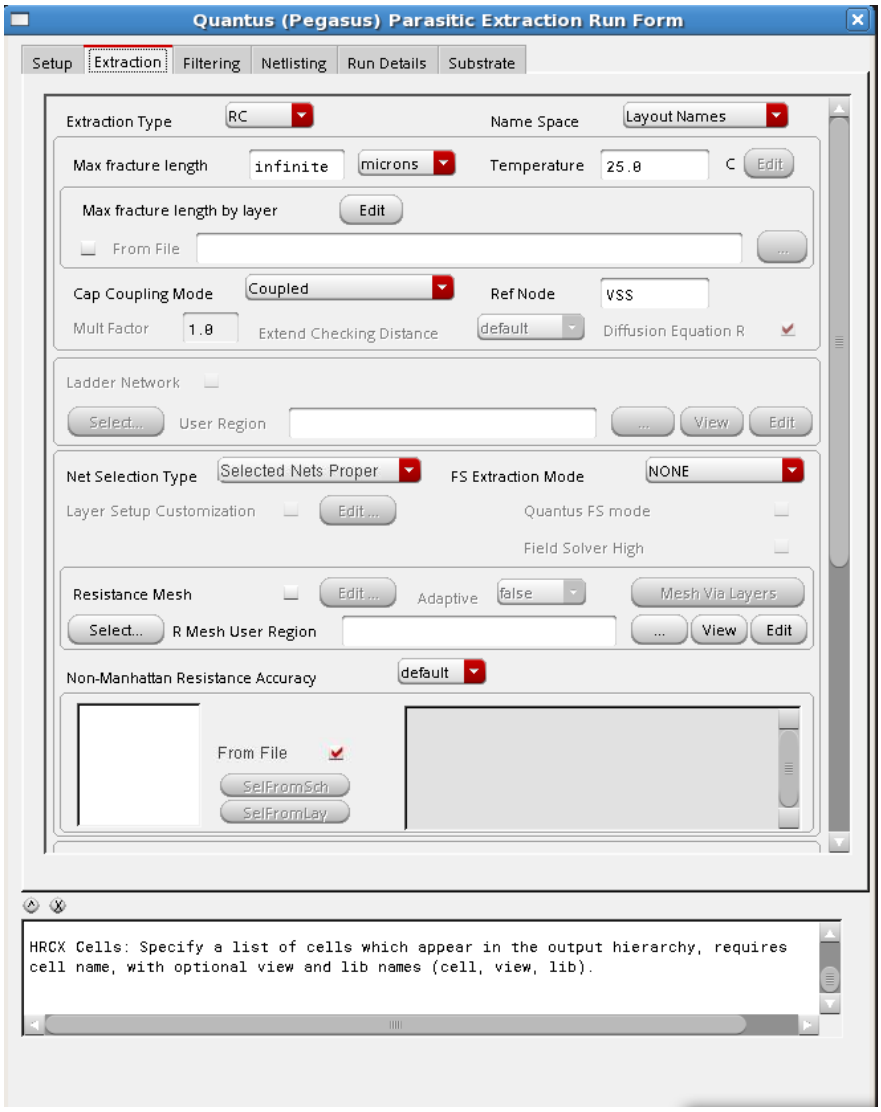
Figure 3-9 QRC (PVS) Parasitic Extraction Run Form – Setup Tab



4. In this form, ensure that the *Output* is set to *Extracted View*.
5. Now, select the *Extraction* tab. The following form opens.

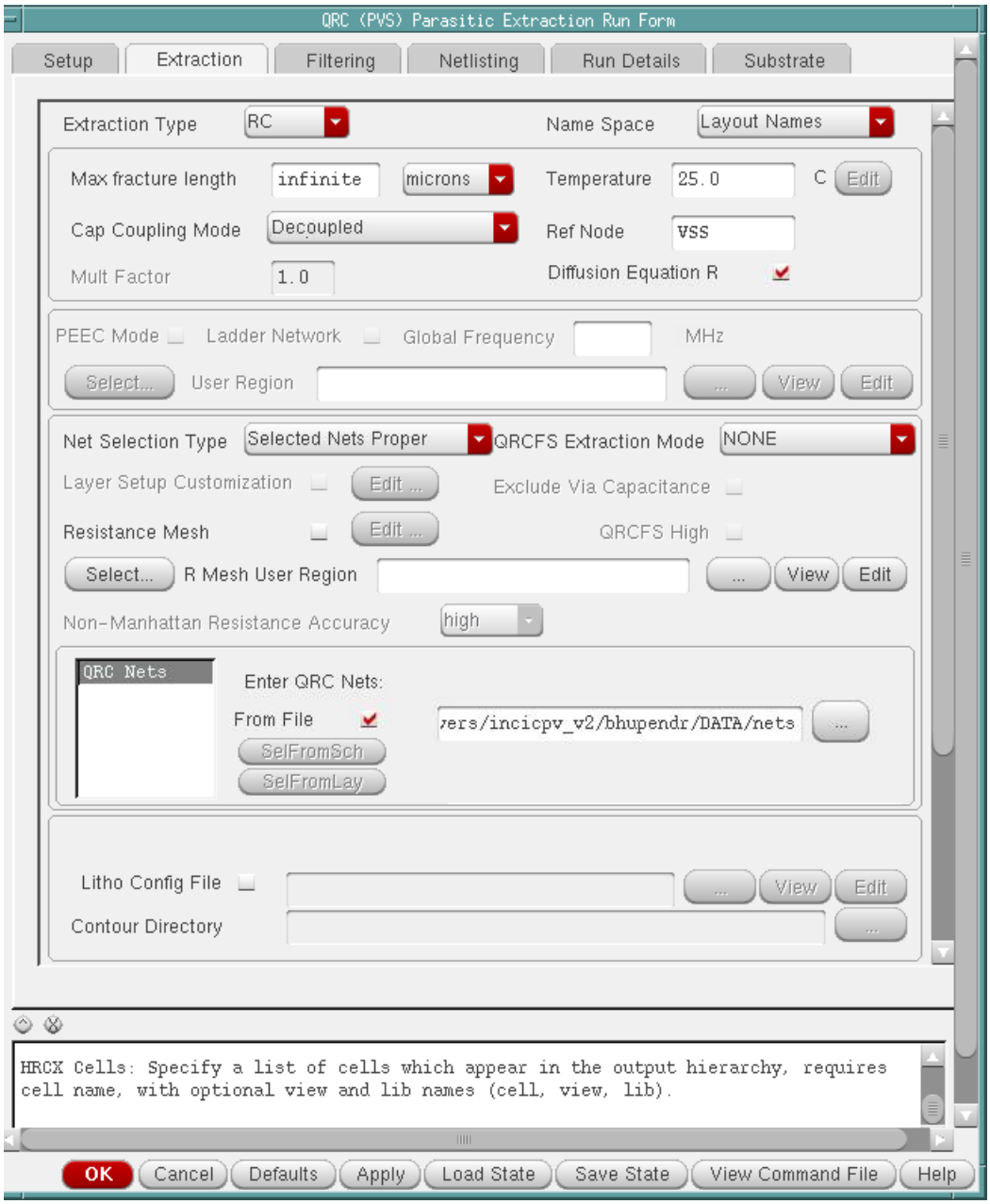
© 2020 All Rights Reserved.

Figure 3-10 Quantus (Pegasus) Parasitic Extraction Run Form – Extraction Tab



© 2020 All Rights Reserved.

Figure 3-11 QRC (PVS) Parasitic Extraction Run Form – Extraction Tab



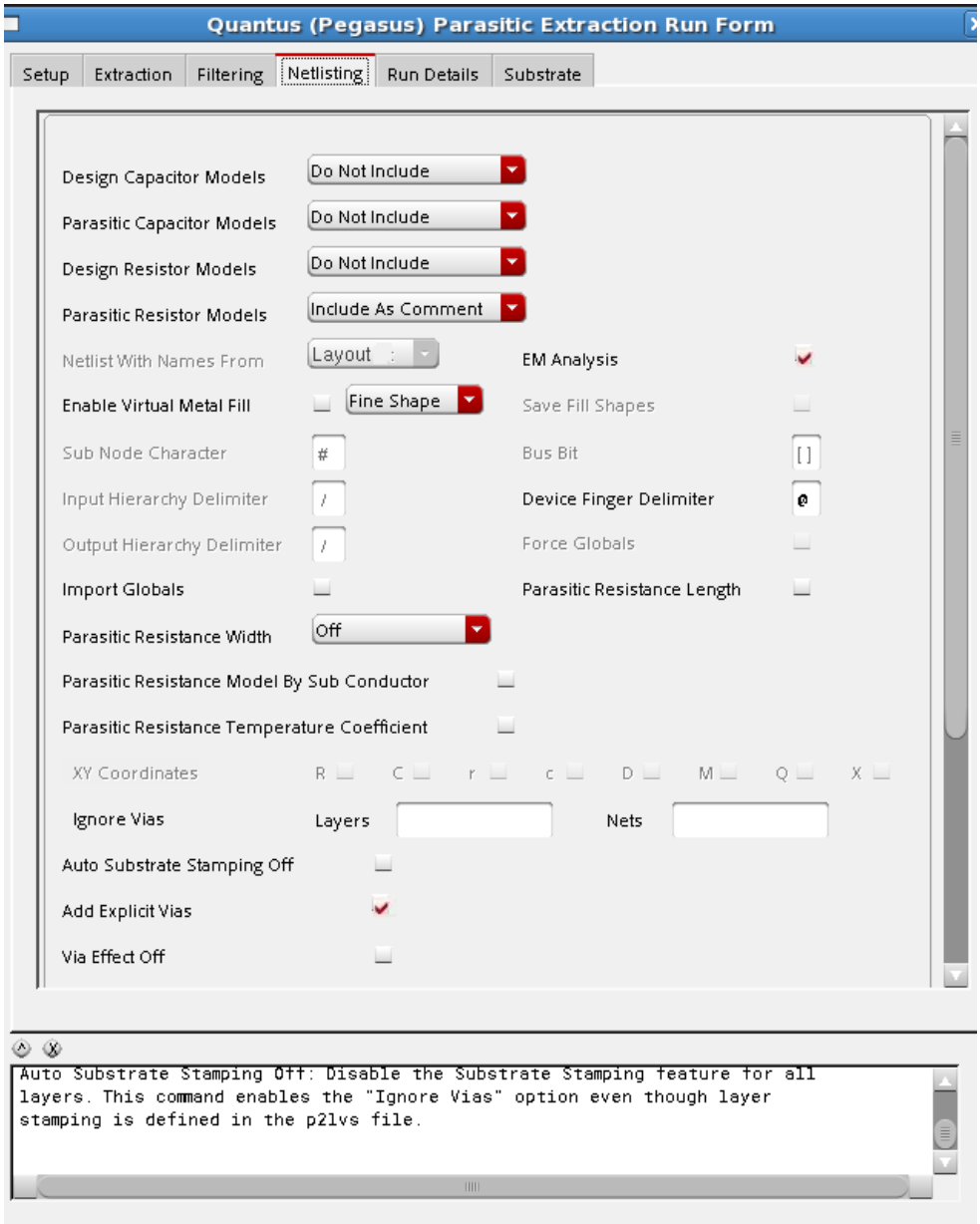
6. You can select the *Extraction Mode* to extract either resistance and capacitance (*RC*) or only resistance (*R only*). When you select the *Extraction Mode* as *R Only*, the simulation time is short. Also, consider the type of simulation run you are planning. If you are performing a DC operating point (DCOP) simulation, then you do not require capacitance values.

© 2020 All Rights Reserved.

Note: You can reduce extraction time in Quantus QRC by selecting specific nets of interest. To do this, select *Net Selection Type – Selected Nets Proper*.

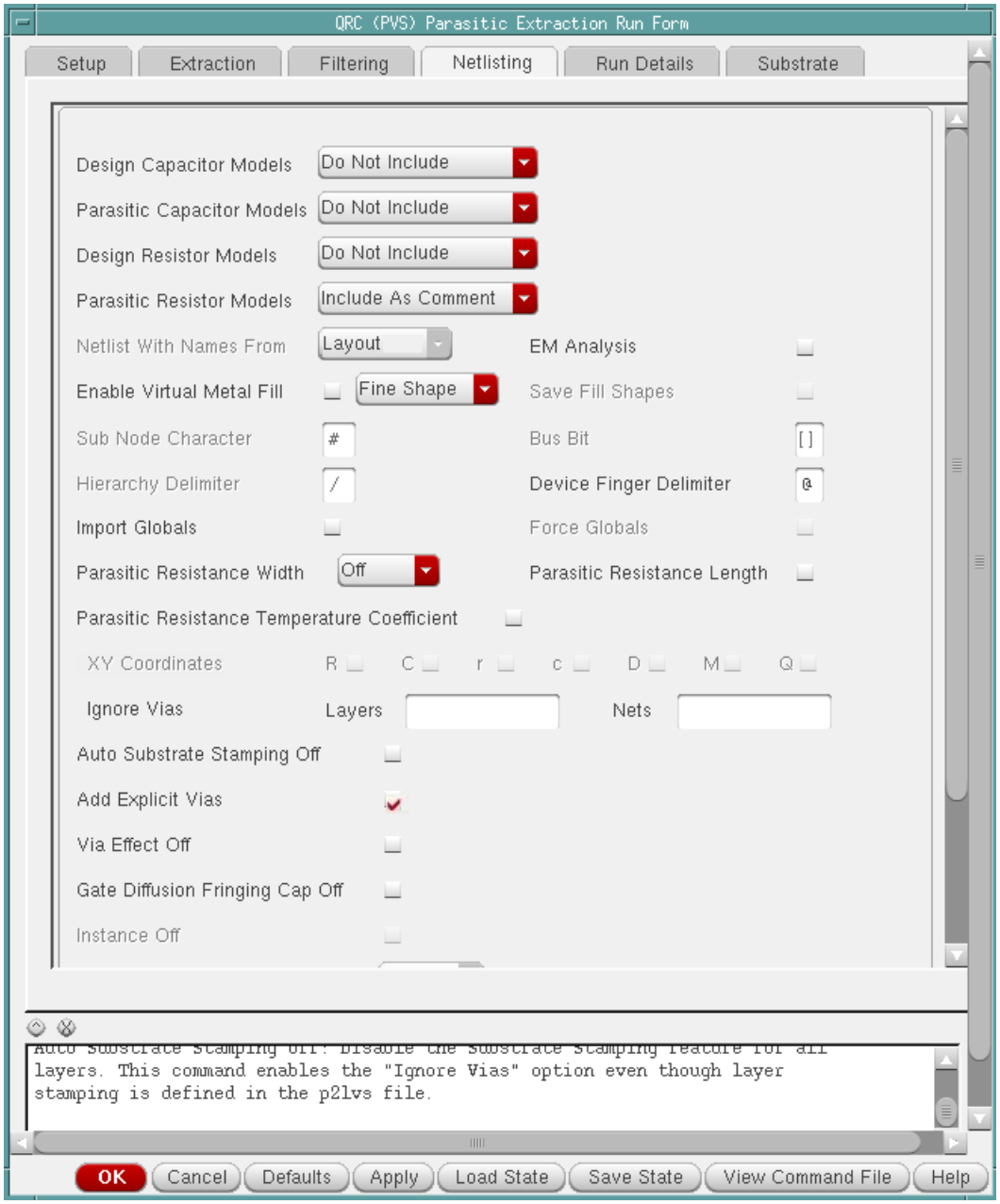
7. Select the *Netlisting* tab. This form is shown below.

Figure 3-12 Quantus (Pegasus) Parasitic Extraction Run Form – Netlisting Tab



© 2020 All Rights Reserved.

Figure 3-13 QRC (PVS) Parasitic Extraction Run form – Netlisting Tab



8. On this 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.

© 2020 All Rights Reserved.

- In the *Parasitic Resistance Width*, select *Drawn* from the options provided in the drop-down menu. The parasitic resistance width option annotates the extracted view with the width of parasitic resistors, which is required to compute the current density.
- Select *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.
- Disable *Import Globals* to enable Quantus QRC to extract resistance from power and ground nets.

Note: The EM analysis performed by Power IR/EM requires the parasitic resistor to be present in the extracted_view. Therefore, you must set the *Parasitic Resistors Models* switch to "`Include Model`" or "`Include as Comment`". If you select "`Include Model`", you will be required to update the `emDataFile.txt` to match model names.

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

Note: You must successfully create the extracted view in Quantus QRC to proceed to the next step.

Running Extraction with Quantus (Calibre) Interface

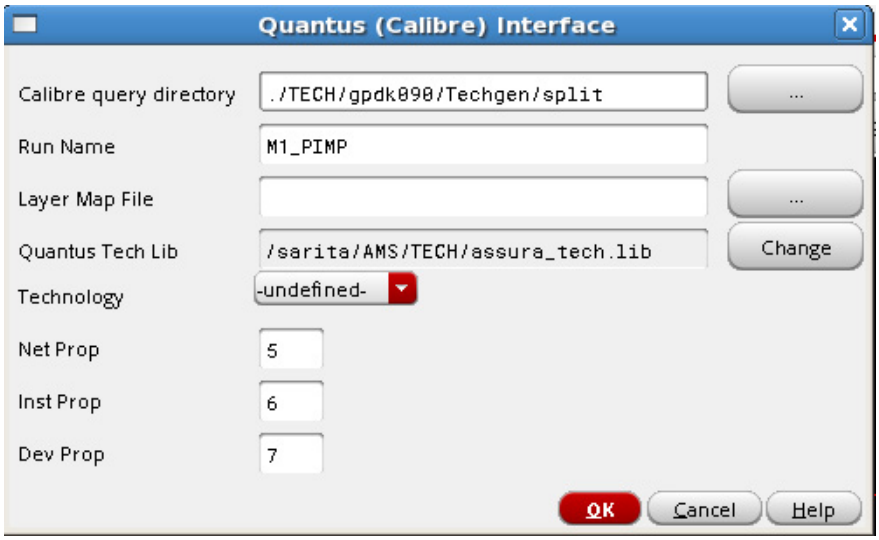
Quantus QRC accepts a Calibre database as input for extraction in place of Pegasus or PVS LVS.

The extracted view includes a standard DFII extracted view, which has parasitic properties back annotated onto a layout design for simulation purposes. The extracted view can be simulated using third-party simulation tools like Spectre, UltraSim, or APS to interpret the Quantus QRC output effectively, and use the simulation results as input to Power IR/EM.

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

© 2020 All Rights Reserved.

Figure 3-14 Quantus (Calibre) Interface Form



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

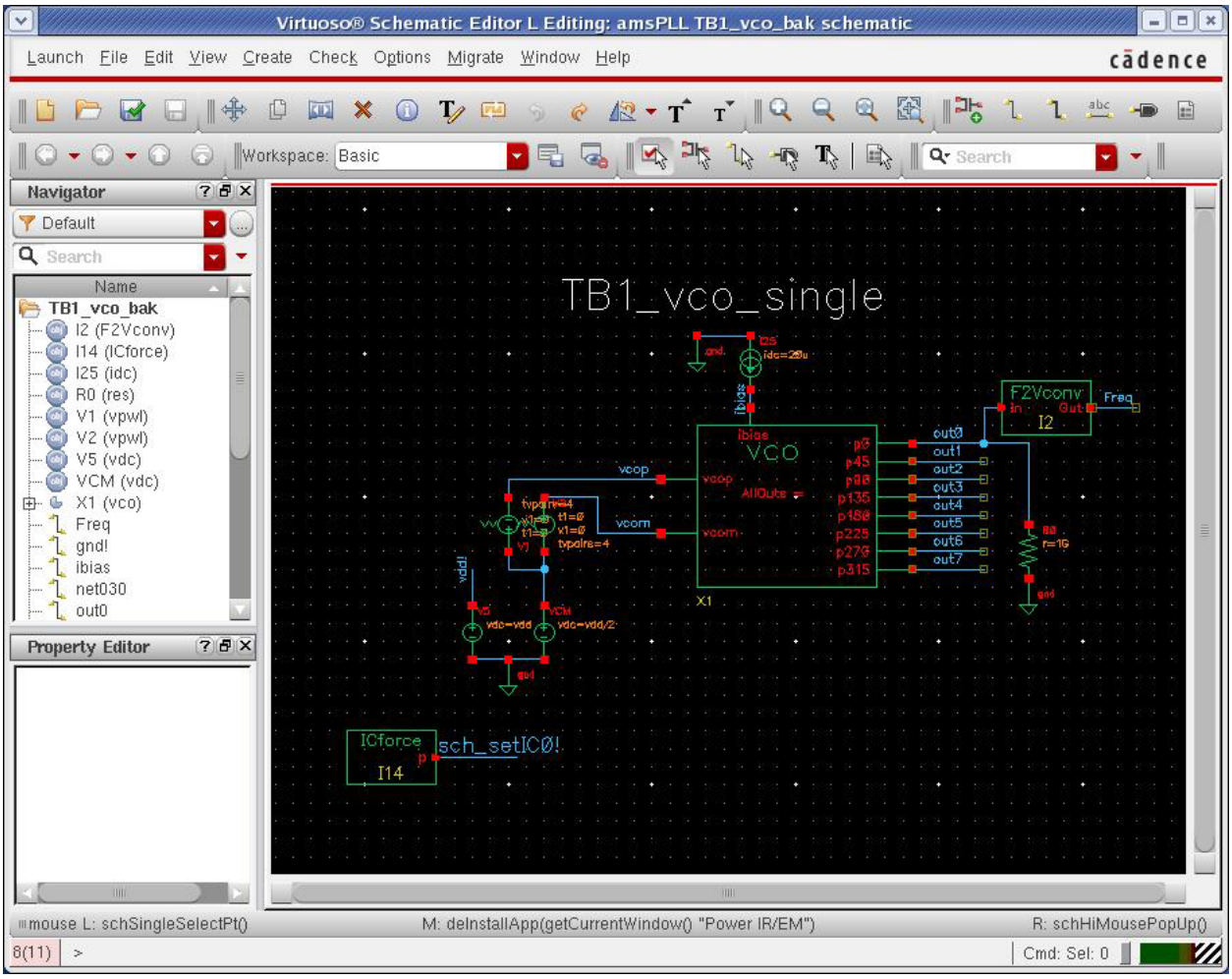
Running a Spectre Simulation

After running extraction, the next step is to use an existing testbench and simulate the extracted view within the Virtuoso® Analog design Environment (ADE) environment. You can use Virtuoso® 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.

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

© 2020 All Rights Reserved.

Figure 3-15 Testbench in Virtuoso Schematic Editor

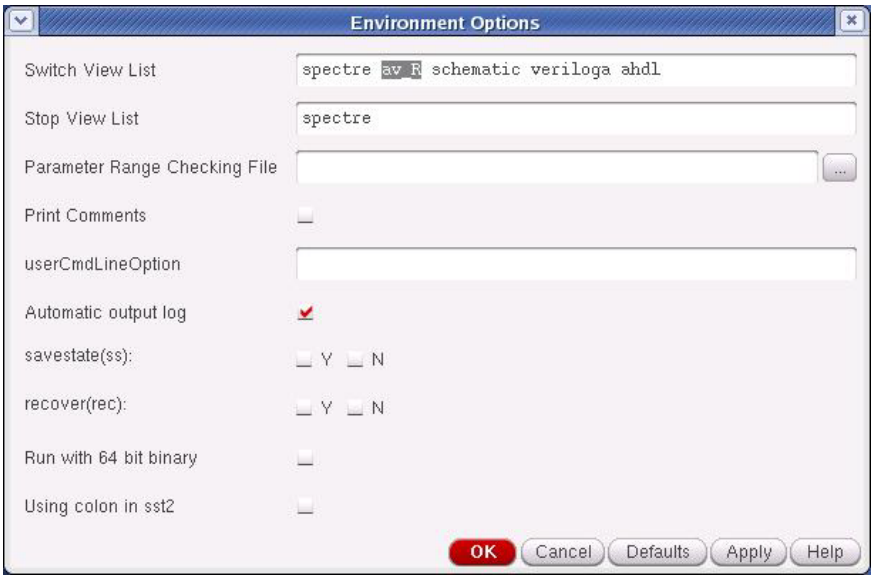


Follow these steps to run Spectre simulation for Power IR/EM:

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

© 2020 All Rights Reserved.

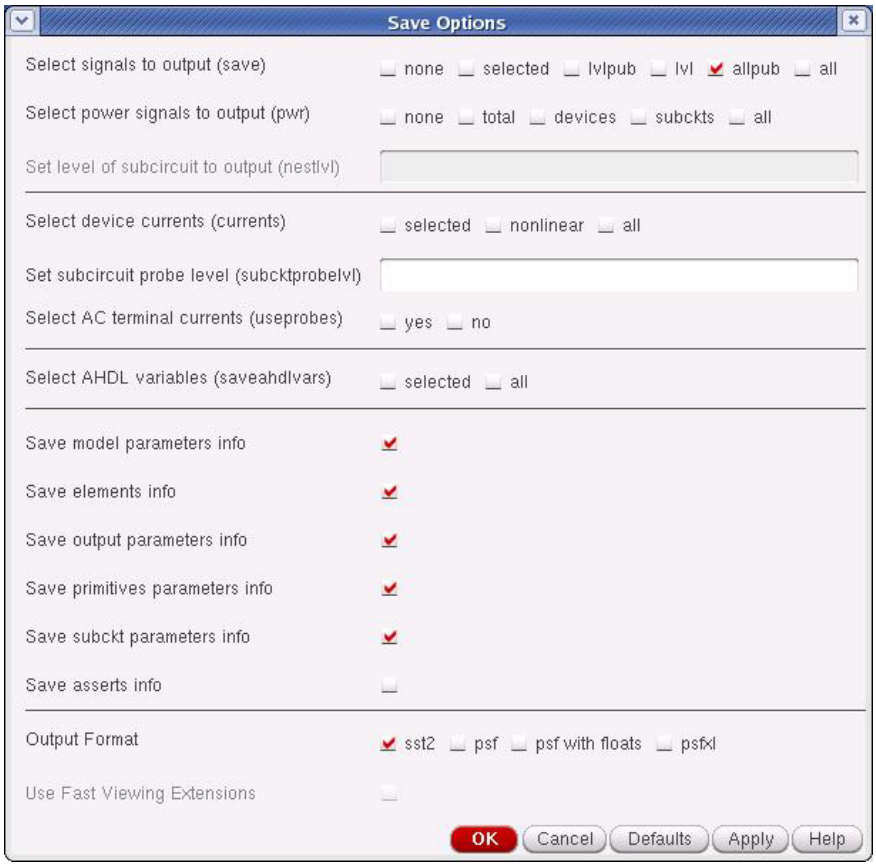
Figure 3-16 Environment Options Form



- In the Save Options form, make sure 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.

© 2020 All Rights Reserved.

Figure 3-17 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 Power IR/EM 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® Accelerated Parallel Simulator (APS). These simulation results can be used as input to Power IR/EM instead of 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 Virtuoso® 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.

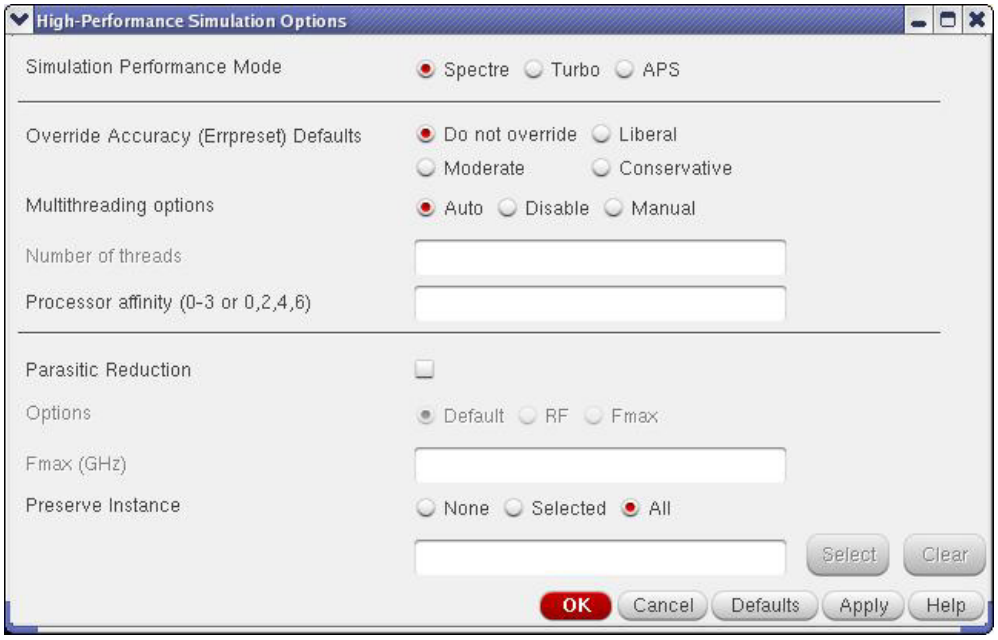
The APS use model is identical to that of Spectre, with the same netlist syntax, device model, analyses, features, and output format support.

© 2020 All Rights Reserved.

Follow these steps to run the APS simulation for Power IR/EM:

- In the Environment Options form, select the Switch view List to specify the extracted view in the simulation. You can also specify config views. This is shown above in the “Running a Spectre Simulation” section.
- In the Save Options form, make sure that the *allpub* option is selected. The Save Options dialog box 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. You do not need to create any `measure` statements.
- In the High-Performance Simulation Options form, ensure that *All* is selected in the *Preserve Instance* field. This is shown in the figure below.

Figure 3-18 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. Power IR/EM Reader can read results in `SST2` database format as well as `PSF/PSF-XL` waveform format.

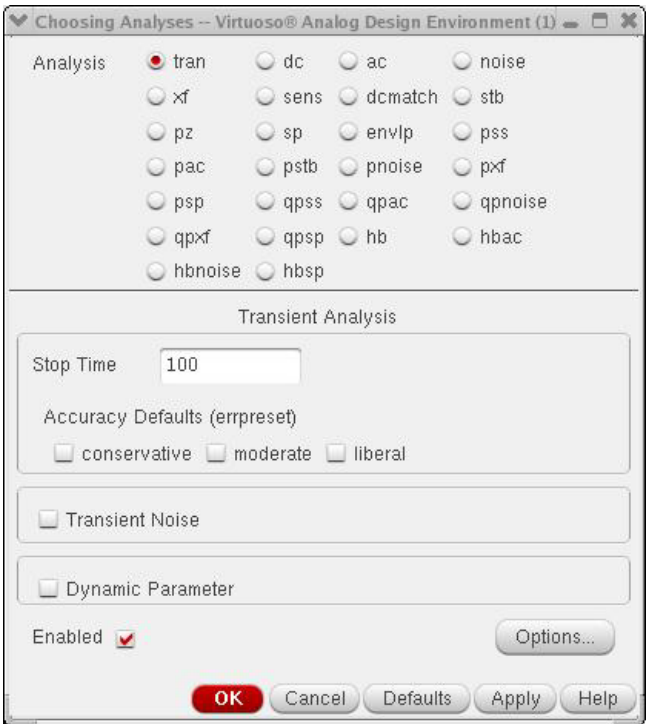
For very large designs, APS can dump the simulation data in a Power IR/EM-specific format. Users can perform average, peak, and RMS analyses in less time by using this data.

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

- In the Virtuoso® Analog Design Environment, select *Analysis – Choose*. The Choosing Analyses form opens.

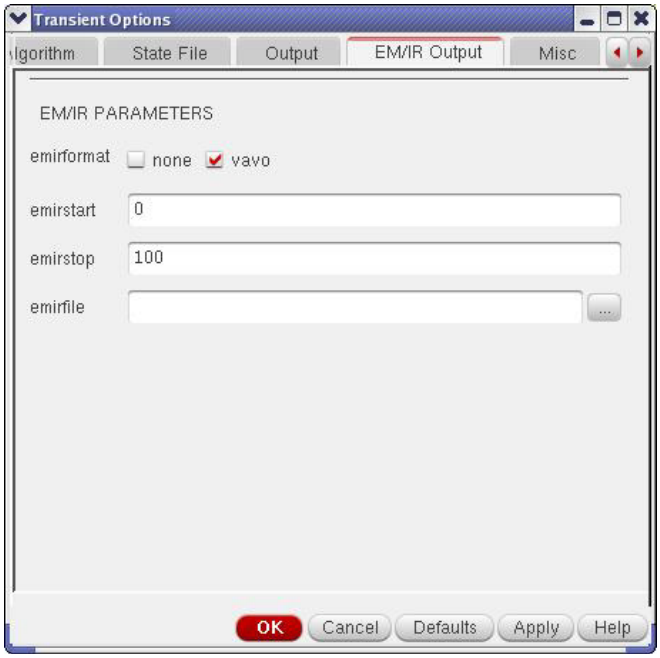
© 2020 All Rights Reserved.

- Select *tran* as the analysis type. This form is shown below.



- Select *Enabled*.
- Click *Options*. the Transient Options form opens. This form is shown below.

© 2020 All Rights Reserved.



- In the Transient Options form, select the *EM/IR Output* tab.
- On the *EM/IR Output* tab, select *vavo* as the *emirformat*.
- Specify the *emirstart* time and *emirstop* time
- Specify the *emirfomat* file and 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`.
`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.

© 2020 All Rights Reserved.

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

Running an UltraSim Simulation

You can use the Virtuoso® UltraSim simulator to perform simulation instead of using Spectre. UltraSim mode of simulation is recommended for large designs because it runs faster.

In general, the following options for UltraSim and Power IR/EM flow are recommended:

1. '`A`' mode without UPS (for smaller cases <5k devices), or
2. UPS with any simulation mode, most likely '`MS`' mode (default), which is the best approach for most designs

Important

For very large designs, UltraSim can dump the simulation data in a Power IR/EM-specific format. Users can perform average, peak, and RMS analyses in less time by using this data. The UltraSim option that enables dumping of simulation data in Power IR/EM-specific format is:

```
usim_emir type=all format=vavo stop=<simulation time>
```

Note: Power IR/EM reads in the simulation results of UltraSim and then tries to match the nodes in the data file with the extracted view. Therefore, you should not use RC reduction in the UltraSim and Power IR/EM flow. For example:

```
usim_opt postl=0 rvshort=0 rshort=0 cgnd=0
```

Also, UltraSim does not save node voltages for the parasitic resistors during the simulation by default. Therefore, you must manually set options to make UltraSim save these node voltages.

© 2020 All Rights Reserved.

Using DFII IC5.1.41 or later with UltraSim

1. Set up normal simulation
2. Under "Outputs" in ADE, select "Save All ..."
3. Turn on "Select all node voltages"
4. Turn on "Preserve All Nodes"
5. Set "Hierarchical Depth" equal to the depth of the extracted view in the schematic hierarchy

© 2020 All Rights Reserved.

© 2020 All Rights Reserved.

4

Setting Up the Power IR/EM Environment

- Setting Up the Environment Variables on page 64
- Manual Net Name Resolution on page 69

© 2020 All Rights Reserved.

Setting Up the Environment Variables

You can set a few environment variables before invoking the EM/IR software to pre-fill the fields of the *I*R (Voltage) Drop Analysis and the Electromigration (EM) Signal Analysis forms.

A few examples of environment variables are:

- `setenv _vsaIRDropSimDir ./simulation`

  Specifies the location of the Spectre simulation directory in the IR Drop Analysis form.

- `setenv _vsaEMSimDir ./simulation`

  Specifies the location of the Spectre simulation directory in the Electromigration Power Analysis form.

- `setenv _vsaEMQRCDir ./qrcDir`

  Specifies the location of the QRC run.

- `setenv _vsaEMDataFile ./emDataFile.txt`

  Specifies the location of the `emDataFile`. This variable is optional. If it is not set, the software looks for a file called `emDataFile.txt` first in the current working directory and then in the DFII technology directory.

- `setenv _vsaEMTechFile path_of_qrcTechFile`

  Specifies the location of `qrcTechFile`. If `emDataFile` is provided, this variable is ignored.

- `setenv _vsaEMMapFile path_of_mapfile`

  Specifies the location of the mapfile that contains layer mapping information between ICT/p2lvsfile/DFII layer names when `vsaEMTechFile` is specified. This variable is optional. If not specified, the tool assumes that the ICT/p2lvsfile/DFII layer names are same.

- `setenv _vsaEMDeltaT 10.1`

  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, Power IR/EM considers a default temperature of 5 degree Celsius.

- `setenv _vsaEMPreExecuteFile "test.il"`

  Specifies the pre execute file for EM analysis in qrcTechFile flow.

© 2020 All Rights Reserved.

- `setenv _vsaP2Lvs ./p2lvsfile`

  Specifies the location of the p2lvsfile used during RCX extraction.

- `setenv _vsaEMColumnViewConfigFile ./filename`

  Specifies the location of the column selection configuration file for displaying customized EM analysis results.

- `setenv _vsaSaveOnlyFailedResults true`

  When this variable is set to "`true`", the software saves EM analysis results for only the failed nets. This setting overrides the default selection of the *Save Only Failed Results* field in the Save Electromigration Analysis to a File form.

- `setenv _vsaLoadOnlyFailedResults true`

  When this variable is set to "`true`", the software loads EM analysis results for only the failed nets. This setting overrides the default selection of the *Load Only Failed Results* in the Select Existing Results File form.

- `setenv _vsaSelectLayers "Via1 m.*"`

  Specifies the list of selected layers for EM analysis.

- `setenv _vsaSelectResistors "Rh.* Re.*"`

  Specifies the list of selected layers for EM analysis.

- `setenv _vsaIgnoreLayers "Via1 m.*"`

  Specifies the list of layers to be ignored for EM analysis.

- `setenv _vsaIgnoreResistors "Rh.* Re.*"`

  Specifies the list of resistors to be ignored for EM analysis.

- `setenv _vsaUseNewLayers false`

  When this variable is set to `false`, Power IR/EM uses the layer names specified in the technology file for highlighting the IR/EM analysis results. By default, this variable is set to `"true"`. When set to "`true`", the tool uses the layer names `vavo0` through `vavo8` for highlighting the IR/EM analysis results.

- `setenv _vsaDrfFile "vavo.drf"`

  Specifies the layer names to be highlighted in the IR/EM analysis results. If this file is not specified, the tool will first look for the file in the current directory and then in the Power IR/EM Installation.

  Note: The new layer names to be highlighted should be specified as `vavo0` through

© 2020 All Rights Reserved.

`vavo8`.

- `setenv _vsaTechFile "vavo_techfile"`

  Specifies the display properties for the layers to be highlighted in the IR/EM analysis results. If this file is not specified, the tool will first look for the file in the current directory and then in the Power IR/EM Installation.

  Note: The above variables, `_vsaDrfFile` and `_vsaTechFile`, are specified together. Refer to *"Technology File and Display Resource File ASCII Syntax Reference Manual"* for more information about the contents of these files and how to modify them.

- `setenv _vsaICTConductorUnit "mA"`

  Specifies the unit for current density of metal layers as mA. The ICT file supports A/cm^2 and mA/um as the units for specifying the current density of metal layers. You can use this variable to specify the current density of these layers in mA.

- `setenv _vsaEnablePWL "1"`

  Specifies that the piece-wise linear (PWL) rule will be applied to the intermediate area values.

- `setenv _vsaSupport28nmRules "true"`

  Enables explicit support for 28-nm rules. By default, the tool supports 28-nm rules when the conditions specific to 28 nm are specified in the `emDataFile`. This variable is helpful in case of a custom function where rules are not specified in the `emDataFile`.

- `setenv _vsaTechnologyNode 40`

  Enables you to configure technology-dependent calculations. For example effective length calculation. You can also specify the `technologyNode = 40` in the `emDataFile`.

- `setenv _vsaEnablePowerGridRules "true"`

  Enables power-rail analysis. By default, power-rail analysis is disabled.

- `setenv _vsaLinearInterpolation "1"`

  Enables linear interpolation for the temperature derating factor.

- `setenv _vsaDisableAllDesignResistorAnalysis "true"`

  Disables analysis of all design resistors that are not connected to the selected pins. By default, the software analyzes all design resistors, irrespective of whether they are connected to the selected pins or not.

- `setenv _vsaLSFMachineString "bsub -q lnx64"`

© 2020 All Rights Reserved.

Specifies the name of the LSF command string that will launch Power IR/EM in the batch mode.

- `setenv _vsaApplyRThreshold "0.05"`

  Specifies the threshold ratio "r" for AC-Peak analysis. The default value of threshold ratio is 0.05. You can also specify the threshold ratio in the following ways:

  - In the Electromigration Edit/View Custom Parameters form, by using the `applyRThreshold` custom variable
  - In the ICT file, by using the `applyR` keyword

  For details, see Rules for Defining Limit-Based Parameters in the "EM Rules Specification" chapter.

- `setenv _vsaRMSMultiplierPin "1.0"`

  Specifies the RMS relaxation factor for EM power-grid analysis. The default value is `1.0`. You can also specify the value in the Edit/View Custom Parameters form by selecting the `RMSMultiplierPin` parameter.

- `setenv _vsaRMSMultiplierSignal "1.0"`

  Specifies the RMS relaxation factor for signal EM analysis. The default value is `1.0`. You can also specify the value in the Edit/View Custom Parameters form by selecting the `RMSMultiplierSignal` parameter.

- `setenv _vsaDisableLastConfig "true"`

  Specifies that the configuration file created in the previous session will not be loaded by default for either IR drop or EM analysis in the current run. When you open either the IR (Voltage) Drop Analysis form or the Electromigration Power Grid Analysis form, the fields are prefilled with the settings specified in the previous session. These settings are saved in the `vsaIRLast.cfg` file for IR drop analysis and in the `vsaEMLast.cfg` file for EM analysis.

- `setenv _vsaUncheckViewFile "true"`

  Specifies that files generated during both IR drop and EM analysis should not open for viewing by default. In the GUI, the *View file* option to view the command files and result files for both IR and EM analyses is checked by default. You can specify this variable if you do not want to view these files.

- `setenv _vsaJmaxLife "5.0"`

  Specifies the scaling factor that applies to the current density limits for different lifetimes. The unit of the lifetime value is the same as that specified using the `em_lifetime_units` parameter defined in the process section of the ICT file. The

© 2020 All Rights Reserved.

default is `5.0`. You can also specify the lifetime value in the Electromigration Edit/View Custom Parameters form by selecting the `JmaxLife` parameter.

- `setenv _vsaMinWarning "true"`

  Specifies that minimum warning messages will be given during a session. When this environment variable is set to `true`, the following warning messages will be suppressed:

  - Duplicate via width information warning message while specifying the `emDataFile` flow in EM analysis. By default, the via width information in the `emDataFile` will override that specified in `qrcTechFile`.
  - Missing current density specifications warning message

- `setenv _vsaIncludeDFIINodeCol "true"`

  Specifies that columns providing information about the DFII node 1 and DFII node 2 should be displayed in the Electromigration Analysis Results form. By default, information about these nodes is not displayed in the form. When this variable is set to `true`, the Customize display of EM results form shows DFII node 1 and DFII node 2 as options available in the Columns section. You can select the options to display these columns in the Electromigration Analysis Results form.

- `setenv _vsaDisableAutoArchive "true"`

  Specifies that the IR drop analysis and EM analysis result files (`IRDrop.results` and `EM.results`) and form configuration files (`IR.cfg` and `EM.cfg`) should not be saved in the working directory. These files are saved in the working directory by default when you click the *OK* or *Apply* buttons in the IR Drop Analysis or the Electromigration Analysis forms.

- `setenv _vsaDBRefPinThreshold` *`value`*

  This variable is only used while performing IR drop peak analysis in the `vavo-db` flow. It is used to specify the maximum value of variation allowed in the reference pin's voltage value (difference between the min and max values). Pins that have a min-max voltage value variation less than the specified value will be considered stable and will be included in the IR drop peak analysis while the pins that have a min-max voltage value variation greater than the specified value will be excluded from the IR drop peak analysis.

  The range of threshold voltage variation values that can be specified is from 0 to 1.

  When "0" is specified, then those pins that have any variations in the min-max voltage values, will be excluded from the IR drop peak analysis.

  When "1" is specified, all pins will be included in the IR drop peak analysis irrespective of the min-max voltage voltage value variations.

© 2020 All Rights Reserved.

The default value of this variable is 0.05, which means the maximum allowed variation in the reference pin's voltage is 5% of its value. The software will include pins with a value less than 5% in the analysis and it will ignore all pins that have a min-max variation of greater than 5% and exclude them from the analysis.

Note: For IR drop peak analysis on transient data for dynamic pins, `vavo-db` simulation data is not recommended. Use the `psf` data instead. However, if you still want to do the IR drop peak analysis in the vavo-db flow, use the `_vsaDBRefPinThreshold` variable to define the threshold value for the stability checks.

Dynamic pins have different min-max pin voltage values for a given period of analysis. For example, in transient analysis, the power supply pin and ground pin may be static pins, while the input and output pins may be dynamic pins.

- `setenv _vsaMicronUnit "true"`

  Specifies that the units for length/width information in the `extview.tmp` file will be considered as micron.

- `setenv _vsaDumpResultDBPath` *`path`*

  Specifies the path of the `VSA_RESULTS` directory that is used for viewing the IR drop and EM analysis results in vsaPlot. By default, this directory is created in the current working directory.

- `setenv _vsaBulkTerminal "r1,r2,.."`

  This variable is only applicable to the design resistor flow. It is used to specify the bulk terminals of the design resistors that are to be ignored while performing EM analysis. You can provide the names of multiple terminals enclosed in commas. When this variable is specified, the value of the `BulkTerminal` parameter in the GUI is updated.

Manual Net Name Resolution

Power IR/EM automatically resolves the net name format used in the PSF/PSF-XL waveform or SST2 database format while importing simulation results. However, sometimes Power IR/EM might fail to correctly identify the net name convention and that will result in an error reading each net while importing the simulation results.

The environment variable `_vsaNetNameFormat` can be used to manually set alternative net name formats to use when importing the simulation results. The possible values of `_vsaNetNameFormat` are:

```
SpicePSF, Spice, SpectreSST, Spectre
```

This environment variable can be used as follows:

© 2020 All Rights Reserved.

1. When encountering excessive net names errors while importing the simulation results, check the currently used name format by typing the following in the Command Interpreter Window (CIW):

   ```
   _vsaGlobalIRDropNameFormat
   ```

   The variable and its current value is displayed in the CIW as follows:

   ```
   _vsaGlobalIRDropNameFormat “Spice”
   ```

2. Set the `_vsaNetNameFormat` environment variable to a net name format other than the one returned in Step 1 above. Type the following in the CIW:

   ```
   setShellEnvVar("_vsaNetNameFormat=SpicePSF")
   ```

   To check the value of the `_vsaNetNameFormat` environment variable, type the following in the CIW:

   ```
   getShellEnvVar("_vsaNetNameFormat")
   ```

3. Now repeat the IR or EM analysis to see if Power IR/EM can import the simulation results. If you still have problems importing the simulation results, you can change the value of `_vsaNetNameFormat` to one of the remaining values.

   When you have discovered the value of `_vsaNetNameFormat` that works for your design, set the environment variable in your setup file so that it will be used whenever you run Power IR/EM.

© 2020 All Rights Reserved.

5

IR Drop Analysis

- Overview on page 72
- Package Effect in IR Drop Analysis on page 72
- IR Drop Analysis on page 73
  - Analyzing IR Drop Across Design Resistors on page 79
  - Writing Out an IR Command File on page 80
  - Opening Power IR/EM Directly from Extracted View on page 82
- IR Drop Results on page 82
  - Importing Results of a Previous Analysis on page 83
  - Selecting Options in the IR (voltage) Drop Results Form to View Results on page 85
  - Customizing Error Layer Display for IR Drop Results on page 91

© 2020 All Rights Reserved.

Overview

IR Drop is the voltage reduction that occurs on power supply networks (VDD) in integrated circuits.

Integrated-circuit design usually assumes the availability of an ideal power supply that can instantly deliver any amount of current to maintain the specified voltage throughout the chip. In reality, however, the combination of increasing current per-unit area on the die and narrowing metal line widths (which increases the power-grid resistance) causes localized voltage drops within the power grid, leading to decreased power supply voltage at transistors.

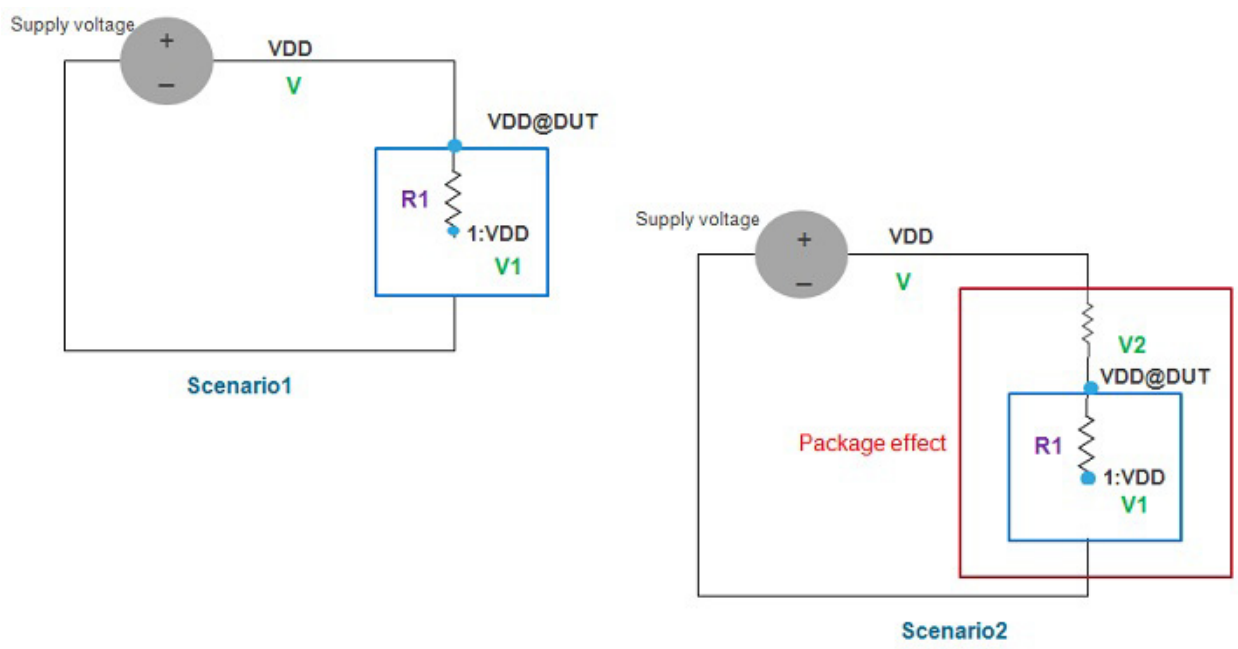
These localized drops in the power supply voltage decrease the operating voltage of the chip, potentially causing timing problems and functional failures.

IR Drop analysis involves analyzing the power grid to identify the IR Drop problems so that they can be corrected.

Package Effect in IR Drop Analysis

The image below depicts how voltage drop is calculated by the software and how package effect is incorporated in IR drop analysis to report accurate IR drop.

Figure 5-1 Calculating IR Drop



© 2020
All Rights Reserved.

In **Scenario1** shown in above image, the IR drop at node, 1:VDD, is calculated using the following equation:

IR drop at 1:VDD = V – V1

In **Scenario2**, there is additional resistance due to package effect. However, by default, the software will calculate the IR drop at node, 1:VDD, using the following equation:

IR drop at 1:VDD = V2 – V1

To incorporate package effect in the above calculation, you can specify the value of “V”. The software will then calculate the IR drop at node, 1:VDD, using the following equation:

IR drop at 1:VDD = V – V1

To specify the voltages, open the IR (voltage) Drop Analysis form and click *Update Voltages for Pins/Signals* to update voltages for pins/signals appearing in the list box. The details are provided in the subsequent section.

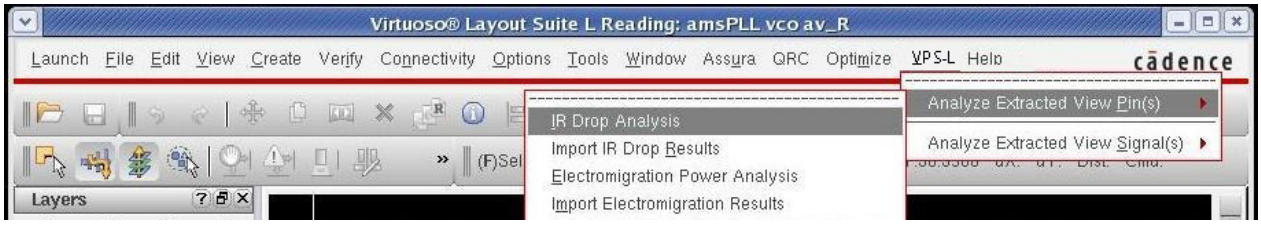
IR Drop Analysis

Power IR/EM is used to create a “thermal map” representation of the IR Drop on selected pins and signals.

To perform IR Drop analysis from the Virtuoso layout window, follow these steps:

- In the Main menu, choose *VPS-L.* Two options*, Analyze Extracted View Pin(s)* and *Analyze Extracted View Signal(s)* appear in the drop-down list.
- Choose *Analyze Extracted View Pin(s)* for analyzing the power pins of the design and *Analyze Extracted View Signal(s)* to analyze the signals of the design.

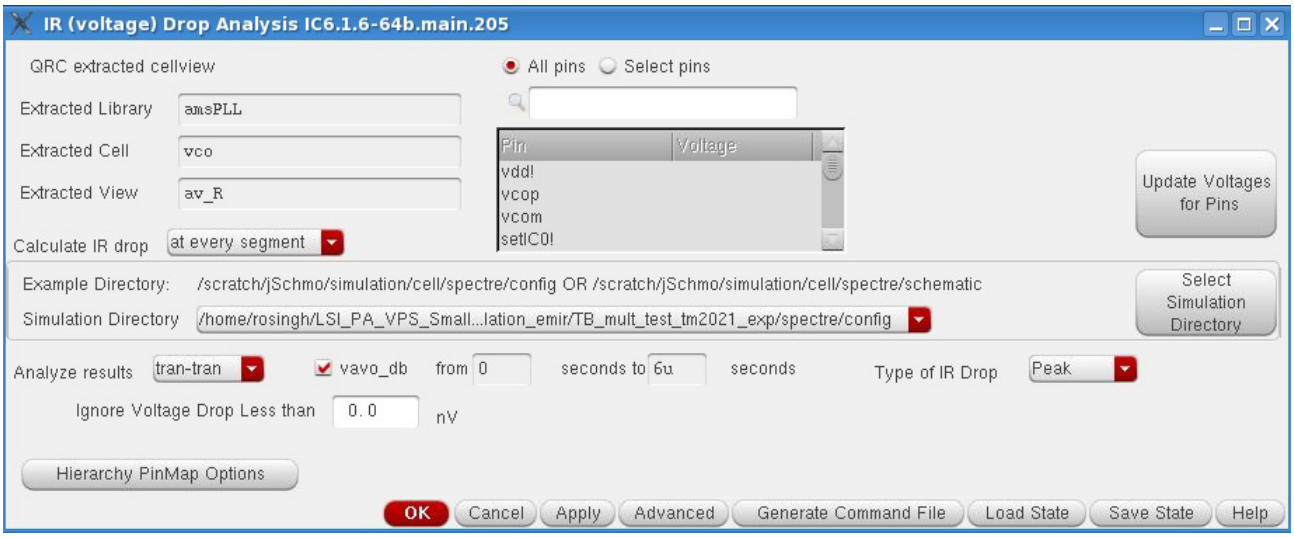
Figure 5-2 Opening IR Drop Analysis Form



- From the drop-down list, select *IR Drop Analysis.* The IR (Voltage) Drop Analysis form opens.

© 2020 All Rights Reserved.

Figure 5-3 Running (Voltage) Drop Analysis



Note: When the above form opens, the settings that were specified in the previous session are displayed. These settings are saved in the `vsaIRLast.cfg` file that is created for every session by default. However, if you do not want the software to load the settings of the previous session by default, set the `-vsaDisableLastConfig` environment variable to `true`.

- In the *QRC extracted cellview*, you can view the *Extracted Library*, *Extracted Cell*, and *Extracted View* name created during the extraction run. See Running Pegasus Quantus QRC in the "IR/EM Analysis Flow" chapter. The default view name for an extracted view is `av_extracted`.
- Click *All pins* to perform IR drop analysis on all pins. The default option is to perform IR drop analysis on all pins.
- Click *Select pins* to select the pin names on which the IR Drop analysis should be performed.

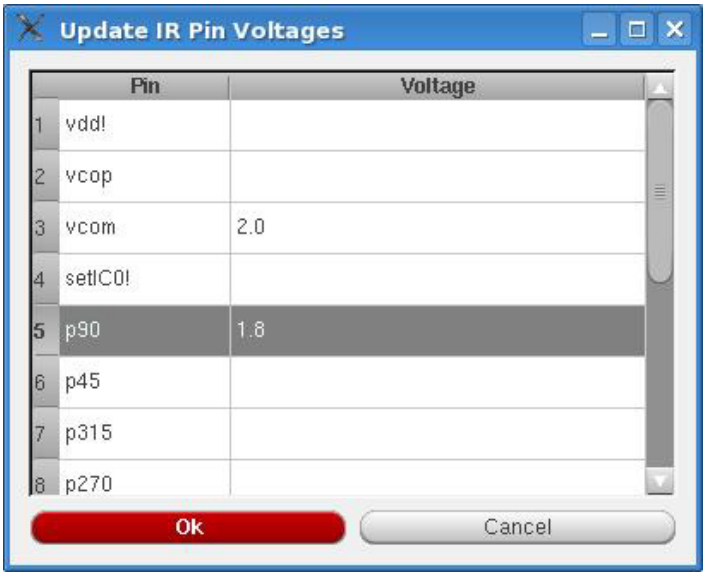
  Note: When *Analyze Extracted View Signal(s)* is selected, the form shows the above two fields as *All signals* and *Select signals* respectively. In this case, use *Select signals* to select the signal names on which the IR Drop analysis should be performed. Pin or signal names can be searched by typing the name of the pin or signal in the search field. The search field also accepts SKILL regular expressions.
- Click *Update Voltages for Pins* to update voltages for pins appearing in the list box. When you click this button, the Update IR Pin Voltages form opens. This form is shown below.

  Note: When *Analyze Extracted View Signal(s)* is selected, the IR (voltage) Drop Analysis form shows an *Update Voltages for Signals* button. Click this button to update

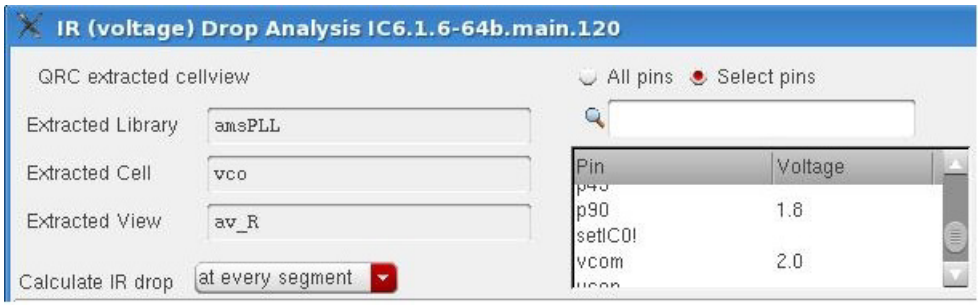
© 2020 All Rights Reserved.

voltages for signals.

Figure 5-4 Updating Voltages for Pins



The above form shows a table with two columns, “Pin” and “Voltage”. Double-click the “Voltage” column to type the updated voltage for any pin. Click *OK*. The updated voltages appear in the list box in the IR (Voltage) Drop Analysis form. This is shown below.



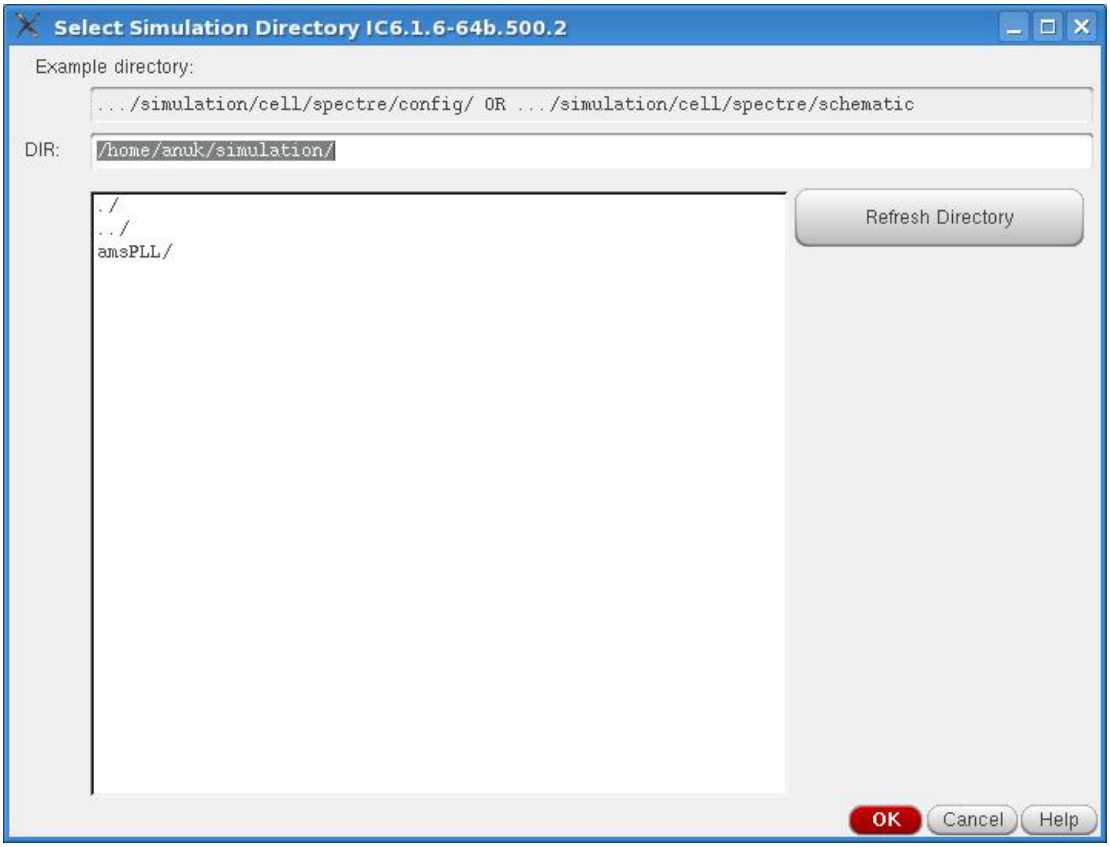
Note: When you click *Update Voltages for Signals*, the Update Signal Voltages form opens. You can update the voltages in this form and click *OK*. The updated voltages appear in the list box in the IR (Voltage) Drop Analysis form.

- *In* the *Calculate IR Drop* cyclic field, choose *at every segment* to include all segments of the net in the IR Drop analysis.
- Choose the *Simulation Directory* to point to the results of the Spectre, UltraSim, or APS simulation that was performed on the extracted view.

© 2020 All Rights Reserved.

- You can also click *Select Simulation directory* to specify the simulation directory. The Select Simulation Directory form opens. This form is shown below.

Figure 5-5 Selecting the Simulation Directory



  - In this form, choose the simulation directory and click *OK*.

    For more information, see Running a Spectre Simulation, Running An UltraSim Simulation, or Running an APS Simulation in the “IR/EM Analysis Flow” chapter.

    Note: The *Example Directory* field shows you how to specify this value.

- In the *Analyze results* cyclic field, choose either *DCOP* or *TRANSIENT* simulation.
  - The DC operating point (DCOP) analysis is recommended for large analog designs that are difficult to simulate in Spectre using transient analysis, especially with all parasitics included.
  - The *TRANSIENT* option sets the simulation time in seconds automatically when the *from/to* fields are selected. It also allows you to specify the following methods of processing results.

© 2020 All Rights Reserved.

- Average—Calculates the current density violations based on the average value of the DC current for metal lines, vias, and contacts.
- Peak—Calculates the current density violations based on the maximum DC current for metal lines, vias, and contacts.

- If your simulation data is in `vavo.db` format, the *vavo_db* option appears in the form. This option is selected by default. To use the PSF database, deselect this option. This option is available only if you are using `vavo.db`. You can also specify this option in the batch mode by using the `_vsa_vavo_db_enable` command. For details of this command, see the "Batch Mode Execution" chapter.
- In the *Type of IR Drop* cyclic field, choose the type of IR Drop analysis to be performed, *Peak* (default) or *Average*. In the results file that is generated, the "Results type" field will indicate whether the *Type of IR Drop* is Peak or Average. The IR Drop Results form shows the type of analysis being performed in the title bar.
- In *Ignore Voltage Drop Less than*, specify the minimum voltage for which you want to view IR drop results. The software will only show IR drop results for voltages greater than that specified in this field.
- Click *Save State* to save the information filled out in the form. The Save IR Form State To A File form opens. This form is shown below.

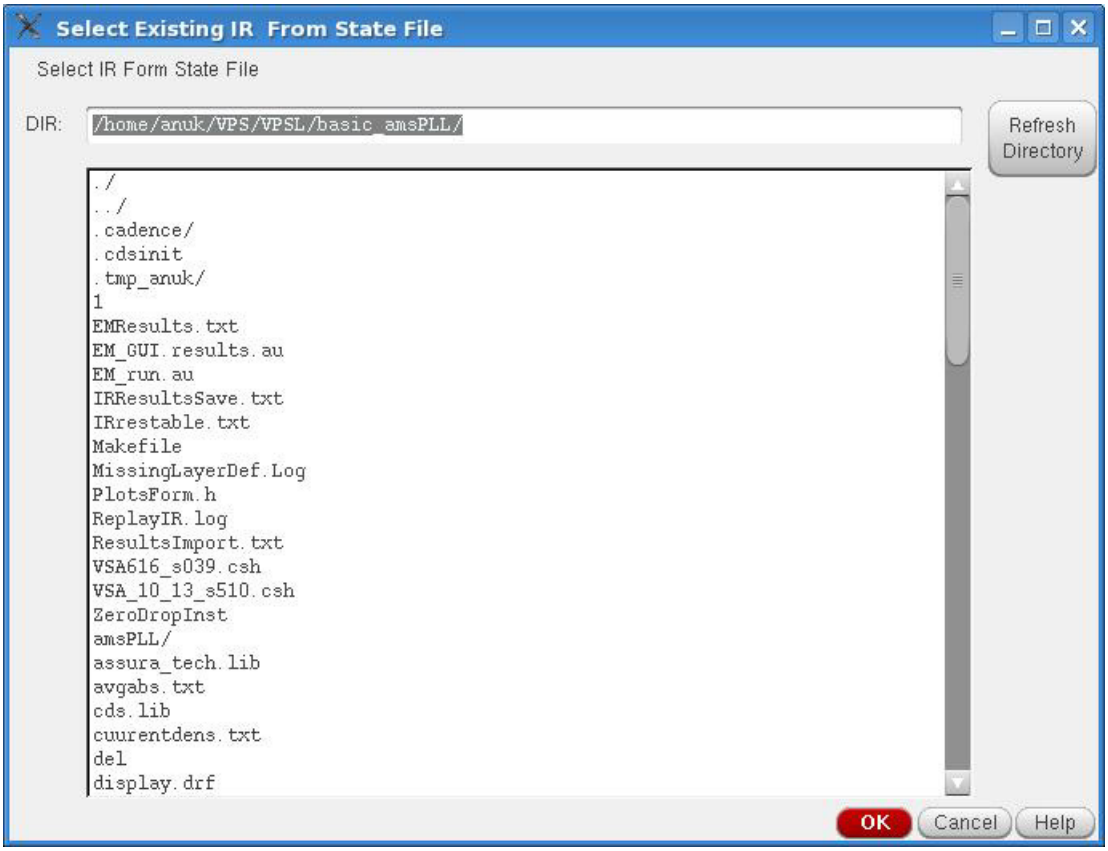
Figure 5-6 Saving IR Form State to a File

- In this form, specify the *Output File Name* and click *OK*.

- Click *Load State* to load previously saved information in the form. The Select Existing IR From State File form opens. This form is shown below.

© 2020 All Rights Reserved.

Figure 5-7 Selecting an Existing IR State



- In this form, select the file and click OK.

- After completing the inputs in the IR Drop Analysis form, click *OK* or *Apply*. When you click *OK* or *Apply* in this form, the result file and the configuration file are stored in the following location by default:

  `working directory/VSA.OUT/CURRENT_DATE/`

  Example:

  `working_dir``/VSA.OUT/MAY_5_1/IRDrop.results and IR.cfg`

  If multiple sessions are run on the same day, the name of the `CURRENT_DATE` folder will be updated sequentially. This is shown below.

  `working_dir``/VSA.OUT/MAY_5_2/IRDrop.results and IR.cfg`

  `working_dir``/VSA.OUT/MAY_5_3/IRDrop.results and IR.cfg`

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

© 2020 All Rights Reserved.

Important

If you do not want to save the above files, set the `_vsaDisableAutoArchive` environment variable to `"true"`.

Analyzing IR Drop Across Design Resistors

By default, Power IR/EM performs IR Drop analysis across parasitic resistors as part of a single net entity and the net is considered to terminate at the pin of the first design device, which might be a resistor. Normally, the signal resumes on the other side of the resistor (the other pin) with a different net name, and is considered by Power IR/EM to be a different signal during IR Drop analysis.

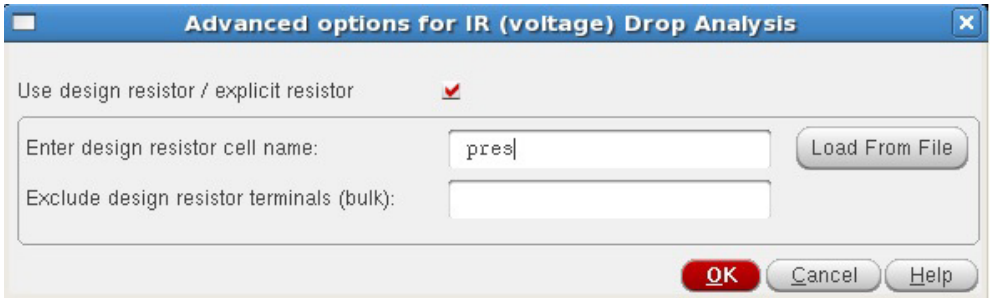
However, in many designs, metal resistors are added to power lines and it is desirable to perform IR Drop analysis on the complete net, even though the signal might traverse both design and parasitic resistors. You can specify the design resistors to include in the IR drop analysis as shown below.

You can also exclude bulk terminals of the design resistors from the IR drop analysis by specifying the names of the terminals in the Advanced options for IR (voltage) Drop Analysis form. This is shown below.

To specify design resistors and to exclude their bulk terminals for IR drop analysis, follow these steps:

- In the IR (Voltage) Drop Analysis form, click *Advanced* at the bottom of the form. The form shown below opens.

Figure 5-8 Specifying Advanced options for IR (Voltage) Drop Analysis Form

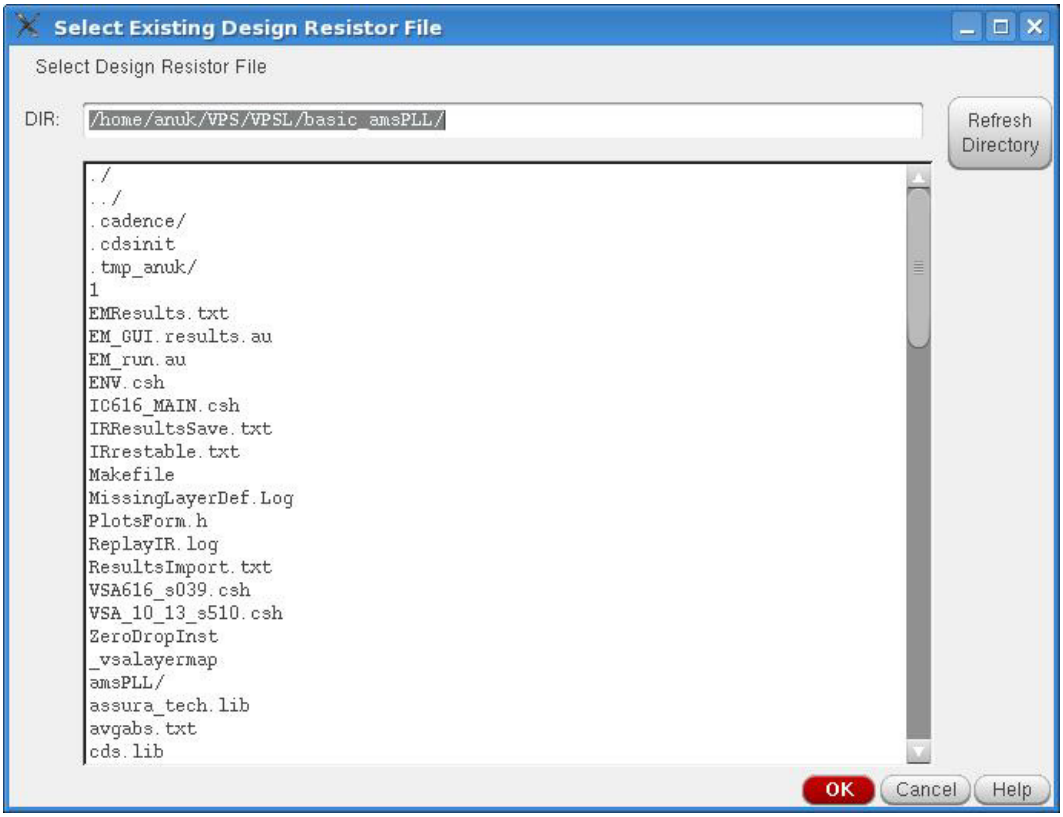


- Click the *Use design resistor / explicit resistor* option to specify that IR Drop analysis should be performed across design resistors.

© 2020 All Rights Reserved.

- In the *Enter design resistor cell name* field, type the cell names of the specific design resistors that are to be included in the IR Drop analysis. Multiple cells can be specified by placing a space between cell names.
- In the *Exclude design resistor terminals (bulk)* field, type the names of the bulk terminals to exclude from the IR drop analysis. Multiple terminals can be specified by placing a space between terminal names.
- Click *Load From File* to specify the file with the cell names of the specific design resistors to be included in the IR drop analysis. The Select an Existing Design Resistor File form opens. This form is shown below.

Figure 5-9 Selecting an Existing Design Resistor File



- In this form, choose the file and click *OK*.

Writing Out an IR Command File

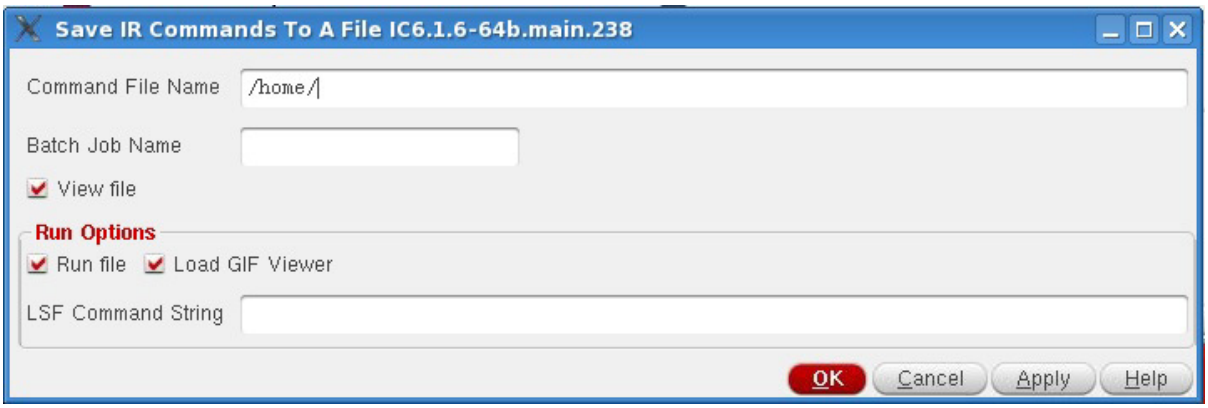
To generate a batch mode command file, follow these steps:

- In the IR (Voltage) Drop Analysis form, select the *Results Directory* and pin name.

© 2020 All Rights Reserved.

- Click *Generate Command File.* The Save IR Commands To a File form opens. This form is shown below.

Figure 5-10 Saving IR Commands To a File



- In this form, specify the *Command File Name*. If the command file name 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 `vsaIRBatch.cmd` as the default command file name.
- 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`.
- 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 Power IR/EM 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.
- Click *Load GIF Viewer* to launch the vsaPlot, which is used for visualizing and customizing the IR drop and EM analysis results. For details, see the "Visualizing Results in vsaPlot" chapter.
- Click *OK*.

© 2020 All Rights Reserved.

Opening Power IR/EM Directly from Extracted View

In the GUI mode, the extracted view can be opened directly, bypassing the need for a top-level schematic. When IR Drop analysis is started, and the testbench schematic is available, a form opens asking for the hierarchical name of the design. Provide the hierarchical name of the design and ensure that the hierarchical name is complete including the delimiter being used, if any. For example /X1 or /IO/I1.

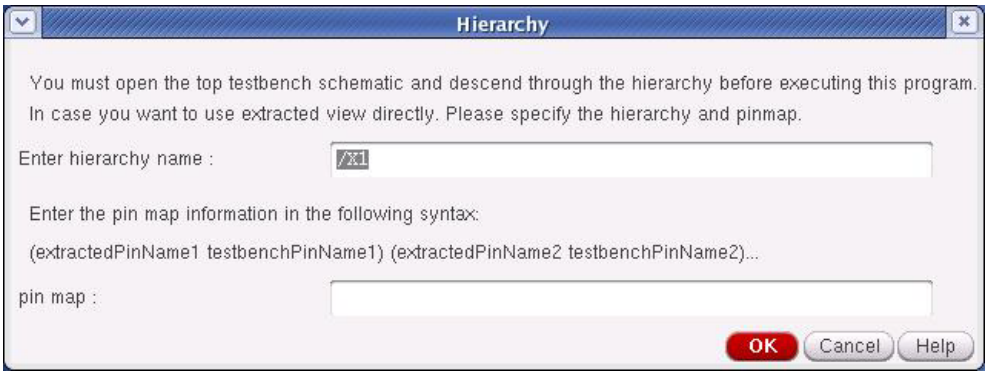
The pin map information can be entered in the same form and is specified in the following format:

```
(extracted_pin_name1 testbench_pin_name1)(extracted_pin_name2
     testbench_pin_name2) ...
```

This information is required to be added for any pins that have different names between the extracted view and testbench schematic view.

This form for specifying hierarchy and pin map information can also be evoked by clicking the *Hierarchy PinMap Options* on the IR (Voltage) Drop Analysis form or Electromigration Power Grid Analysis form depending upon the type of analysis you want to perform. This provides the option to change hierarchy or pin map information. This form is shown below.

Figure 5-11 Hierarchy Form



IR Drop Results

You can view the results of the IR drop analysis by selecting the various options available in the IR (voltage) Drop Results form. You can either import the results of an IR drop analysis performed earlier or view results for the current IR drop analysis run by using this form. Both methods are detailed below.

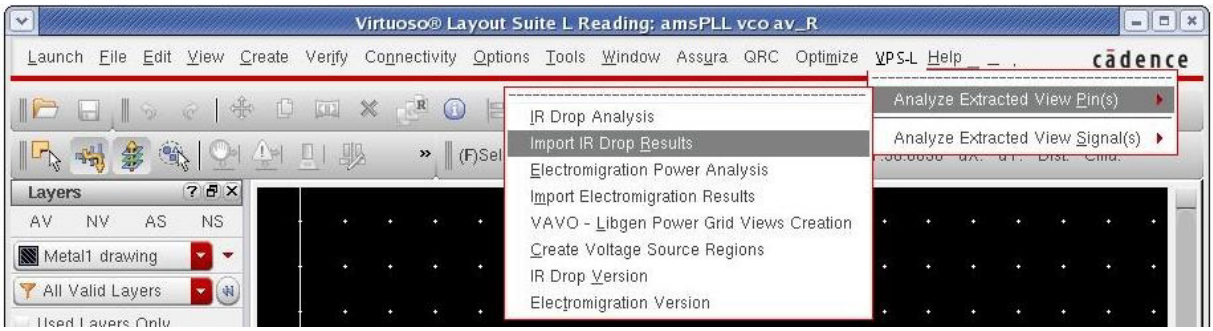
© 2020 All Rights Reserved.

Importing Results of a Previous Analysis

To import the results of an IR Drop analysis performed earlier, perform the following steps:

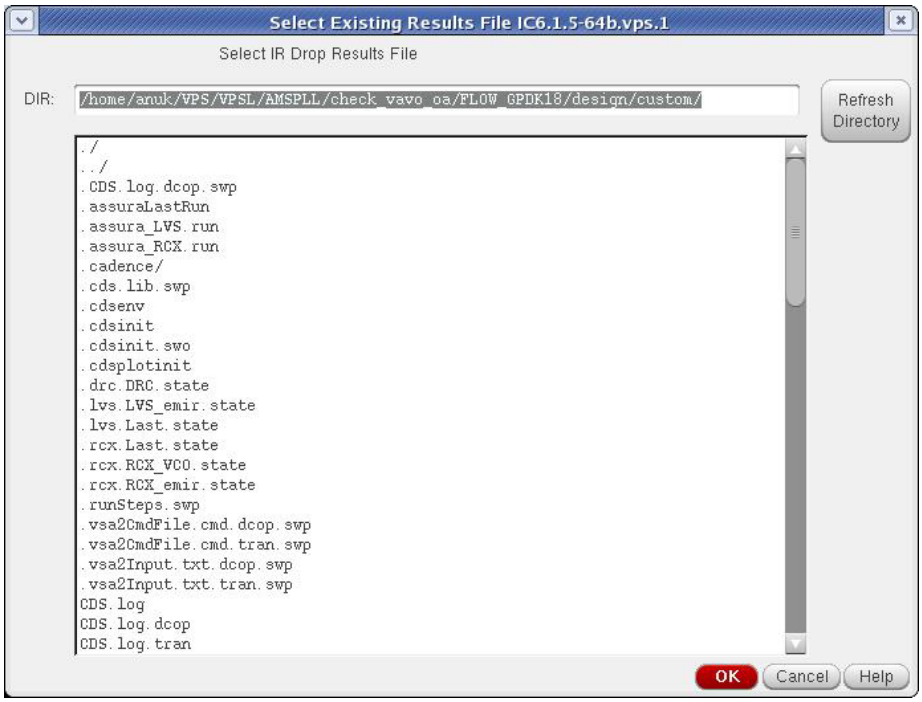
- Select *Analyze Extracted View Pin(s) or Analyze Extracted View Signal(s) – Import IR Drop Results.*

Figure 5-12 Opening Import IR Drop Results Form



- The *Select* Existing Results File form opens.

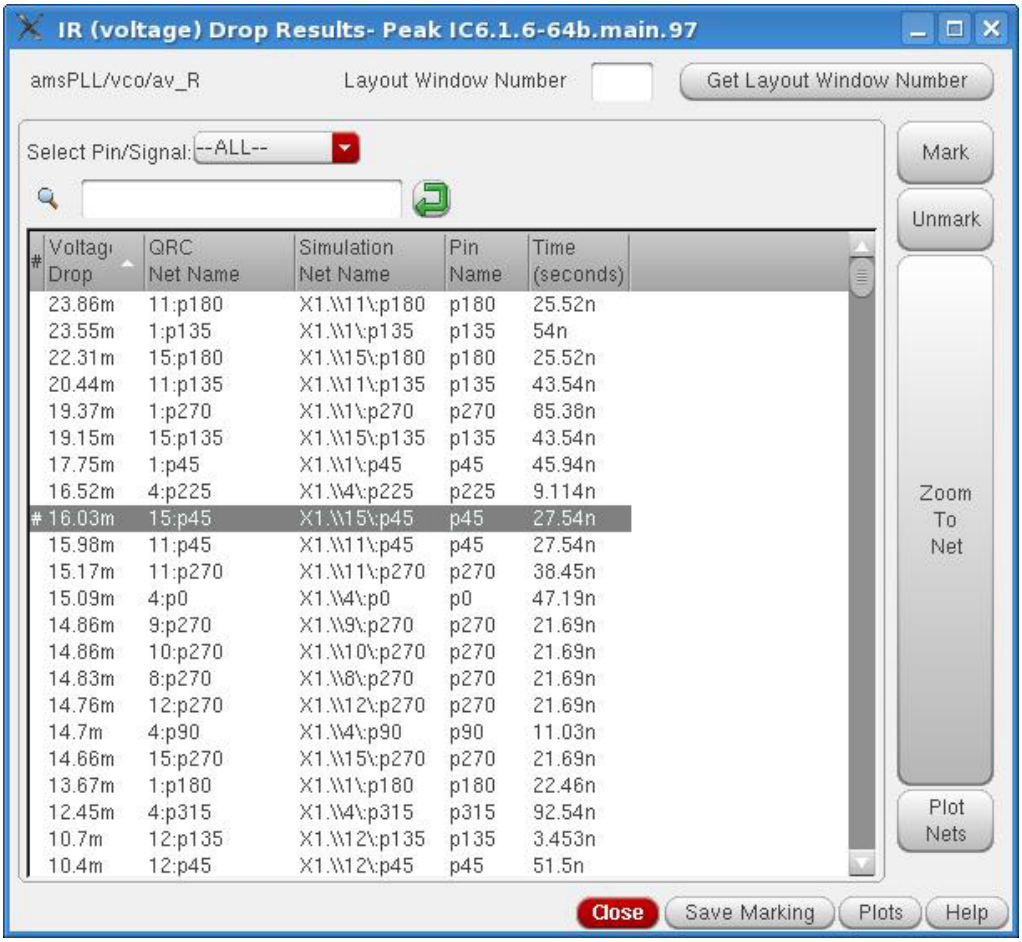
Figure 5-13 Importing Existing IR Drop Results



- Select the file from the list and click *OK.* The IR (voltage) Drop Results form opens.

© 2020 All Rights Reserved.

Figure 5-14 Opening IR (voltage) Drop Results Form when IR Drop Results are Imported



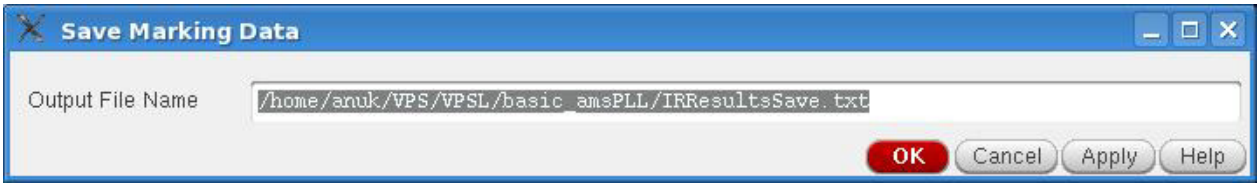
#	Voltage Drop	QRC Net Name	Simulation Net Name	Pin Name	Time (seconds)
	23.86m	11:p180	X1.\\11\:p180	p180	25.52n
	23.55m	1:p135	X1.\\1\:p135	p135	54n
	22.31m	15:p180	X1.\\15\:p180	p180	25.52n
	20.44m	11:p135	X1.\\11\:p135	p135	43.54n
	19.37m	1:p270	X1.\\1\:p270	p270	85.38n
	19.15m	15:p135	X1.\\15\:p135	p135	43.54n
	17.75m	1:p45	X1.\\1\:p45	p45	45.94n
	16.52m	4:p225	X1.\\4\:p225	p225	9.114n
#	16.03m	15:p45	X1.\\15\:p45	p45	27.54n
	15.98m	11:p45	X1.\\11\:p45	p45	27.54n
	15.17m	11:p270	X1.\\11\:p270	p270	38.45n
	15.09m	4:p0	X1.\\4\:p0	p0	47.19n
	14.86m	9:p270	X1.\\9\:p270	p270	21.69n
	14.86m	10:p270	X1.\\10\:p270	p270	21.69n
	14.83m	8:p270	X1.\\8\:p270	p270	21.69n
	14.76m	12:p270	X1.\\12\:p270	p270	21.69n
	14.7m	4:p90	X1.\\4\:p90	p90	11.03n
	14.66m	15:p270	X1.\\15\:p270	p270	21.69n
	13.67m	1:p180	X1.\\1\:p180	p180	22.46n
	12.45m	4:p315	X1.\\4\:p315	p315	92.54n
	10.7m	12:p135	X1.\\12\:p135	p135	3.453n
	10.4m	12:p45	X1.\\12\:p45	p45	51.5n

Note: When existing results for IR drop analysis are imported, the IR (voltage) Drop Results form provides a *Save Marking* button instead of *Save Results to A File* button. This is shown in the above form.

- Click *Save Marking* to save the marked results in a file for future reference. The Save Marking Data form opens. The results can be reloaded in subsequent sessions. This is shown below.

© 2020 All Rights Reserved.

Figure 5-15 Saving the Marking Data



In this form, specify the *Output File Name* and click *OK*.

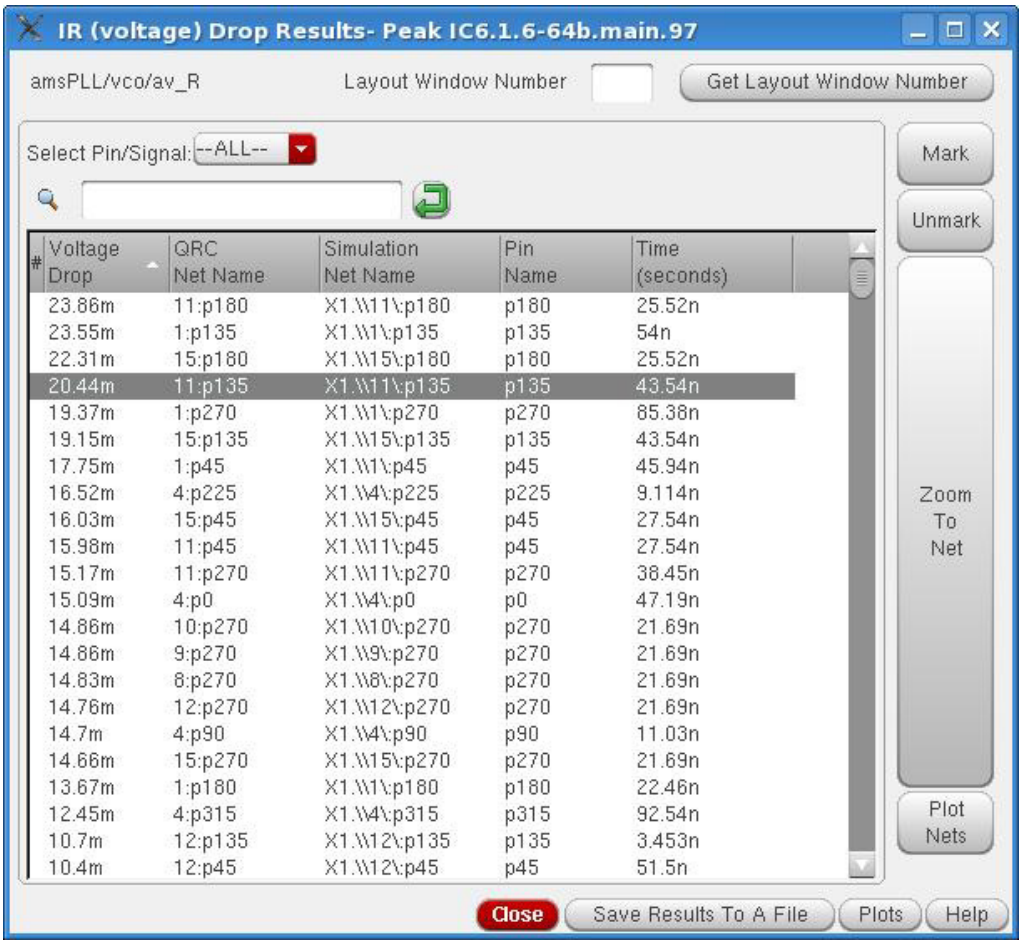
Selecting Options in the IR (voltage) Drop Results Form to View Results

The IR drop analysis results can be viewed using the various options provided in the IR (voltage) Drop Results form.

After performing IR drop analysis, when you click OK on the IR (voltage) Drop Analysis form, the IR (voltage) Drop Results form opens. This form is shown below.

© 2020 All Rights Reserved.

Figure 5-16 Opening IR (voltage) Drop Results Form for the Current IR Drop Analysis Run



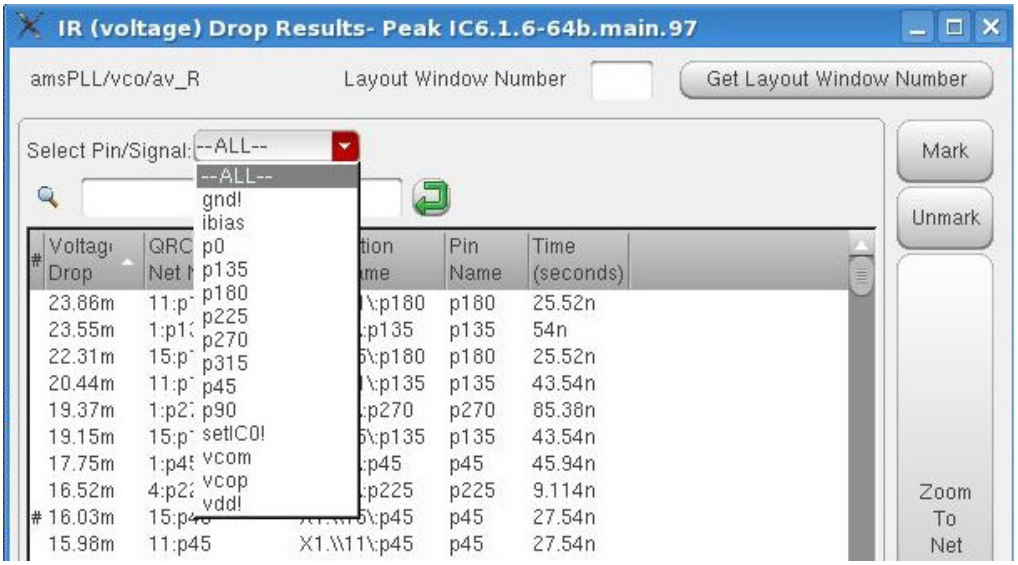
#	Voltage Drop	QRC Net Name	Simulation Net Name	Pin Name	Time (seconds)
	23.86m	11:p180	X1.\\11\:p180	p180	25.52n
	23.55m	1:p135	X1.\\1\:p135	p135	54n
	22.31m	15:p180	X1.\\15\:p180	p180	25.52n
	20.44m	11:p135	X1.\\11\:p135	p135	43.54n
	19.37m	1:p270	X1.\\1\:p270	p270	85.38n
	19.15m	15:p135	X1.\\15\:p135	p135	43.54n
	17.75m	1:p45	X1.\\1\:p45	p45	45.94n
	16.52m	4:p225	X1.\\4\:p225	p225	9.114n
	16.03m	15:p45	X1.\\15\:p45	p45	27.54n
	15.98m	11:p45	X1.\\11\:p45	p45	27.54n
	15.17m	11:p270	X1.\\11\:p270	p270	38.45n
	15.09m	4:p0	X1.\\4\:p0	p0	47.19n
	14.86m	10:p270	X1.\\10\:p270	p270	21.69n
	14.86m	9:p270	X1.\\9\:p270	p270	21.69n
	14.83m	8:p270	X1.\\8\:p270	p270	21.69n
	14.76m	12:p270	X1.\\12\:p270	p270	21.69n
	14.7m	4:p90	X1.\\4\:p90	p90	11.03n
	14.66m	15:p270	X1.\\15\:p270	p270	21.69n
	13.67m	1:p180	X1.\\1\:p180	p180	22.46n
	12.45m	4:p315	X1.\\4\:p315	p315	92.54n
	10.7m	12:p135	X1.\\12\:p135	p135	3.453n
	10.4m	12:p45	X1.\\12\:p45	p45	51.5n

This form displays the IR drop information for the specified extracted view. In the form above, the extracted view selected is `av_R`. In this form, specify the following:

- In *Layout Window Number, s*pecify the window ID for the layout, if a layout is open for the design. Click *Get Layout Window Number* to enable the software to get the layout window number automatically from an already open window. If no layout window is open, a new window will be opened with a layout view.
- In the drop-down list provided against the *Select Pin/Signal* field, select the pin or signal for which you want to view the results.

© 2020 All Rights Reserved.

Figure 5-17 Selecting Pins/Signals from the Drop-down List



You can also search for pin or signal names by typing the name in the search field and pressing the enter button – the button with the green arrow – or by using SKILL regular expressions. The pin or signal names can also be searched on the basis of IR drop by providing the IR drop limit with a comparison operator. For example, to view results for pins or signals with voltage drop greater than 15.00m, type `>15` in the search field.

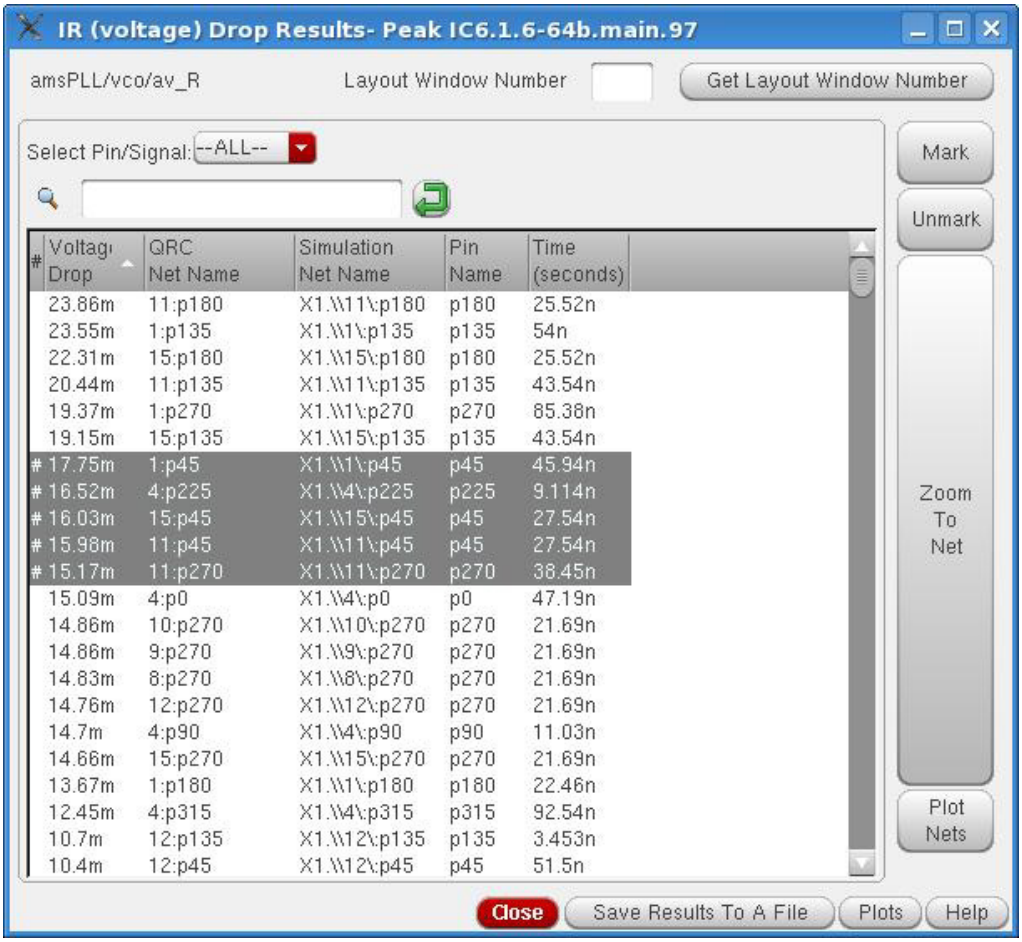
- In the table, filter out results for the selected pins or signals on the basis of the following:
  - Voltage Drop
  - QRC Net Name
  - Simulation Net Name
  - Pin Name
  - Time (seconds) – This column is not available when the type of IR drop analysis performed is average.

  Note: To sort by heading, double-click the headings.

- Click *Mark* and *Unmark* for marking and unmarking nets. Select the nets and click *Mark*. A hash (#) symbol gets added in the beginning of the rows in which the IR drop analysis results of the nets are displayed. The result file displays the IR drop analysis results for all marked nets. Similarly, click *Unmark* to unmark a net. Nets showing false errors can be unmarked so they are not reported in the result file. This helps save time spent viewing and debugging all the false errors reported in the result file.

© 2020 All Rights Reserved.

Figure 5-18 Marking Nets in the IR Drop Results Form

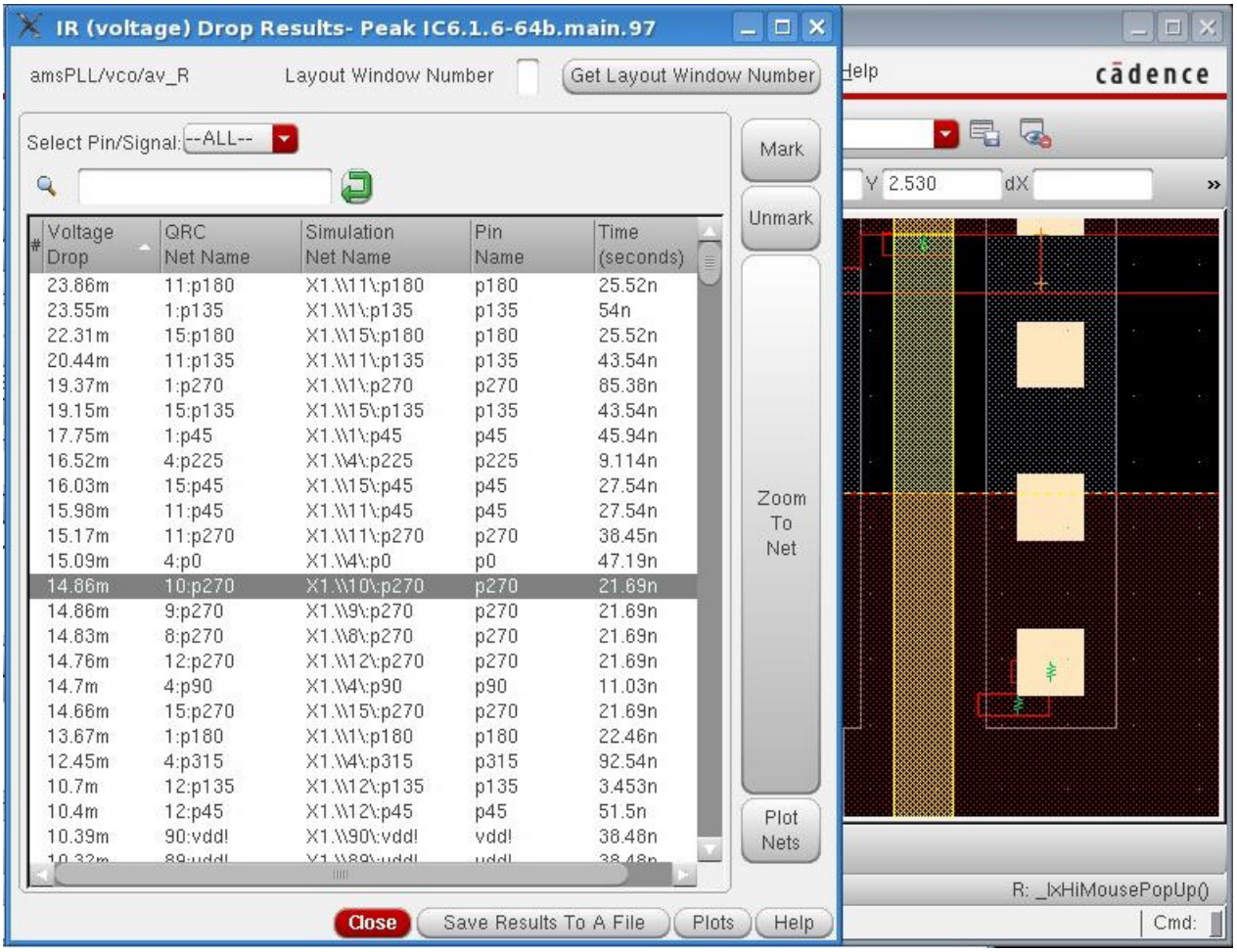


#	Voltage Drop	QRC Net Name	Simulation Net Name	Pin Name	Time (seconds)
	23.86m	11:p180	X1.\\11\:p180	p180	25.52n
	23.55m	1:p135	X1.\\1\:p135	p135	54n
	22.31m	15:p180	X1.\\15\:p180	p180	25.52n
	20.44m	11:p135	X1.\\11\:p135	p135	43.54n
	19.37m	1:p270	X1.\\1\:p270	p270	85.38n
	19.15m	15:p135	X1.\\15\:p135	p135	43.54n
#	17.75m	1:p45	X1.\\1\:p45	p45	45.94n
#	16.52m	4:p225	X1.\\4\:p225	p225	9.114n
#	16.03m	15:p45	X1.\\15\:p45	p45	27.54n
#	15.98m	11:p45	X1.\\11\:p45	p45	27.54n
#	15.17m	11:p270	X1.\\11\:p270	p270	38.45n
	15.09m	4:p0	X1.\\4\:p0	p0	47.19n
	14.86m	10:p270	X1.\\10\:p270	p270	21.69n
	14.86m	9:p270	X1.\\9\:p270	p270	21.69n
	14.83m	8:p270	X1.\\8\:p270	p270	21.69n
	14.76m	12:p270	X1.\\12\:p270	p270	21.69n
	14.7m	4:p90	X1.\\4\:p90	p90	11.03n
	14.66m	15:p270	X1.\\15\:p270	p270	21.69n
	13.67m	1:p180	X1.\\1\:p180	p180	22.46n
	12.45m	4:p315	X1.\\4\:p315	p315	92.54n
	10.7m	12:p135	X1.\\12\:p135	p135	3.453n
	10.4m	12:p45	X1.\\12\:p45	p45	51.5n

- Click *Zoom To Net* to enlarge the view of the currently selected net in the list box on the layout. This is shown below.

© 2020 All Rights Reserved.

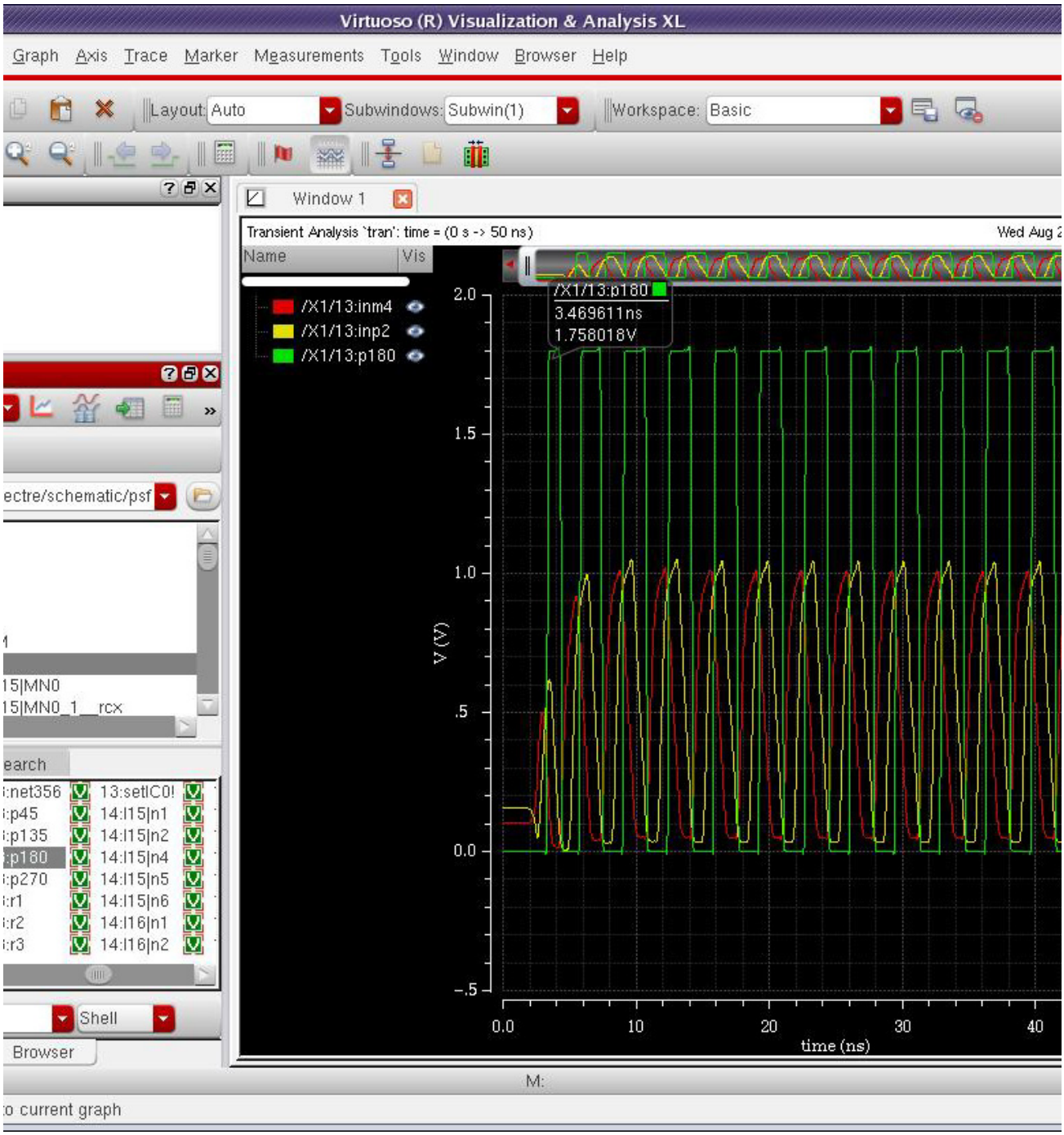
Figure 5-19 Zooming to Selected Net in the Layout



- Click *Plot Nets* to display the current waveform on the selected net in the Virtuoso (R) Visualization & Analysis XL window. This is shown below.

© 2020 All Rights Reserved.

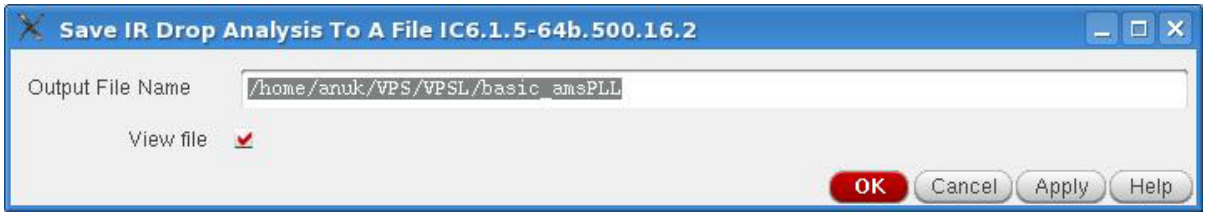
Figure 5-20 Viewing Current Waveforms on the Selected Nets in Virtuoso (R) Visualization & Analysis XL



- Click *Save Results To A File* to save the IR drop analysis results as a text file. The Save IR Drop Analysis To A File form opens. This form is shown below.

© 2020 All Rights Reserved.

Figure 5-21 Saving IR Drop Analysis Results to a File



- In this form, specify the *Output File Name*.
- Click *View file* to view the result file. This option is selected by default. If you do not want to view the result file, you can change the default selection of this option by setting the `_vsaUncheckViewFile` environment variable to `true`.
- Click *OK.*

Customizing Error Layer Display for IR Drop Results

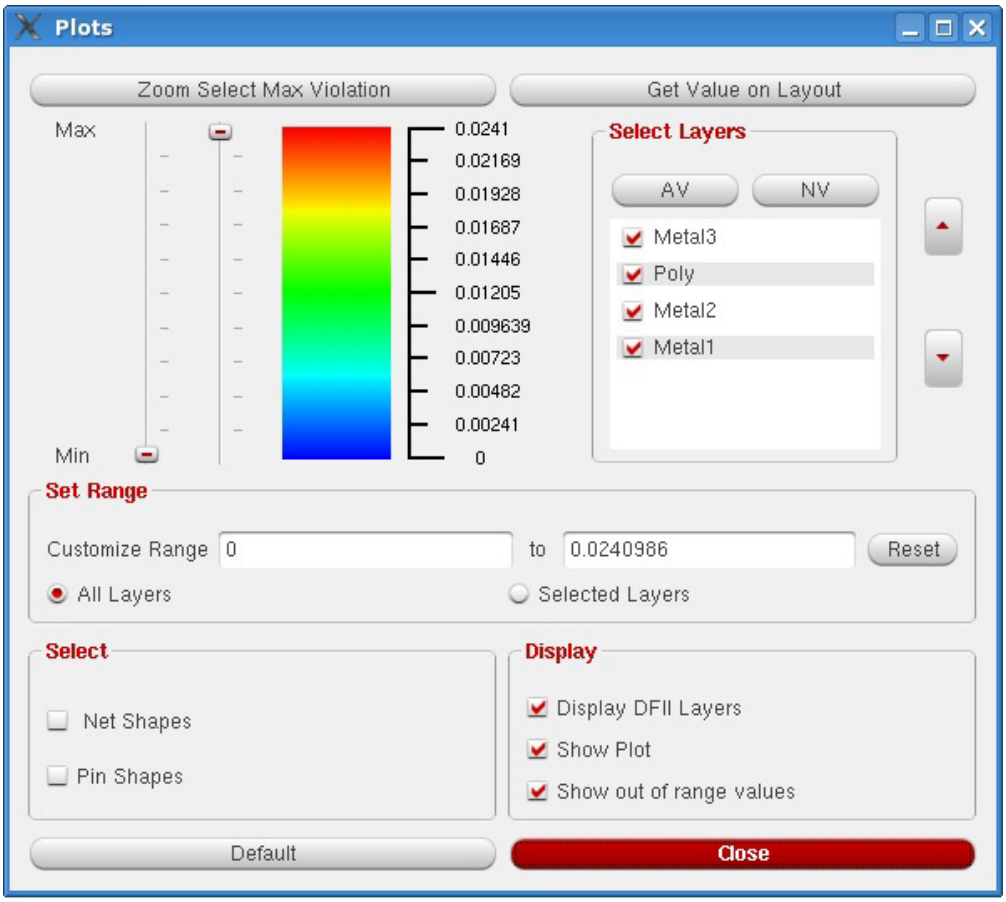
Power IR/EM lets you customize IR drop result displays on the Virtuoso layout. For this, you can customize settings in the Plots form.

In this form, you can customize the violation voltage ranges using a continuos RGB gradient. The vast range of color options provided by the RGB gradient makes it easier to view a specific range of violations.

Click *Plots* in the IR (voltage) Drop Results form. The form shown below opens.

© 2020 All Rights Reserved.

Figure 5-22 Displaying Highlight Color Spectrum and Options for Customizing Display of IR Drop Plots

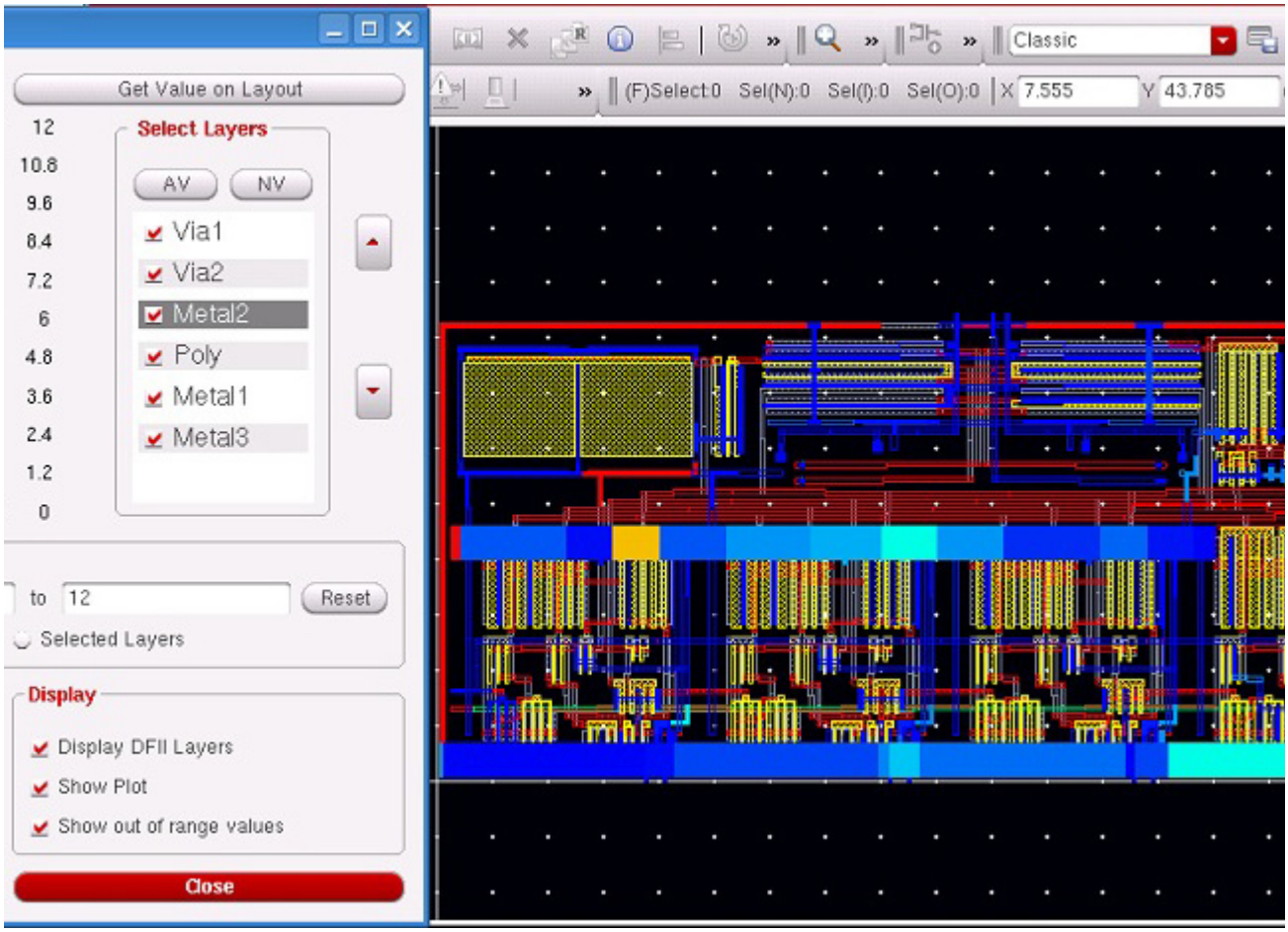


In the Select Layers group box:

- Select the layers for which you want to view the violations. Click *AV* or *NV* to select or deselect all layers respectively. Click the layer names to select specific layers. When the Plots form opens, all layers are selected by default.
- Two buttons, with up and down arrows, are provided next to the layers listed in the list box. You can change the order of layers by selecting any layer and then clicking the up or down arrow button to move the layer up or down in the list. The updated layer order is reflected in the plot highlights on the extracted view. The images below show the change in plot highlights on the extracted view when the order of Metal2 layer is changed.

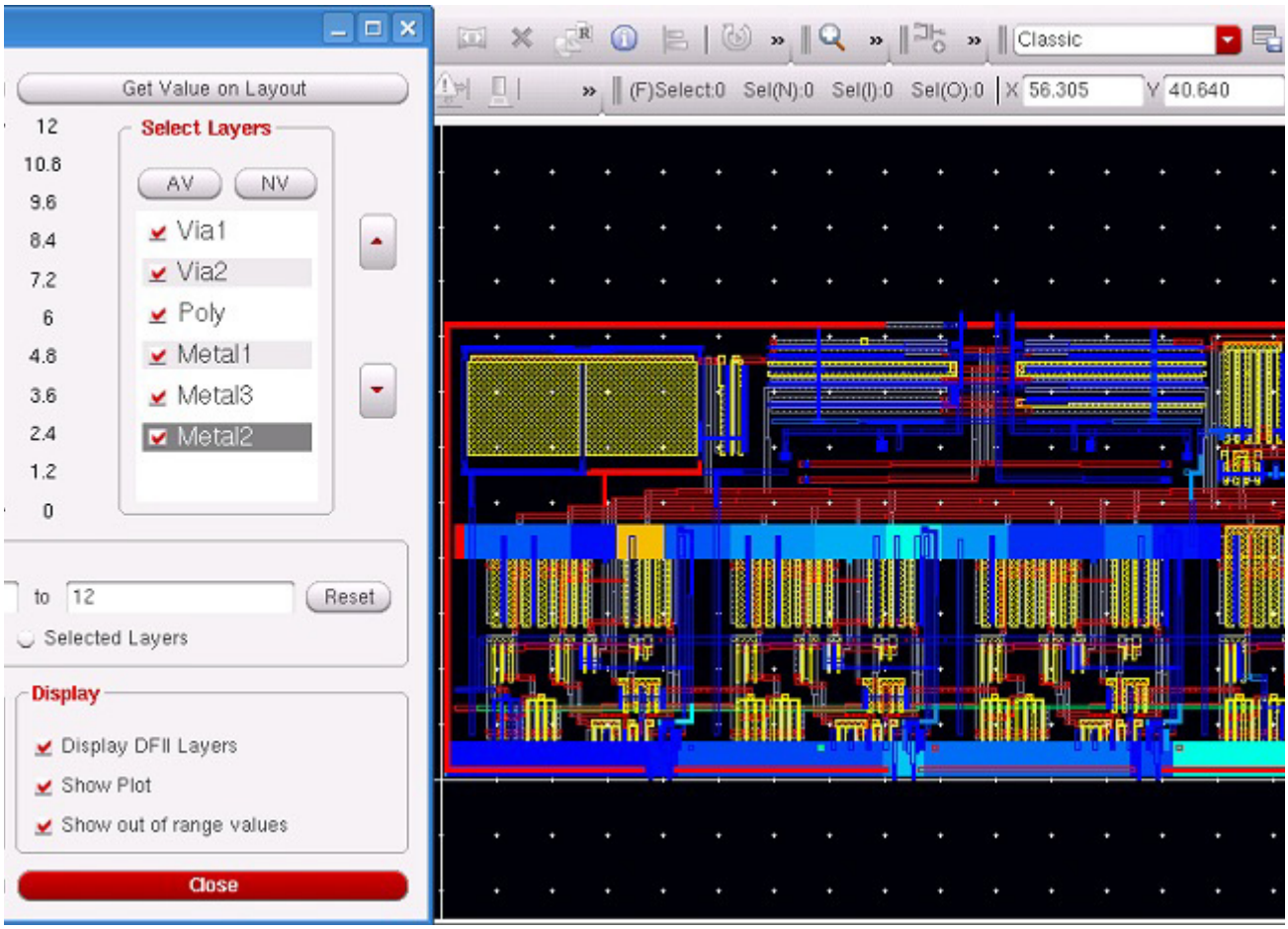
© 2020 All Rights Reserved.

Figure 5-23 Changing the Order of Layers



© 2020
All Rights Reserved.

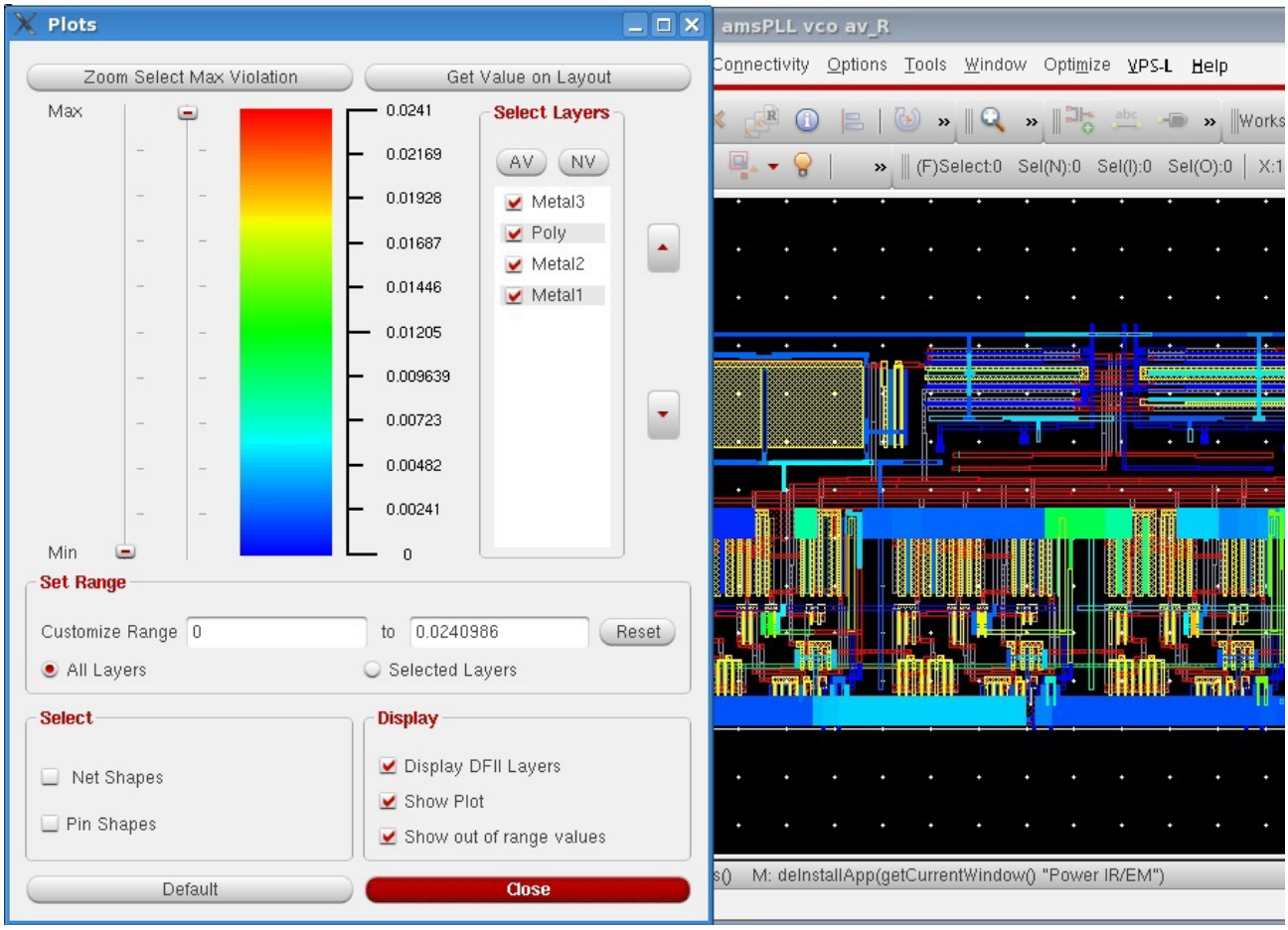
Figure 5-24 Changing the Order of Layers



- The RGB color spectrum is provided for viewing violations in different colors. The ruler range of IR drop values for all or selected layers is provided next to the color spectrum. The image below shows the layout display of total violation ranges for all layers.

© 2020 All Rights Reserved.

Figure 5-25 Displaying Total Violation Range for All Layers on the Layout



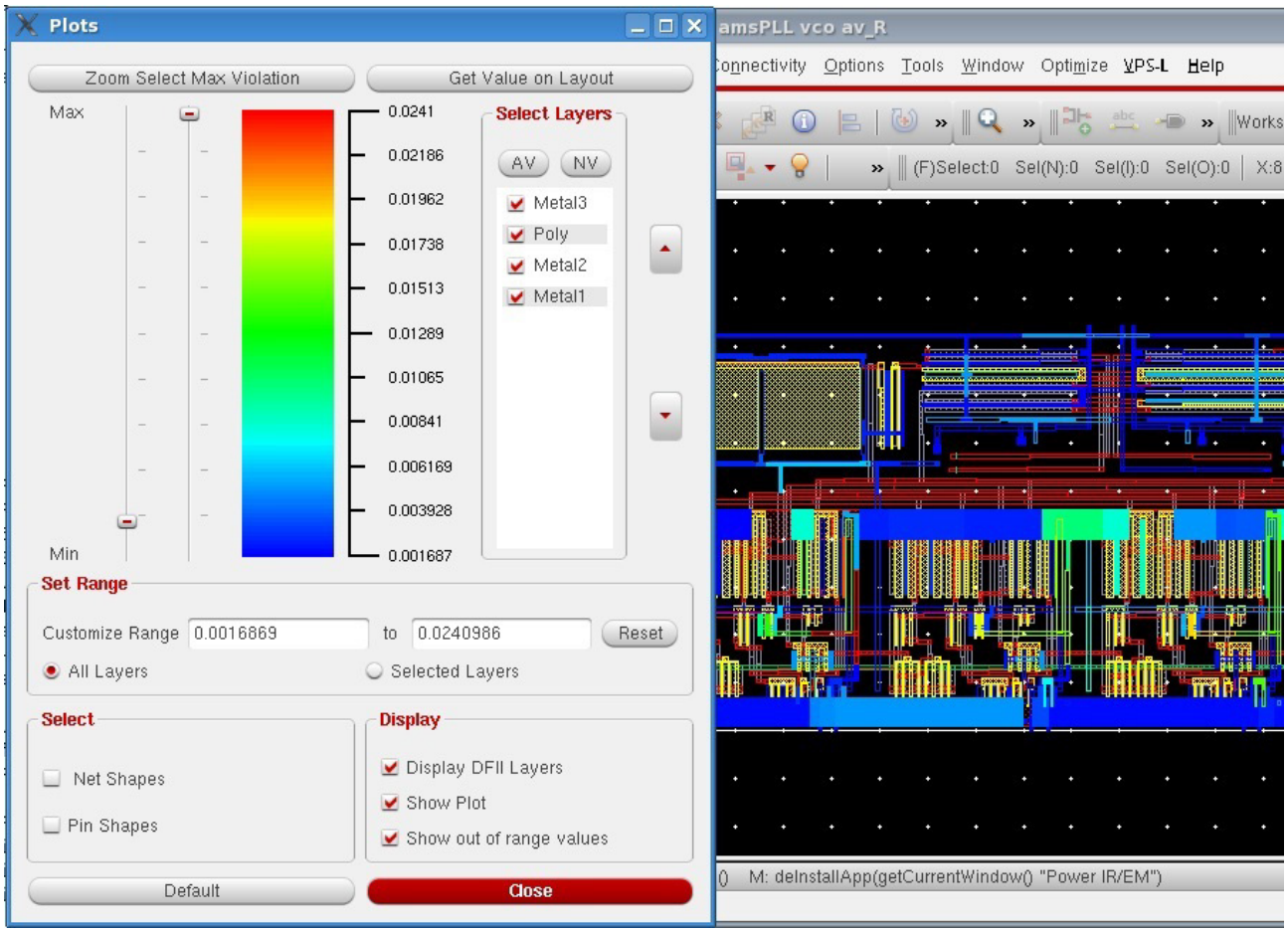
- *Min* and *Max* sliders are provided for customizing the ruler range for viewing the violations that fall within the specified *Min Max* slider range. You can move the Min Max sliders up and down to update the ruler ranges according to the new slider positions.

  When *Min Max* sliders are used to specify the ruler range for violations, the selection is updated in the *Customize Range* fields.

  The violations above and below the slider marks are displayed in “deep red” and “deep blue” respectively. This is shown in the image below.

© 2020 All Rights Reserved.

Figure 5-26 Customizing the Ruler Range for Viewing IR Drop Violations



In the Set Range group box:

- 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* group box.
- Click *Selected Layers* to select the violation range for layers selected in the *Select Layers* group box. The range is updated both on the slider and in the *Customize Range* fields for the violations of the selected layers. The layout is highlighted accordingly.

  Note: The software saves the last state for *All Layers.* When you switch back to *All Layers* after using *Selected Layers*, the *Min Max* slider values, the ruler range, and the values in the *Customize Range* fields are restored to those in the previously-saved state. The display on the layout is updated accordingly.

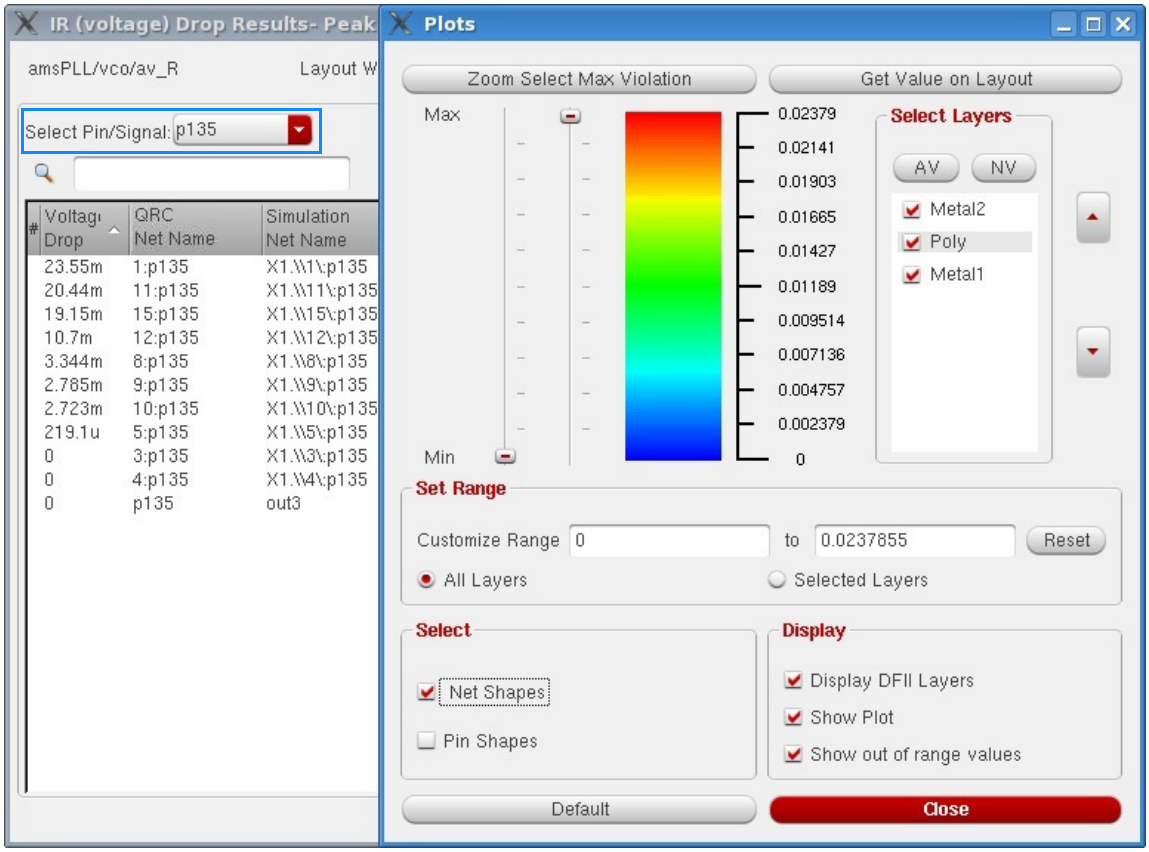
© 2020 All Rights Reserved.

- Click *Customize Range* to enter the minimum and maximum values of the required range in the text fields and press enter. The specified range becomes the new ruler range.
- Click *Reset* to restore the ruler range to the default values. For *All Layers*, when you click *Reset*, the ruler range moves back to the total violation range. For *Selected Layers,* when you click *Reset,* the ruler range moves back to the total violation range for selected layers only.

In the *Select* group box:

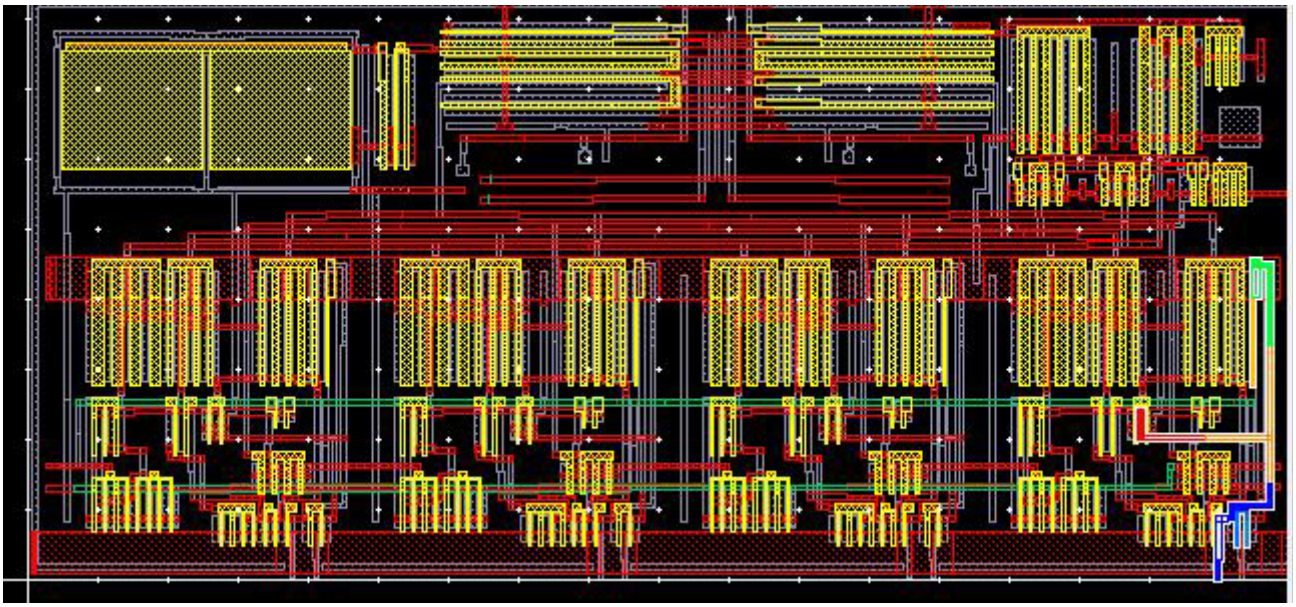
- Click *Net Shapes* to highlight the specified nets on the layout. For example, to highlight a net, select the net in the IR (voltage) Drop Results form and then click *Net Shapes* in the Plots form. The selected net is highlighted in the layout as seen in below images.

Figure 5-27 Selecting a Net in the IR (voltage) Drop Results Form



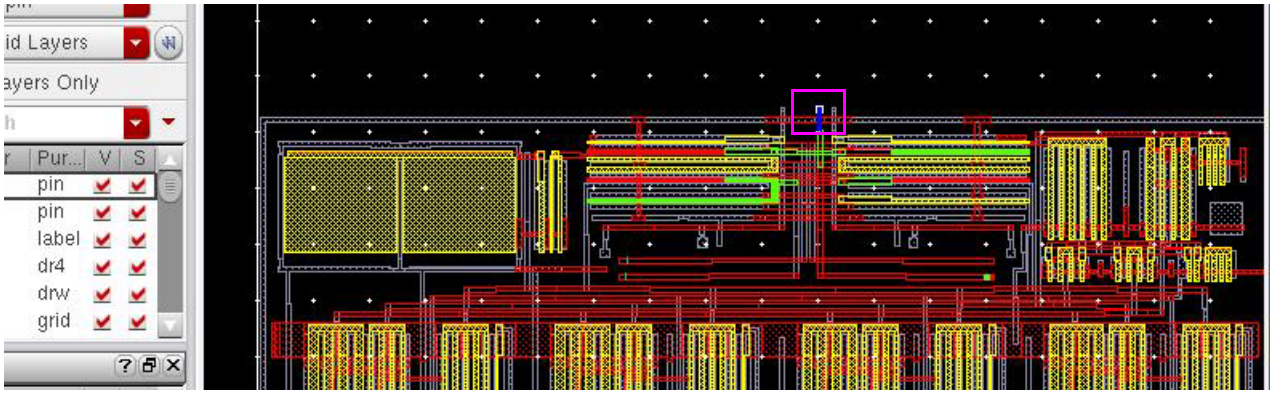
© 2020 All Rights Reserved.

Figure 5-28 Highlighting Net Shapes on the Layout



- Click *Pin Shapes* to highlight the specified pins on the layout. First, select the net in the IR (voltage) Drop Results form and then click *Pin Shapes* in the Plots form. The pin shape is highlighted in the layout as seen in below image.

Figure 5-29 Highlighting Selected Pin Shapes on the Layout

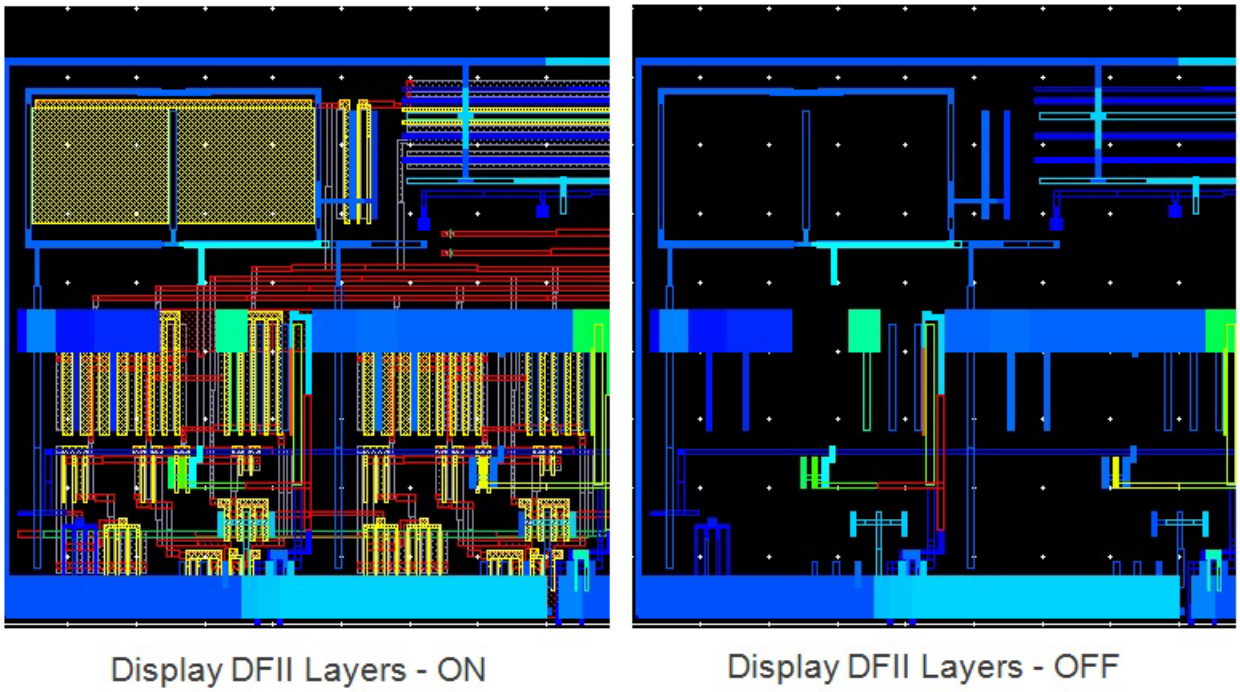


Note: In the above image, a pink textbox is used to show the highlighted pin clearly for your reference.

- Select or deselect *Display DFII Layers* to toggle the visibility of the DFII layers on the layout. This is shown in the image below.

© 2020 All Rights Reserved.

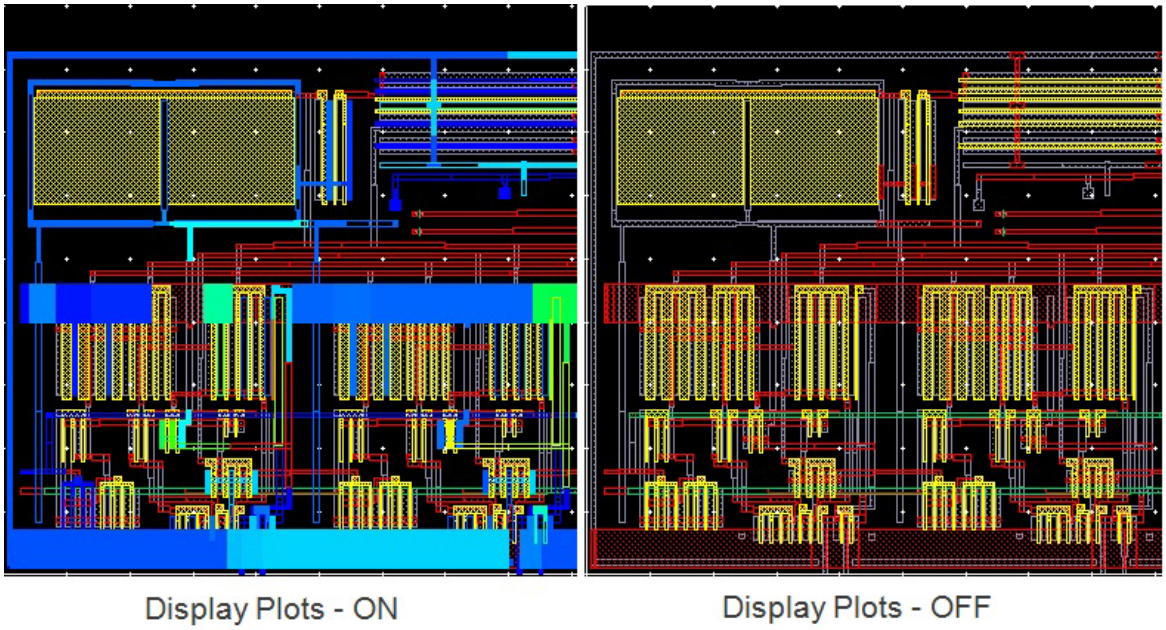
Figure 5-30 Toggling Display of DFII Layers on the Layout



- Select and deselect *Show Plots* to toggle the highlights on the layout. This is shown in the image below.

© 2020 All Rights Reserved.

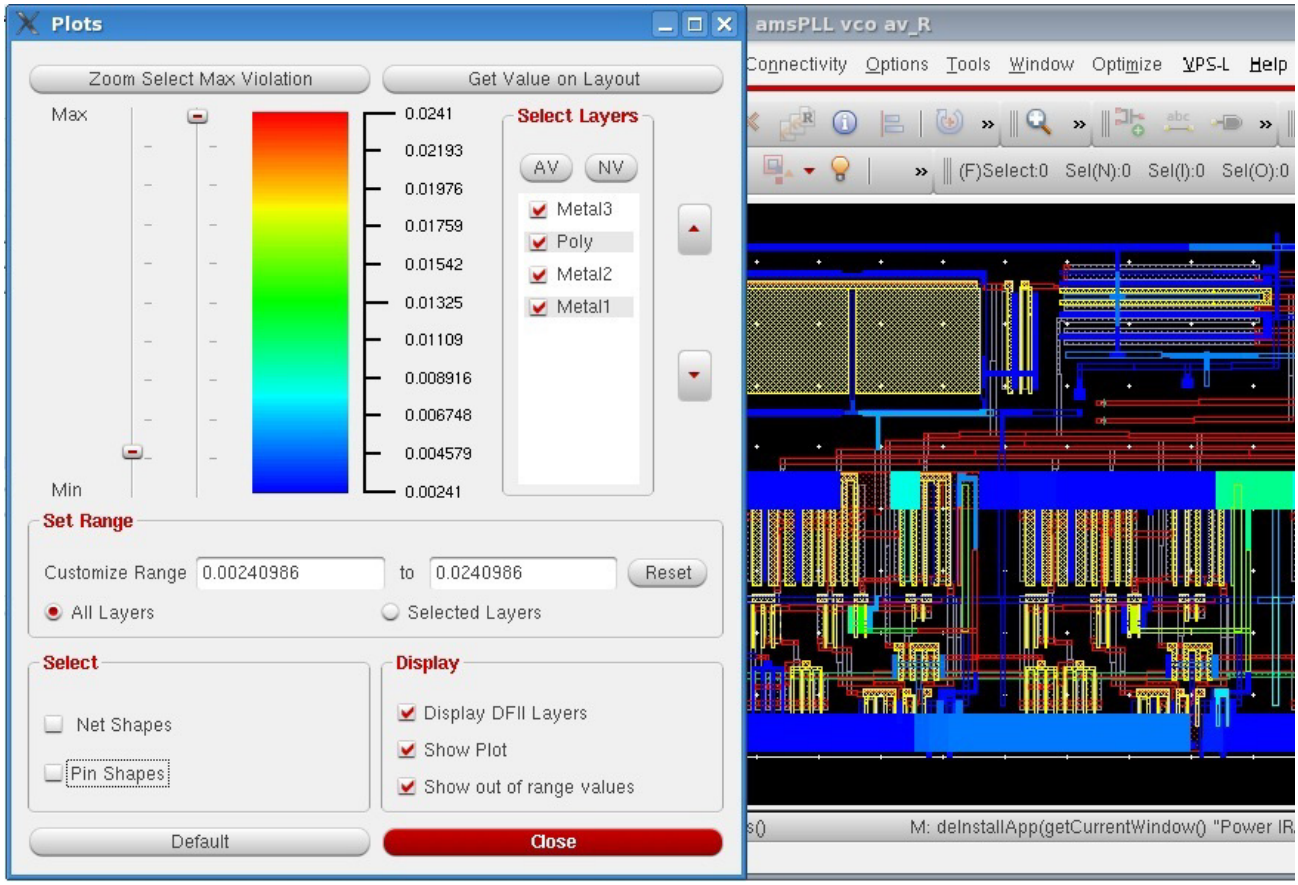
Figure 5-31 Toggling Highlights on the Layout



- Click *Show out of range values* to show violations with values outside the specified ruler range. For example, in the image below, the *Min* slider is set at 0.004579. When you click *Show out of range values*, the layout shows violations below the specified minimum range in deep blue.

© 2020 All Rights Reserved.

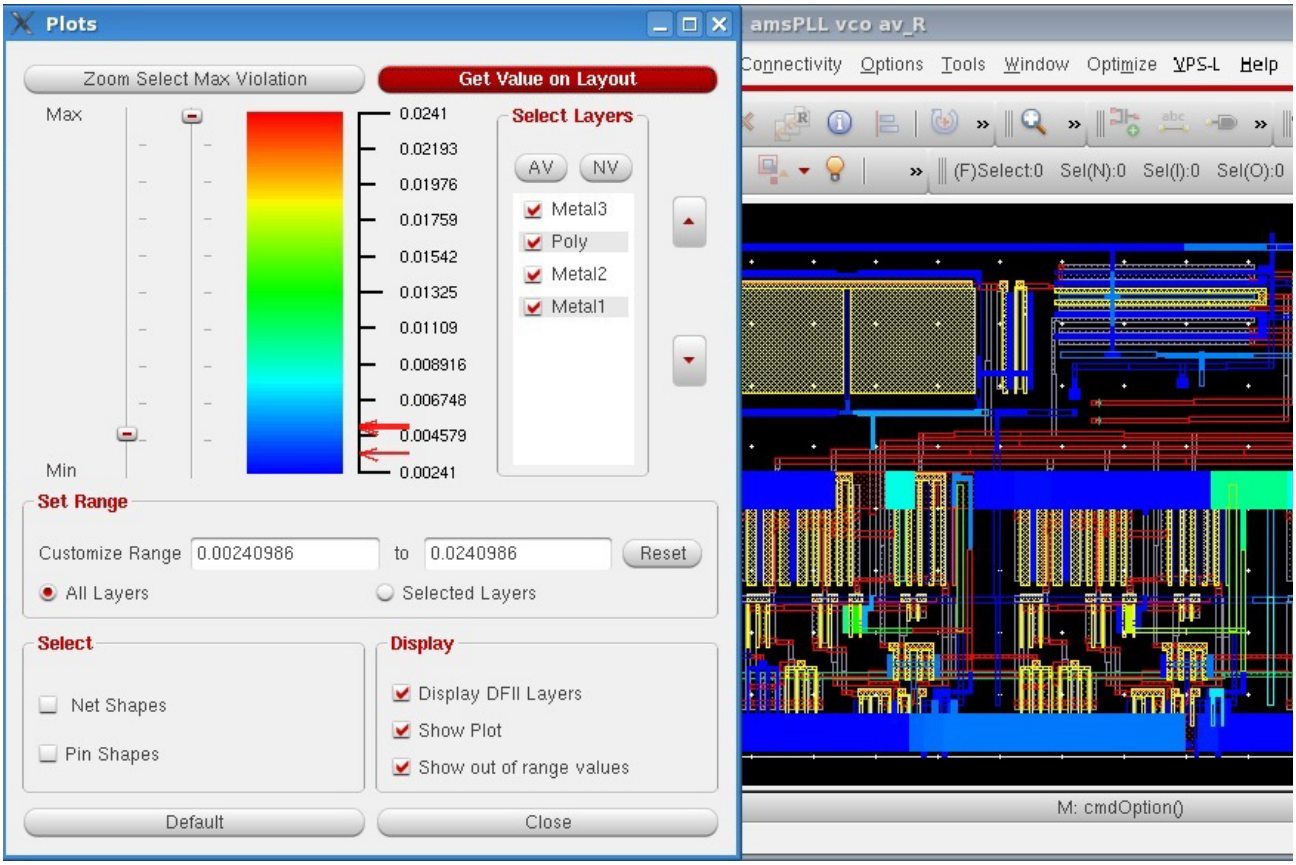
Figure 5-32 Showing Out of Range Values



- Click *Get Value on Layout* to retrieve the values from the extracted view. For this, first click *Get Value on Layout* in the *Plots* form and then click on the layout to select a point. The violations at that point are marked on the ruler with red arrows and are highlighted in the IR (voltage) Drop Results form. The range of violations is listed in the *Customize Range* fields. This is shown in the image below.

© 2020 All Rights Reserved.

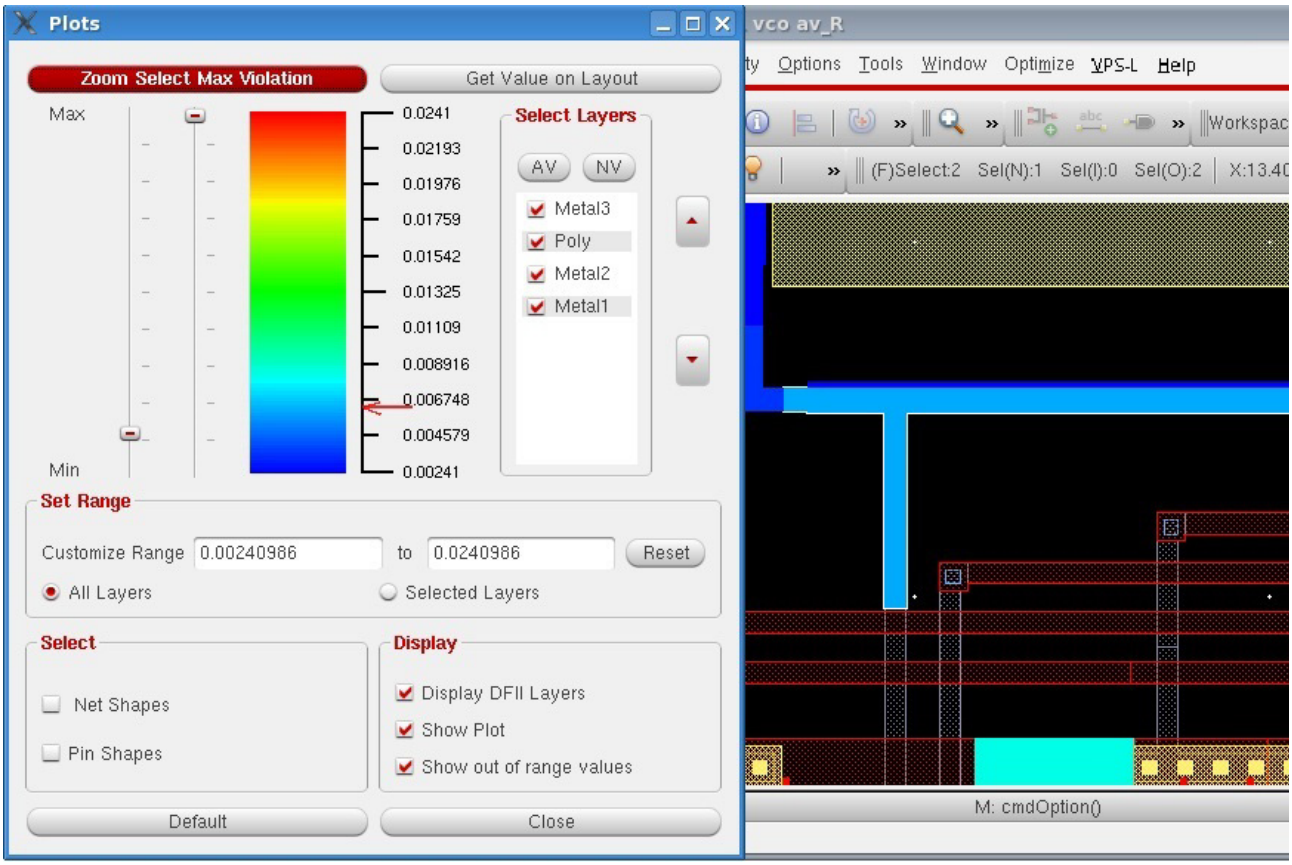
Figure 5-33 Retrieving Values from the Extracted View



- Click *Zoom Select Max Violation* to highlight the maximum violation in the selected area. For this, select an area on the layout to locate the maximum violation in that area. The maximum violation in the selected area is marked on the ruler with a red arrow and highlighted in the IR (voltage) Drop Results form, and the layout zooms to the maximum violation. This is shown in the image below.

© 2020 All Rights Reserved.

Figure 5-34 Zooming into the Maximum Violation in the Selected Area



- Click *Default* to restore the default settings.
- Click *Close* to exit.

© 2020 All Rights Reserved.

© 2020 All Rights Reserved.

6

EM Analysis

- Overview on page 106
- EM Analysis of the Power Grid on page 106
  - Layermap File for EM Analysis on page 123
  - Analyzing EM Across Design Resistors on page 124
  - Writing Out an EM Command File on page 125
  - Opening Power IR/EM Directly from Extracted View on page 126
- EM Analysis of Signal Nets on page 126
- EM Analysis Results on page 128
  - Importing Results of a Previous EM Analysis on page 128
  - Viewing Electromigration Analysis Results for the Current Run on page 131
  - Customizing Error Layer Display for EM Analysis Results on page 140
  - Customizing the EM Analysis Results Display on page 142
  - Using EM Analysis Results for Cross-Probing in Power IR/EM on page 145

© 2020 All Rights Reserved.

Overview

Electromigration (EM) is used as a general term to describe failure mechanisms in the metal wires of a computer chip caused by the movement of metal atoms in a wire because of high current stress. As electrons move through a metal wire, they collide with the atoms in that wire. These collisions cause wires to become heated, and if enough electrons collide with a metal atom over a period of time, the metal atom can move in the direction of the electron flow, causing the following failure mechanisms:

- First, if enough atoms move, the wire breaks and becomes an open circuit.
- Second, if enough atoms move to the same location, a short to an adjacent metal wire is created. This phenomenon is commonly known as fusing.

Either of these mechanisms changes the functionality of the chip.

EM analysis is performed to analyze the power grid and signals of the design for any current density violations of the *DC current* through the metal conductor lines, vias, and contacts. These violations can then be corrected by limiting the current density in metal conductor lines, vias, and contacts.

EM Analysis of the Power Grid

Before running Electromigration 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 “Reading EM Rules from qrcTechFile” and “Reading EM Rules from emDataFile” sections in the “EM Rules Specification” chapter.

By default, in Power IR/EM, the extraction, simulation, and EM analysis flows are run at the same temperature, which is called the nominal temperature. However, you can choose to perform simulation at non-nominal temperatures using `TC1` and `TC2`. You can also perform IR/EM analysis without creating multiple `emDataFiles` for different temperature ranges.

This is possible because even while performing extraction at nominal temperature, you can store the `TC1` and `TC2` (non-nominal temperature) values. The simulation and analysis flows then modify the resistance values internally as per the temperature values specified by you. So, when you perform EM analysis, the software reads in the data and calculates the new resistance using the temperature values specified by you and uses this resistance value for EM analysis.

© 2020 All Rights Reserved.

This means that you only need to maintain one `emDataFile` corresponding to nominal temperature and you can use the same file for all temperature ranges.

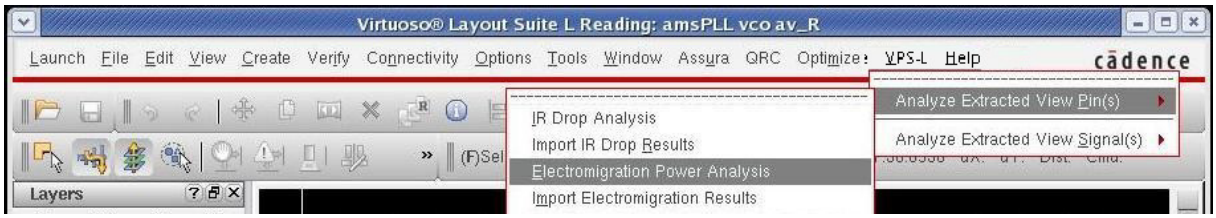
To store `TC1` and `TC2` parameters in the extracted view, select *Parasitic Resistance Temperature Coefficient* in the *Netlisting* tab of the Quantus (Pegasus) Parasitic Extraction Run form while generating the extracted view.

For details, see the *QRC Extraction Users Manual.*

To perform EM analysis of the power grid from the Virtuoso layout window, follow these steps:

1. In the Main menu, choose *VPS-L.* Two options*, Analyze Extracted View Pin(s)* and *Analyze Extracted View Signal(s),* appear in the drop-down list.
2. Choose *Analyze Extracted View Pin(s)* for analyzing the power pins of the design and *Analyze Extracted View Signal(s)* to analyze the signals of the design.
3. The procedure below is detailed for EM power-grid analysis. Choose *Analyze Extracted View Pin(s) - Electromigration Power Analysis* from the drop-down list. This is shown below.

Figure 6-1 Opening Electromigration Power Grid Analysis Form

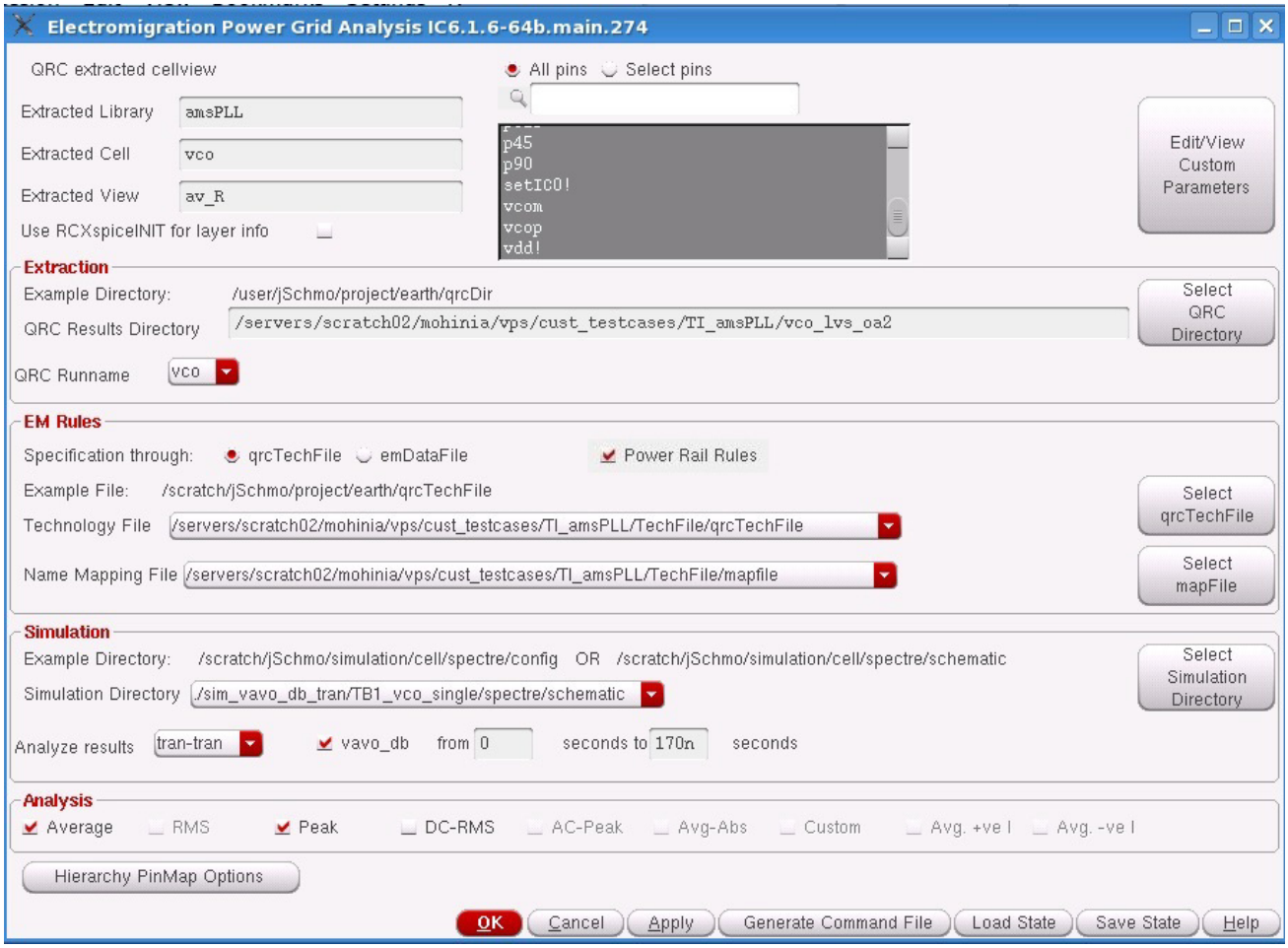


4. The Electromigration Power Grid Analysis form opens. This form is shown below.

   Note: For information about performing EM analysis on signal nets, see EM Analysis of Signal Nets.

© 2020 All Rights Reserved.

Figure 6-2 Running Electromigration Power Grid Analysis



Note: When the above form opens, the fields are prefilled with the settings that were specified in the previous session. These settings are saved in the `vsaEMLast.cfg` file that is created for every session by default. However, if you do not want the software to load the settings of the previous session by default, then set the `-vsaDisableLastConfig` environment variable to `true`.

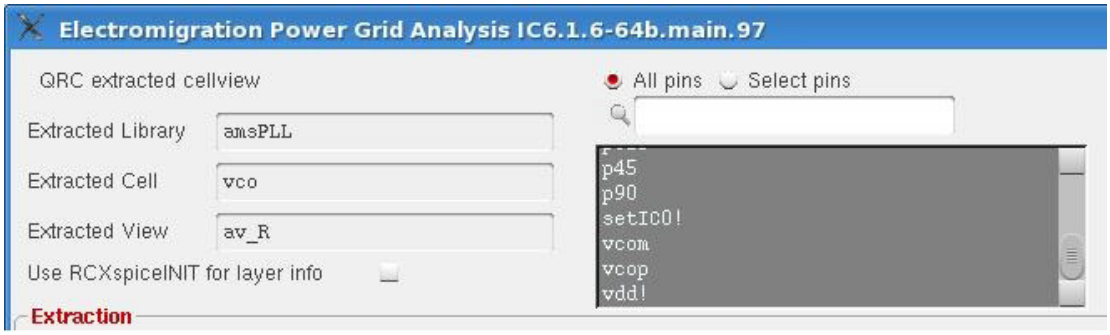
- In the *QRC extracted cellview*, you can view the *Extracted Library*, *Extracted Cell*, and *Extracted View* name created during the extraction run. See Running Pegasus Quantus QRC in the "IR/EM Analysis Flow" chapter. The default view name for an extracted view is `av_extracted`.
- Alternatively, you can select *Use RCXspiceINIT for layer info* so that Power IR/EM can read the RCXspiceINIT file instead of the extracted view.
- Click *All pins* to perform EM analysis on all pins. Click *Select pins* to select the pin names on which the EM analysis should be performed. You can also search for pin

© 2020 All Rights Reserved.

names by typing the name in the search field. The search field also accepts SKILL regular expressions.

Note: When you open the Electromigration Power Grid analysis form, all pins listed in the list box are selected by default. This is shown in the image below.

Figure 6-3 Selecting Pins from the Extracted Cellview

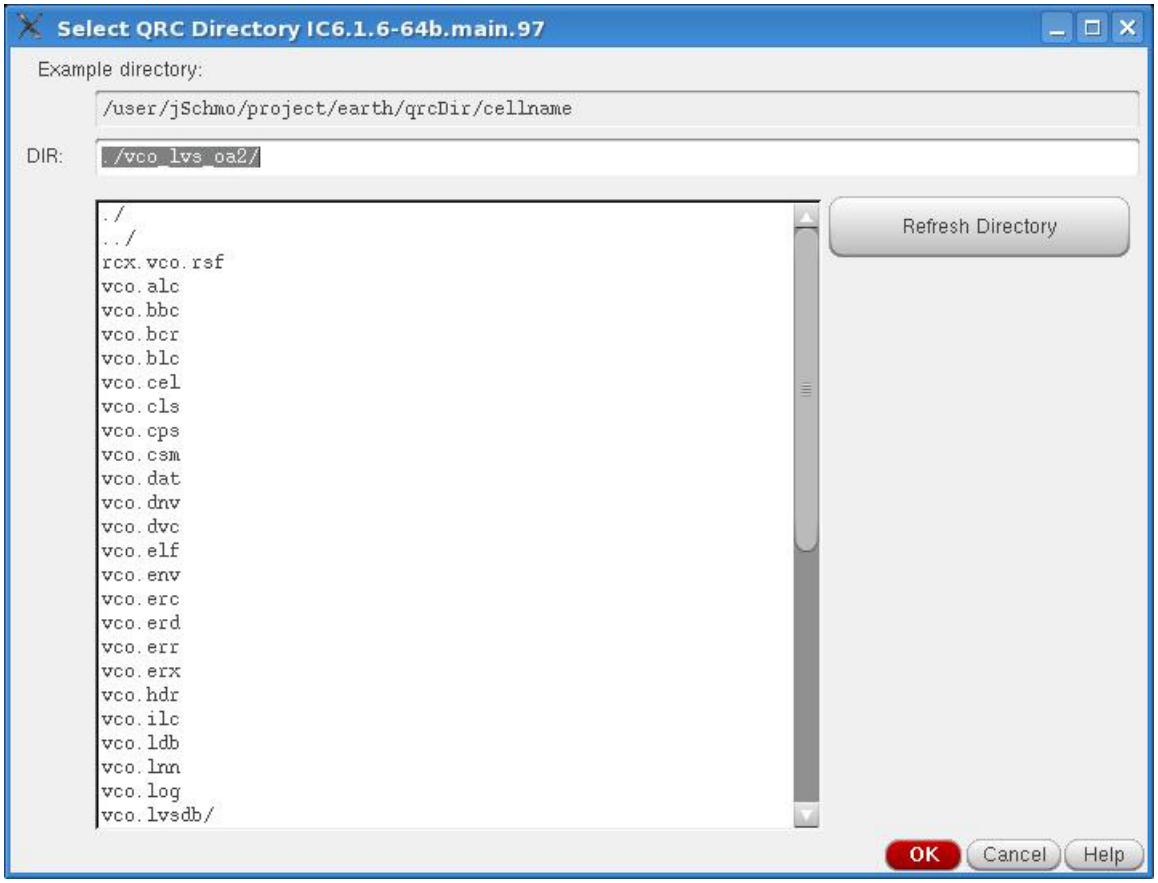


- In the *Extraction* group box:
  - Specify the directory name in the *QRC Results Directory* where the RCXspiceINIT file exists. Select the signal names on which the EM analysis is to be performed.

    Note: The *Example Directory* field shows you how to specify this value.
  - Click *Select QRC Directory* to open the Select QRC Directory form. This form is shown below.

© 2020 All Rights Reserved.

Figure 6-4 Select QRC Directory

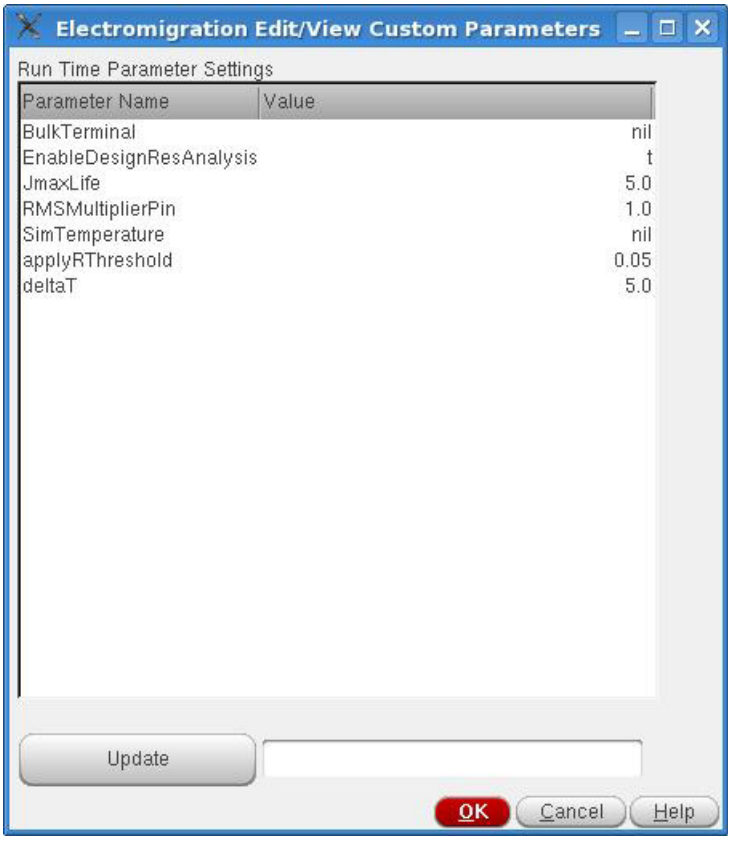


For more information, see Running Pegasus or PVS LVS in the “IR/EM Analysis Flow” chapter.

- Specify the name of the QRC run in the *QRC Runname* cyclic field. See Running Pegasus/PVS Quantus QRC in the “IR/EM Analysis Flow” chapter.
- Click *Edit/View Custom Parameters* to specify custom parameters during the run time and set their values in the `emDataFile`. The Electromigration Edit/View Custom Parameters form opens. This form is shown below.

© 2020 All Rights Reserved.

Figure 6-5 Opening the Electromigration Edit/View Custom Parameters Form



- This form lists parameter names and their values. You can change the value of a parameter in the text field provided at the bottom of the form and click *Update*.
  - The `BulkTerminal` parameter is only applicable to the design resistor flow. This parameter is used to specify the bulk terminals of the design resistors that are to be ignored while performing EM analysis. You can provide the names of multiple terminals separated by a comma. You can also specify the terminals by using the `_vsaBulkTerminal` environmental variable. When this environment variable is specified, the value of the `BulkTerminal` parameter in the GUI is updated.
  - The `EnableDesignResAnalysis` parameter is used to enable or disable the analysis of all design resistors. It is set to `true` by default. Set to `nil` to disable analysis of all design resistors.
  - `JmaxLife` is used to specify the scaling factor that applies to the current density limits for different lifetimes. You can also specify the lifetime value using the environment variable, `_vsaJmaxLife`.

© 2020 All Rights Reserved.

- The `RMSMultiplierPin` and `RMSMultiplierSignal` parameters are used to specify the RMS relaxation factor for EM power-grid and signal analysis respectively. You can also set the `_vsaRMSMultiplierPin` and `_vsaRMSMultiplierSignal` environment variables to `true`.
- `SimTemperature` parameter is used to specify the simulation temperature to be used for EM analysis. When this temperature value is specified, it overrides the temperature value specified in the simulation directory.
- The `applyRThreshold` parameter is used to specify the threshold ratio "r" for AC-Peak analysis. You can also specify the ratio using the `_vsaApplyRThreshold` environment variable.
- `deltaT` is used to specify the maximum rise in temperature in degree Celsius, caused due to Joule Heating. For details, see `deltaT` it in the EM Rules Specification chapter.

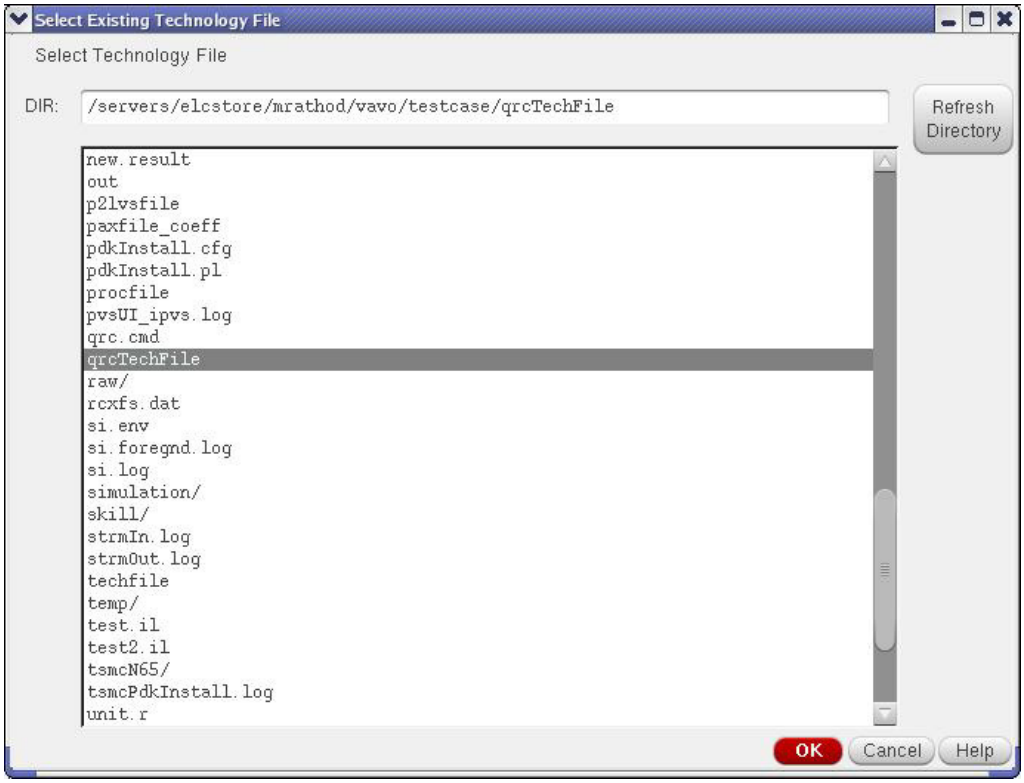
- Click *OK*. EM analysis will be performed using the new values.

For details, see Rules for Defining Limit-Based Parameters in the "EM Rules Specification" chapter.

- In the EM Rules group box:
  - Click *Power Rail Rules* to enable power-grid analysis.
  - Click *qrcTechfile* to specify `qrcTechFile` flow. When you click *qrcTechfile, Technology file* and *Name Mapping File* cyclic fields appear in the form. Select the relevant technology file and name mapping file to be used for EM analysis. For details of the layermap file used for EM analysis, see Layermap File for EM Analysis.
  - You can also click *Select qrcTechFile* to specify the relevant `qrcTechFile`. The Select Existing Technology File form opens.

© 2020 All Rights Reserved.

Figure 6-6 Selecting the Existing Technology File

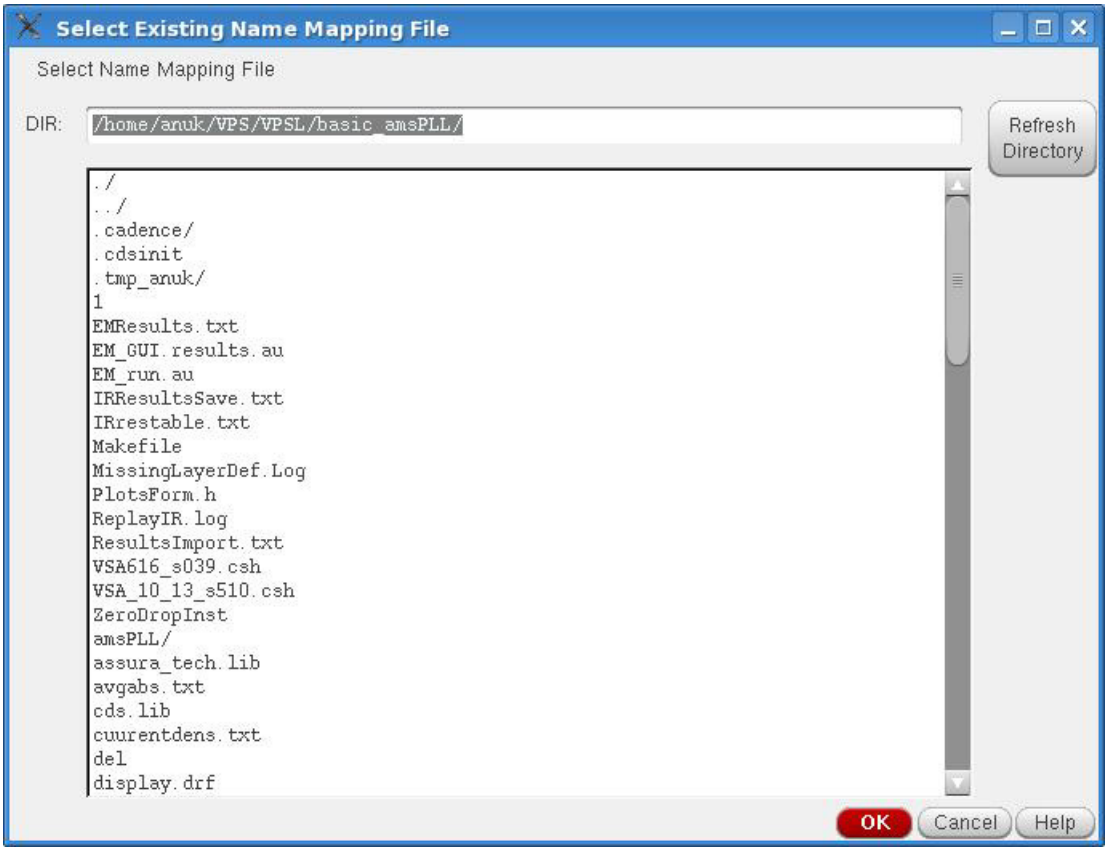


In this form, select the file and click *OK.*

- Click *Select mapFile* to select the relevant layermap file. The Select Existing Name Mapping File form opens. The form is shown below.

© 2020 All Rights Reserved.

Figure 6-7 Selecting an Existing Mapfile



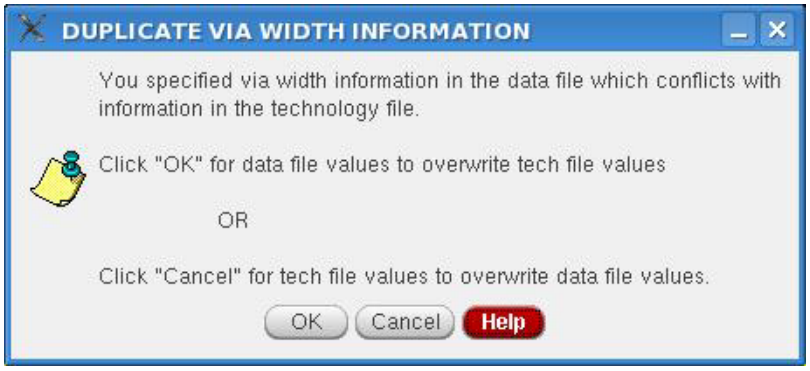
In this form, select the file and click *OK.*

- Click *emDataFile* to specify the `emDataFile` flow. When you click *emDataFile, the EM Data File* cyclic field appears in the form. From the cyclic field, specify an existing `emDataFile` – available from previous runs – for performing EM analysis.

  When you select the `emDataFile` flow, a pop-up window with the duplicate via width information warning message appears. This is shown below.

© 2020 All Rights Reserved.

Figure 6-8 Duplicate Via Width Information Warning

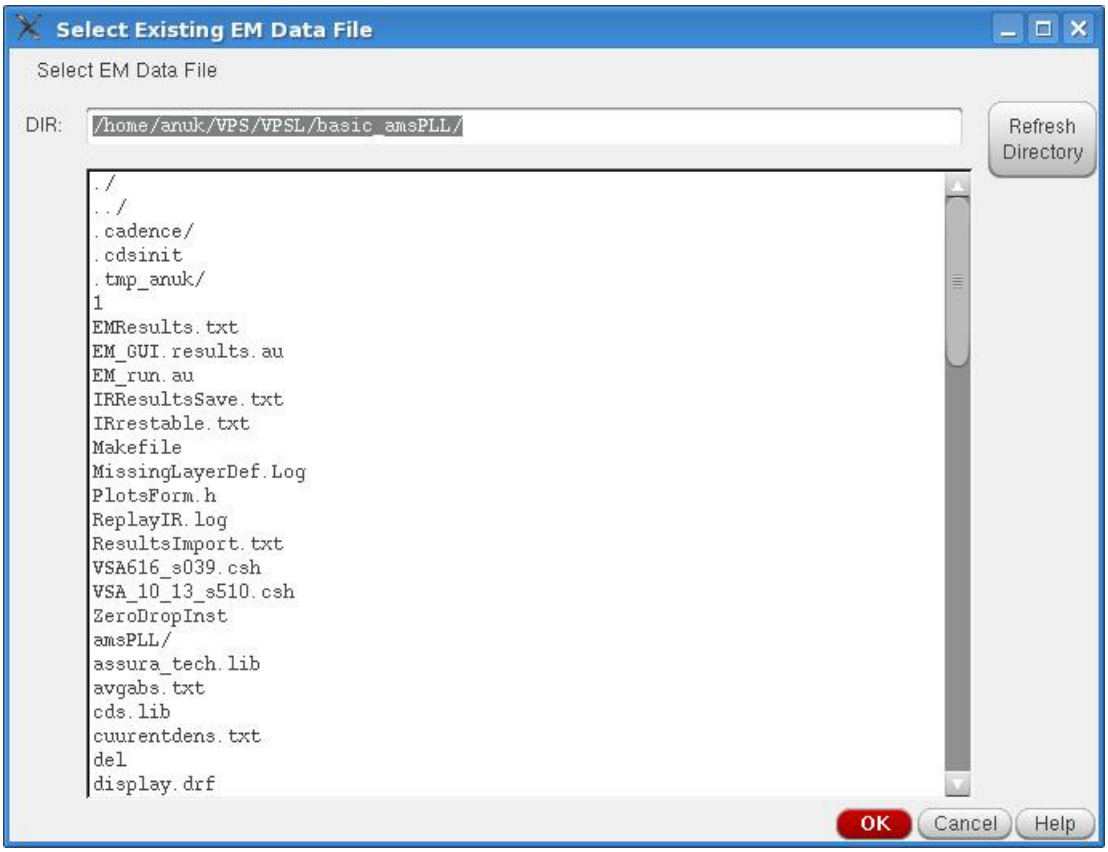


In the above pop-up window, click *OK* to use the via width information from the `emDataFile`. Click *Cancel* to use the via width values from `qrcTechFile`. To suppress this warning message, set the `_vsaMinWarning` environment variable to `true`. In this case, OK will be considered as default, which means that the via width information specified in the `emDataFile` will override that specified in `qrcTechFile`.

- You can also click *Select emDatafile* to specify an existing `emDataFile`. The Select Existing EM Data File form opens. This form is shown below.

© 2020 All Rights Reserved.

Figure 6-9 Selecting an Existing EM Data File

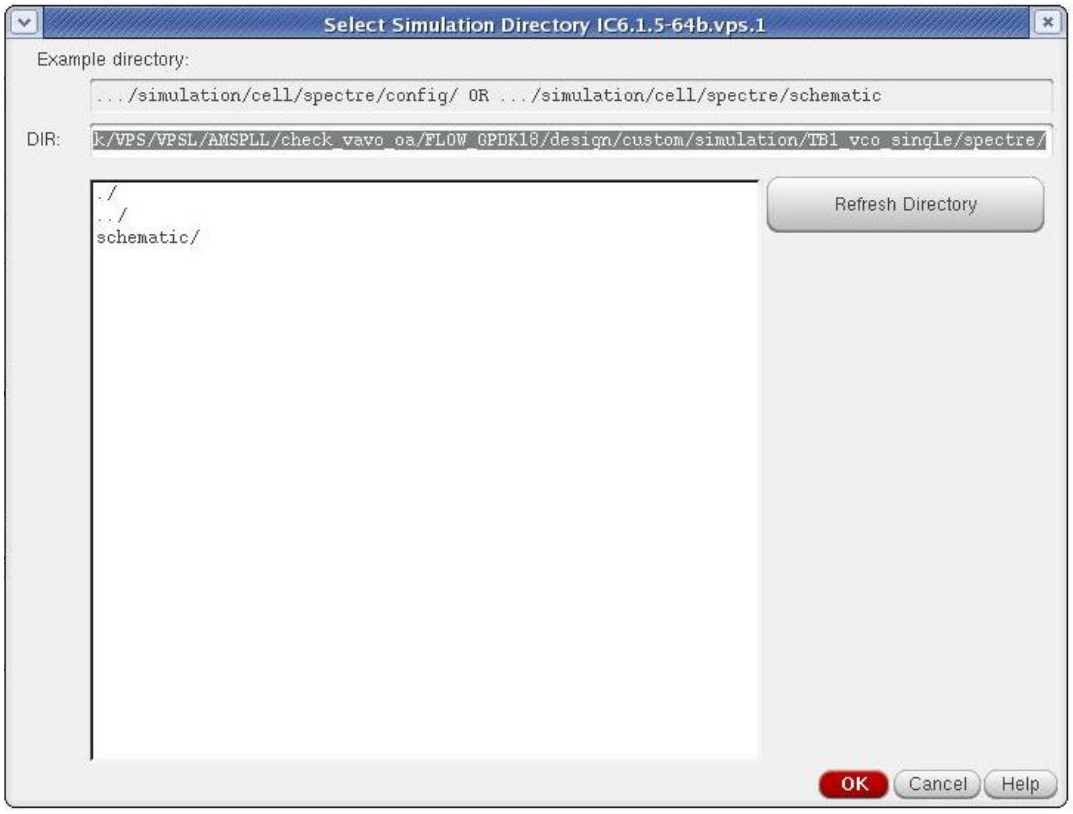


- In this form, select the file and click *OK*.

- In the Simulation group box:
  - Specify the *Simulation Directory* to point to the results of the Spectre, UltraSim, or APS simulation that was performed on the extracted view.
  - You can also click *Select Simulation Directory* to open the Select Simulation Directory form. This form is shown below.

© 2020 All Rights Reserved.

Figure 6-10 Selecting the Simulation Directory



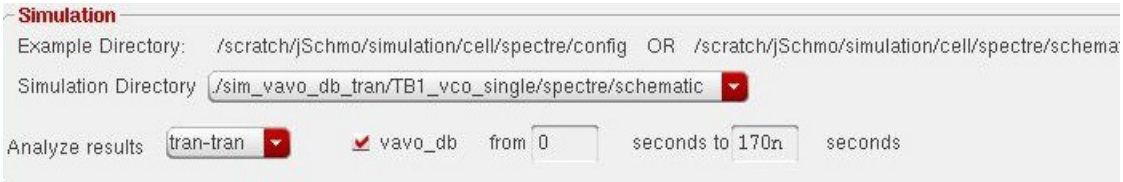
In this form, select the simulation directory and click OK.

For more information, see Running a Spectre Simulation, Running An UltraSim Simulation, or Running an APS Simulation in the “IR/EM Analysis Flow” chapter.

Note: The *Example Directory* field shows you how to specify this value.

- In the *Analyze results* cyclic field, you can specify the following:

Figure 6-11 Specifying the Type of Analysis



  - Select *DCOP* or DC operating point analysis, which is recommended for large analog designs that are difficult to simulate in Spectre using transient analysis, especially when all parasitics are included in the analysis.

© 2020 All Rights Reserved.

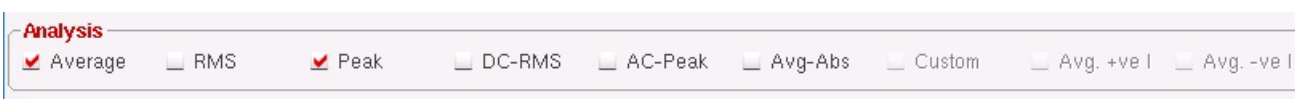
- Select *TRANSIENT* analysis, which automatically sets the simulation time in seconds when the *from/to* fields are selected. You can select a subset of this value for transient analysis by changing the value in the *from/to* fields.

  If the simulation data is in the `vavo.db` format, you cannot change the value in the *from/to* fields. This is because all currents are pre-processed and stored by the simulator before Power IR/EM is run. To change these values, change the `emirstart/emirstop` options in the simulator tran statement. For more information, see Running an APS Simulation in the "IR/EM Analysis Flow" chapter.

- If your simulation data is in `vavo.db` format, the *vavo_db* option appears in the form. This option is selected by default. To use the PSF database, deselect this option. This option is available only if you are using `vavo.db`. You can also specify this option in the batch mode by using the `_vsa_vavo_db_enable` command.

■ In the *Analysis* group box, specify the method for processing results. The available options are detailed below.

Figure 6-12 Specifying the Electromigration Power Grid Analysis Options



Note: If no analysis type is specified, the software performs *Average* and *Peak* analysis by default.

- **Average**—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.

© 2020 All Rights Reserved.

- **RMS**—Calculates the current density violations based on the root mean square (RMS) of the peak 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.

- **Peak**—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

- **DC-RMS**—Calculates the current density violations based on the RMS of the maximum DC current for metal lines, vias, and contacts.

Note: The RMS analysis type is specified using the parameter, `em_jmax_ac_rms` in qrcTechFile while the DC-RMS analysis type is specified using the `em_jmax_dc_rms` parameter.

- **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.

$I_{peak_ac} = I_{peak}$

The above values are from simulation.

$I_{peak_ac(limit)} = I_{peak_dc(limit)} / (r)^{1/2}$

where the $I_{peak_dc(limit)}$ is specified in the ICT file

and duty ratio “r” is:

r = $\mathbf{I}_{abs\text{-}avg}$ / $\mathbf{I}_{peak}$, by default

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

And

r = T_d/ T_{total}, when the `dynamicACPeak` variable is set.

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

T_{total} = total transient time

© 2020 All Rights Reserved.

Note: The default value of “r” is 0.05. You may need to modify this parameter in scenarios where the value of “r’ is very small. A small “r” value results in an unreasonable increase in the I_{peak_ac} limit. You can avoid such scenarios in any of the following ways; by using the `applyR` keyword to reset the value of “r” in the ICT file, by using the `applyRThreshold` custom variable in the Electromigration Edit/View Custom Parameters form, or by using the `_vsaApplyRThreshold` environment variable.

- **Avg-Abs**—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.

- **Custom**—Uses the user-defined current calculation method. In this case, a SKILL function named “`vsaUserDefinedICalc`” must be added before starting the analysis.

This function is detailed below:

```
procedure(vsaUserDefinedICalc(@key
(Ipeak 0)
(Iavg 0)
(Iavgabs 0)
(Irms 0)
(Ipeakac 0)
(width 0)
(resistance 0)
(temperature 0)
(layer 'nil)
(pResistorName 'nil)
(Hierarchy 'nil)
(parameters 'nil)

))
```

Argument	Description	Type	Units
`Ipeak`	peak current	float	Ampere
`Iavg`	average current	float	Ampere

© 2020 All Rights Reserved.

Argument	Description	Type	Units
`Iavgabs`	absolute average current	float	Ampere
`Irms`	root mean square current	float	Ampere
`width`	width of presistor	float	Micron
`resistance`	resistance value	float	Ohm
`temperature`	simulation temperature	float	Centigrade
`layer`	layer name	string	Text
`pResistorName`	parasitic resistance	string	Text
`Hierarchy`	hierarchy	string	Text
`parameters`	extra property of resistance	table	-

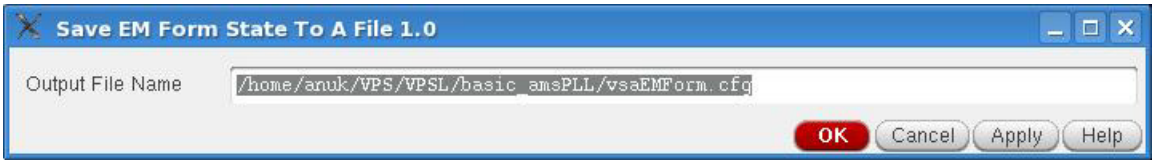
Note: You can write any logic inside your function for finding the customized value of current. The procedure must return the value of current in Ampere and the type as float.

Important

If the current density specification is not available for any specific analysis mode, for example, average or peak, then their corresponding check boxes will be disabled in the Electromigration Power Grid Analysis form.

- Click *Save State* to save the information filled out in the form. The Save EM Form State To A File form opens.

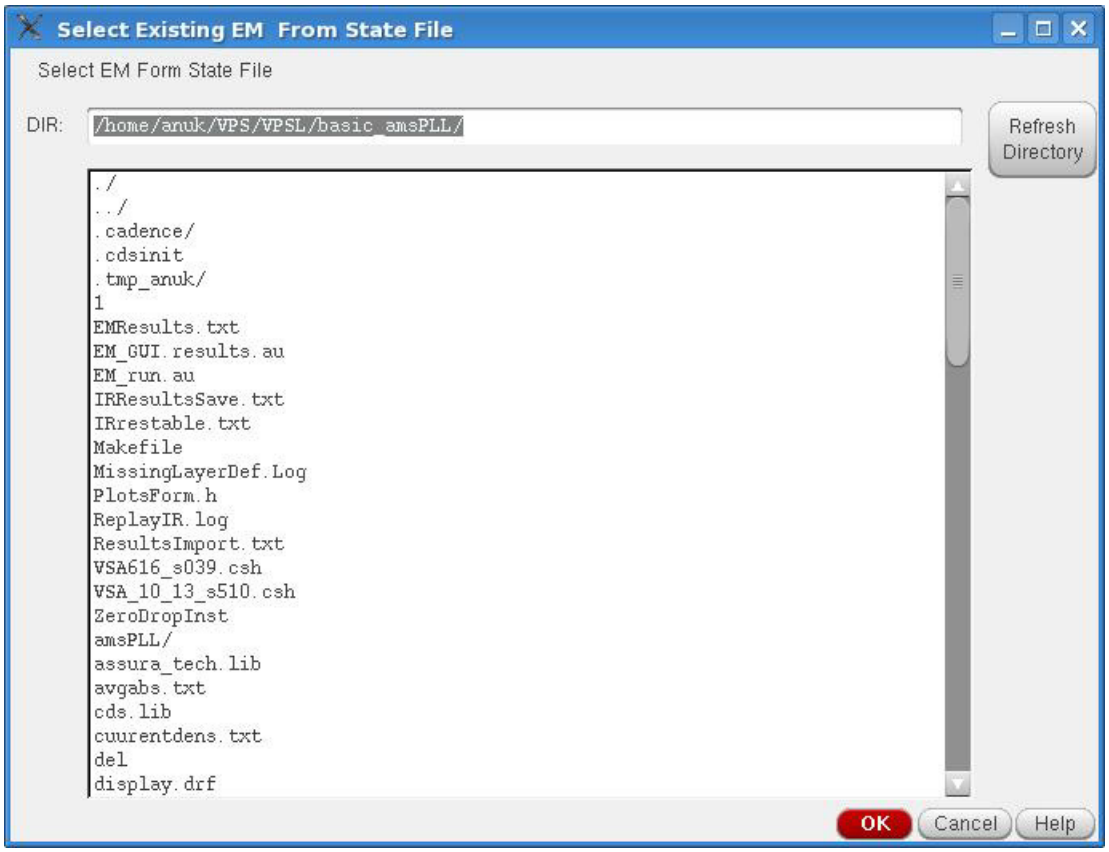
Figure 6-13 Saving EM Form State to a File



- Click *Load State* to load previously saved information in the form. The Select Existing EM From State File form opens.

© 2020 All Rights Reserved.

Figure 6-14 Selecting Existing EM from State File



- After completing the inputs in the Electromigration Power Grid Analysis form, click *OK* or *Apply*. When you click *OK* or *Apply* in this form, the result file and the configuration file are stored in the following location by default:

  `working_dir/VSA.OUT/CURRENT_DATE/`

  Example:

  `working_dir/VSA.OUT/MAY_5_1/EM.results and EM.cfg`

  If multiple sessions are run on the same day, the name of the `CURRENT_DATE` folder will be updated sequentially. This is shown below.

  `working_dir/VSA.OUT/MAY_5_2/EM.results and EM.cfg`

  `working_dir/VSA.OUT/MAY_5_3/EM.results and EM.cfg`

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

© 2020 All Rights Reserved.

Important

If you do not want to save the above files, set the `_vsaDisableAutoArchive` environment variable to `"true"`.

Layermap File for EM Analysis

The layermap file or the mapping file specified for EM analysis, provides the mapping between the ICT file layer names, the extraction layer names and the DFII layer names. A sample layermap file is provided below.

```
#ictName      #extractionName      #dfIIName
mt1              mt1                 Metal1
mt2              mt2                 Metal2
Via1             Via1                Via1
Via2             Via2                Via2
```

You can also provide the device resistor information in the layermap file. In this case, the "`#extractionName`" column will have the device resistor information in the following format:

```
DEVICE_RES:<cellName of the device resistor>:<resistor propertyName>:<width
property name>:<length property Name>:<width unit>:<length unit>
```

It is important that all properties are specified in the order, from left to right, shown in the above format. This means, to specify a property listed on the right, you must first specify the properties listed on the left. For example, to specify the width property name, you must specify the resistor property name and to specify the length property name, you must specify both the width property name and the resistor property name. For example:

```
mt1            DEVICE_RES:nplusres:r:wSim:lSim         Metal1
```

Where,

`nplusres` is the device resistor cell name

`r` is the resistor property name

`wSim` is the width property name

`lSim` is the length property name

Note: All properties other than the resistor cell name are optional. When a property name or unit is not specified, the software uses the default value for that property. For example, if the unit for either the width or the length property name is not specified, the software will use the default unit, which is "meter".

© 2020 All Rights Reserved.

Analyzing EM Across Design Resistors

Power IR/EM supports EM analysis of design resistors. This is done by specifying information in the `emDataFile`.

You can use the `DesignResistors` parameter to provide the list of cell names. For each cell name, you can specify the resistance property name, width property name, length property name, and units of width and length property names of the design resistors for which you want EM to be analyzed.

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.

After specifying the above, append the details for current density specifications for the above design resistors in the `emDataFile` in the following manner.

```
xrefLayers = (
( "CellName" ("emruleName" "DFIILayer"))
)
```

Example:

```
xrefLayers  = (
( “nplusres” (“emrule_nplus” “diff”))
)
```

Add the rule for the design resistors in the current density rules specification section.

```
avgPositiveCurrentDensSpecList = (
(nil layer “emrule_nplus" minW 0.0 maxW -1.0 currentDensity ((1.6, 20) (1.61,110)
(1.62,125)) )
```

There can be some resistors in the design that do not belong to any of the pins selected for analysis. By default, all the design resistors are analyzed irrespective of the pins selected for analysis. To analyze only those design resistors that are connected to selected pins, set the `_vsaDisableAllDesignResistorAnalysis` environment variable to `true`.

To disable analysis of all design resistors, set the `EnableDesignResAnalysis` parameter in the Electromigration Edit/View Custom Parameters form to `nil`. This parameter is set to `true` by default.

For details of the syntax and examples of the parameters to be added to the `emDataFile`, see Reading EM Rules from emDataFile in the “EM Rules Specification” chapter.

Note: Within the `PSF/PSF-XL` flow, the design resistor analysis is currently only supported for the `emDataFile` flow and not for `qrcTechFile` flow.

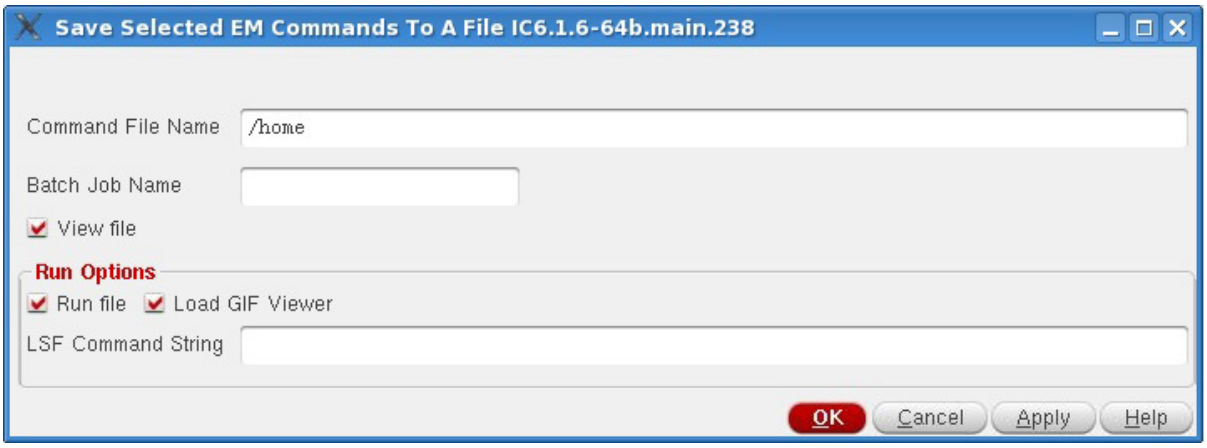
© 2020 All Rights Reserved.

Writing Out an EM Command File

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

- In the Electromigration Power Grid Analysis form, specify the *QRC Results Directory* and the pin name.
- Click *Generate Command File.* The Save Selected EM Commands To a File form opens. This form is shown below.

Figure 6-15 Save Selected EM Commands To a File



- In this form, specify the *Command File Name*. If the command file name 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 `vsaEMBatch.cmd` as the default command file name.
- 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.
- 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 Power IR/EM 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.

© 2020 All Rights Reserved.

- Click *Load GIF Viewer* to launch the vsaPlot, which is used for visualizing and customizing the IR drop and EM analysis results. For details, see the "Visualizing Results in vsaPlot" chapter.
- 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`.
- Click *OK*.

Opening Power IR/EM Directly from Extracted View

The procedure for opening Power IR/EM directly from extracted view for EM analysis is the same as described for IR Drop analysis. For detailed information, see the same section in the "IR Drop Analysis" chapter.

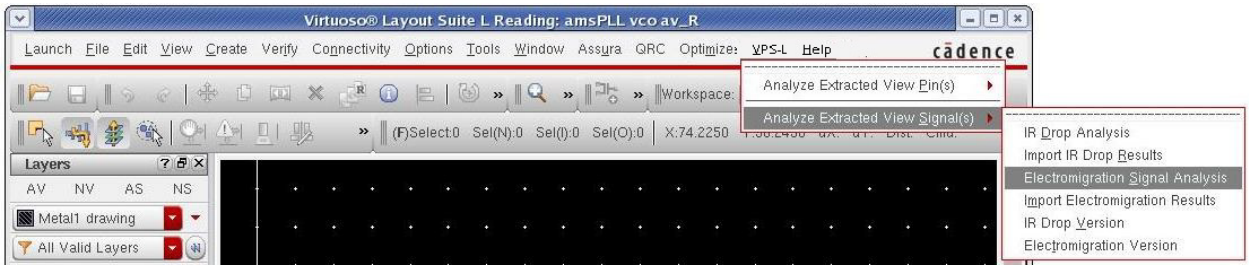
EM Analysis of Signal Nets

EM analysis of signal nets is similar to the EM analysis of power nets.

To perform EM analysis on signal nets, follow these steps:

- Choose *VPS-L – Electromigration Signal Analysis* from the menu bar in the extracted view. This is shown below.

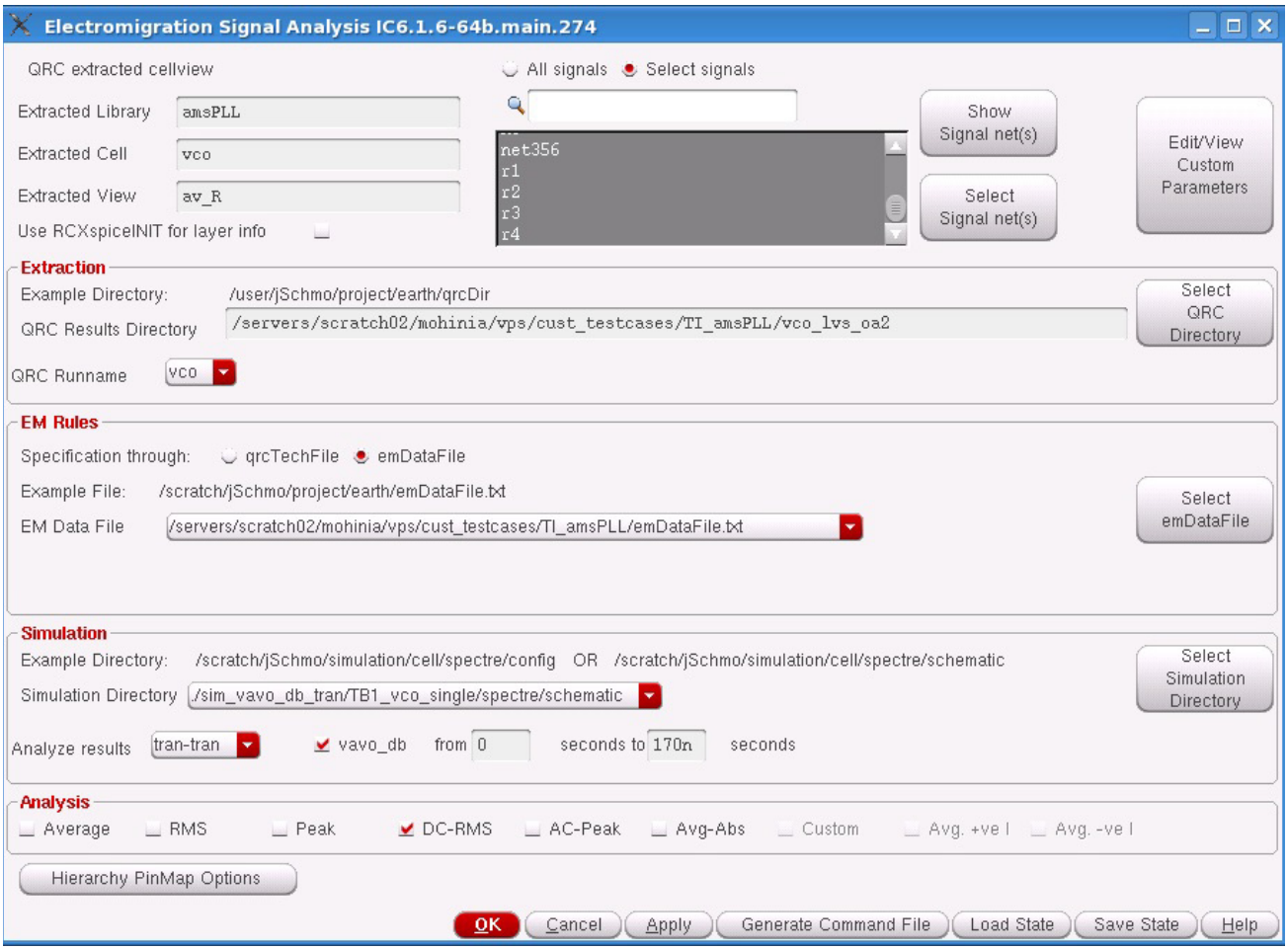
Figure 6-16 Opening Electromigration Signal Analysis Form



- The Electromigration Signal Analysis form opens. This is shown below.

© 2020 All Rights Reserved.

Figure 6-17 Running Electromigration Signal Analysis



- Click *All signals* to perform EM analysis on all signals.This is the default option. This does not include the power or ground nets that must be analyzed as part of the *Electromigration Power Analysis.* See EM Analysis of the Power Grid.
- Click *Select signals* to specify signals on which EM analysis will be performed. Search for signal names by typing the signal name in the search field. The search field also accepts SKILL regular expressions.

  Note: Any signal that is connected to a pin is considered a power net and should be analyzed using *Electromigration Power Grid Analysis.*

- When you click *Select signals*, a list box is enabled that shows all the signal names, the Show *Signal net(s)* option, and the *Select Signal nets(s)* option.
  - Select the nets in the list box and click *Show Signal net(s)* to view the selected signals highlighted in the extracted view.

© 2020 All Rights Reserved.

- ❑ Click*Select Signal net(s)* to select signals from the extracted view. Click *Select Signal Net(s)* and then select the signals in the extracted view. The selected signals are highlighted. After selecting the signals, press the `Esc` key. The selected nets are highlighted in the list box.

  EM analysis will be performed for only those *presistors* that are on the selected signal nets.

- ■ The remaining fields to be filled out in the Electromigration Signal Analysis form are the same as those explained in EM Analysis of the Power Grid.

EM Analysis Results

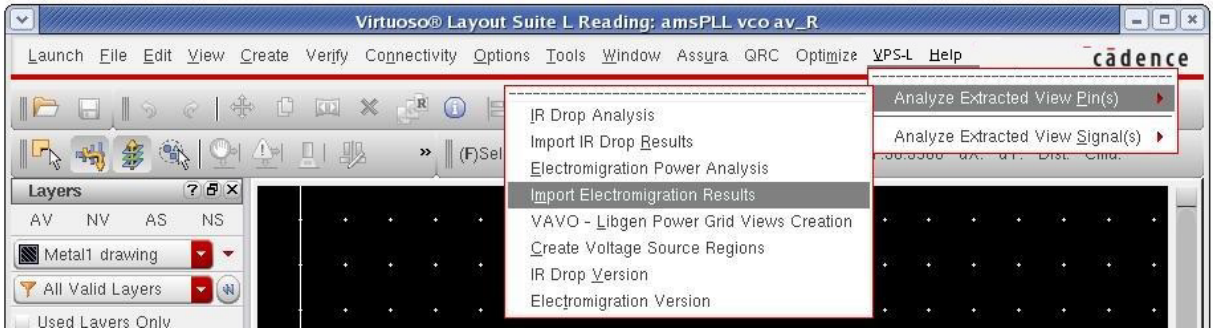
After completing EM analysis for pins or signals, the results can either be imported from the EM analysis performed earlier or they can be displayed by selecting options in the Electromigration Analysis Results form. Both methods are detailed below.

Importing Results of a Previous EM Analysis

To import the results of an EM analysis performed earlier, perform the following steps:

- ■ Choose *Analyze Extracted View Pin(s) or Analyze Extracted View Signal(s) – Import Electromigration Results.*

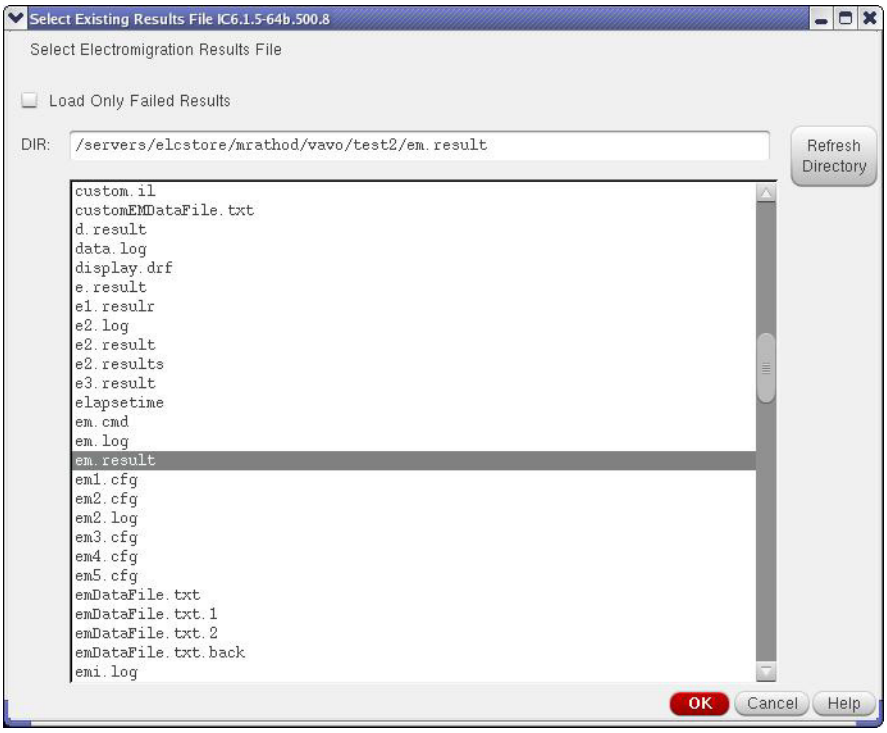
Figure 6-18 Opening Import Electromigration Results Form



- ■ The Select Existing Results File form opens.

© 2020 All Rights Reserved.

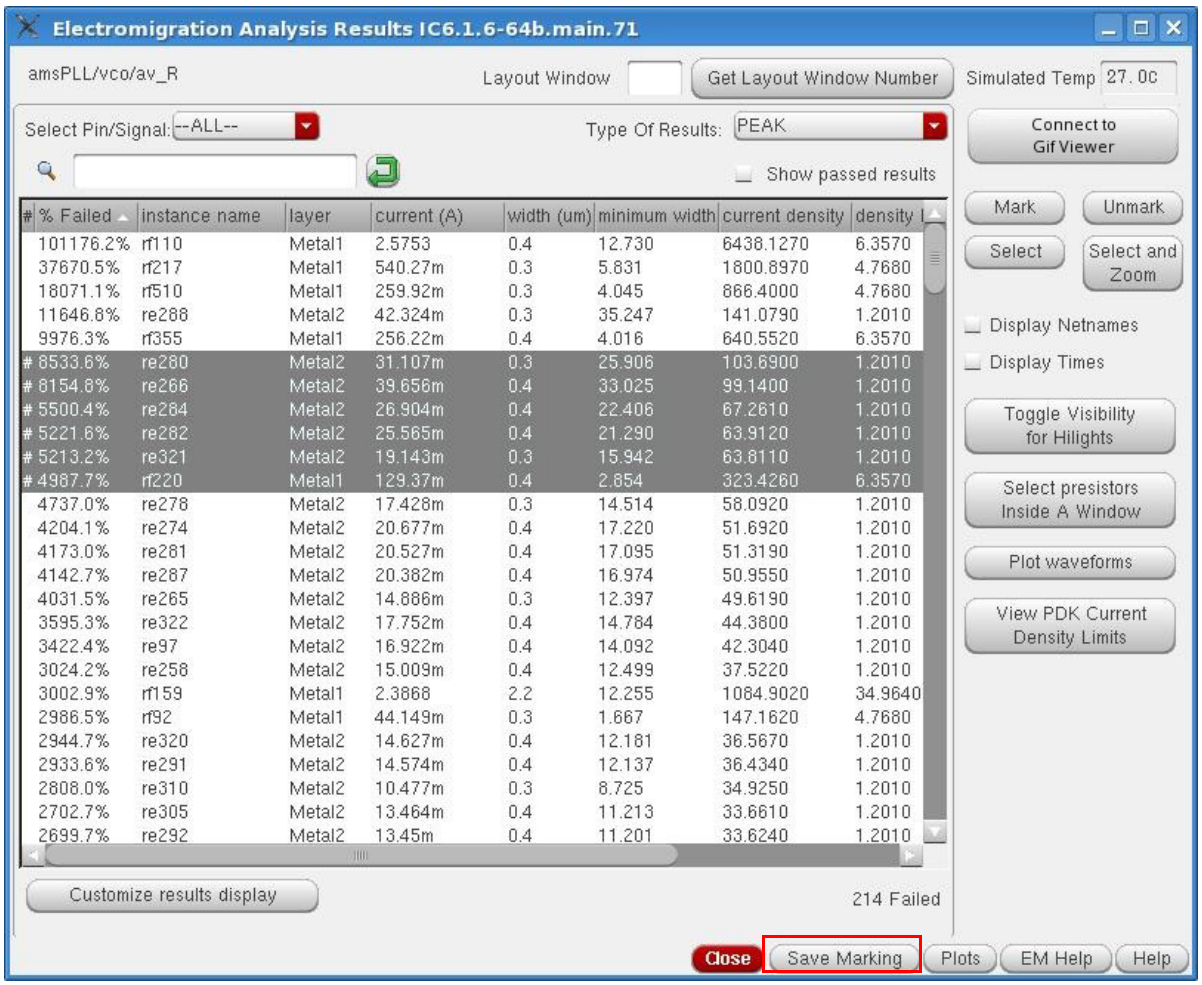
Figure 6-19 Importing Existing Electromigration Results



- In the above form, click *Load Only Failed Results* to view EM results for only failed resistors.
- Select the file from the list and click *OK*. The Electromigration Analysis Results form opens.

© 2020 All Rights Reserved.

Figure 6-20 Opening Electromigration Analysis Results Form when EM Analysis Results are Imported



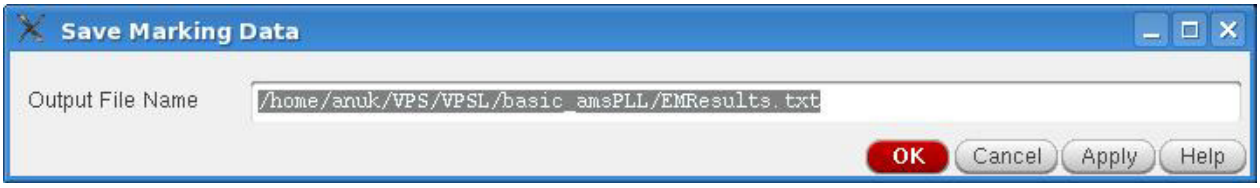
#	% Failed	instance name	layer	current (A)	width (um)	minimum width	current density	density
	101176.2%	rf110	Metal1	2.5753	0.4	12.730	6438.1270	6.3570
	37670.5%	rf217	Metal1	540.27m	0.3	5.831	1800.8970	4.7680
	18071.1%	rf510	Metal1	259.92m	0.3	4.045	866.4000	4.7680
	11646.8%	re288	Metal2	42.324m	0.3	35.247	141.0790	1.2010
	9976.3%	rf355	Metal1	256.22m	0.4	4.016	640.5520	6.3570
#	8533.6%	re280	Metal2	31.107m	0.3	25.906	103.6900	1.2010
#	8154.8%	re266	Metal2	39.656m	0.4	33.025	99.1400	1.2010
#	5500.4%	re284	Metal2	26.904m	0.4	22.406	67.2610	1.2010
#	5221.6%	re282	Metal2	25.565m	0.4	21.290	63.9120	1.2010
#	5213.2%	re321	Metal2	19.143m	0.3	15.942	63.8110	1.2010
#	4987.7%	rf220	Metal1	129.37m	0.4	2.854	323.4260	6.3570
	4737.0%	re278	Metal2	17.428m	0.3	14.514	58.0920	1.2010
	4204.1%	re274	Metal2	20.677m	0.4	17.220	51.6920	1.2010
	4173.0%	re281	Metal2	20.527m	0.4	17.095	51.3190	1.2010
	4142.7%	re287	Metal2	20.382m	0.4	16.974	50.9550	1.2010
	4031.5%	re265	Metal2	14.886m	0.3	12.397	49.6190	1.2010
	3595.3%	re322	Metal2	17.752m	0.4	14.784	44.3800	1.2010
	3422.4%	re97	Metal2	16.922m	0.4	14.092	42.3040	1.2010
	3024.2%	re258	Metal2	15.009m	0.4	12.499	37.5220	1.2010
	3002.9%	rf159	Metal1	2.3868	2.2	12.255	1084.9020	34.9640
	2986.5%	rf92	Metal1	44.149m	0.3	1.667	147.1620	4.7680
	2944.7%	re320	Metal2	14.627m	0.4	12.181	36.5670	1.2010
	2933.6%	re291	Metal2	14.574m	0.4	12.137	36.4340	1.2010
	2808.0%	re310	Metal2	10.477m	0.3	8.725	34.9250	1.2010
	2702.7%	re305	Metal2	13.464m	0.4	11.213	33.6610	1.2010
	2699.7%	re292	Metal2	13.45m	0.4	11.201	33.6240	1.2010

Note: When existing results for EM analysis are imported, the Electromigration Analysis Results form provides a *Save Marking* button instead of *Save Results to A File* button. This is shown in the above form.

- Click *Save Marking* to save the marked results in a file for future reference. The Save Marking Data form opens. The results can be reloaded in subsequent sessions. This is shown below.

© 2020 All Rights Reserved.

Figure 6-21 Saving the Marking Data



In this form, specify the *Output File Name* and click *OK*.

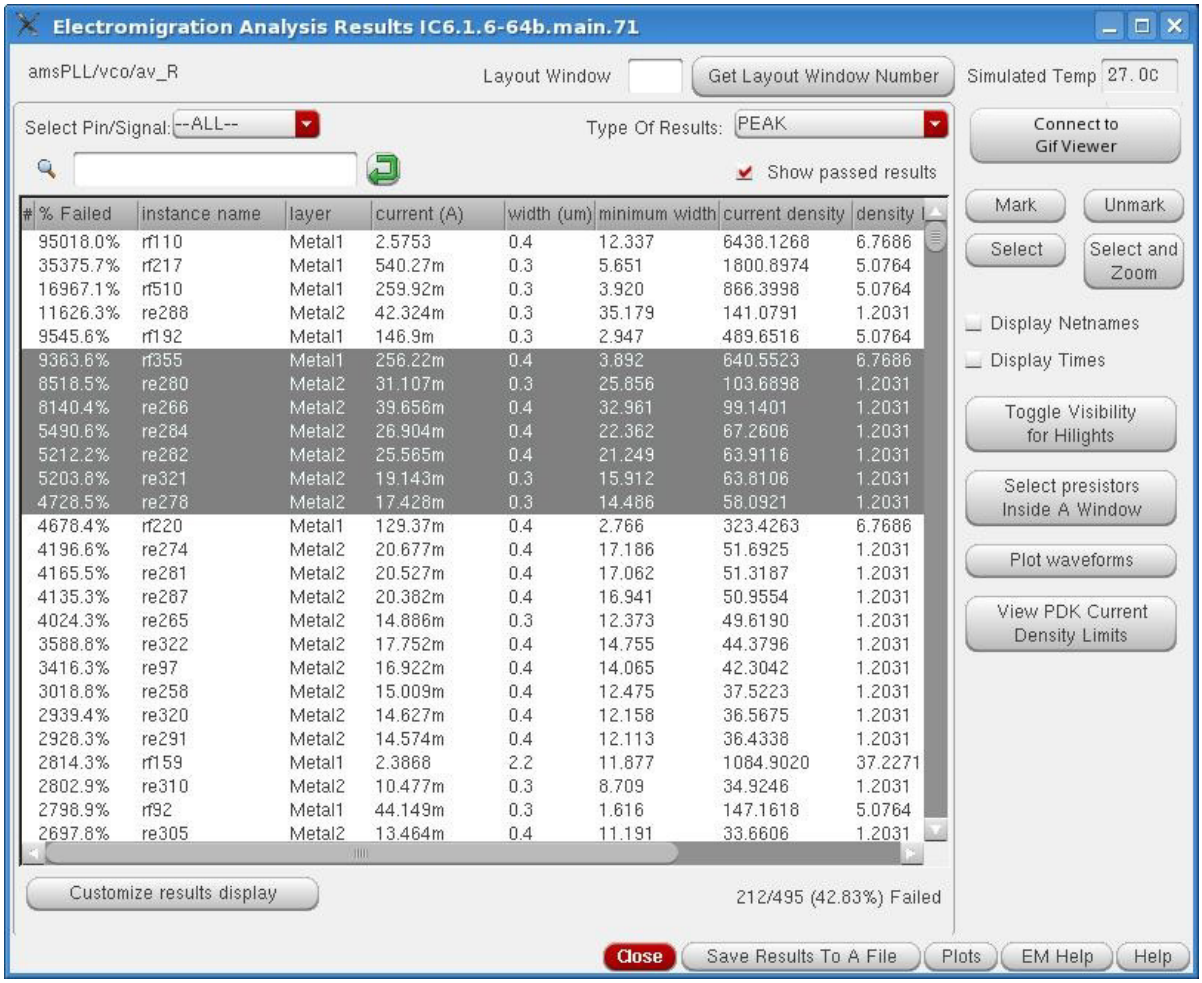
Viewing Electromigration Analysis Results for the Current Run

The EM analysis results for the current run can be viewed using the various options provided in the Electromigration Analysis Results form.

After running EM analysis for either pins or signals, when you click *OK* on the Electromigration Power Grid Analysis or Electromigration Signal Analysis form, the Electromigration Analysis Results form opens. This form is shown below.

© 2020 All Rights Reserved.

Figure 6-22 Opening Electromigration Analysis Results Form for the Current EM Analysis Run



#	% Failed	instance name	layer	current (A)	width (um)	minimum width	current density	density
	95018.0%	rf110	Metal1	2.5753	0.4	12.337	6438.1268	6.7686
	35375.7%	rf217	Metal1	540.27m	0.3	5.651	1800.8974	5.0764
	16967.1%	rf510	Metal1	259.92m	0.3	3.920	866.3998	5.0764
	11626.3%	re288	Metal2	42.324m	0.3	35.179	141.0791	1.2031
	9545.6%	rf192	Metal1	146.9m	0.3	2.947	489.6516	5.0764
	9363.6%	rf355	Metal1	256.22m	0.4	3.892	640.5523	6.7686
	8518.5%	re280	Metal2	31.107m	0.3	25.856	103.6898	1.2031
	8140.4%	re266	Metal2	39.656m	0.4	32.961	99.1401	1.2031
	5490.6%	re284	Metal2	26.904m	0.4	22.362	67.2606	1.2031
	5212.2%	re282	Metal2	25.565m	0.4	21.249	63.9116	1.2031
	5203.8%	re321	Metal2	19.143m	0.3	15.912	63.8106	1.2031
	4728.5%	re278	Metal2	17.428m	0.3	14.486	58.0921	1.2031
	4678.4%	rf220	Metal1	129.37m	0.4	2.766	323.4263	6.7686
	4196.6%	re274	Metal2	20.677m	0.4	17.186	51.6925	1.2031
	4165.5%	re281	Metal2	20.527m	0.4	17.062	51.3187	1.2031
	4135.3%	re287	Metal2	20.382m	0.4	16.941	50.9554	1.2031
	4024.3%	re265	Metal2	14.886m	0.3	12.373	49.6190	1.2031
	3588.8%	re322	Metal2	17.752m	0.4	14.755	44.3796	1.2031
	3416.3%	re97	Metal2	16.922m	0.4	14.065	42.3042	1.2031
	3018.8%	re258	Metal2	15.009m	0.4	12.475	37.5223	1.2031
	2939.4%	re320	Metal2	14.627m	0.4	12.158	36.5675	1.2031
	2928.3%	re291	Metal2	14.574m	0.4	12.113	36.4338	1.2031
	2814.3%	rf159	Metal1	2.3868	2.2	11.877	1084.9020	37.2271
	2802.9%	re310	Metal2	10.477m	0.3	8.709	34.9246	1.2031
	2798.9%	rf92	Metal1	44.149m	0.3	1.616	147.1618	5.0764
	2697.8%	re305	Metal2	13.464m	0.4	11.191	33.6606	1.2031

In this form, specify the following:

- In this form, select the pin or signal for which you want to view the results, from the *Select Pin/Signal* cyclic field. You can search for pin or signal names by typing the name in the search field and pressing enter, the button with the green arrow. The search field also accepts SKILL regular expressions.
- Click *Show passed results* to view the presistors that passed the current density limits. This field is enabled by default when the Electromigration Analysis Results form is opened after completing the Electromigration Power Grid Analysis or Electromigration Signal Analysis form and clicking *OK*.

  When you deselect this option, the *Select Pin/Signal* cyclic field is updated to list only those pins and signals that have failed the current density limits. This is useful in reducing

© 2020 All Rights Reserved.

the debugging time when there are many pins or signals but only few have failed the current density limits. Instead of viewing all nets, both passed and failed, you can deselect this option to view only the failed nets and proceed with debugging.

Note: This field is disabled by default if *Import Electromigration Results* submenu is selected from the *VPS-L* menu.

- In *Layout Window, s*pecify the window ID for the layout, if a layout is open for the design. Click *Get Layout Window Number* to let the software get the layout window number automatically from an already open window. If no layout window is open, a new window will be created with a layout view and that one will be used.
- Click *Connect to Gif Viewer* to open the vsaplot window.
- From the *Type Of Results* cyclic field, select *Average*, *RMS*, *PEAK,* or *AC-Peak* values of the current from the Spectre simulation. This field shows options based on the options selected for transient analysis in the Electromigration Signal Analysis or Electromigration Power Grid Analysis form.
- The *Simulated Temp* field displays the simulation temperature in degree Celsius. In the above form, the simulated temperature is 27.0 degree celsius.
- Click *Mark* and *Unmark* for marking and unmarking presistors. Select the presistors and click *Mark*. A # symbol gets added at the beginning of the row in which the EM analysis result of the presistor is displayed. The result file displays the EM analysis results for all marked presistors. Similarly, click *Unmark* to unmark a presistor. Presistors showing false errors can be unmarked so that they are not reported in the result file. This helps save time spent viewing and debugging all the false errors reported in the result file.
- A text box in the form provides information about the presistors on which the EM analysis is being performed. There are several columns in the text field. The *% Failed* column indicates whether a presistor passed or failed the current density limit (EM failure). The measured width is the width of the polygon, and the minimum width is the width that is required to avoid an EM failure.

  Note: The fraction of presistors that failed the current density limits as compared to the total number of presistors and the percentage failure are displayed at the bottom-right corner of the form. For example, the form above shows `212/495 (43.23%) Failed`.

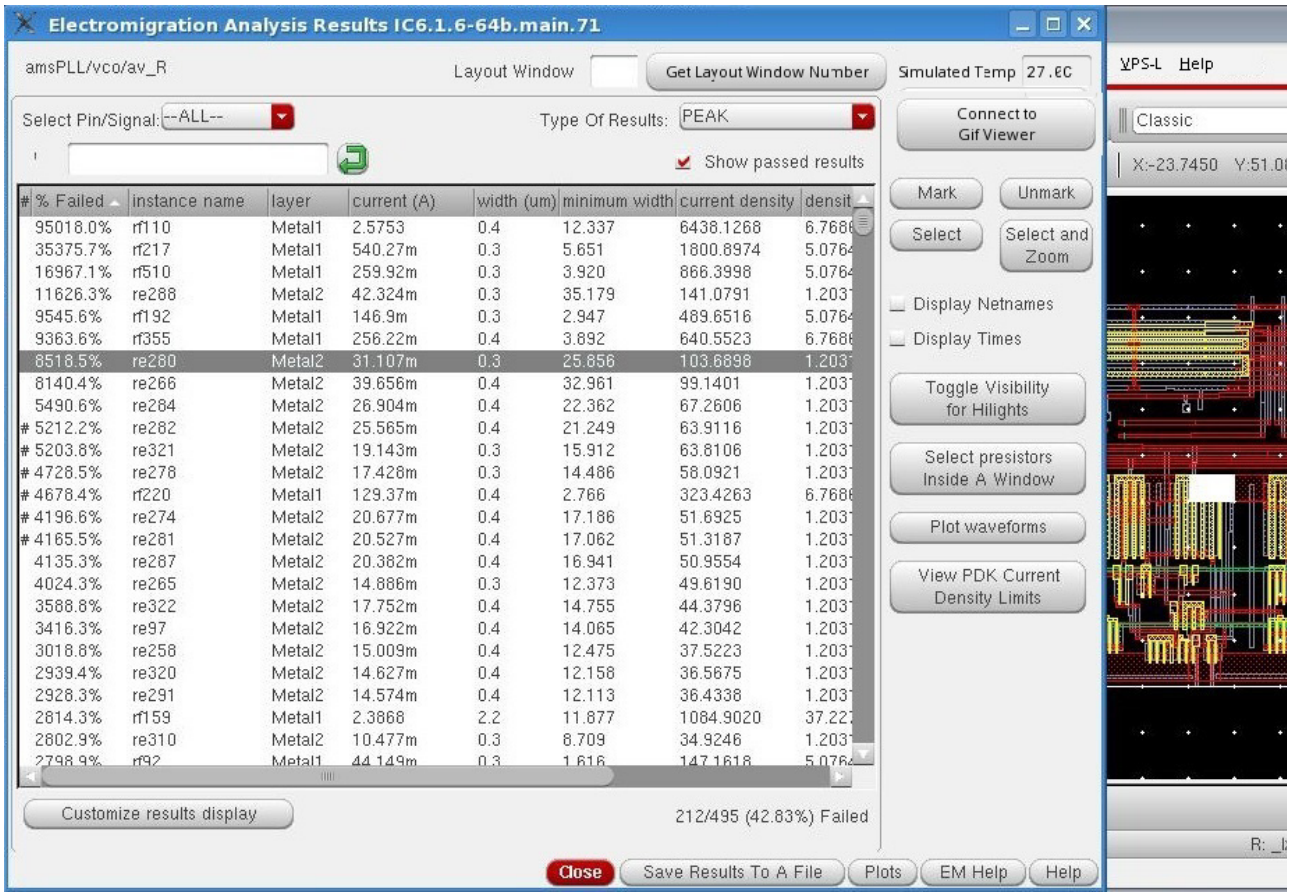
Tip

In case all the presistors in the design pass the EM analysis, the results are sorted in descending order beginning with the presistors that come closest to the established threshold.

© 2020 All Rights Reserved.

- Click *Toggle Visibility for Hilights* to toggle the highlights of the selected presistors on and off.
- Click *Select* to select a presistor of interest and highlight it. When a presistor is selected, all previous presistor highlights are removed and only the current selection is highlighted. The presistor is highlighted in white and blinks. However, you can set your color preference for highlighting a presistor. For this, use the `_vsaDrfFile` environment variable to specify the `vavo8` highlight layer in the `vavo.drf` file.

Figure 6-23 Selecting a Presistor and Highlighting it on the Layout

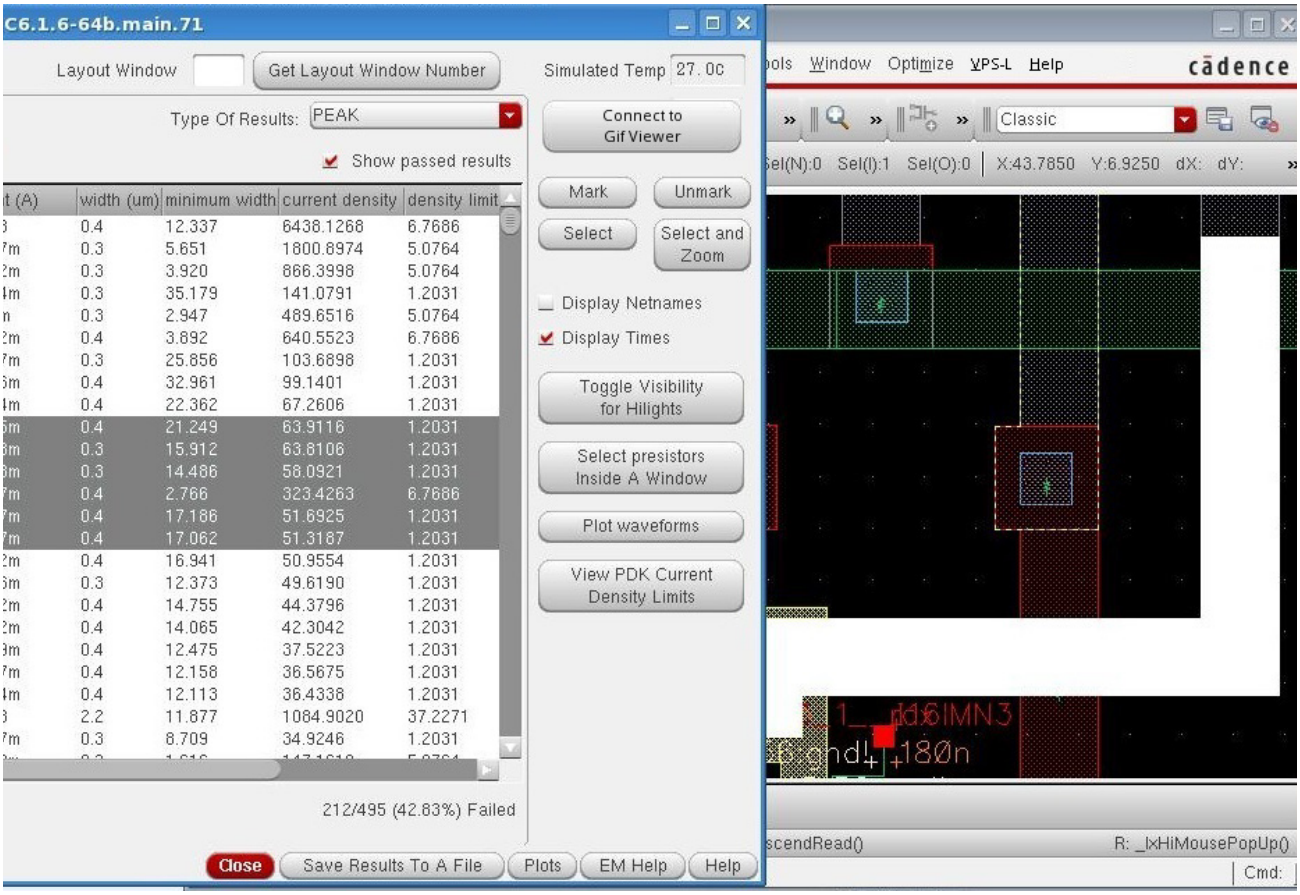


#	% Failed	instance name	layer	current (A)	width (um)	minimum width	current density	densit
	95018.0%	rf110	Metal1	2.5753	0.4	12.337	6438.1268	6.7688
	35375.7%	rf217	Metal1	540.27m	0.3	5.651	1800.8974	5.0764
	16967.1%	rf510	Metal1	259.92m	0.3	3.920	866.3998	5.0764
	11626.3%	re288	Metal2	42.324m	0.3	35.179	141.0791	1.2037
	9545.6%	rf192	Metal1	146.9m	0.3	2.947	489.6516	5.0764
	9363.6%	rf355	Metal1	256.22m	0.4	3.892	640.5523	6.7688
	8518.5%	re280	Metal2	31.107m	0.3	25.856	103.6898	1.2037
	8140.4%	re266	Metal2	39.656m	0.4	32.961	99.1401	1.2037
	5490.6%	re284	Metal2	26.904m	0.4	22.362	67.2606	1.2037
#	5212.2%	re282	Metal2	25.565m	0.4	21.249	63.9116	1.2037
#	5203.8%	re321	Metal2	19.143m	0.3	15.912	63.8106	1.2037
#	4728.5%	re278	Metal2	17.428m	0.3	14.486	58.0921	1.2037
#	4678.4%	rf220	Metal1	129.37m	0.4	2.766	323.4263	6.7688
#	4196.6%	re274	Metal2	20.677m	0.4	17.186	51.6925	1.2037
#	4165.5%	re281	Metal2	20.527m	0.4	17.062	51.3187	1.2037
	4135.3%	re287	Metal2	20.382m	0.4	16.941	50.9554	1.2037
	4024.3%	re265	Metal2	14.886m	0.3	12.373	49.6190	1.2037
	3588.8%	re322	Metal2	17.752m	0.4	14.755	44.3796	1.2037
	3416.3%	re97	Metal2	16.922m	0.4	14.065	42.3042	1.2037
	3018.8%	re258	Metal2	15.009m	0.4	12.475	37.5223	1.2037
	2939.4%	re320	Metal2	14.627m	0.4	12.158	36.5675	1.2037
	2928.3%	re291	Metal2	14.574m	0.4	12.113	36.4338	1.2037
	2814.3%	rf159	Metal1	2.3868	2.2	11.877	1084.9020	37.227
	2802.9%	re310	Metal2	10.477m	0.3	8.709	34.9246	1.2037
	2798.9%	rf92	Metal1	44.149m	0.3	1.616	147.1618	5.0764

- Click *Select and Zoom* to select a presistor of interest and zoom in on it.

© 2020 All Rights Reserved.

Figure 6-24 Selecting and Zooming into a Presistor on the Layout

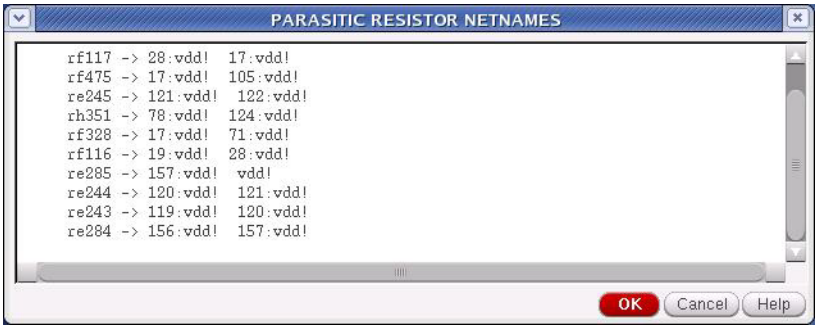


In the extracted view, when a selected net is zoomed into, the layout editor also zooms to the same location. Any layout editing to correct EM violations has to be done in the layout view, after which the design needs to be re-extracted to verify the corrections.

- Click *Display Netnames* to display the net names for the selected parasitic resistors. For this, first click *Display Netnames*, then select the presistors and click *Select* or *Select and Zoom*. The PARASITIC RESISTOR NETNAMES form opens to list all the net names connected to the selected parasitic resistors. This form lists the presistor instance names and their corresponding net names.This is shown below.

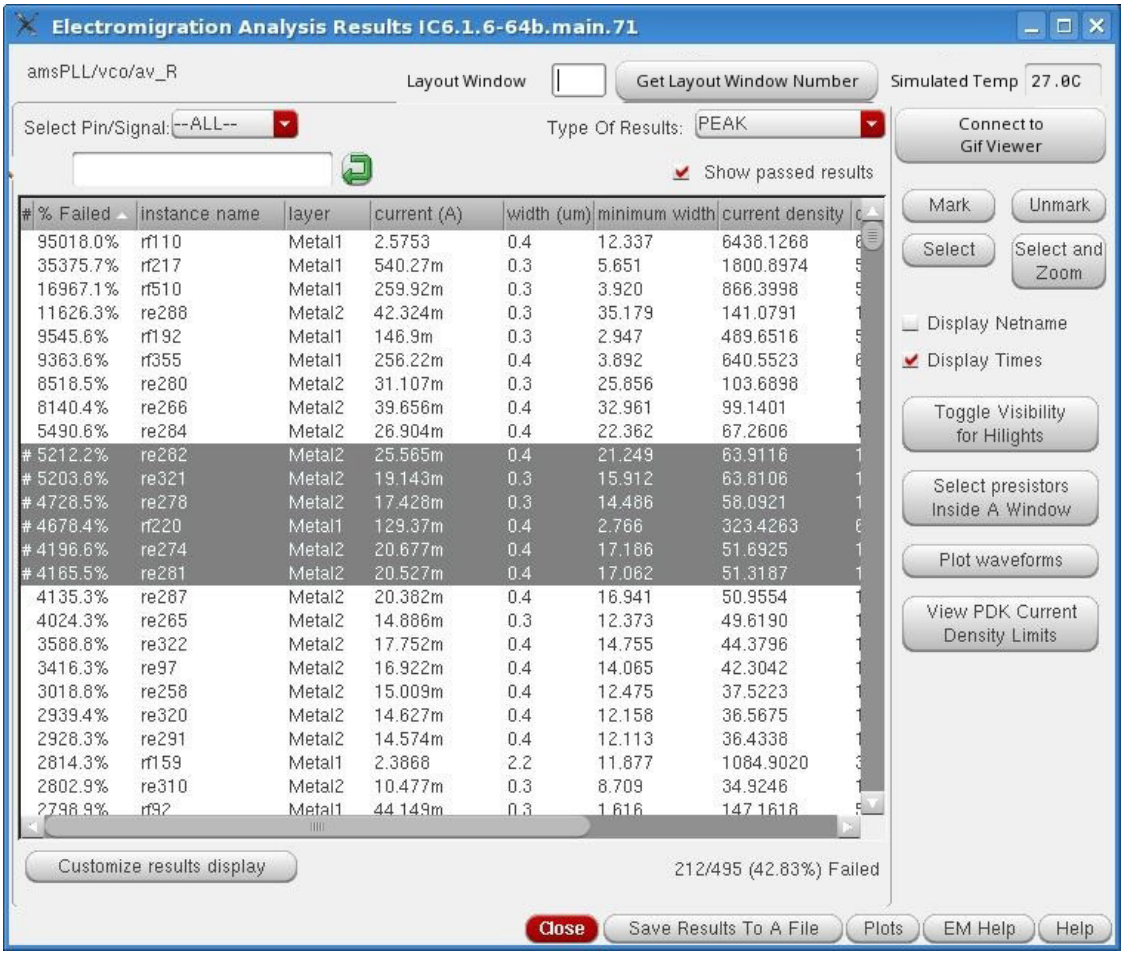
© 2020 All Rights Reserved.

Figure 6-25 PARASITIC RESISTOR NETNAMES Form



- Click *Display Times* to find out when the current density violation occurs. This option is available only when the *Type of Results* selected is *Peak*. For this, first click *Display Times*, then select the presistors and click *Select* or *Select and Zoom*.

Figure 6-26 Displaying Times for Selected Presistors

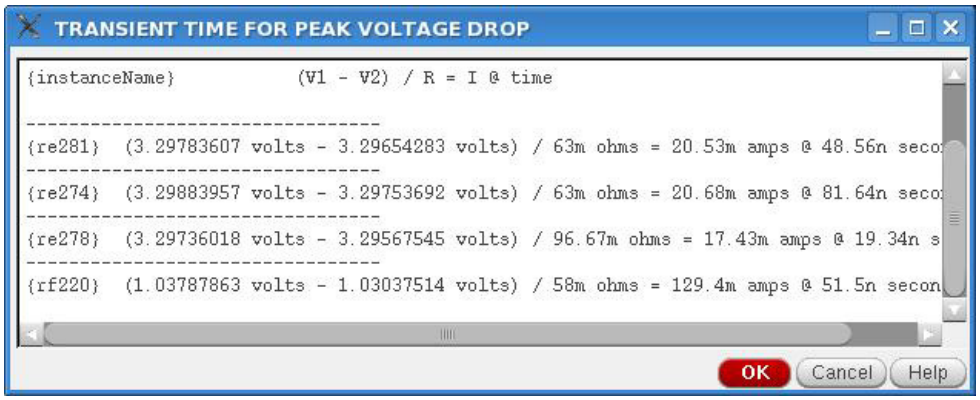


#	% Failed	instance name	layer	current (A)	width (um)	minimum width	current density
	95018.0%	rf110	Metal1	2.5753	0.4	12.337	6438.1268
	35375.7%	rf217	Metal1	540.27m	0.3	5.651	1800.8974
	16967.1%	rf510	Metal1	259.92m	0.3	3.920	866.3998
	11626.3%	re288	Metal2	42.324m	0.3	35.179	141.0791
	9545.6%	rf192	Metal1	146.9m	0.3	2.947	489.6516
	9363.6%	rf355	Metal1	256.22m	0.4	3.892	640.5523
	8518.5%	re280	Metal2	31.107m	0.3	25.856	103.6898
	8140.4%	re266	Metal2	39.656m	0.4	32.961	99.1401
	5490.6%	re284	Metal2	26.904m	0.4	22.362	67.2606
#	5212.2%	re282	Metal2	25.565m	0.4	21.249	63.9116
#	5203.8%	re321	Metal2	19.143m	0.3	15.912	63.8106
#	4728.5%	re278	Metal2	17.428m	0.3	14.486	58.0921
#	4678.4%	rf220	Metal1	129.37m	0.4	2.766	323.4263
#	4196.6%	re274	Metal2	20.677m	0.4	17.186	51.6925
#	4165.5%	re281	Metal2	20.527m	0.4	17.062	51.3187
	4135.3%	re287	Metal2	20.382m	0.4	16.941	50.9554
	4024.3%	re265	Metal2	14.886m	0.3	12.373	49.6190
	3588.8%	re322	Metal2	17.752m	0.4	14.755	44.3796
	3416.3%	re97	Metal2	16.922m	0.4	14.065	42.3042
	3018.8%	re258	Metal2	15.009m	0.4	12.475	37.5223
	2939.4%	re320	Metal2	14.627m	0.4	12.158	36.5675
	2928.3%	re291	Metal2	14.574m	0.4	12.113	36.4338
	2814.3%	rf159	Metal1	2.3868	2.2	11.877	1084.9020
	2802.9%	re310	Metal2	10.477m	0.3	8.709	34.9246
	2798.9%	rf92	Metal1	44.149m	0.3	1.616	147.1618

© 2020 All Rights Reserved.

In the above form, the *Display Times* field is checked and the presistors are selected. When you click *Select* or *Select to Zoom*, the TRANSIENT TIME FOR PEAK VOLTAGE DROP form opens. The form displays the transient times when the maximum current passes through each selected parasitic resistor. This form is shown below.

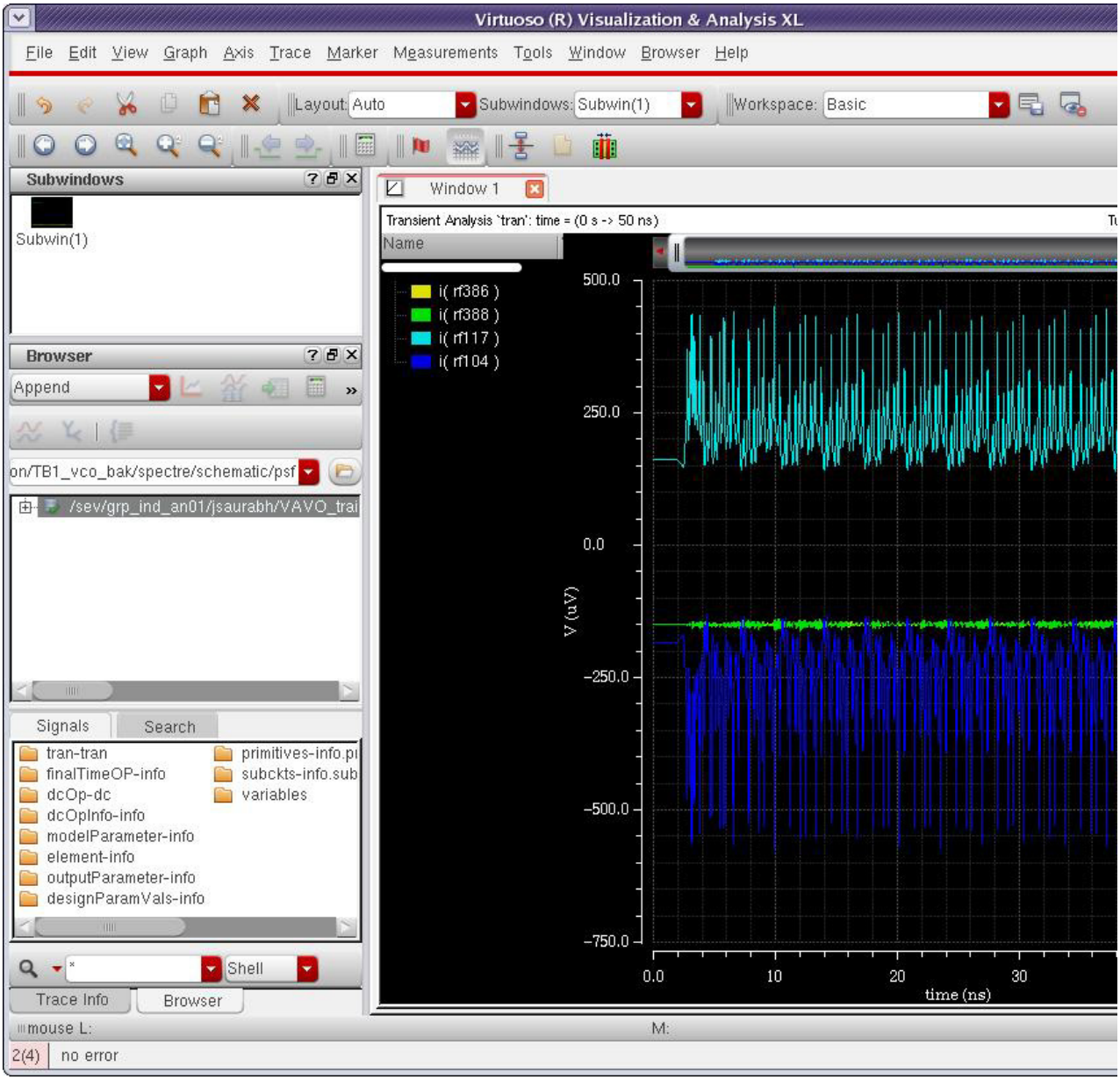
Figure 6-27 Opening the TRANSIENT TIME FOR PEAK VOLTAGE DROP Form



- Click *Select presistors Inside A Window* to highlight a region of the design and select the parasitic resistors in that region. The corresponding lines are highlighted in the Electromigration Analysis Results form. This option allows you to locate all presistors within a region.
- Click *Plot waveforms* to display the current waveform on the selected presistor in the Virtuoso (R) Visualization & Analysis XL window. This is shown in the figure below.

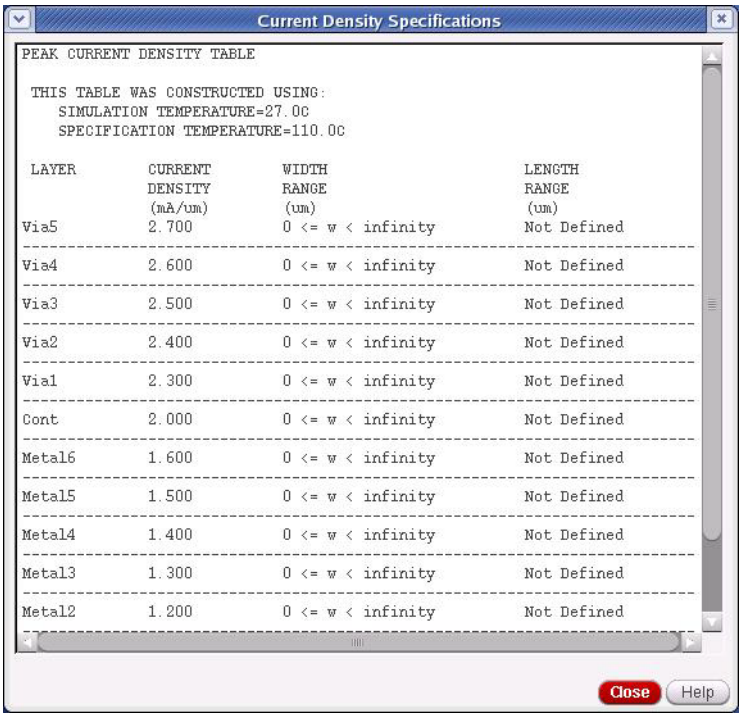
© 2020 All Rights Reserved.

Figure 6-28 Displaying Current Waveforms for Selected Presistors in the Virtuoso (R) Visualization & Analysis XL



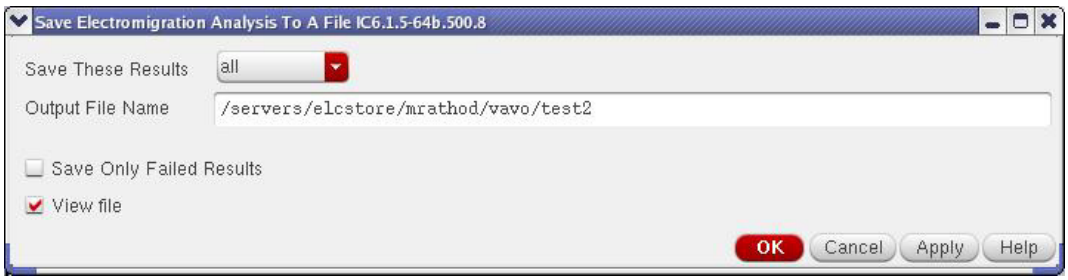
© 2020 All Rights Reserved.

- Click *View PDK Current Density Limits* to view the current density numbers from the `emDataFile`, as shown in the example below.



- Click *Save Results To A File* to save the EM analysis results. The Save Electromigration Analysis to A File form opens. This is shown below. In this form, you can click *Save Only Failed Results* to save EM analysis results of only the failed nets.

Figure 6-29 Saving Electromigration Analysis to A File



In this form:

- Specify the *Output File Name* in which you want to save the results.
- Click *Save Only Failed Results* option to save EM analysis results of only the failed nets.

© 2020 All Rights Reserved.

- Click *View file* to view the result file. This option is selected by default. If you do not want to view the result file, you can change the default selection of this option by setting the `_vsaUncheckViewFile` environment variable to `true`.

Customizing Error Layer Display for EM Analysis Results

In addition to viewing results as described above, Power IR/EM provides the option to customize EM analysis result displays.

Click *Plots* in the Electromigration Analysis Results form. The Plots form opens. In this form, you can customize the EM violation ranges 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.

Note: The forms for plotting IR drop and EM analysis results are similar. The options that are specific to the EM Plots are only detailed below.

- For details of all the available options that you can set in the Plots form, see the section titled, Customizing Error Layer Display for IR Drop Results, in the “IR Drop Analysis” chapter.

© 2020 All Rights Reserved.

Figure 6-30 Displaying Highlight Color Spectrum and Options for Customizing Display of EM Plots

- In this form, the following options are provided for the types of plots that can be displayed:
  - Current: displays the current violations for all or selected layers
  - Current Density: displays the current density violations for all or selected layers
  - Current Density Ratio: displays the current density ratio violations for all or selected layers

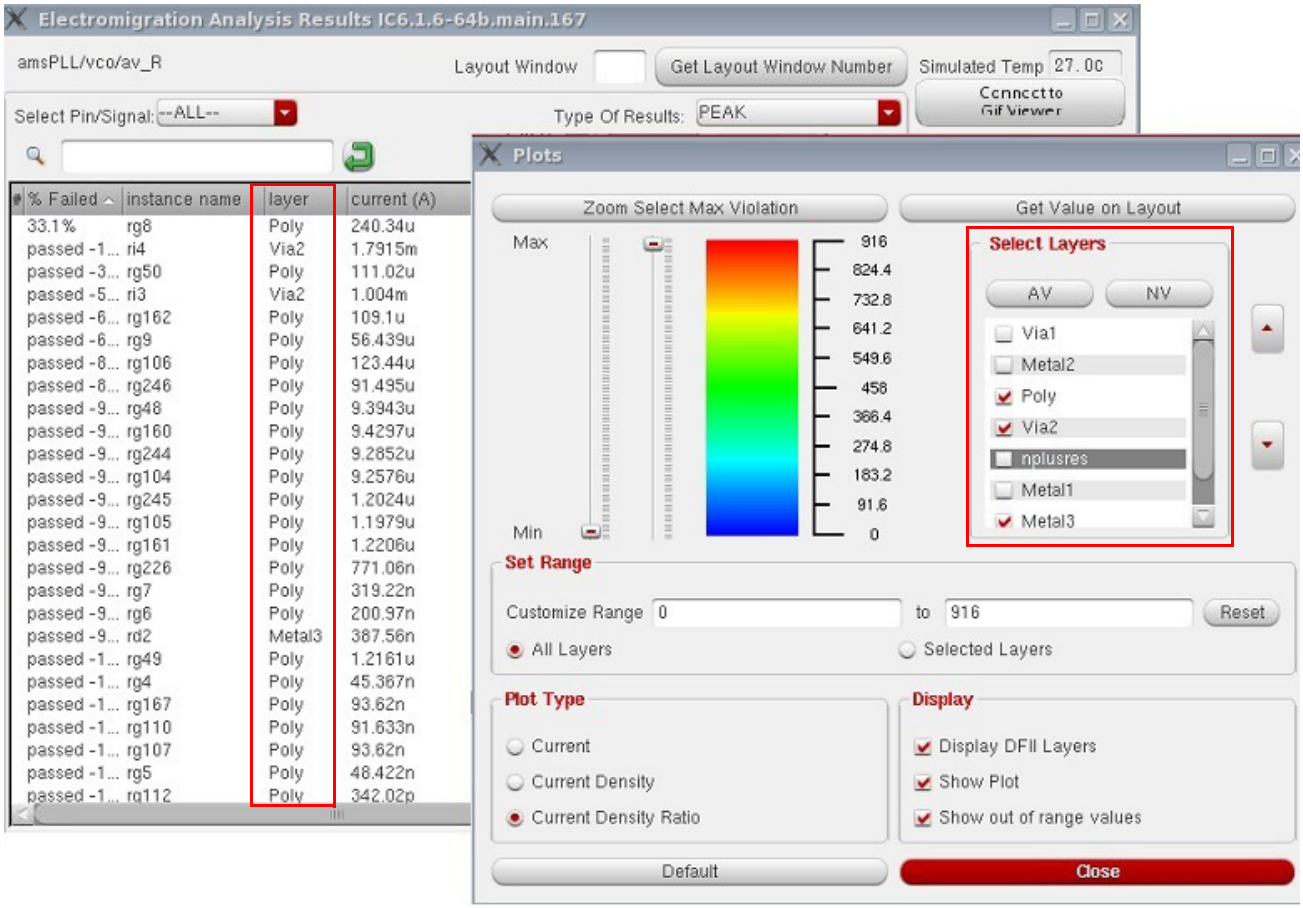
    Note: The Current Density Ratio plot is selected by default.

Important

When you update the layers for which you want to view plots in the *Select Layers* cyclic field, the layers listed in the text box in the Electromigration Analysis Results form are updated to display information for only those layers that are selected in this form.

© 2020 All Rights Reserved.

Figure 6-31 Layer Selection in Plots Form Updated in Results Form



Note: For detailed descriptions of all the other fields and options available in the Plots form shown above, see the section titled, Customizing Error Layer Display for IR Drop Results, in the “IR Drop Analysis” chapter.

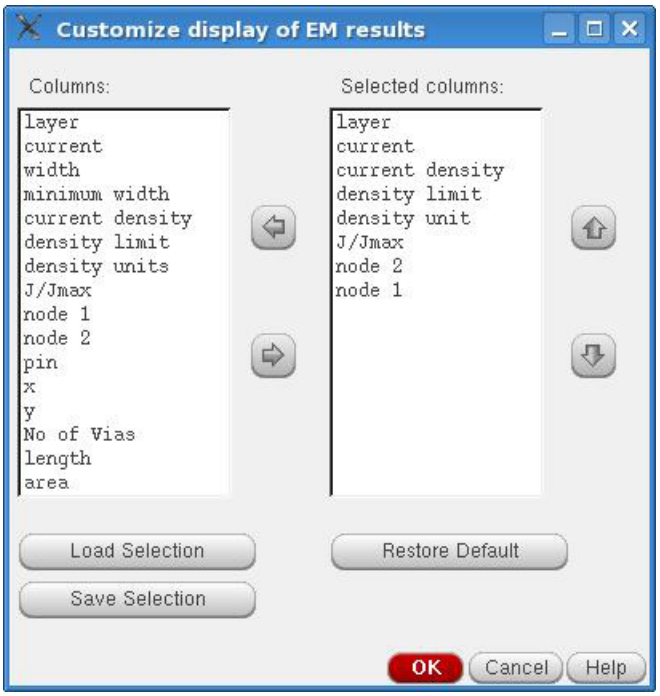
- The last state for *All Layers* is saved for each plot type and switching back to the previous plot type when *All Layers* is selected will restore the last state.

Customizing the EM Analysis Results Display

The information displayed in the list box of the Electromigration Analysis Results form can be customized to meet your specific needs, or display the information most important to you. In the form, click *Customized results display*. The Customize display of EM results form opens. This form is shown below.

© 2020 All Rights Reserved.

Figure 6-32 Customize display of EM results Form



This form lets you customize the information displayed in the Electromigration Analysis Results form. Any number of columns can be displayed in the result form.

- Select the items in the *Columns* field and select the right arrow to add the items in the *Selected columns* field. Use the left arrow to remove items from the *Selected columns* list.
- Click *Restore Default* to restore the default selected columns information in the form.
- Click *Save Selection* to save your customized selected columns, configuration to a file to be used later. The Save EM Result Column View To A File form opens.

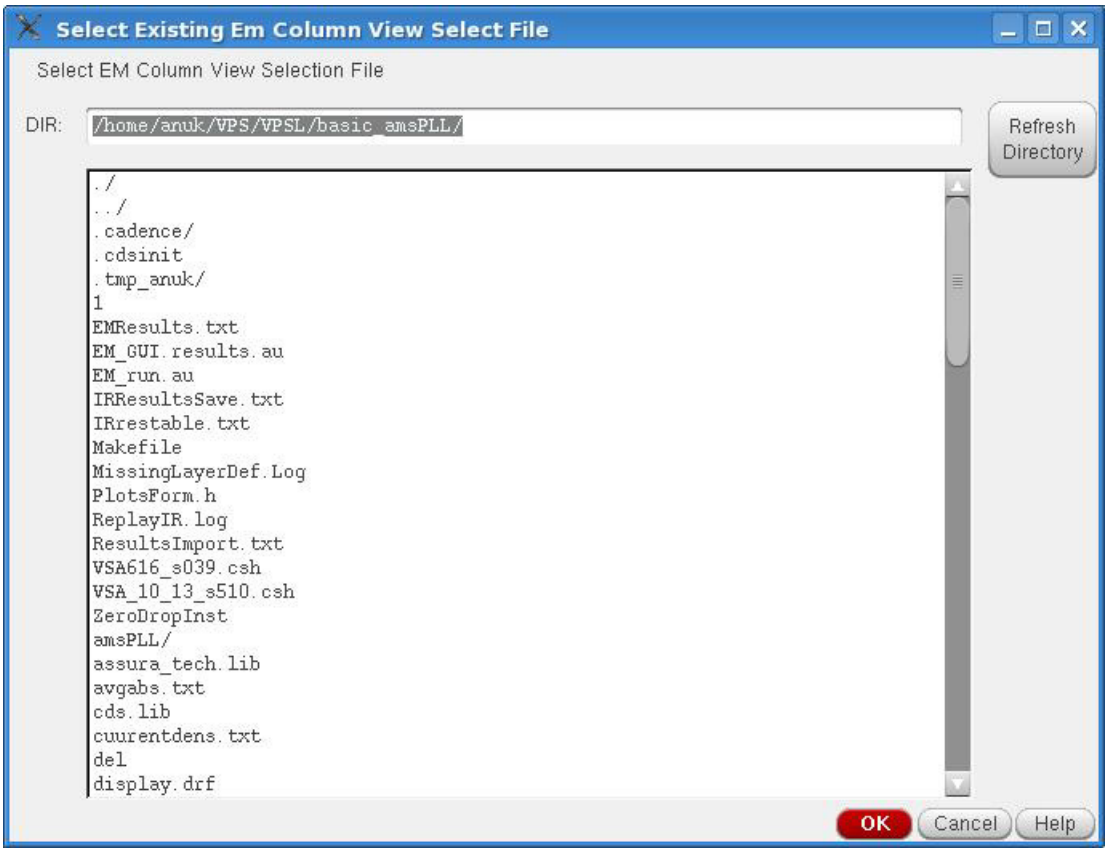
Figure 6-33 Saving the Customized Selection

- In this form, specify the *Output File Name* and click *OK*.
- Click *Load Selection* to select a configuration file with the selected columns information saved earlier. The Select Existing Em Column View Select File form opens.

© 2020 All Rights Reserved.

Figure 6-34 Selecting Existing EM Column View Select File

■

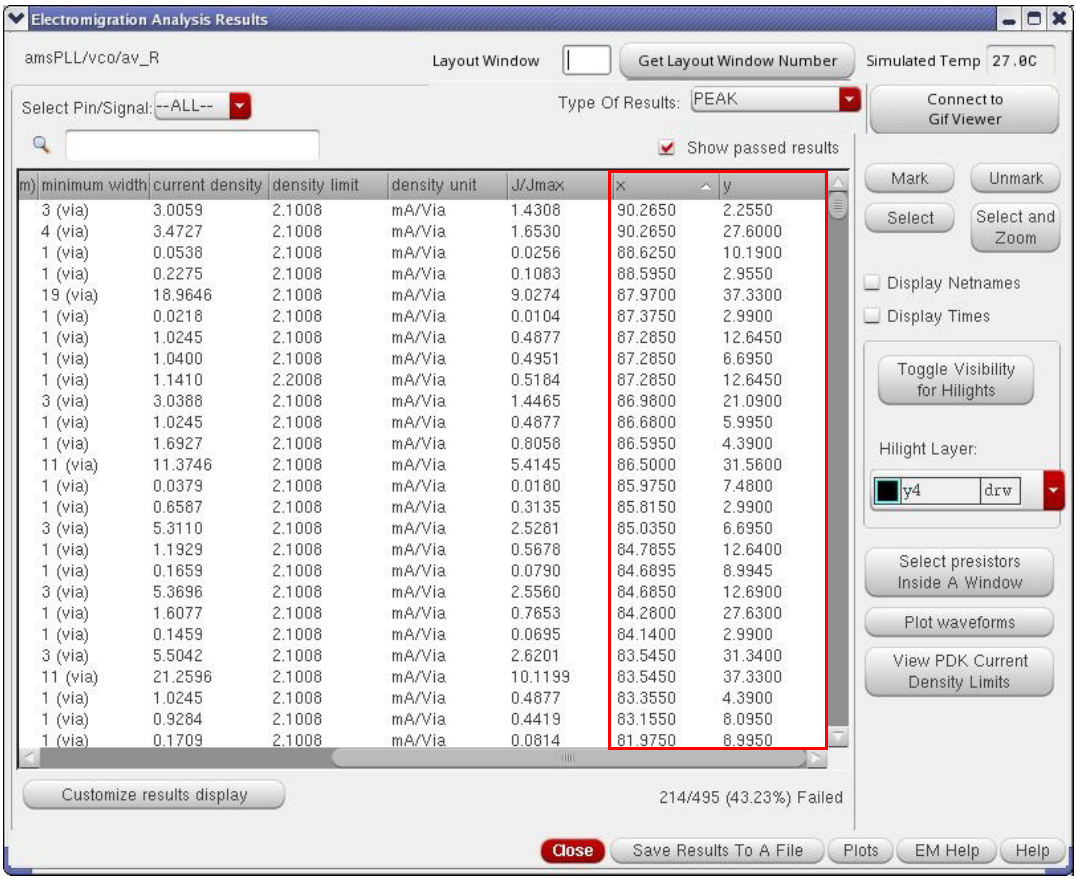


■ In this form, select the file and click *OK*.

Note: The results of adding x and y coordinates can be viewed in the figure below. The x and y coordinates are always saved in the EM results file.

© 2020 All Rights Reserved.

Figure 6-35 Customized EM Results Displaying x and y Coordinates



m)	minimum width	current density	density limit	density unit	J/Jmax	x	y
	3 (via)	3.0059	2.1008	mA/Via	1.4308	90.2650	2.2550
	4 (via)	3.4727	2.1008	mA/Via	1.6530	90.2650	27.6000
	1 (via)	0.0538	2.1008	mA/Via	0.0256	88.6250	10.1900
	1 (via)	0.2275	2.1008	mA/Via	0.1083	88.5950	2.9550
	19 (via)	18.9646	2.1008	mA/Via	9.0274	87.9700	37.3300
	1 (via)	0.0218	2.1008	mA/Via	0.0104	87.3750	2.9900
	1 (via)	1.0245	2.1008	mA/Via	0.4877	87.2850	12.6450
	1 (via)	1.0400	2.1008	mA/Via	0.4951	87.2850	6.6950
	1 (via)	1.1410	2.2008	mA/Via	0.5184	87.2850	12.6450
	3 (via)	3.0388	2.1008	mA/Via	1.4465	86.9800	21.0900
	1 (via)	1.0245	2.1008	mA/Via	0.4877	86.6800	5.9950
	1 (via)	1.6927	2.1008	mA/Via	0.8058	86.5950	4.3900
	11 (via)	11.3746	2.1008	mA/Via	5.4145	86.5000	31.5600
	1 (via)	0.0379	2.1008	mA/Via	0.0180	85.9750	7.4800
	1 (via)	0.6587	2.1008	mA/Via	0.3135	85.8150	2.9900
	3 (via)	5.3110	2.1008	mA/Via	2.5281	85.0350	6.6950
	1 (via)	1.1929	2.1008	mA/Via	0.5678	84.7855	12.6400
	1 (via)	0.1659	2.1008	mA/Via	0.0790	84.6895	8.9945
	3 (via)	5.3696	2.1008	mA/Via	2.5560	84.6850	12.6900
	1 (via)	1.6077	2.1008	mA/Via	0.7653	84.2800	27.6300
	1 (via)	0.1459	2.1008	mA/Via	0.0695	84.1400	2.9900
	3 (via)	5.5042	2.1008	mA/Via	2.6201	83.5450	31.3400
	11 (via)	21.2596	2.1008	mA/Via	10.1199	83.5450	37.3300
	1 (via)	1.0245	2.1008	mA/Via	0.4877	83.3550	4.3900
	1 (via)	0.9284	2.1008	mA/Via	0.4419	83.1550	8.0950
	1 (via)	0.1709	2.1008	mA/Via	0.0814	81.9750	8.9950

By default, for displaying column information in the Electromigration Analysis Results form, the software searches for the column configuration file at the following locations, in the order in which they are listed below:

- The configuration file specified using the `_vsaEMColumnViewConfigFile` environment variable
- The `emColumnView.cfg` file in the current working directory
- The `emColumnView.cfg` file in SKILL paths

If the file is not available in any of the above three locations, the software uses the configuration from the previous customized display.

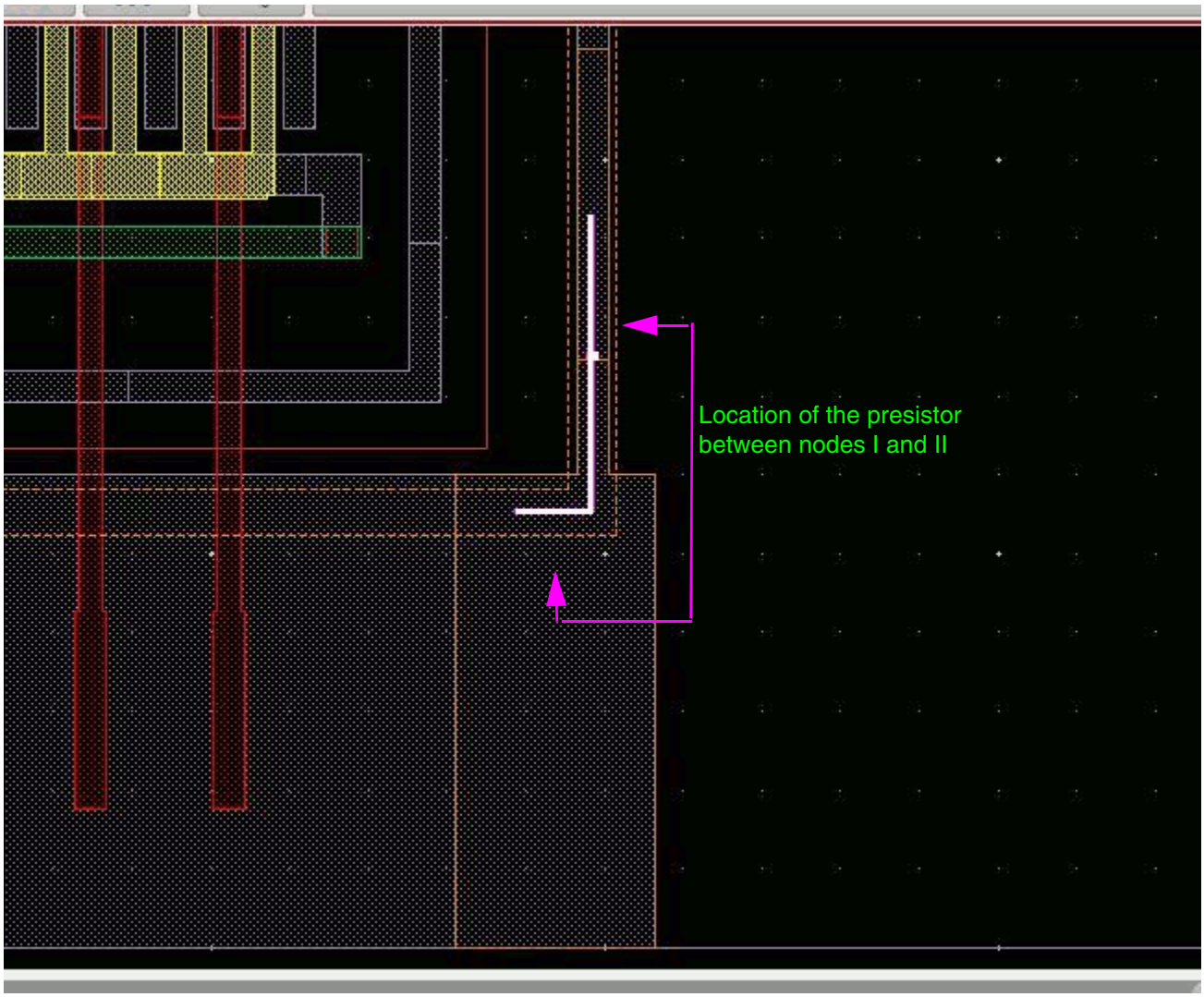
Using EM Analysis Results for Cross-Probing in Power IR/EM

The node 1 and node 2 columns in the *EM Analysis Results* screen represent the actual nodes of the parasitic resistor. These are used for Power IR/EM cross -probing wherein Power

© 2020 All Rights Reserved.

IR/EM displays the location of the parasitic resistor as a white line in the extracted view. This is shown below.

Figure 6-36 Virtuoso Layout Suite - Displaying the Location of the Presistor



© 2020 All Rights Reserved.

7

Viewing Results in vsaPlot

- Overview on page 148
- Launching vsaPlot on page 148
  - Launching vsaPlot from the Shell on page 148
  - Launching vsaPlot from Power IR/EM on page 149
- Viewing Results in vsaPlot on page 152
  - Main Menus of vsaPlot on page 154
  - Toolbar Widgets on page 161
  - Main Components of vsaPlot on page 164
- Querying IR Drop and EM Results on page 170
  - Zooming into the Worst Violation in the Selected Area on page 170
  - Retrieving the Values of All the Violations at a Specified Point on the Layout on page 171
  - Displaying the Colormap for Point-to-Point and Region-to-Region Selection on page 172
- Synchronizing Multiple vsaPlot Windows on page 173
  - Opening a Sub-Window to Initiate Synchronism on page 174
  - Synchronizing Options on page 174
- Synchronizing vsaPlot with the Layout or Extracted View on page 176

© 2020 All Rights Reserved.

Overview

vsaPlot is used to view the results of the IR drop and EM analysis that is performed in Power IR/EM. You can access vsaPlot both from within and outside the Virtuoso environment. Both these methods of accessing the tool are described in subsequent sections.

The tool 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 environment**: 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. This is beneficial for those who do not have Virtuoso.
2. **Increased speed of visualization**: 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.

Launching vsaPlot

You can launch vsaPlot in two ways, from the *VPS-L* menu, or directly from the UNIX, shell, or xterm window.

- Launching vsaPlot from the Shell
- Launching vsaPlot from Power IR/EM

Launching vsaPlot from the Shell

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

```
vsaplot <Result_DB>
```

Where:

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

© 2020 All Rights Reserved.

Note: You can also run vsaPlot by typing just `vsaplot` 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 Power IR/EM

You can launch vsaPlot from Power IR/EM GUI in the following ways:

1. From the *Connect to VsaPlot* submenu of the VPS-L Main 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.

Figure 7-1 Launching vsaPlot from VPS-L Main Menu

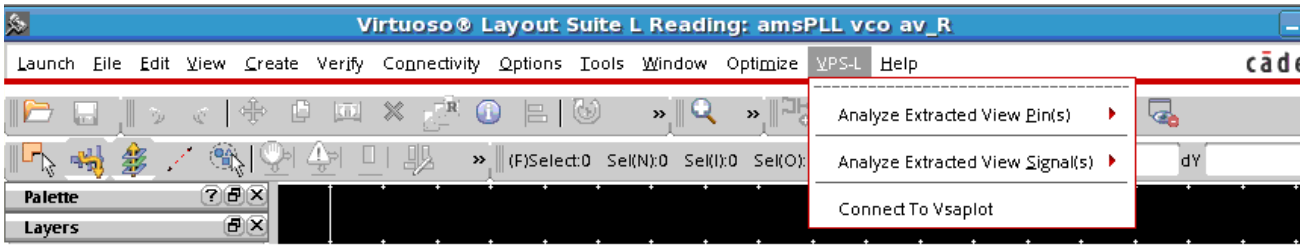
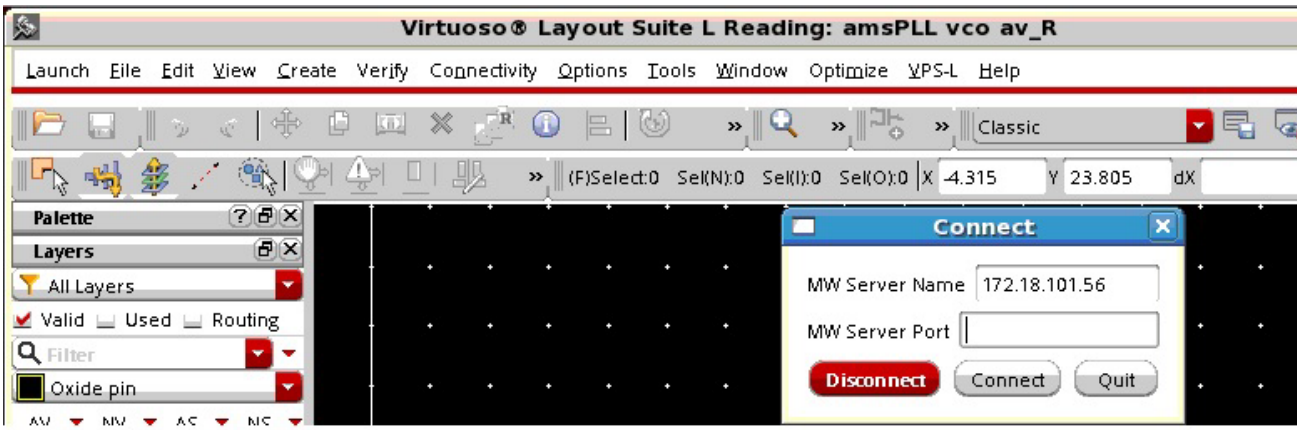


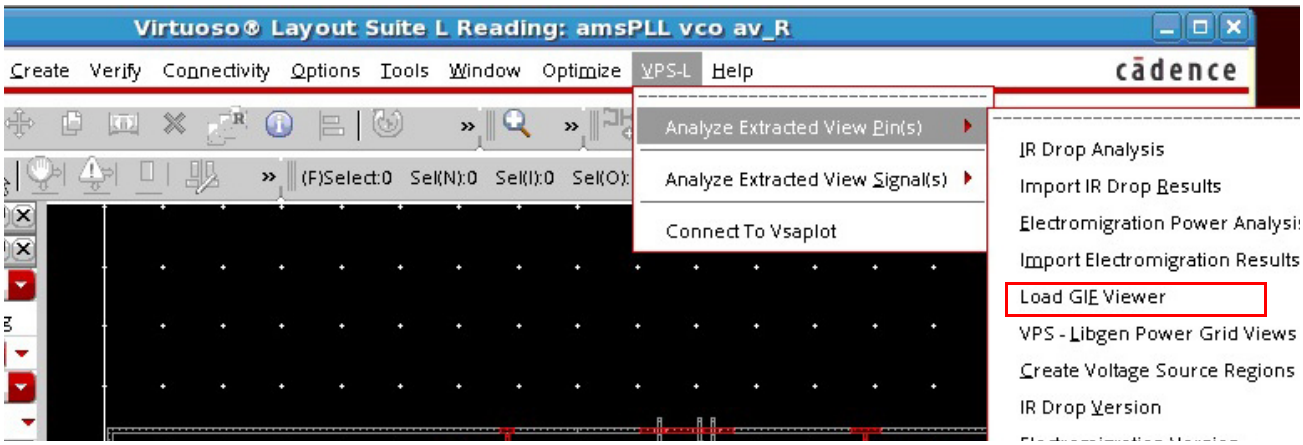
Figure 7-2 Opening the Connect Pop-Up Window



2. From the *Load GIF Viewer* submenu in the *Analyze Extracted View Pin(s)/ Signal(s)* menus: When the vsaPlot window opens, you will need to load the analysis results by specifying the results directory. This is shown in the figure below.

© 2020 All Rights Reserved.

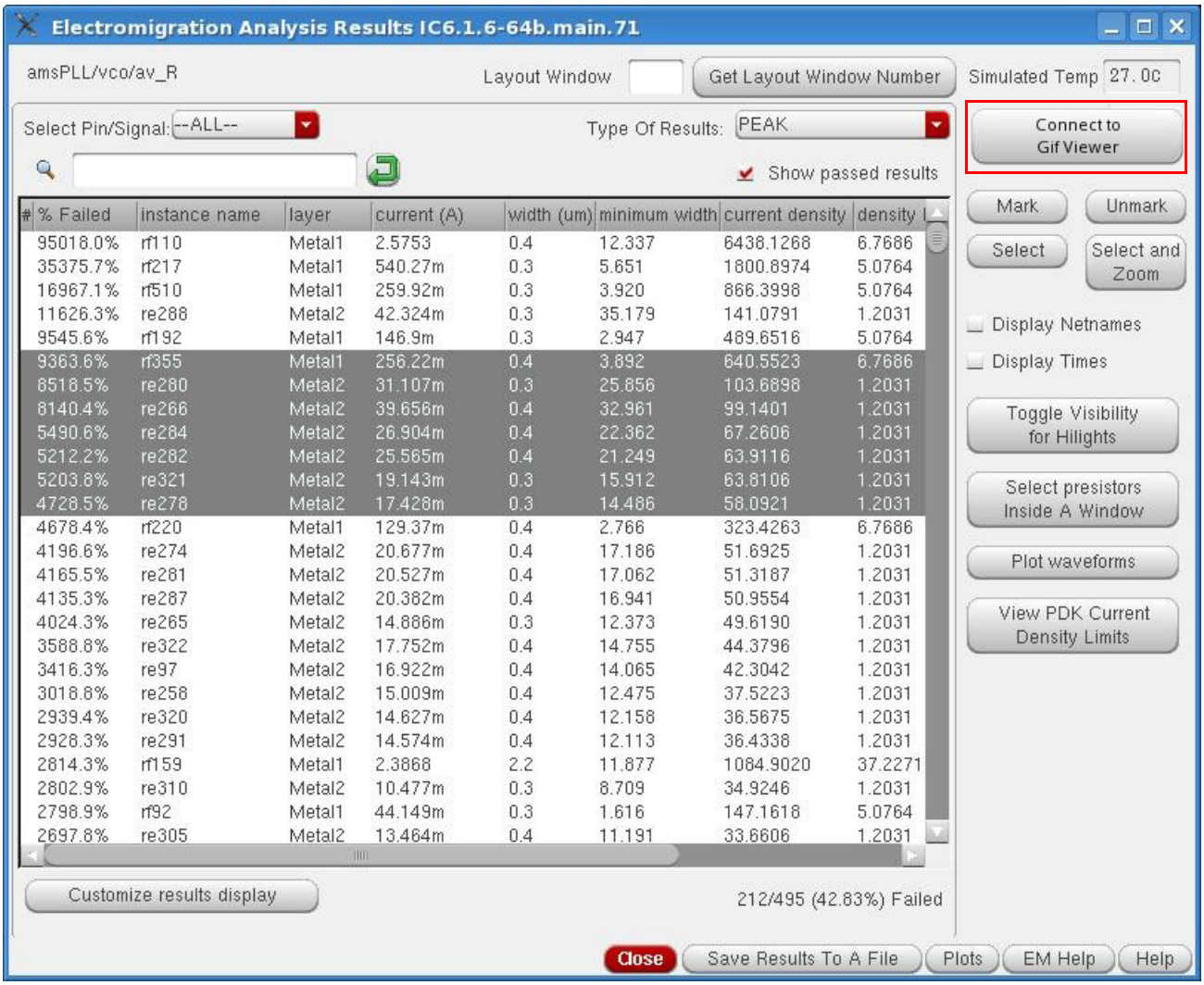
Figure 7-3 Launching vsaPlot from Analyze Extracted View Pins/Signal (s) Form



3. From the *Load Gif Viewer* option provided in the Electromigration Analysis Results form. The vsaPlot window opens. This is shown below.

© 2020 All Rights Reserved.

Figure 7-4 Launching vsaPlot from the EM Results Form

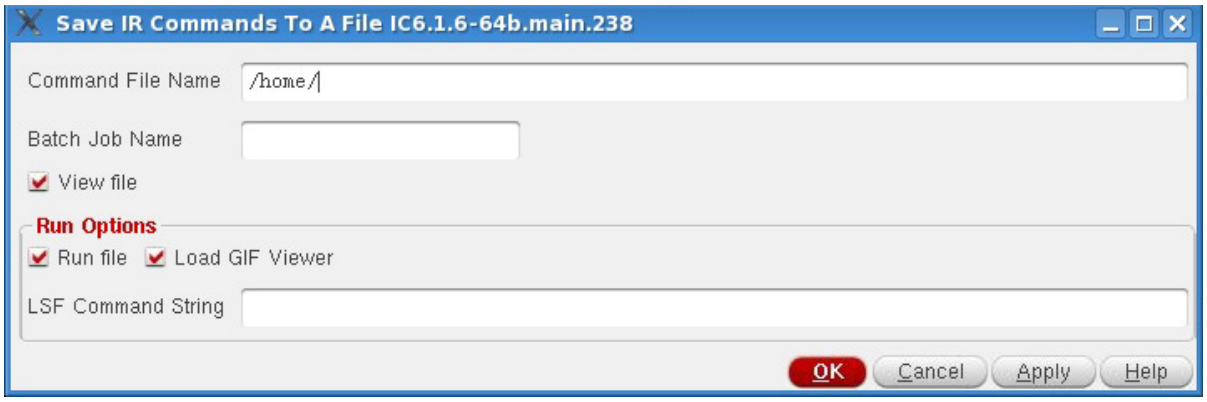


#	% Failed	instance name	layer	current (A)	width (um)	minimum width	current density	density
	95018.0%	rf110	Metal1	2.5753	0.4	12.337	6438.1268	6.7686
	35375.7%	rf217	Metal1	540.27m	0.3	5.651	1800.8974	5.0764
	16967.1%	rf510	Metal1	259.92m	0.3	3.920	866.3998	5.0764
	11626.3%	re288	Metal2	42.324m	0.3	35.179	141.0791	1.2031
	9545.6%	rf192	Metal1	146.9m	0.3	2.947	489.6516	5.0764
	9363.6%	rf355	Metal1	256.22m	0.4	3.892	640.5523	6.7686
	8518.5%	re280	Metal2	31.107m	0.3	25.856	103.6898	1.2031
	8140.4%	re266	Metal2	39.656m	0.4	32.961	99.1401	1.2031
	5490.6%	re284	Metal2	26.904m	0.4	22.362	67.2606	1.2031
	5212.2%	re282	Metal2	25.565m	0.4	21.249	63.9116	1.2031
	5203.8%	re321	Metal2	19.143m	0.3	15.912	63.8106	1.2031
	4728.5%	re278	Metal2	17.428m	0.3	14.486	58.0921	1.2031
	4678.4%	rf220	Metal1	129.37m	0.4	2.766	323.4263	6.7686
	4196.6%	re274	Metal2	20.677m	0.4	17.186	51.6925	1.2031
	4165.5%	re281	Metal2	20.527m	0.4	17.062	51.3187	1.2031
	4135.3%	re287	Metal2	20.382m	0.4	16.941	50.9554	1.2031
	4024.3%	re265	Metal2	14.886m	0.3	12.373	49.6190	1.2031
	3588.8%	re322	Metal2	17.752m	0.4	14.755	44.3796	1.2031
	3416.3%	re97	Metal2	16.922m	0.4	14.065	42.3042	1.2031
	3018.8%	re258	Metal2	15.009m	0.4	12.475	37.5223	1.2031
	2939.4%	re320	Metal2	14.627m	0.4	12.158	36.5675	1.2031
	2928.3%	re291	Metal2	14.574m	0.4	12.113	36.4338	1.2031
	2814.3%	rf159	Metal1	2.3868	2.2	11.877	1084.9020	37.2271
	2802.9%	re310	Metal2	10.477m	0.3	8.709	34.9246	1.2031
	2798.9%	rf92	Metal1	44.149m	0.3	1.616	147.1618	5.0764
	2697.8%	re305	Metal2	13.464m	0.4	11.191	33.6606	1.2031

4. From the *Load GIF Viewer* option provided in the Save IR Commands To a File and Save Selected EM Commands to a File forms: When the vsaPlot window opens, the results of the IR drop/EM analysis will be automatically loaded. This is shown in the figure below.

© 2020 All Rights Reserved.

Figure 7-5 Launching vsaPlot from the Save IR/EM Commands To A File Form



Viewing Results in vsaPlot

To view results of IR drop and EM analysis in vsaPlot, first specify the results directory. When you perform IR drop and EM analysis in Power IR/EM, the results of the analyses are stored in the `VSA_RESULTS` directory, which is located in the current directory, by default. If, however, the results directory is not located in the default location, you can specify the path of the result directory by using the following environment variable:

```
setenv _vsaDumpResultDBPath path
```

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.

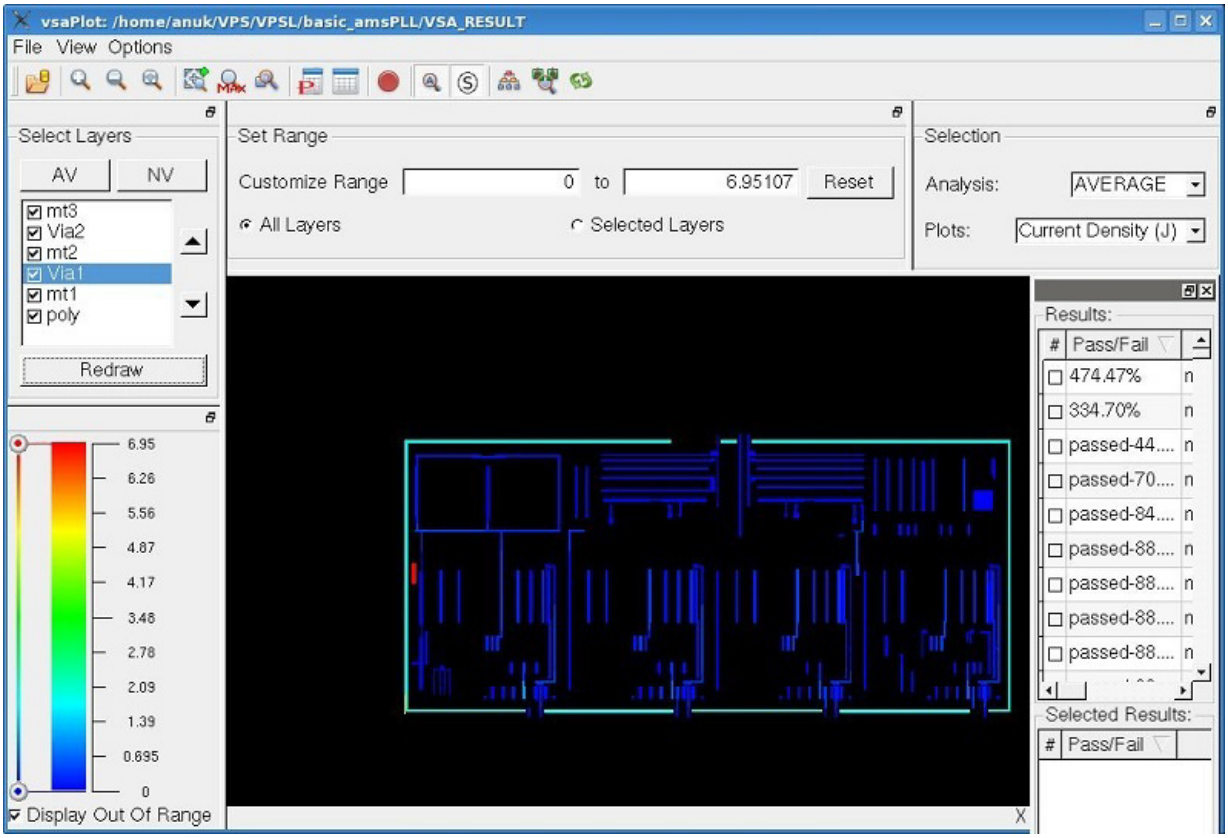
The following topics are covered in this section:

- Main Menus of vsaPlot
- Toolbar Widgets
- Main Components of vsaPlot

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

© 2020 All Rights Reserved.

Figure 7-6 The vsaPlot Window



© 2020 All Rights Reserved.

Main Menus of vsaPlot

There are three main menus in vsaPlot.

- File
- View
- Options

These menus are detailed in subsequent sections.

File Menu

The File menu has the following submenus:

- Open
- Sync
- Save
- Quit

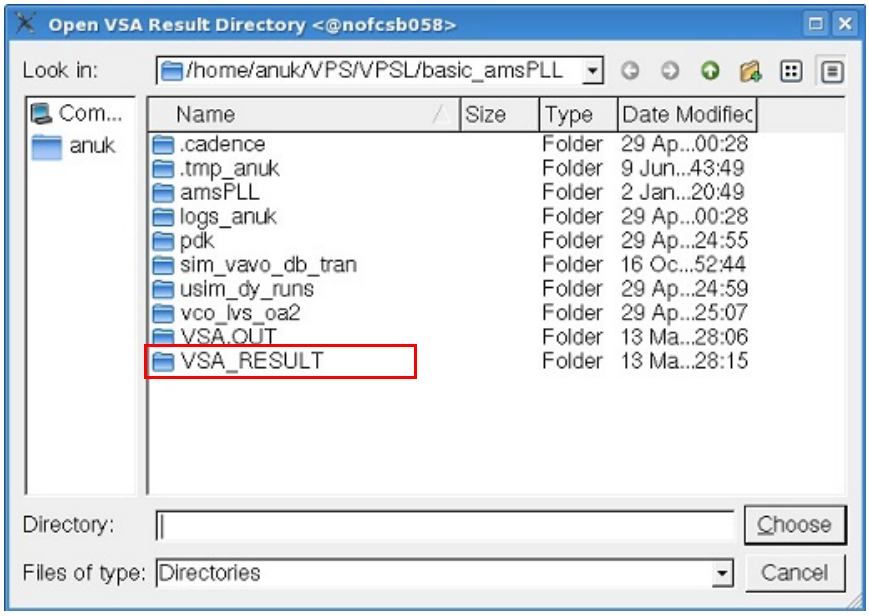
These submenus are detailed below.

- 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*. This is shown below.

    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>` command in the shell.

© 2020 All Rights Reserved.

Figure 7-7 The Open VSA Result Directory Form

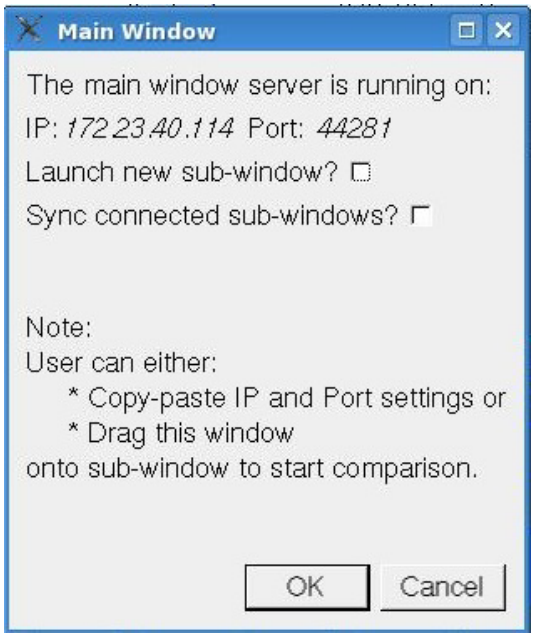


The extracted view of the design will open in the display area or the layout area of vsaPlot.

- *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 that lets you launch a sub-window and synchronize connected windows. This is shown below.

© 2020 All Rights Reserved.

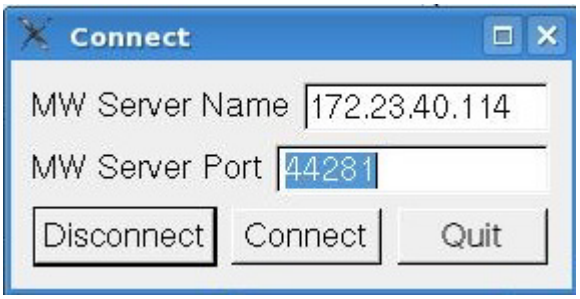
Figure 7-8 The Main Window



- *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:. The following options are available:
  - *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.

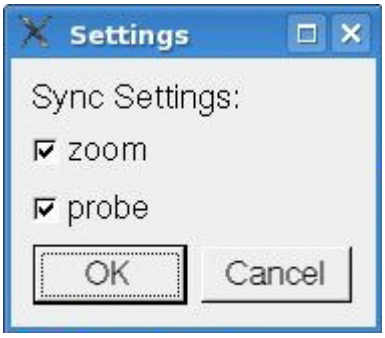
Figure 7-9 The Connect Window



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

© 2020 All Rights Reserved.

Figure 7-10 The Settings Window



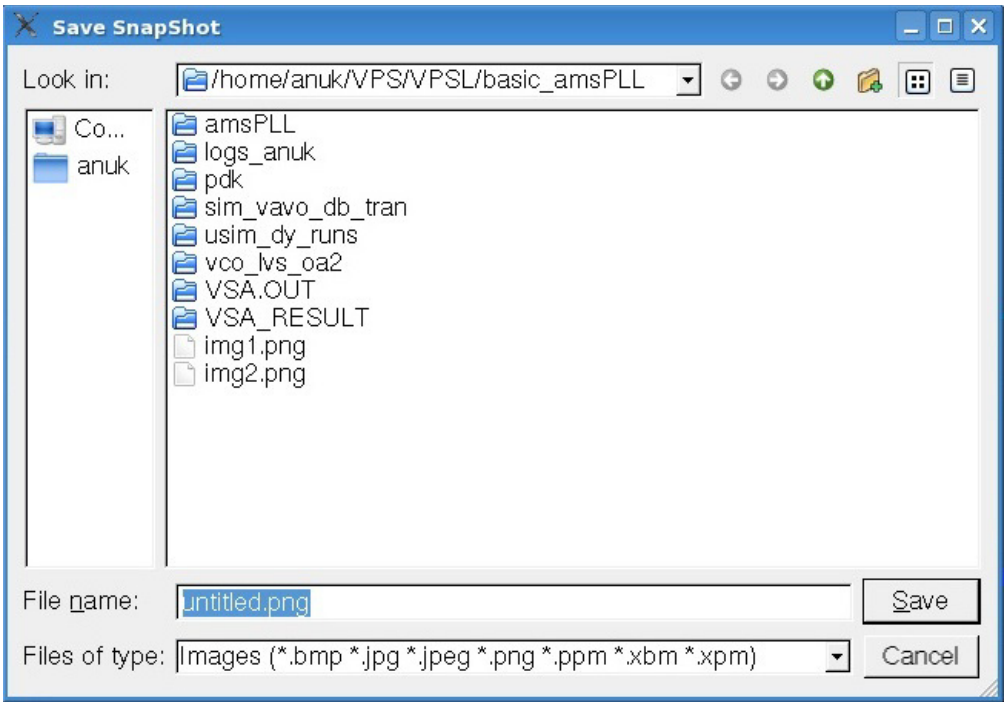
For details of the synchronism function in vsaPlot, see Synchronizing Multiple vsaPlot Windows.

- 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.

© 2020 All Rights Reserved.

Figure 7-11 The Save Snapshot Form



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

View Menu

The *View* menu has the following submenus:

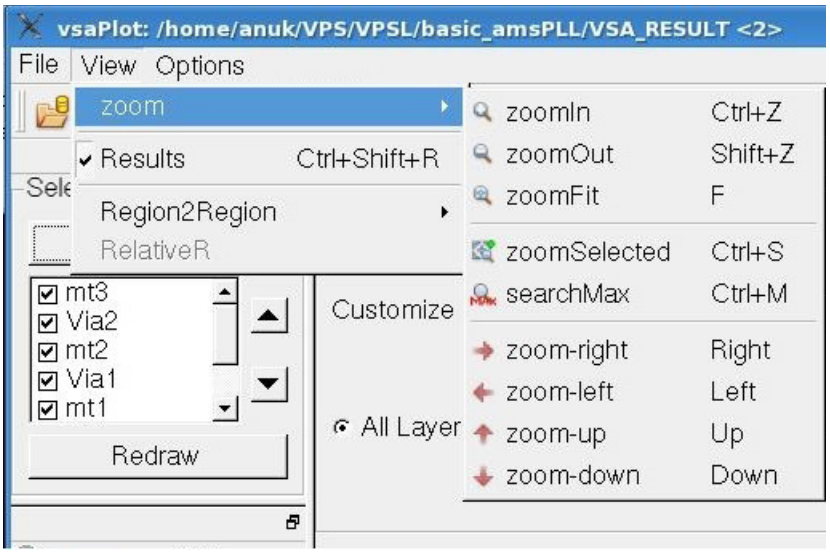
- zoom
- Results
- Region2Region

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.

© 2020 All Rights Reserved.

Figure 7-12 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 of your mouse 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 of your mouse 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.

© 2020 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 on the right side of 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.

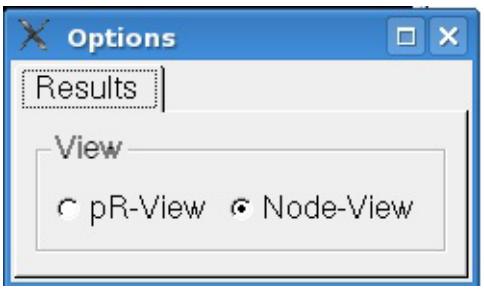
  For details, see Displaying the Colormap for Point-to-Point and Region-to-Region Selection.

  Note: This option is enabled only if a particular net is specified in the *Net(s):* field. It is disabled if ALL nets are selected.

Options Menu

When you click the *Options* menu, the Options form opens. This 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. This is because the Options form lets you specify whether you want to view the analysis results in the pR-View (presistor view) or the Node-View (node view), and IR drop violations only have a node view. Therefore, this form is not relevant for IR drop analysis results. This form is shown below.

Figure 7-13 The Options form

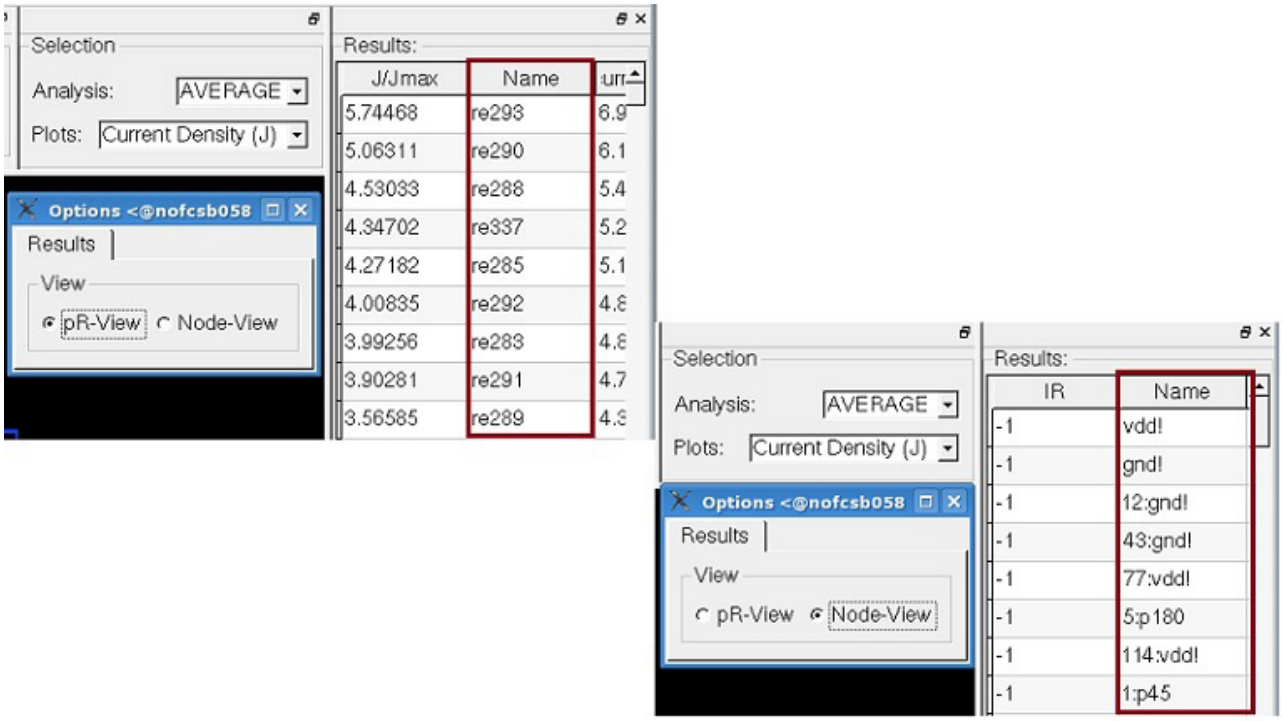


In this form, you can specify whether you want to view the analysis results in the pR-View (presistor view) or in the Node-View (node view). By default, the Results window displays results in the node view.

© 2020 All Rights Reserved.

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.

Figure 7-14 Viewing Results in Presistor View or Node View



Toolbar Widgets

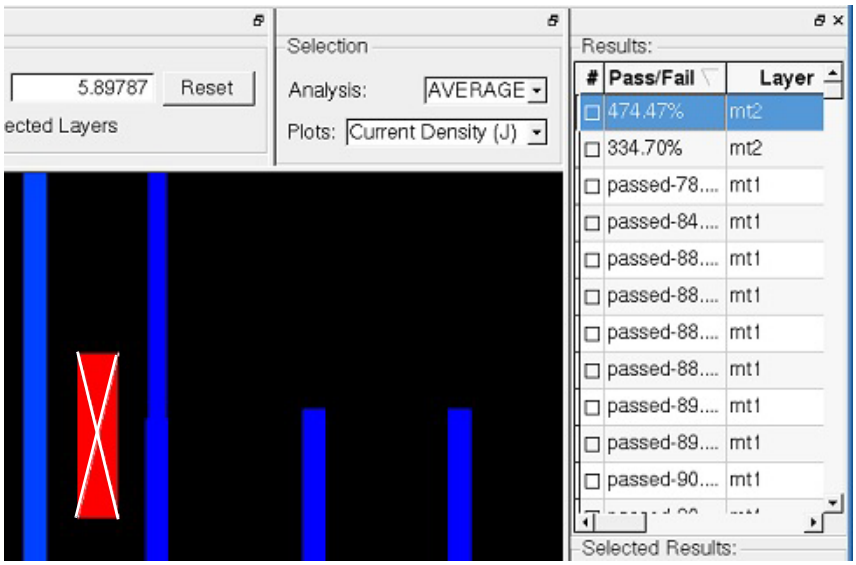
The following row of widgets, located below the menus and above the design display area, includes shortcuts for the menu options detailed in the above section.

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

© 2020 All Rights Reserved.

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. For details, see Displaying the Colormap for Point-to-Point and Region-to-Region Selection.
	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 columns are displayed. This is shown in Figure 7-21 on page 170. By default, the Results window opens in partial view.
	Open/Close Results – toggles the Results window.

© 2020 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. This is 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.

© 2020 All Rights Reserved.

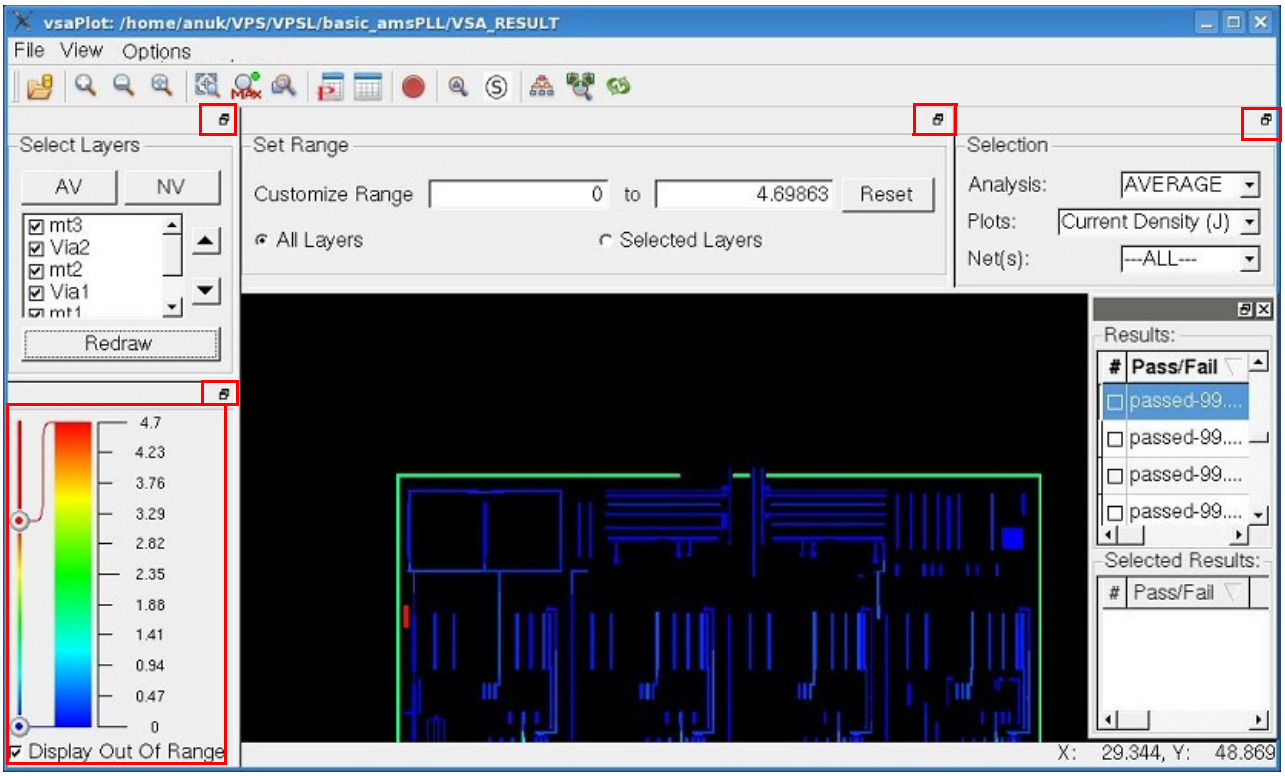
Main Components of vsaPlot

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 7-15 Main Components of vsaPlot



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

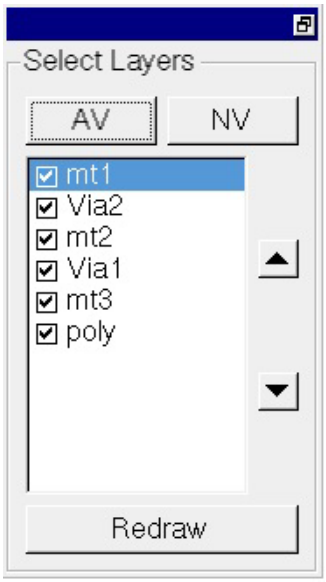
The sub-windows are described in detail below.

© 2020 All Rights Reserved.

Selecting the Layers

The *Select Layers* window is shown below.

Figure 7-16 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.

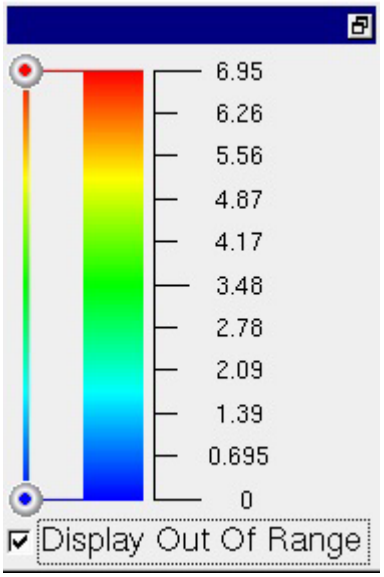
Specifying the Slider Range

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

© 2020 All Rights Reserved.

values for all or selected layers is provided next to the color spectrum. This is shown in the figure below.

Figure 7-17 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.

- 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

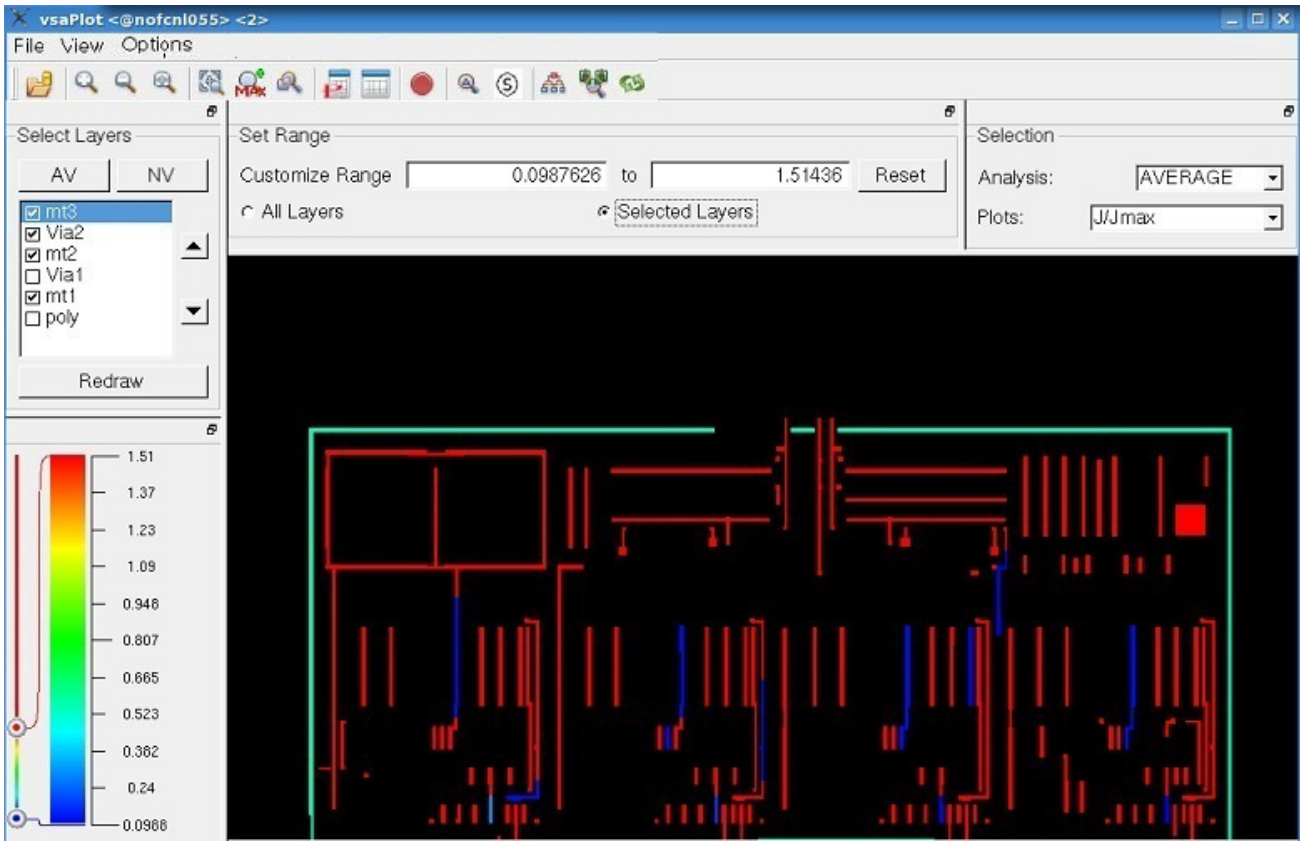
© 2020 All Rights Reserved.

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 current density violations for the selected layers within the specified slider range.

Figure 7-18 Customizing the Slider Range for Viewing Current Density Violations for the Selected Layers



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

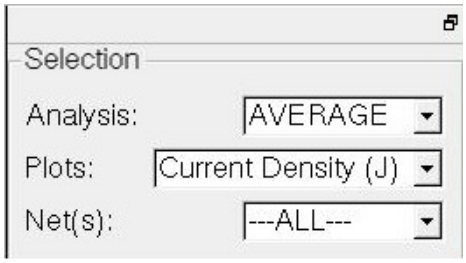
- *Analysis:* Types of analyses that can be performed
- *Plots:* Types of plots available for the the different analysis types

© 2020 All Rights Reserved.

- *Net(s):* Nets on which the analysis is to be performed.

These are shown in the figure below.

Figure 7-19 The Selection Window



The following analysis types are available in the Analysis cyclic field:

- AVERAGE: Displays the current density violations based on the average value of the DC current for metal lines, vias, and contacts.
- PEAK: Displays the current density violations based on the maximum DC current for metal lines, vias, and contacts.

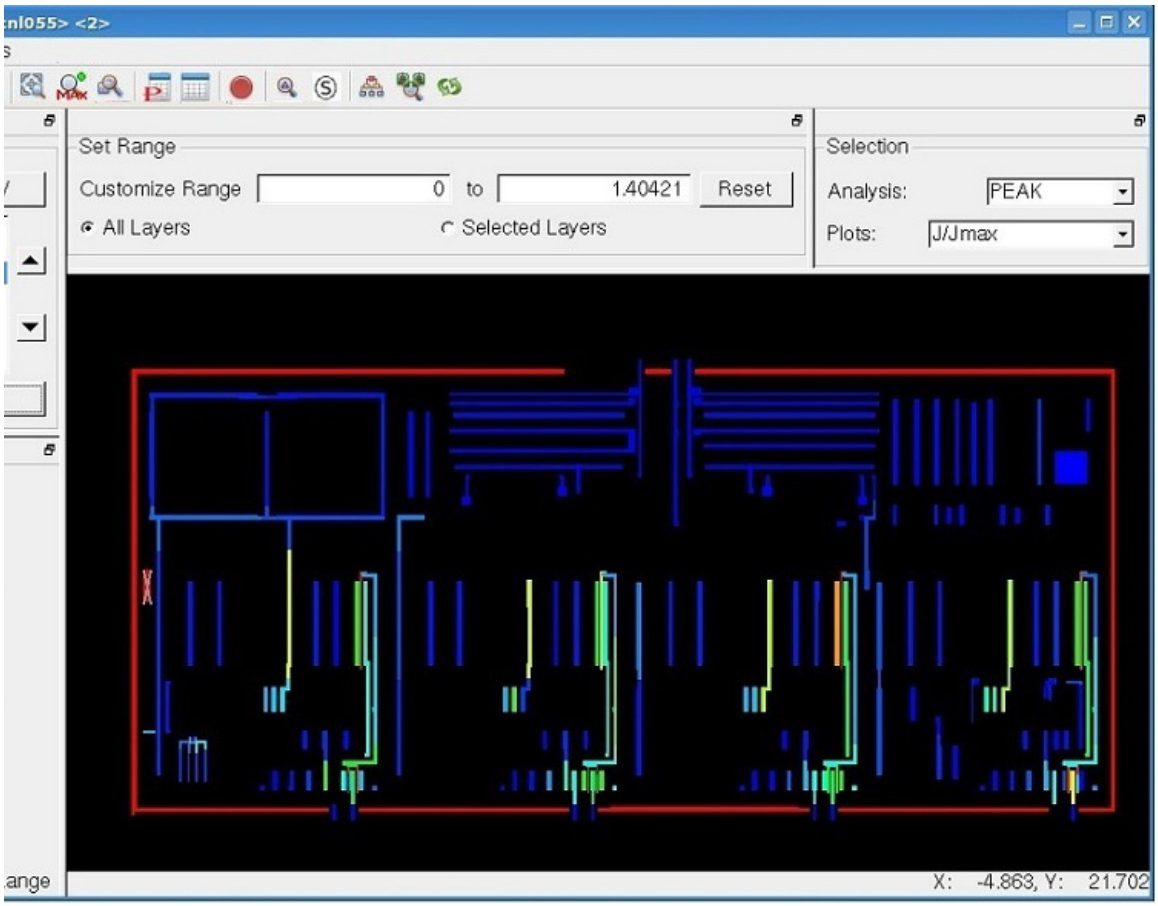
The following types of plots are available for the above analyses in the Plots cyclic field:

- 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 peak current for all layers is shown in the figure below. The range of violations is customized using the slider.

© 2020 All Rights Reserved.

Figure 7-20 Viewing Current Density Violations for the Peak Current for All 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 Pass/Fail, Layer, and 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.

© 2020 All Rights Reserved.

Figure 7-21 The Results Window Displaying Full Results in pR-View

Pass/Fail	Layer	J/Jmax	Name	entDensity (mA/	Current (A)	Density Limit	R(ohms)	Width(um)	MinWidth(
ssed-41.06%	V2	0.589368	rnb15512	0.0777966	0.000388983	0.132	0.12	5	2.947
ssed-41.07%	CS_P_D_40031	0.589285	rbb9572	0.0777856	7.0007e-05	0.132	1	0.9	0.53
ssed-41.08%	V2	0.589226	rnb14820	0.0777778	0.000388889	0.132	0.12	5	2.946
ssed-41.08%	V2	0.589168	rnb14548	0.0777702	0.000388851	0.132	0.12	5	2.946
ssed-41.09%	V2	0.589092	rnb14308	0.0777602	0.000388801	0.132	0.12	5	2.945
ssed-41.11%	V2	0.588947	rnb15756	0.077741	0.000388705	0.132	0.12	5	2.945

Tip

You can save your markings in the Results window. Check the 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 in vsaPlot.

- Zooming into the Worst Violation in the Selected Area
- Retrieving the Values of All the Violations at a Specified Point on the Layout
- Displaying the Colormap for Point-to-Point and Region-to-Region Selection

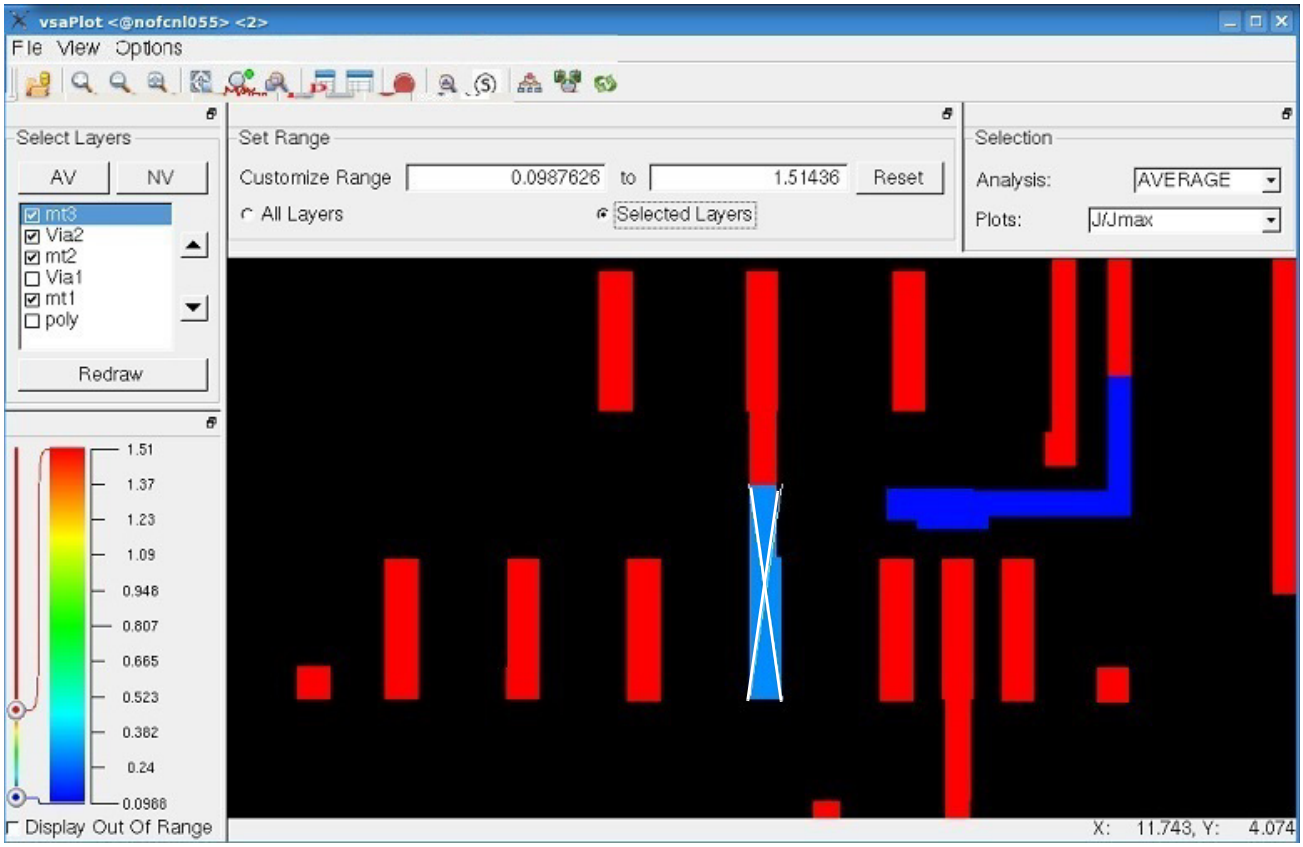
Zooming into the Worst Violation in the Selected Area

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. This is shown below.

© 2020 All Rights Reserved.

Figure 7-22 Zooming into the Worst Violation in the Selected Area

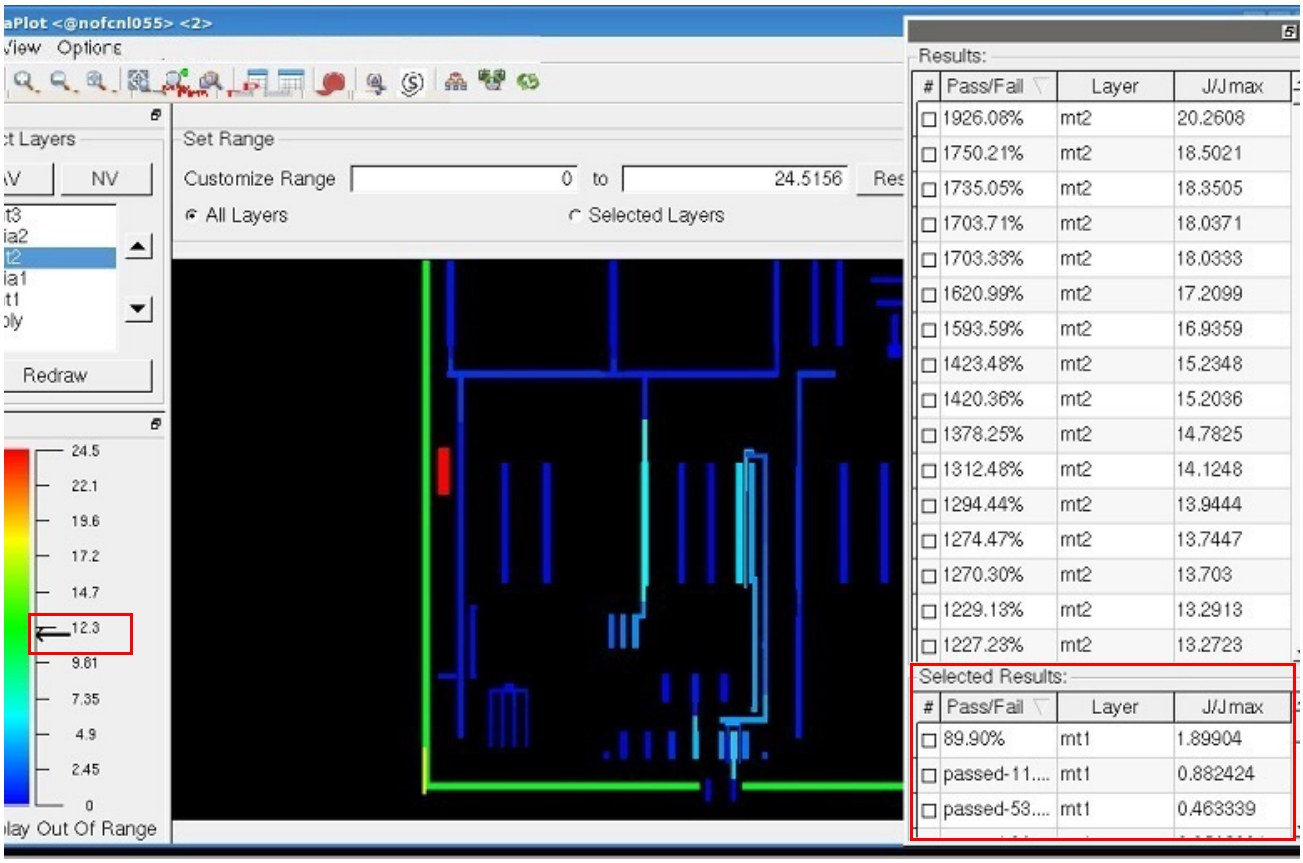


Retrieving the Values of All the Violations at a Specified Point on the Layout

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. This is shown below.

© 2020 All Rights Reserved.

Figure 7-23 Retrieving the Values of the Violations at a Specified Point



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

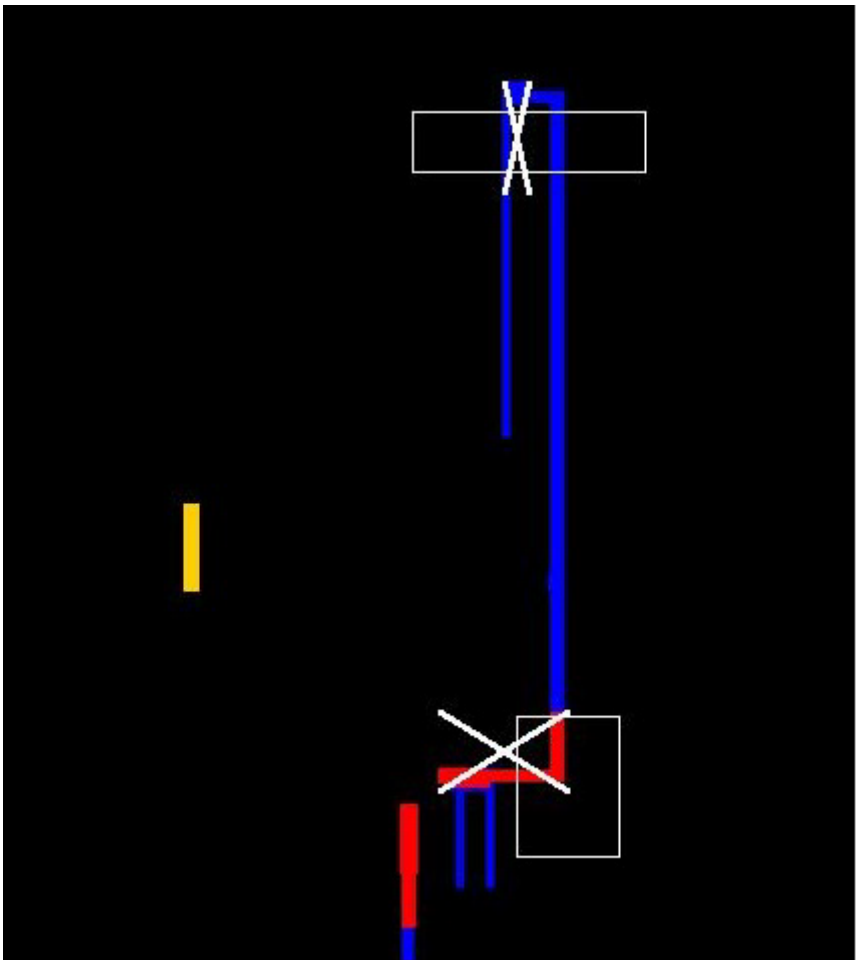
vsaPlot supports the display of the colormap between two points or regions selected on the layout. For this, the *Region2Region* submenu of the *View* menu is provided with four options; *Point*, *Rectangle*, *Ellipse*, and *Clear*. The *Clear* option is used to clear the selection.

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.

© 2020 All Rights Reserved.

Figure 7-24 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.

Synchronizing Multiple vsaPlot Windows

vsaPlot 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.

© 2020 All Rights Reserved.

- 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.

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 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 window pop-up 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

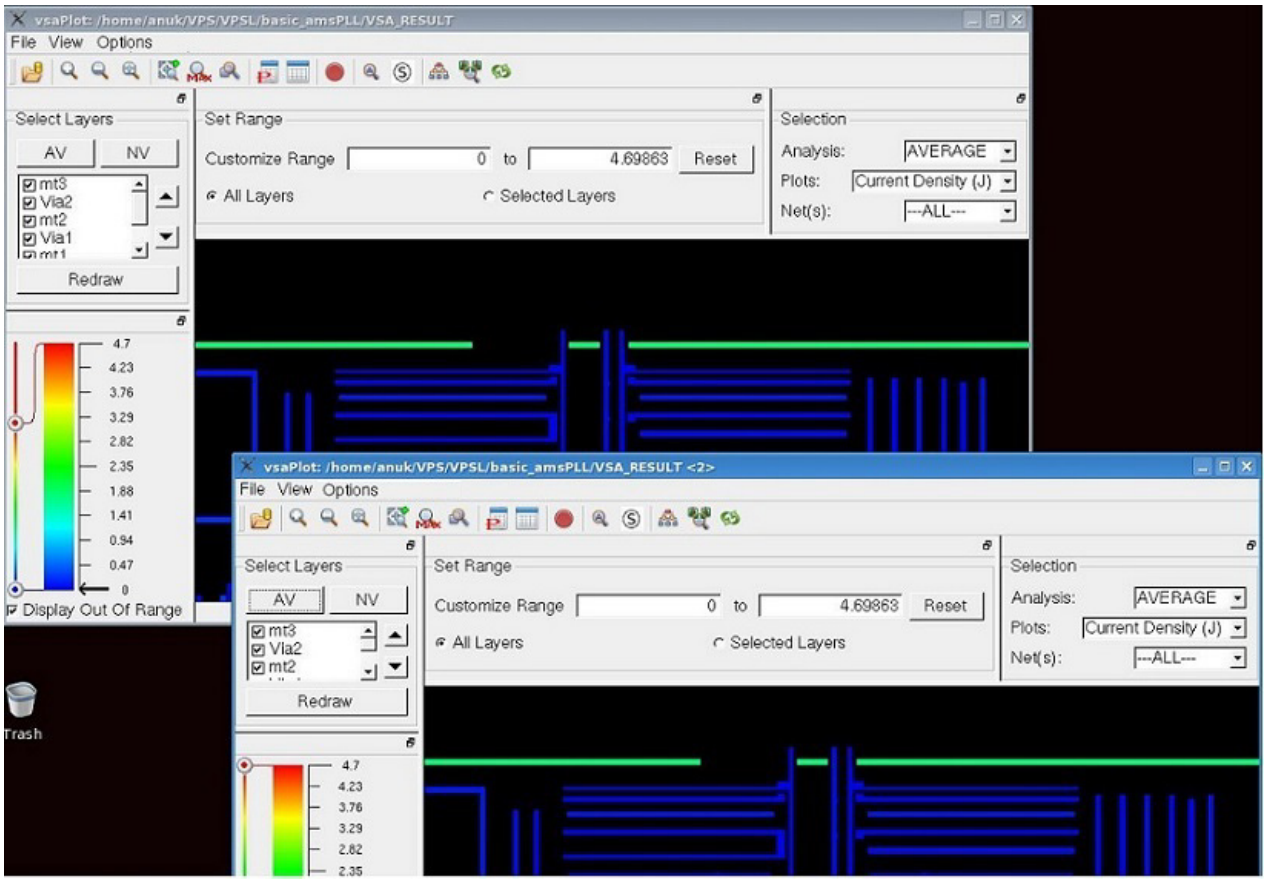
Once the windows are open for comparison, you can select the options for synchronism depending upon your requirements. The following options are available:

- 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

© 2020 All Rights Reserved.

window, you have to manually synchronize the sub-window from the main window again to bring the two windows back into the same state. This is shown below.

Figure 7-25 Opening a Sub-Window

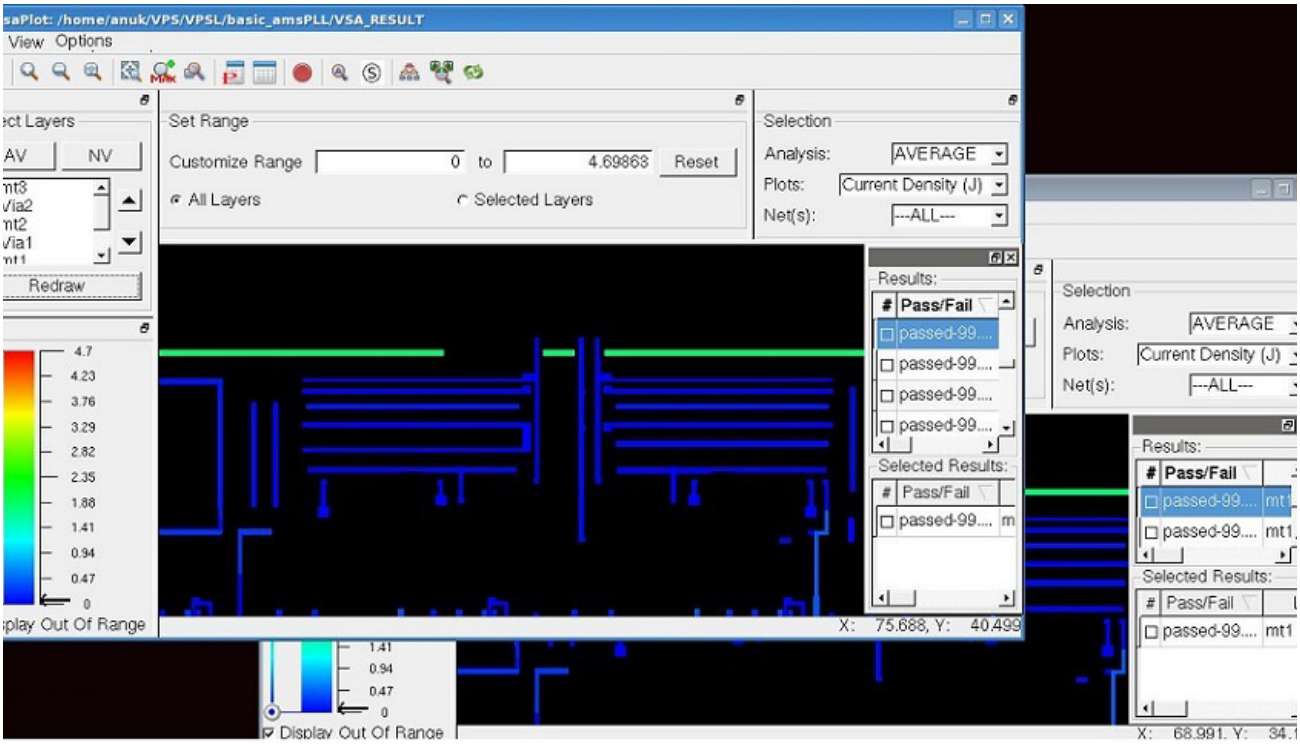


- Dynamic synchronism: Properties such as zooming and probing are supported dynamically. You can filter the instruction received by the sub-window from the main window by using the synchronization settings. Open the *Synch* submenu in the *File* menu. Click Settings. The Settings pop-up window opens. 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. This is shown below. By default, both zoom and probe actions are synchronized.

© 2020 All Rights Reserved.

Figure 7-26 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:

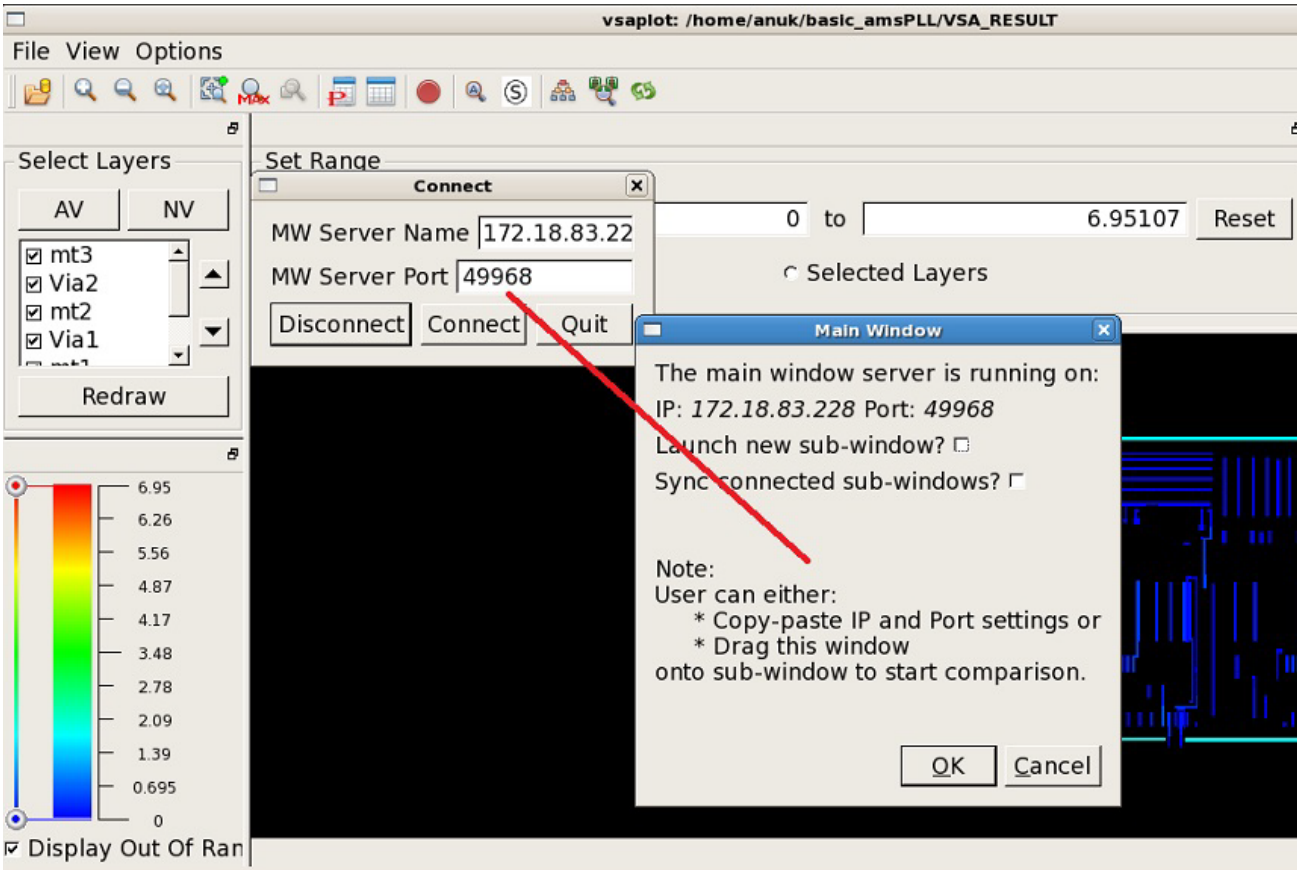
- Load the Results directory, VSA_RESULT, using the following command:

  ```
  vsaplot VSA_RESULT
  ```

- In the *Open* submenu of the *File* menu, click New Window and Sync. The Main Window pop-up 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 Server Name* field of the Connect window. The port information is populated in the *MW Server Port* field.This is shown below.

© 2020 All Rights Reserved.

Figure 7-27 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.

© 2020 All Rights Reserved.

© 2020 All Rights Reserved.

8

Power-Grid View Creation

- Overview on page 180
- Data Requirements and Flow for PGV Creation on page 180
- Creating Power-Grid Views on page 182
- Writing Out a PGV Command File on page 185
  - Sample Batch Command file on page 186
- Outputs of PGV Generation on page 187
- File Formats on page 187
  - DSPF Layermap File on page 187
  - LEF Layermap File on page 188
  - PGDB Layermap File on page 189

© 2020 All Rights Reserved.

Overview

Increasing operating frequency and density of IC designs increases the need for IR drop and Electromigration (EM) power-grid analysis. Power-grid signoff analysis is provided by Voltus in the digital domain, and Power IR/EM in the analog domain.

Power IR/EM provides the ability to create power-grid views (PGVs) of an analog design block that can be used in Voltus for mixed-signal analysis. PGVs contain the following:

- A model view of the design
- Information about the power ports of the cell or block
- Information about the internal power grid, intrinsic capacitance, and the tap current distribution within the cell or block

PGVs provide an accurate characterization of capacitance and currents, and power-grid extraction. Voltus uses these views to model power rail and power distribution information for each instance of the cell in the design for dynamic or static full-chip power-grid analysis.

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

Data Requirements and Flow for PGV Creation

The following table shows the required inputs for generating power-grid views in Power IR/EM. The inputs are categorized into the following: 1) from the design; and 2) from the user.

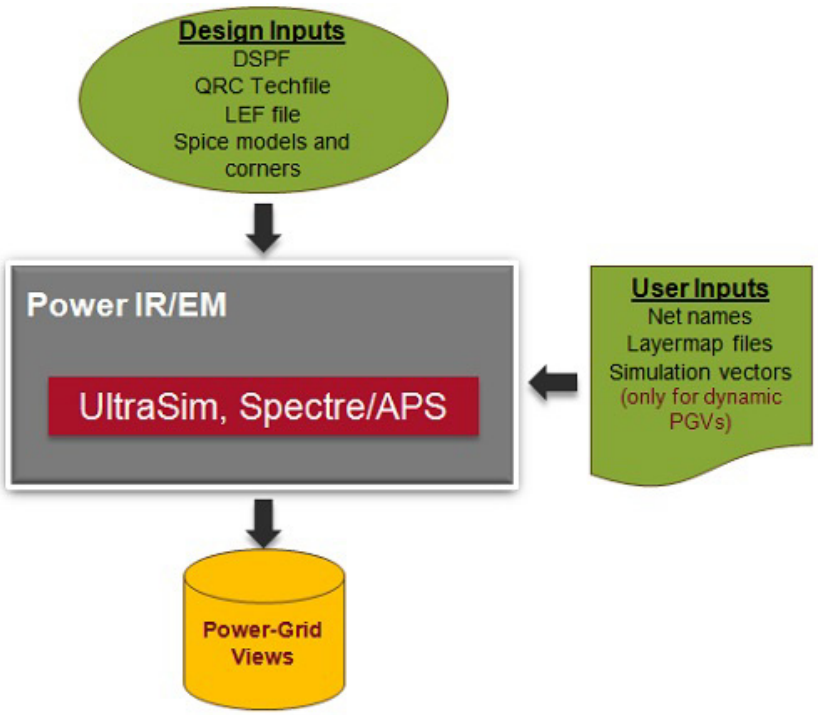
© 2020 All Rights Reserved.

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 for the following: ■ Layer names in the LEF file to layer names in the technology file ■ PGDB layer names to the layer names in the technology file
	`Stimulus file`	Required for providing information about simulation vectors, start time/stop time, and interval **Note:** This is required only for dynamic PGV generation.

© 2020 All Rights Reserved.

The following diagram illustrates the flow of PGV creation in Power IR/EM.

Figure 8-1 Power-Grid View Generation Flow in Power IR/EM



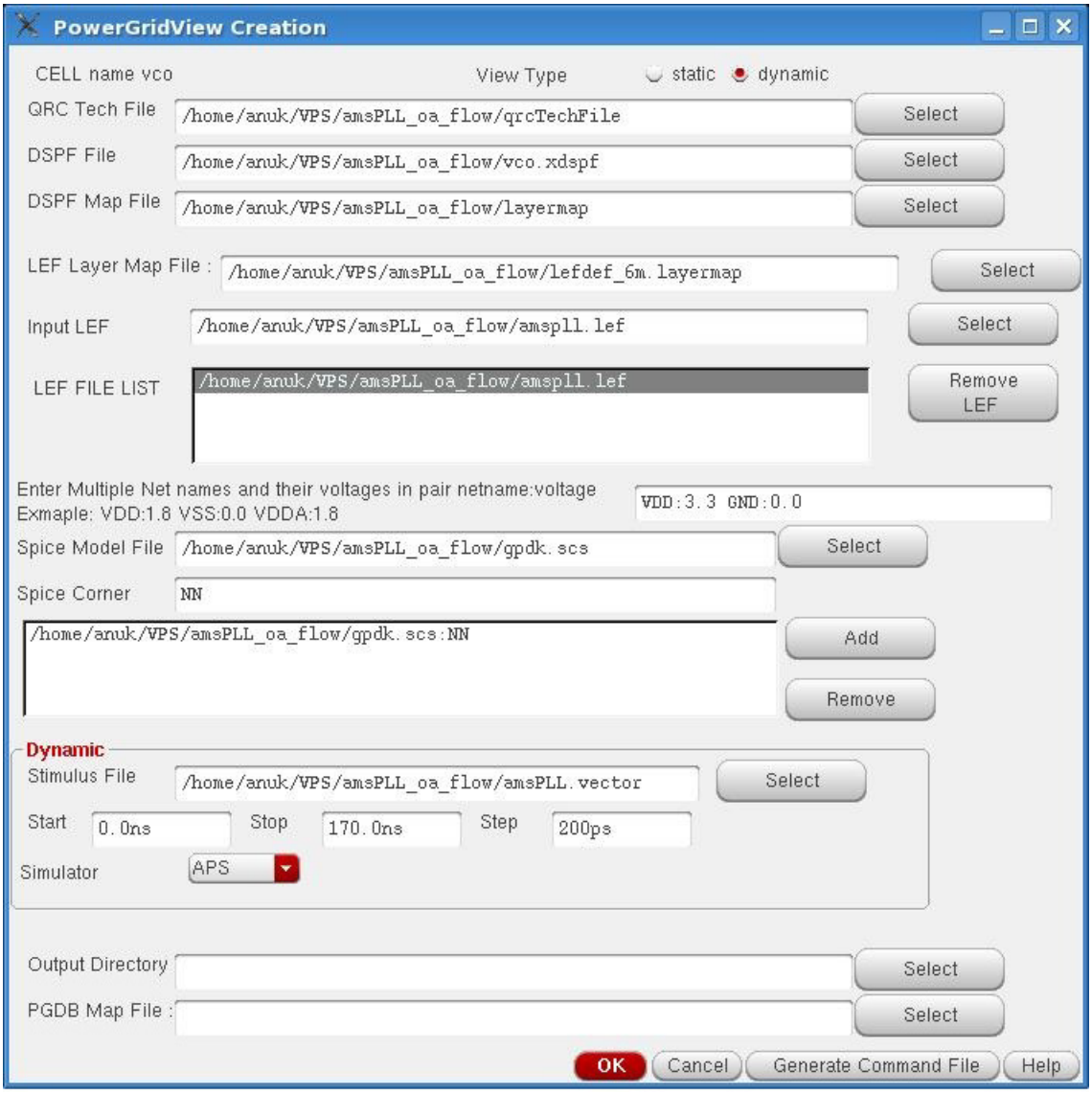
Creating Power-Grid Views

To create PGVs in Power IR/EM, perform the following steps:

- In the Virtuoso Layout Suite L, select *VPS-L- Analyze Extracted View Pin(s).*
- From the drop-down list, select VPS-*LibGen Power Grid Views Creation.* The PowerGridView Creation form opens. This form is shown below.

© 2020 All Rights Reserved.

Figure 8-2 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.
- 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 QRC technology file that will be used for PGV generation.
- Select the *DSPF File*, which is the the third party or xDSPF netlist file that is to be imported.

© 2020 All Rights Reserved.

- Select the *DSPF Map File*, which is the layermap file that has 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. An example of the DSPF map file is shown in the File Formats section.

Important

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 PGDB will not be created.

- Specify the *LEF Layer Map File*, which contains the mapping information for the layer names in the LEF file to the layer names in the technology file. An example file is shown in the File Formats section.
- Select the technology LEF files and the Macro LEF files in the *Input LEF* field. The technology LEF files contain 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.
  - Select the *Stimulus File*, which is the input file for the simulator. Choose this option to specify the vector information to be used for 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.

© 2020 All Rights Reserved.

- Choose either APS or Ultrasim in the *Simulator* cyclic field. By default, APS is selected.

- 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 between the PGDB layer names and the layer names in the technology file. This file is required only when the layer names in the PGDB are different from the layer names in the technology file. An example of the PGDB layermap file is shown in the File Formats section.

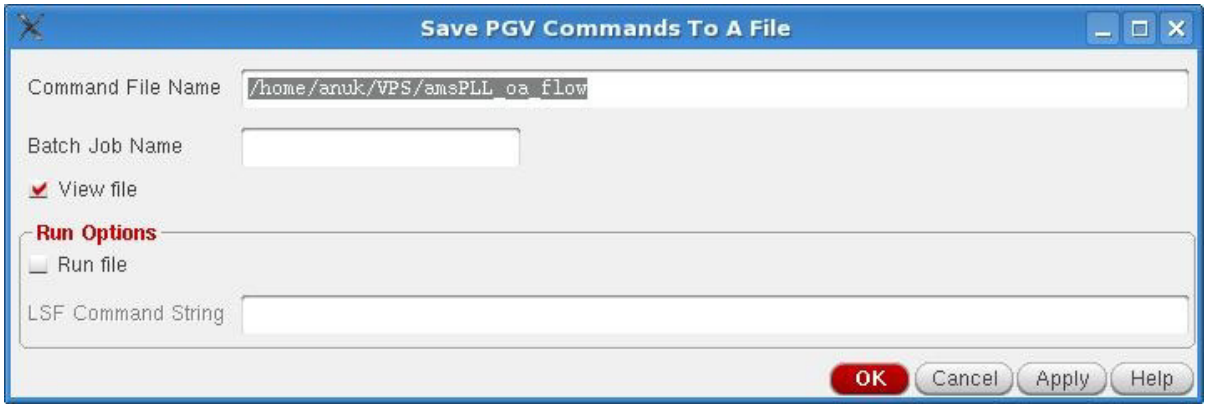
- Click *OK*.

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 8-3 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.

© 2020 All Rights Reserved.

- 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`.
- 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 Power IR/EM 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.
- Click *OK*.

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
```


© 2020 All Rights Reserved.

Outputs of PGV Generation

Following are the outputs of PGV generation in Power IR/EM:

- 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.

File Formats

This section lists the file formats of the layermap files specified during PGV generation.

- 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
```


© 2020 All Rights Reserved.

```
#  ---------   ------------
    metal       m4
    via         via3
    metal       m3
    via         via2
    metal       m2
    via         via1
    metal       m1
    metal       po
    metal       mdtap
```

LEF Layermap File

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. It can be poly, via, metal, or diff.

`technology_layer_name` specifies the name of the library layer that is to be mapped to the LEF layer.

`lefdef` specifies the keyword for the LEF layer mapping.

`lef_layer_name` specifies the name of the LEF layer to map to the library layer.

Example:

```
#type   technology_layer_name   lefdef    lef_layer_name
#---    ---------------------   ------    --------------
metal   M1                      lefdef    M1
via     VIA1                    lefdef    VIA1
metal   M2                      lefdef    M2
via     VIA2                    lefdef    VIA2
metal   M3                      lefdef    M3
via     VIA3                    lefdef    VIA3
metal   M4                      lefdef    M4
```


© 2020 All Rights Reserved.

```
via     VIA4                    lefdef    VIA4
metal   M5                      lefdef    M5
via     VIA5                    lefdef    VIA5
metal   M6                      lefdef    M6
via     VIA6                    lefdef    VIA6
metal   M7                      lefdef    M7
via     VIA7                    lefdef    VIA7
metal   M8                      lefdef    M8
via     VIA8                    lefdef    VIA8
metal   M9                      lefdef    M9
poly    PO                      lefdef    PO
via     polyCont                lefdef    CO
```

PGDB Layermap File

A layer mapping file is required for the Rail Analysis library generation flow. The PGDB layermap file contains the mapping information to map the PGDB layer names to the layer names in the technology file.

In the GUI, you can specify this file in the *PowerGridView Creation* form.

Sample file name: `pgdb.map`

Format:

```
type    technology_layer_name    pgdb    pgdb_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 PGDB layer name.

`pgdb` specifies the keyword for the PGDB layer mapping.

`pgdb_layer_name` specifies the name of the PGDB layer.

Example:

```
#type   technology_layer_name   pgdb    pgdb_layer_name
#---    ---------------------   ----    ---------------
metal   M4                      pgdb    M4
via     VIA3                    pgdb    VIA3
metal   M3                      pgdb    M3
```


© 2020 All Rights Reserved.

```
via     VIA2                    pgdb    VIA2
metal   M2                      pgdb    M2
via     VIA1                    pgdb    VIA1
metal   M1                      pgdb    M1
poly    PO                      pgdb    PO
via     polyCont                pgdb    CO
```


© 2020 All Rights Reserved.

9

EM Rules Specification

- Overview on page 193
- Reading EM Rules from qrcTechFile on page 193
  - Supported Limit-Based Parameters on page 193
  - Supported JMAX Parameters on page 194
  - Rules for Defining Limit-Based Parameters on page 195
- EM Rule Selection on page 200
- Reading EM Rules from emDataFile on page 202
- User Procedures for Electromigration JMAX Checking on page 215
- Custom Parameters in emDataFile on page 218
- SKILL Functions on page 220
  - vsaDefVariable on page 220
  - vsaGetVariable on page 221
  - vsaWarn on page 222
  - vsaError on page 222
  - vsaDbType on page 223
  - vsaRegisterEMDataFile on page 224
  - vsaRegisterQrcTechFile on page 224
  - vsaRegisterLayerMapFile on page 225
  - vsaGetR on page 226
  - vsaResGetIrms on page 227
  - vsaResGetIpeak on page 227

© 2020 All Rights Reserved.

- vsaResGetIavg on page 228
- vsaResGetX on page 229
- vsaResGetY on page 230
- vsaResGetLayer on page 230
- vsaGetNode on page 231
- vsaNodeGetIR on page 232
- vsaEMGetQRCDirAndRunName on page 233
- vsaEMGetSimDirAndAnalysisType on page 233
- vsaIRGetSimDirAndAnalysisType on page 234

■ Call-Back Functions on page 236

- vsaEMInitUserCallBack on page 236
- vsaEMUserCallBack on page 236
- vsaIRUserCallBack on page 237

■ Sample emDataFile on page 239

© 2020 All Rights Reserved.

Overview

Before running Power EM analysis, the EM reliability rules need to be specified. For example, the rule specifying different current densities with respect to width, length, and temperature.

These rules can either be read from the QRC technology file (`qrcTechFile`) or from the `emDatafile`, which includes user-specified rules.

For more information about the syntax of limit-based EM models, refer to the EM_Models section in the “Creating the ICT File” chapter in *QRC Techgen Reference Manual*.

Reading EM Rules from qrcTechFile

Currently, Power IR/EM only supports parameters for limit-based EM analysis.

For more information about the syntax of limit-based EM models, refer to the section titled, EM_Models, in the “Creating the ICT File” chapter in *QRC Techgen Reference Manual*.

Supported Limit-Based Parameters

The following parameters are supported:

- `em_vcwidth <value>`

  Optional parameter for via layers that defines the width of an individual standard via layer in microns. 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_segment_length`

  A process definition parameter that determines how to calculate the length of a wire segment. This parameter determines whether the segment length is of the type `perimeter` or `centerline`. The difference between the perimeter and centerline segment length types is as follows:

© 2020 All Rights Reserved.

- `perimeter`—specifies that the length calculation is along the wire outer edge boundary. This value is used for 28nm process nodes.
- `centerline`—specifies that the length is extended to the edge of the segment along the centerline path. This value is used for 20nm process nodes.

The following example shows how to use the `em_segment_length` parameter:

```
em_segment_length centerline
```

- `em_recover`

  Optional parameter that specifies the recovery factor used for calculating the average current density with recovery. You can set the recovery factor to -1.0 when you want to account for the average of the absolute current (`avg(abs(current))`) so that the negative current does not cancel the positive current values in arriving at the average current result.

Supported JMAX Parameters

For current density limit (jmax) analysis, the following parameters are supported:

- `em_jmax_dc_avg*`

  Optional parameter that specifies the AVG analysis in Power IR/EM.

- `em_jmax_dc_peak*`

  Optional parameter that specifies PEAK analysis in Power IR/EM.

- `em_jmax_dc_rms*`

  Optional parameter that specifies DC RMS analysis in Power IR/EM.

- `em_jmax_ac_rms*`

  Optional parameter that specifies AC RMS analysis in Power IR/EM.

- `em_jmax_ac_peak*`

  Optional parameter that specifies AC_PEAK analysis in Power IR/EM.

Note: Star ‘ * ’ implies that ‘`_w`’ and ‘`_n`’ rules are also supported.

Important

jmax parameter, `em_jmax_ac_rec*` is not currently supported in Power IR/EM.

© 2020 All Rights Reserved.

Rules for Defining Limit-Based Parameters

Below is an example of the EM rule for defining the `em_jmax_ac_avg` parameter.

```
[em_jmax_ac_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]
[via_range <value>]
[conditions]
[single]
[power_rail]
```

All limit-based 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 limit-based `em_model` parameters:

- Equations in EM Model

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

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

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

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

  Spaces are permitted to make an equation more readable.

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

© 2020 All Rights Reserved.

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

Order Dependency

The actual current density limit (value, PWL pairs, or equation) should follow immediately after the current density keyword (`em_jmax_ac_avg`). Any of the qualifiers (`jmax_factor`, `current_direction`, `single,` or `via_range`) 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/cm2 or mA/um for metal layers (width based)
  - A/via or mA/via for via layers (per via)

Note: You can also specify the current density of metal layers in mA instead of in A/cm2 or mA/um by using the environment variable, `_vsaICTConductorUnit`.

- For a PWL table:
  - A/cm2 or mA/um, with the width specified in microns for metal layers
  - A/cm2 or mA, with the area specified in square microns for via layers

Note: By Default, PWL points are discrete points and not a continuous line. Therefore, the area values that fall between the two points do not match the rule. To apply PWL rules to intermediate area values, use the environment variable, `_vsaEnablePWL`. In this case, the tool interpolates the value for the intermediate area. For area values above the maximum area defined in the PWL, the tool will do constant interpolation. For area values below the minimum area defined in the PWL, the tool will do linear interpolation from <0 0> to the minimum value pair that is specified.

- JMAX Factors

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

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

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

© 2020 All Rights Reserved.

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

- JMAX Life

```
jmax_life <life1> <scale1> [<life2> <scale2>.......]
```

`jmax_life` provides the ability to set the scaling factor that applies to the current density limits for different lifetimes. Based on the lifetime value that you specify (using the `em_lifetime_units` parameter defined in the process section of the ICT file), the appropriate multiplication factor is applied to the nominal current density limit.

For more information, see the Rules for Defining Limit-Based Parameters section in the "Creating the ICT File" chapter in *QRC Techgen Reference Manual*.

You can specify the lifetime value by using the `JmaxLife` custom variable in the Electromigration Edit/View Custom Parameters form.

You can also specify the lifetime value using the environment variable, `_vsaJmaxLife`.

- Format of the Conditions

`Condition = [L/W/Lu/Wu/Lb/Wb/Lv2v/`T_d`/r < | <= | > | >= <value in microns>] | [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 only for vias.

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

Where,

`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

© 2020 All Rights Reserved.

Power IR/EM supports metal length or 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.

For more information, refer to the Rules for Defining Limit-Based Parameters section in the “Creating the ICT File” chapter in *QRC Techgen Reference Manual*.

- Single

  The keyword `single` can be specified to differentiate between a single square cut via and a 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
```

- Current Direction

  `current_direction [up | down | both]`

  `current_direction` can be specified for via current direction. The syntax of `current_direction` is:

  - `up` - means the current is from down to top and specifies how to use the length and width of the metal below Lb/Wb for the VIA Jmax factor
  - `down` - means the current is from top to down and specifies how to use the length and width of the metal above Lu/Wu for VIA Jmax factor
  - `both` - This rule is only applicable when the current direction is uncertain. This rule is unlikely to be applied since the software calculates the current direction

- Via Range

  `via_range <value>`

  `via_range` is an optional parameter which can be used with the via layers. The value of `via_range` denotes the number of vias for which the calculated limits are valid. `via_range` is specified as a positive integer.

  Example:

```
via Via1 {
   em_model {
      em_jmax_dc_avg 1.11 via_range 1
      em_jmax_dc_avg 2.22 via_range 4
      em_jmax_dc_avg 3.33 via_range 8
   }
   ...
```


© 2020 All Rights Reserved.

```
}
Current density value of 1.11 mA/via will be applied for viaCount=1
Current density value of 2.22 mA/via will be applied for viaCount=4
Current density value of 3.33 mA/via will be applied for viaCount=8
```

- The keyword `power_rail` can be specified to enable the particular rule for power-rail analysis. When power-rail analysis is enabled, a rule with the `power_rail` 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
```

- 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 I_{peak_ac} limit. You can avoid such scenarios by resetting the value of “r” in the ICT file. For example:

```
em_jmax_ac_peak EQ 4.0 r < 0.01 applyR 0.01
```

  You can specify the threshold ratio by using the `applyRThreshold` custom variable in the Electromigration Edit/View Custom Parameters form .

  You can also specify the threshold ratio using the environment variable, `_vsaApplyRThreshold`.

- In `qrcTechFile` flow, `deltaT` can be specified in the GUI using the Edit/View Custom Parameters form. This is used to specify 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, Power IR/EM considers a default temperature of 5 degrees Celsius.

  For more information, see the Edit/View Custom Parameters section in the “EM Analysis” chapter.

Important

Advanced EM rules with `power_rail` and `current_direction` keywords require a VPS-XL license.

© 2020 All Rights Reserved.

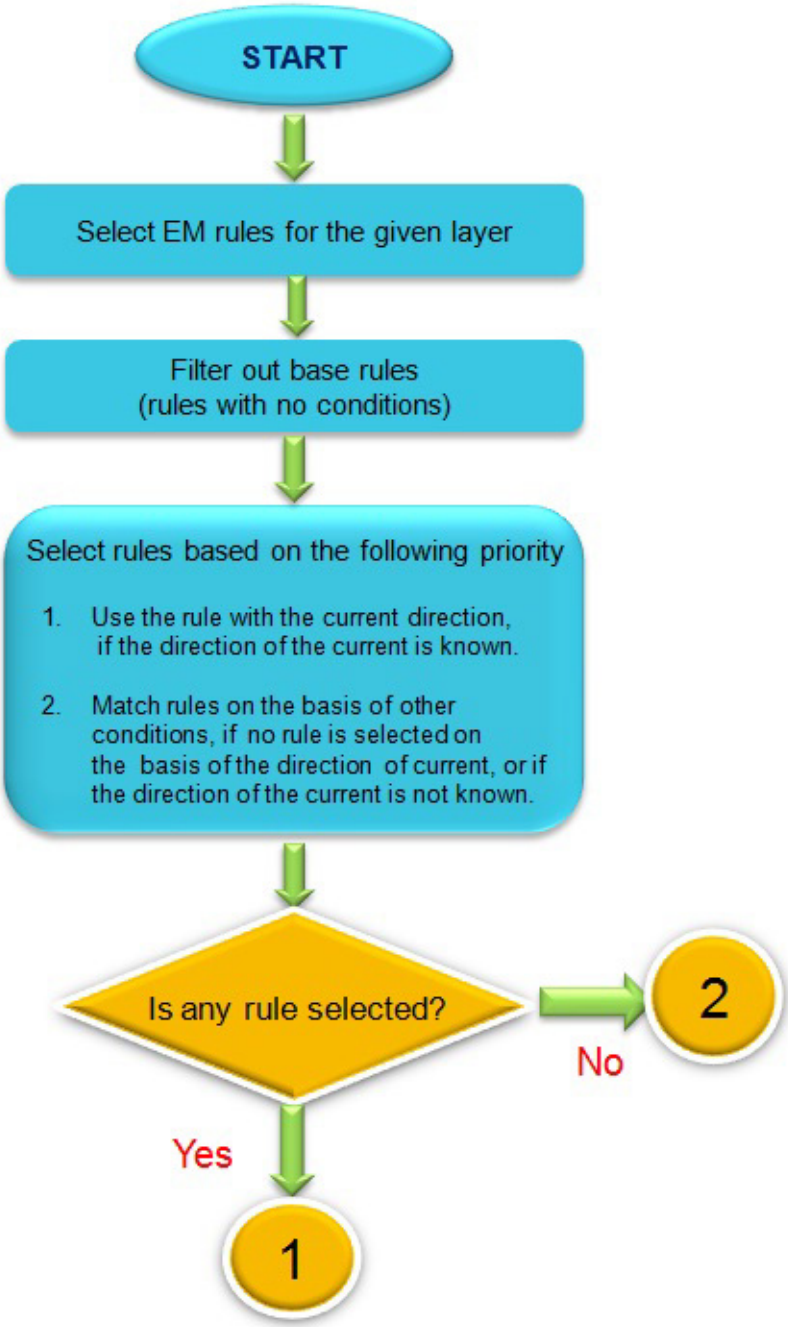
EM Rule Selection

The earlier EM selection rule was that the software matched rules in the top-to-bottom order in a given sequence. The first rule that matched was selected. This behavior is changed to support the 20NM rules.

The following diagrams outline the flow of EM selection rule in Power IR/EM.

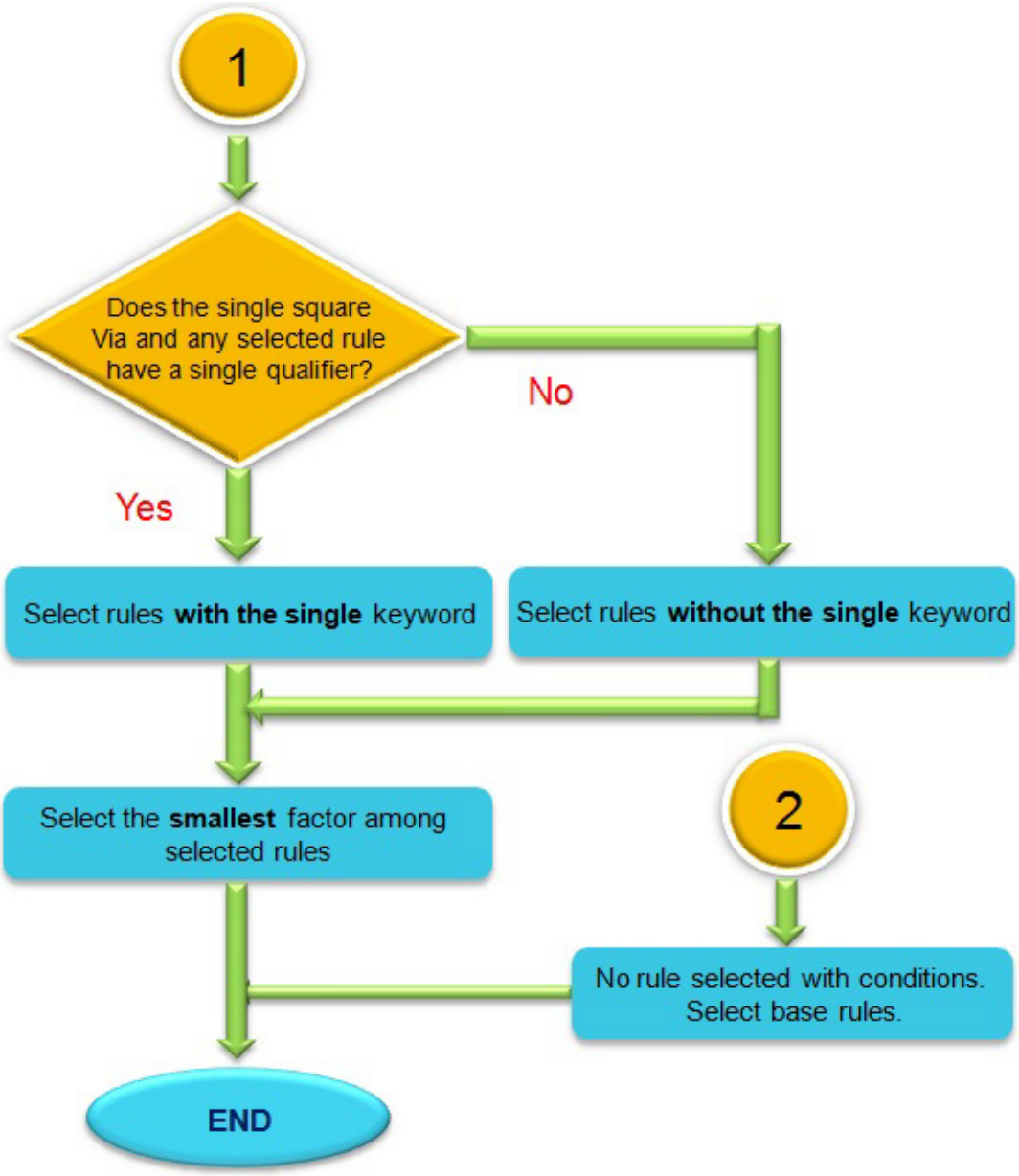
© 2020 All Rights Reserved.

Figure 9-1 EM Rule Selection Flow



© 2020 All Rights Reserved.

Figure 9-2 EM Rule Selection Flow (continued)



Reading EM Rules from emDataFile

The following sections detail the EM Rules specification file syntax.

© 2020 All Rights Reserved.

EM Rules Specification File Syntax

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

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

- The Current Density Milliamps Per Via keyword (`currentDensityMPV`) allows you to define the current density specifications in terms of milliamps per via instead of the standard milliamps per micron. Most foundries specify via and contact current densities in milliamps per via, and the use of this keyword saves you from having to manually convert the foundry specification.

  ```
  currentDensityMPV = true
  ```

Important

If the keyword `currentDensityMPV` is used, then all current density specifications must be stated in terms of milliamps per via.

Note: When `currentDensityMPV` is enabled, `minW` and `maxW` are defined as the number of vias. For example,

```
(nil layer "myVia" minW 1 maxW 4 currentDensity ((10,85))
(nil layer "myVia" minW 4 maxW 8 currentDensity ((20,85))
```

Current density=`10mA/um` if number of vias is `>=1` and `<4`
Current density=`20mA/um` if number of vias is `>=4` and `<8`

When `currentDensityMPV` is disabled, `minW` and `maxW` are defined as the total width of a via array. For example,

Current density=`10mA/um` if width of via array is `>=1` and `<4`
Current density=`20mA/um` if width of via array is `>=4` and `<8`

Width of via array is calculated as (`number of vias` x `minimum width of via`)

- The `meshR` keyword is set to `true` when the extraction is done using the `meshR` option. This option is used to specify regions for mesh resistance extraction. When `meshR` is specified, the software evaluates the node current direction based on this flag.

  ```
  meshR = true
  ```

- `deltaT` is used to specify 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, Power IR/EM 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`

© 2020 All Rights Reserved.

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)))
```

- `preExecuteFiles`

  Takes the list of custom files that are pre-loaded when reading the `emDataFile`.

  Examples:

  ```
  1 preExecuteFiles=("custom.il")
  2 preExecuteFiles=( getShellEnvVar("myPreExecuteFile"))
  ```

  where the filename is defined in the environment variable as `"myPreExecuteFile"`. You can specify any name as the environment filename.

  ```
  preExecuteFiles=("customFile1.il" "customFile2.il")
  ```

- Minimum Current (`minI`) can be used to limit the reporting of small currents which are of little or no interest during either Power or Signal EM analysis. The value of `minI` is specified in Amps (A). Any wire segments that have current values less than `minI` will not be displayed in the EM results windows or reported in the EM result file. The default value of `minI` is 0, so if it is not specified in the `emDataFile` it will have no effect on the analysis.

  ```
  minI = 1e-9
  ```

- The EM `recoveryFactor` is used to calculate 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

  Note: The tool errors out if the `recoveryFactor` value is not equal to 1.0 and when simulation data is in the Power IR/EM-specific format. This is because there is not enough simulation data available in this format for calculating the `recoveryFactor`. However, `recoveryFactor` is supported with simulation data in the Power IR/EM-specific format if the simulator version used is Spectre/APS 11.1 ISR19 and above.

© 2020 All Rights Reserved.

- The `recoveryFactorList` is used to specify 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.

- The `durationTime` is used to specify the maximum time period for which a violation is allowed. For example, if you set the duration time to 1u(sec), then any violations that occur for durations less than 1u are ignored. This helps you eliminate short duration violations that do not have an impact on electromigration. This, in turn, reduces the verification time taken for EM analysis.

  ```
  durationTime = <value>
  ```

- The `cadGrid` specifies the minimum width that Power IR/EM uses when a parasitic resistor (presistor) does not contain width information or has a width that is less than the specified `cadGrid`. This ensures that no `presistor` will have a width less than `cadGrid`. When Power IR/EM changes the width of a `presistor` to meet the `cadGrid` a warning message is written to the `cds.log`. The `cadGrid` can also be specified in the tech lib. Syntax to specify `cadGrid` is:

  ```
  cadGrid = 0.002
  ```

  The unit of measurement is micrometer (um) or micron.

- 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")
  ```

- The via layers are the names of the DFII layers. Look for `viaLayers` in the Virtuoso technology file, for example:

  ```
  ( 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")
  ```

- The cross-reference layers (`xrefLayers`) contain a list of layer names in the following format:

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

  - For conductor layers, *`QRCName`* refers to the `pro_layer` name in the QRC `layerSetupFile` or `p2lvsfile`. For via layers, *`QRCName`* refers to the `ext_cont` layer names. For example:

    ```
    === layerSetupFile ===
    pro_layer=myM1 ext_layer=m1_par, m1
    ```


© 2020 All Rights Reserved.

```
pro_layer=myM2 ext_layer=m2_par,m2 model=M2mod
pro_cont=myV1 ext_cont=v1_par, v1
pro_cont=myV2 ext_cont=v2_par, v2 model=V2mod

=== emDataFile ===
xrefLayers=(
( myM1 (M1 M1))
( M2mod (M2 M2))
( v1_par (V1 V1))
( v1 (V1 V1))
( V2mod (V2 V2))
```

- *currentDensityName* is the layer name used in the current density specification declared by the foundry.
- *DFIIName* is the layer name in the DFII technology directory.

The above three layer names might all be same or different depending on how the rules files are written. For example:

```
xrefLayers = (
( "poly1con" ("cw" "cw"))
( "poly" ("poly1" "poly1")))
```

Note: In general, *QRCName* should match the parasitic resistor model names that are found in the `extview.tmp` file in the QRC run directory. However, if the layer name matching fails for the specified *QRCName*, the matching will be done using *currentDensityName* and *DFIIName*.

If the same via layer has different resistance values, for example, diffusion contact and poly contact, then they should be mapped under different `currentDensityNames` so that the software can use the appropriate resistance values. For example:

```
xrefLayers(
(diffCont (CONTdiff CONT))
(pcCont (CONTpc CONT))
)
(nil layer "CONTdiff" minW 0.0 maxW -1.0 minL 0.0 maxL 1.0 res 2.0
currentDensity ((1.0, 27))
(nil layer "CONTpc" minW 0.0 maxW -1.0 minL 0.0 maxL 1.0 res 3.0 currentDensity
((1.0, 27))
OR
(nil layer "CONTdiff" minW 0.0 maxW -1.0 minL 0.0 maxL 1.0 currentDensity
((1.0, 27))
(nil layer "CONTpc" minW 0.0 maxW -1.0 minL 0.0 maxL 1.0 currentDensity ((1.0,
27))
```

Note: If "res" is not explicitly specified, it will be retrieved from the QRC run directory which is assumed to be within the specified LVS run directory.

© 2020 All Rights Reserved.

- `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.

  There are seven Current Density Specifications in the `emDataFile`: `avgCurrentDensSpecList, avgAbsCurrentDensSpecList, rmsCurrentDensSpecList, DCrmsCurrentDensSpecList, peakCurrentDensSpecList, ACpeakCurrentDensSpecList,` and `userDefinedCurrentDensSpecList.`

  The DC operating point (DCOP) analysis will use the `avgCurrentDensSpecList`. `ACpeakCurrentDensSpecList` is used when you select *AC-Peak* analysis on the EM form. See EM Analysis of the Power Grid, in the "EM Analysis" chapter. Each specification in the `emDataFile` can be unique with its own currentDensity and temperature.

  An example of a current density specification for a given layer is as follows:

```
(nil layer "v1" minW 0.0 maxW -1 res 0.68 currentDensity ((1.2100,100)
(1.2110,110) (1.2125,125)))
```

  - *`nil`* is a reserved keyword indicating the start of the line.
  - *`layer "poly1"`* is the `layer` keyword followed by the specific layer name to apply the current density line. The layer name is the current density layer name from the layer cross-reference section of the `emDataFile`.
  - *`minW`* is the minimum line width in microns.
  - *`maxW`* is the maximum line width. When set to -1.0, it represents infinity.
  - *`res`* is the resistance in ohms.
    - For contacts and via layers res should be set to the total resistance of a minimum width contact.
  - *`currentDensity`* is specified in milliamps per micron at a specified junction temperature. This can be represented in multiple (currentDensity, temperature) pairs.

    currentDensity ((1.2100,100) (1.2110,110) (1.2125,125))

    - Most foundries specify via and contact current densities in milliamps per contact or milliamps per square micron. The foundry specification must be converted to milliamps per micron. For example, if the foundry specification provides a current density of 0.4`mA/via` and the via is 0.2micron X 0.2micron, then the currentDensity value is 0.4 / 0.2 = 2mA/um

  Note: With the introduction of the `currentDensityMPV` keyword, current densities can now be specified in milliamps per contact as provided by the foundry

© 2020 All Rights Reserved.

so no manual conversion is required.

- *temperature* is specified in degree Celsius.

  The software compares the simulation temperature with this temperature. When the simulation temperature is not in the temperature range specified above, the software behaves in the following manner:

  - If the simulation temperature is less than the specified minimum temperature, the current density value is picked for the specified minimum temperature and not for the simulation temperature.
  - If the simulation temperature is greater than the specified maximum temperature, the current density value is picked for the specified maximum temperature and not for the simulation temperature.

  When the simulation temperature falls within the specified temperature range, the software interpolates the value using a formula that is provided in the example below.

  Note: The tool uses temperature in Kelvin while doing the calculation using the interpolation equation.

  Examples:

  In these examples, `T` is the simulation temperature, `T1` is the specified minimum temperature, and `T2` is the specified maximum temperature.

  currentDensity ((RT1, T1) (RT2, T2)) where T1 < T2

  If `T < T1`; current density will be taken for temperature T1

  If `T > T2`; current density will be taken for temperature T2

  If `T1 < T < T2`; The tool will interpolate for the temperature derating factor that is derived from the formula: `TTF=A*J^(-n)*exp(Ea/kT)`

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

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

  Note: You can enable linear interpolation for RT factor at temperature T by setting the `_vsaLinearInterpolation` variable to true.

- *currentDensity* can also be specified as an equation containing the variables `'w'` for width and `deltaT` for temperature difference:

  currentDensity ((14*w*deltaT,100) (1.434 *3.9*w,110) (5.3,125))

© 2020 All Rights Reserved.

- You do not need to specify the value of `'w'` because it is found in the actual design data. Minimum width is calculated for all the presistors. Current density equations using `'w'` will be evaluated for each metal line width. This means, for example, that `"m1"` layer will have a different current density spec for line widths 0.16um as compared to 0.25um.

  If "`w`" is used in the equation for the current density specification, the width is resolved for all passed or failed presistors. However, the level of accuracy will be different for them.

  For failed presistors, the minimum width will be accurate up to 0.001 micron. However, the minimum width reported will be on the positive side. That means that it is the width for which it is certain that the EM violation will not happen.

  For passed presistors, if the EM violations are passed by more than 20 percent, the accuracy of the minimum width will be 10 percent of the minimum width reported on the positive side. For others, the accuracy will be 1 percent of the value of the minimum width that is reported.

  For presistors having zero current value, the minimum width will be reported as 0.

- When an equation is used in the *`currentDensity`* spec, “unknown” can be written to the results of the analysis if the equation cannot be resolved.

- *`currentDensity`* can also be specified as an equation using the following operators: *`+, -, *, /,sqrt(), and exp()`*:

  currentDensity ((sqrt(4.9100),100) (4.9110,110) (4.9125,125))

  - The square root function takes a positive floating point number as input.

- `min()` function in the current density limits allows you to specify the minimum value of current density to be used for EM results. You can specify multiple values as shown in the example below. The software selects the minimum value of the current density from the list.

```
((min(sqrt(0.84*w*(w+1.604)),1.0, 2.0, 3.0),110)
(min(sqrt(0.84*w*(w+1.604)),1.0, 2.0, 3.0),125))
```

The following is an example of the `peakCurrentDensSpecList` parameter, where the currentDensity is dependent on the width of the presistor:

```
peakCurrentDensSpecList = (
(nil layer "m1" minW 0.0 maxW 0.16 currentDensity ( (1.434*3.9*w,100) (3.9,110)
(0.387*3.9,125)))
(nil layer "m1" minW 0.16 maxW -1.0 currentDensity ( (14*w*deltaT,100) (7.0-
0.16,110) (5.3,125)))
```


© 2020 All Rights Reserved.

In the preceding example, the `currentDensity` for layer “m1” will be calculated by the equation using the actual line width depending upon whether it is greater than the 0.16 maxW specified as the breakpoint between the two lines.

The following is an example of the `peakCurrentDensSpecList` parameter, where the currentDensity is dependent on length of the presistor:

```
(nil layer "Metal1" minW 0.0 maxW -1.0 minL 0.0 maxL 1.0
currentDensity ((1.0, 27) (1.01,110) (1.02,125)) )
(nil layer "Metal1" minW 0.0 maxW -1.0 minL 1.0 maxL 2.0
currentDensity ((3.1*L*deltaT, 27) (4.11*L*deltaT,110) (4.32,125)) )
(nil layer "Metal1" minW 0.0 maxW -1.0 minL 2.0 maxL -1.0
currentDensity ((1.2, 27) (1.21,110) (1.22,125)) )
```

The following is an example of the `peakCurrentDensSpecList` parameter, where the currentDensity is dependent on the length of the routing layers connecting to the via:

In the following example, the `minW` and `maxW` values are for Via layers while `minL` and `maxL` values are for the metal layers adjacent to the via layers:

```
(nil layer "Via1" minW 0.0 maxW -1.0 minL 0.0 maxL 1.0
res 2.0 currentDensity ((1.0, 27) (1.01,110) (1.02,125)) )
(nil layer "Via1" minW 0.0 maxW -1.0 minL 1.0 maxL 2.0
res 2.0 currentDensity ((3.1*L*deltaT, 27) (4.11*L*deltaT,110) (4.32,125)) )
(nil layer "Via1" minW 0.0 maxW -1.0 minL 2.0 maxL -1.0
res 2.0 currentDensity ((1.2, 27) (1.21,110) (1.22,125)) )
)
```

The `minL` and `maxL` specified in the `emDataFile` refer to the effective path length and not to the individual presistor length. The Jmax will be picked based on the path length of presistor and temperature. In case the Jmax equation contains the L variable it will be replaced with the path length.

Note: Effective path length is the longest path length in a segment. If there is a loop in the segment, the total length of the segment is taken as the effective path length.

The current density rules can be specified in terms of both width and length. For example the current limits for layer M1 can be described as below:

Metal length in um	Jmax
L >= 20um	1.509 x (w-0.016)
20 > L > 5	(20/L) x 1.509 x (w-0.016)
L <= 5	4 x 1.509 x (w-0.016)

© 2020 All Rights Reserved.

In the above case, the "`w`" will be replaced with the presistor width and "`L`" will be replaced with the path length of the metal route of which the presistor is a part.

- Use the `technologyNode = 40` parameter to specify the technology-dependent calculation. If the `technologyNode >= 40` then the effective length is calculated by the earlier method in which all the lengths are added.

  Note: You can also specify this using the environment variable, `_vsaTechnologyNode 40`. If both are specified, the `technologyNode` parameter specified in the `emDataFile` will have a higher priority than the corresponding environment variable.

- In case of via, you can specify upper metal layers and bottom metal layers in terms of both length and width. For example:

  Metal length/width in um jmax

  ```
  Lu < 5.0um                          1.2
  Lb < 7.0um                           1.2
  Wu < 4.0um                           1.2
  Wb < 6.0um                          1.2
  ```

  where:

  Lu = Upper metal length

  Lb = Bottom metal length

  Wu = Upper metal width

  Wb = lower metal width

  For example:

  ```
  (nil layer "Via1" minW 0.0 maxW -1.0 minL 2.0 maxL -1.0
  minLu 0.0 maxLu 5.0 minLb 0.0 maxLb 7.0 minWu 0.0 maxWu 4.0 minWb 0.0 maxWb 6.0
  res 2.0 currentDensity ((1.2, 27) (1.21,110) (1.22,125)) )
  ```

  Note: The (min/max) and (Lu/Lb/Wu/Wb) are optional parameters. If not specified, than min* is taken as 0.0 and max* is -1.0.

- 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

© 2020 All Rights Reserved.

- The Ignore Missing Spec (`ignoreMissingSpec`) flag helps you to ignore layers for which current densities are missing.

  ```
  ignoreMissingSpec= true
  ```

  The default value of the flag is `false`.

  Note: If `ignoreMissingSpec` is not specified, the tool selects the default current density as 0.9 for missing current densities.

- The Via Width list (`viaWidthList`) specifies the minimum via width. In the following example:

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

  - *`Cont`* is the DFII name of the via layer.
  - 0.2 is the minimum width of the default via for this layer.

Tip

If the via width information is not found in the DFII technology file, it must be included in the `emDataFile`. In this case, it must include all of the vias specified in the `viaLayers` section of the `emDataFile` or an error will occur.

- The Via Width list (`extViaWidthList`) specifies the minimum via width. In the following example:

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

  - *`extCont`* is the QRC (Extraction) name of the via layer.
  - 0.2 is the minimum width of the default via for this layer.

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

- The Via Length list (`viaLengthList`) specifies the via length for use in calculating the area of vias that are not square. This feature is only required when `currentDensityMPV` is set to "`true`". If the vias in the design are square, then you can specify `viaWidthList/extViaWidthList` without `viaLengthList` or `extViaLengthList`.

  ```
  viaLengthList = (Via1 0.12)
  ```

  - `Via1` is the DFII name of the via layer.

- The Via Length List (`extViaLengthList`) specifies the via length for use in calculating the area of vias that are not square. This feature is only required when currentDensityMPV is set to "`true`". If the vias in the design are square, then you can specify `viaWidthList/extViaWidthList` without `viaLengthList` or `extViaLengthList`.

© 2020 All Rights Reserved.

```
extViaLengthList = (extVia1 0.12)
```

- ❑ `extVia1` is the QRC (Extraction) name of the via layer.

Note: `viaLengthList` will be ignored if `extViaLengthList` is specified.

- The `dfiiUnits` variable specifies the units of the via length to the tool, if the tool is unable to identify the length units by itself. Valid values for this variable are `"micron"` and `"meter"`.

```
dfiiUnits = "micron" or dfiiUnits = "meter"
```

- The following filtering options allow you to filter resistors based on either resistor names or layer names:

  - ❑ `selectLayers`

    Selects resistors based on the extracted layer name. All resistors on other layers will be ignored for the analysis.

    Example:

```
selectLayers  =  ("Via1" "mt1" "mt2")
selectLayers  =  ("Via1" "m.*")
```

    SKILL regular expressions are supported.

    You can also provide the list of selected layers using the `_vsaSelectLayers` environment variable.

  - ❑ `selectResistors`

    Selects resistors by their names. All other resistors will be ignored for the analysis.

    Example:

```
selectResistors  =  ("rh12" "re45" "rf25")
selectResistors  =  ("rh.*" "re.*")
```

    SKILL regular expressions are supported.

    You can also provide the list of selected resistors using the `_vsaSelectResistors` environment variable.

  - ❑ `ignoreLayers`

    Ignores resistors based on the extracted layer name. All resistors on other layer will be selected for the analysis.

    Example:

```
ignoreLayers  =  ("Via1" "mt1" "mt2")
ignoreLayers  =  ("Via1" "m.*")
```


© 2020 All Rights Reserved.

SKILL regular expressions are supported.

You can also provide the list of ignored layers using the `_vsaIgnoreLayers` environment variable.

- `ignoreResistors`

  Ignore resistors by their name. All other resistors will be selected for the analysis.

  Example:

  ```
  selectResistors  =  ("rh12" "re45" "rf25")
  selectResistors  =  ("rh.*" "re.*")
  ```

  SKILL regular expressions are supported.

  You can also provide the list of ignored resistors using the `_vsaIgnoreLayers` environment variable.

  Note: The parameters specified in `emDataFile` have higher priority than their corresponding environment variables. If both are specified, the parameters specified in the `emDataFile` will be selected over the corresponding environment variable values.

  Also, the selection list has a higher priority than the ignore list. If `selectLayers` or `selectResistors` parameters are already provided, then the values specified using `ignoreLayers` and `ignoreResistors` parameters will be ignored.

- `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.*")
  ```

  SKILL regular expressions are supported.

  Example:

  ```
  DesignResistors = (
  ```


© 2020 All Rights Reserved.

```
                ("nplusres"        "r" ("wSim" "meter") ("lSim" "meter"))
                 ("pplusres"       "r" "wSim" "lSim")
                 ("qplusres"        )
)
```

Where:
`nplusres, pplusres,` and `qplusres` refer to the design resistor cell names
`r` = resistor property name
`wSim =` width property name
`meter` = unit for width
`lSim =` length property name
`meter` = unit for length

Note: The default unit for both the width and length property name is meter. However, the software also accepts “micron”, “millimeter”, “centimeter”, “mill”, and “inch” as units for these property names.

For details, see Analyzing EM Across Design Resistors in the “EM Analysis” chapter.

User Procedures for Electromigration JMAX Checking

Power IR/EM has its own methodology to perform average, peak, and RMS analyses. But it also provides you options to define your own methodology to calculate current densities.

In EM flow, after you calculate the current density of a presistor, you can compare it with the current density limits to find violations. The current density limits are read from the `emDataFile` that you provide. In addition to providing the `emDataFile`, you can also specify that the current density limits be calculated dynamically.

Power IR/EM provides the following procedures to the users for `rms, DCrms, avg, avgAbs, peak, peakAC, and custom` analysis respectively:

```
vsaGetRmsJmax
vsaGetDCRmsJmax
vsaGetAvgJmax
vsaGetAvgAbsJmax
vsaGetPeakJmax
vsaGetPeakACJmax
vsaGetCustomJmax
```


© 2020 All Rights Reserved.

For example, if you think that peak current density limits cannot be specified using the `emDataFile`, you can write a SKILL function `vsaGetPeakJmax`() in a .il file and load that before starting the EM analysis. The parameters passed to this function are fixed and the function returns a float value for Jmax. Also, this function will be called for each presistor so that you can specify different values for Jmax.

An example of a SKILL function is shown below:

```
procedure(vsaGetPeakJmax(@key
(r 1.0)
(w 1.0)
(temp 25)
(layer "METAL1")
(pResistorName lowerCase(resName)
(Hierarchy hierarchy)
(parameters 'nil)

; 1.0, 1.0, 25, METAL1, lowerCase(resName), and hierarchy are default values used
only for SKILL syntax
```

Argument	Description	Type	Units
`r`	resistor value	float	Ohm
`w`	width	float	Micron
`temp`	simulation temperature	float	Centigrade
`layer`	layer name	string	Text
`pResistorName`	presistor name	string	Text
`Hierarchy`	hierarchy	string	Text
`parameters`	extra property of resistance	table	-

```
    ; Following code is the body of procedure
    prog((
        ; define local variables here
        cdLimit
        )
        ; start writing the logic and equations using r,w, temp and layer
        ; to calculate the Jmax
        ; an example formula is used below
        if(layer == "METAL2" then
```


© 2020 All Rights Reserved.

```
            cdLimit = sqrt(w) + r
        else
            cdLimit = sqrt(w) / r
        )
        return(cdLimit)
        )
)
```

You can define the logic inside the function to calculate Jmax based on your requirements. The function returns a float value that will be used by Power IR/EM to get the EM violations.

An example of a custom function for providing Jmax is shown below:

```
procedure vsaGetCustomJmax(@key
(r 0)
(w 0 )
(layer 'nil )
(temp 0 )
(pResistorName 'nil )
(Hierarchy 'nil)
(paramaters 'nil)
```

The extra properties of resistance are provided as a `skillTable`. You can access the value of a property by specifying `parameters["property_name"]`. If the entry is not present with a particular `'property_name',` the tool will return `'nil`.

By default, the parameter table is not passed to the custom function in order to improve the run time and memory efficiency of the tool. You can enable this by setting the environment variable, `vsaSupport28nmRules` to "`true`". The tool enables this automatically when the EM rules specified in the `emDataFile` have conditions specific to 28nm.

Example:

```
Upper_metal_length = parameters["Lu"]
```

The list of properties include:

Property Name	Description	Type	Units
`Lu`	Upper Metal Segment Length	Float	Micron
`Lb`	Lower Metal Segment Length	Float	Micron

© 2020 All Rights Reserved.

Property Name	Description	Type	Units
`Wu`	Upper Metal Segment Width	Float	Micron
`Wb`	Lower Metal Segment Width	Float	Micron
`a`	Via Area	Float	Micron ^{2} or cm^2

Custom Parameters in emDataFile

You can specify the custom parameters or define the variables for use at run time to set values being used in the `emDataFile`. You can also edit or view custom parameters in the GUI. An *Edit/View Custom Parameters* button is provided in the EM Analysis form.

For details, see EM Analysis of the Power Grid in the “EM Analysis” chapter.

Example: Provided below is an example to set the via resistance based on a min, typ, max corner parameter:

```
--- User file ( test.il )  ---
          vsaDefVariable(“Corner” “nominal”)
          vsaDefVariable(“CDM”      1.0)

procedure( getResPerCut()    ; user defined skill function
          prog( (
               resVal
          )
          case( vsaGetVariable(“Corner”)
               (“min”    resVal = 0.25)
               (“max”    resVal = 0.35)
               (t              resVal = 0.30)
          ) ; case

          return( resVal )
          )
```


© 2020 All Rights Reserved.

```
          )

procedure( getCurrDensityMultiplier()     ; user defined skill function
          prog( (
               )



          return( evalstring(vsaGetVariable("CDM")
          )
          )
--- emdataFile.txt   ---
(
preExecuteFiles = ("test.il")   ;  Read user files only once.

res_per_cut                        =  getResPerCut()
curr_density_multiplier =  getCurrDensityMultiplier()
......
avgCurrentDensSpecList = (
            ...
         (nil layer "via1" minW  0.000 maxW -1.000 res res_per_cut  currentDensity
     (0.335 *  curr_density_multiplier * w), 100)
...
)
......
)
```


© 2020 All Rights Reserved.

SKILL Functions

- vsaDefVariable
- vsaGetVariable
- vsaWarn
- vsaError
- vsaDbType
- vsaRegisterEMDataFile
- vsaRegisterQrcTechFile
- vsaRegisterLayerMapFile
- vsaGetR
- vsaResGetIrms
- vsaResGetIpeak
- vsaResGetIavg
- vsaResGetX
- vsaResGetY
- vsaResGetLayer
- vsaGetNode
- vsaNodeGetIR
- vsaEMGetQRCDirAndRunName
- vsaEMGetSimDirAndAnalysisType
- vsaIRGetSimDirAndAnalysisType

vsaDefVariable

```
vsaDefVariable(
     ?varName parameter_name
     ?value parameter_value
     )
```


© 2020 All Rights Reserved.

Description

Defines a new variable or custom parameter and provides its value. If the variable is already defined, this variable overwrites the current value of the variable with the new value.

Argument

varName	Specifies the name of the variable.
value	Specifies the value of the variable.

Values Returned

None.

vsaGetVariable

```
vsaGetVariable(
     ?varName parameter_name
     )
     => string | nil
```

Description

Gets the value of the variable or custom parameter.

Argument

varName	Specifies the name of the variable.

Value Returned

`string`	Returns the value of the custom parameter as a string.
`nil`	Returns `nil` when the custom parameter is not defined.

© 2020 All Rights Reserved.

Note: Here custom parameters are different from the SKILL variables. You can access it only through the APIs `vsaDefVariable` and `vsaGetVariable`.

vsaWarn

```
vsaWarn(
     ?message warning_message
     ?mode display_mode
     )
```

Description

Creates a warning message in the command interpreter window (CIW) or the dialog box, or both.

Arguments

message	Displays the warning message as a string.
mode	Specifies the display mode (`"all"/ "ciw" / "dbox"`). Default mode is `all`. `all` - message is displayed in the CIW and the dialog box `ciw` - message is displayed in the CIW `dBox` - message is displayed in the dialog box

Values Returned

None.

vsaError

```
vsaError(
     ?message error_message
     ?mode display_mode
     )
```


© 2020 All Rights Reserved.

Description

Creates an error message in the CIW or the dialog box, or both.

Arguments

message	Displays the error message as a string.
mode	Specifies the display mode (`"all"/ "ciw"/ "dbox"`). Default mode is `all`. `all` - message is displayed in the CIW and the dialog box `ciw` - message is displayed in the CIW `dBox` - message is displayed in the dialog box

Values Returned

None.

vsaDbType

```
vsaDbType(
     ?simDir simulation_dir
     )
     => 32/64/-1
```

Description

Considers the simulation directory as the input and returns the mode of `vavo.db` file.

Argument

simDir	Specifies the simulation directory path. This is the same as the one entered in the *Simulation Directory* field in the EM analysis form.

© 2020 All Rights Reserved.

Values Returned

32	If vavo-db is in 32 bit mode
64	If vavo-db is in 64 bit mode
-1	If there is any error or the software is unable to detect any mode

vsaRegisterEMDataFile

```
vsaRegisterEMDataFile(
     ?emDataFile file_name
     )
```

Description

Adds the specified `emDataFile` to the list of available `emDataFiles`. The specified `emDataFile` will appear in the drop-down list in the Electromigration Power Grid Analysis and Electromigration Signal Analysis forms for the user to select for EM analysis.

Argument

emDataFile	Specifies the name of the `emDataFile` to be added to the list of available emDataFiles.

Values Returned

None.

vsaRegisterQrcTechFile

```
vsaRegisterQrcTechFile(
     ?qrcTechFile @optional (mapFile nil)
     )
```


© 2020 All Rights Reserved.

Description

Adds the specified `qrcTechfile` to the list of available QRC technology files. You can also add the corresponding `mapFile`. The specified `qrcTechfile` and `mapFile` will appear in the drop-down list in the Electromigration Power Grid Analysis and Electromigration Signal Analysis forms for you to select for EM analysis.

Note: Specifying the `mapFile` is optional. If the `mapFile` is specified here, the tool will auto-sync the `mapFile` with `qrcTechFile` when `qrcTechFile` is changed in the GUI.

Arguments

`qrcTechFile`	Specifies the name of `qrcTechFile` to be added to the list of available QRC technology files.
`mapFile`	Specifies the name of the corresponding `mapFile` to be added to the list of available map files.

Values Returned

None.

vsaRegisterLayerMapFile

```
vsaRegisterLayerMapFile(
     ?mapFile file_name
     )
```

Description

Adds the specified `mapFile` to the list of available name mapping files. The specified `mapFile` will appear in the drop-down list in the Electromigration Power Grid Analysis and Electromigration Signal Analysis forms and can be selected for EM analysis.

Argument

`mapFile`	Specifies the name of the `mapFile` to be added to the list of available name mapping files.

© 2020 All Rights Reserved.

Values Returned

None.

vsaGetR

```
vsaGetR(
     ?x x_coordinate
     ?y y_coordinate
     ?layer layerName
     [?netName netName]
     )
     => string | nil
```

Description

Returns the name of the presistor of the specified net, on the specified layer, near the specified x and y coordinates.

Argument

`x`	Specifies the x coordinate of the search point.
`y`	Specifies the y coordinate of the search point.
`layer`	Specifies the name of the layer on which to search for the presistor.
`netName`	Filters the presistor based on the specified net name. This is an optional argument.

Values Returned

`string`	Returns the name of the presistor as a string.
`nil`	Returns `nil` when the presistor is not found.

Example

```
vsaGetR(1.1 2.2 "M1" "VDD")   =>  "R1"
```


© 2020 All Rights Reserved.

In this example, the software returns the name of the presistor, R1, found on metal layer, M1, on net, VDD, at the specified x and y coordinates.

vsaResGetIrms

```
vsaResGetIrms(
     ?R resistor_name
     )
     => f_current | nil
```

Description

Returns the value of the rms current for the specified presistor.

Argument

`R`	Specifies the name of the presistor for which the rms current value is to be returned.

Value Returned

`f_current`	Returns the value of rms current in ampere. This is a floating-point number.
`nil`	Returns `nil` when no data is available.

Example

```
vsaResGetIrms("R1")   =>  1.23e-03
```

vsaResGetIpeak

```
vsaResGetIpeak(
     ?R resistor_name
     )
     => f_current | nil
```


© 2020 All Rights Reserved.

Description

Returns the value of the peak current for the specified presistor.

Argument

`R`	Specifies the name of the presistor for which the peak current value is to be returned.

Value Returned

`f_current`	Returns the value of peak current in ampere. This is a floating-point number.
`nil`	Returns `nil` when no data is available.

Example

```
vsaResGetIpeak("R1")   =>  1.23e-03
```

vsaResGetIavg

```
vsaResGetIavg(
     ?R resistor_name
     )
     => f_current | nil
```

Description

Returns the value of the average current for the specified presistor.

Argument

`R`	Specifies the name of the presistor for which the average current value is to be returned.

© 2020 All Rights Reserved.

Value Returned

`f_current`	Returns the value of average current in ampere. This is a floating-point number.
`nil`	Returns `nil` when no data is available.

Example

```
vsaResGetIavg("R1")   =>  1.23e-03
```

vsaResGetX

```
vsaResGetX(
     ?R resistor_name
     )
     => f_xcoordinate | nil
```

Description

Returns the value of the x coordinate for the specified presistor.

Argument

`R`	Specifies the name of the presistor for which the value of the x coordinate is to be returned.

Value Returned

`f_xcoordinate`	Returns the value of the x coordinate for the specified presistor.
`nil`	Returns `nil` when no presistor is found.

Example

```
vsaResGetX("R1")   =>  1.23
```


© 2020 All Rights Reserved.

vsaResGetY

```
vsaResGetY(
     ?R resistor_name
     )
     => f_ycoordinate
```

Description

Returns the value of the y coordinate for the specified presistor.

Argument

`R`	Specifies the name of the presistor for which the value of the y coordinate is to be returned.

Value Returned

`f_ycoordinate`	Returns the value of the y coordinate for the specified presistor.
`nil`	Returns `nil` when no presistor is found.

Example

```
vsaResGetY("R1")   =>  4.56
```

vsaResGetLayer

```
vsaResGetLayer(
     ?R resistor_name
     )
     => s_layername | nil
```

Description

Returns the layer name for the specified presistor.

© 2020 All Rights Reserved.

Argument

`R`	Specifies the name of the presistor for which the layer name is to be returned.

Value Returned

`s_layername`	Returns the name of the layer for the specified presistor as a string.
`nil`	Returns `nil` when no data is available.

Example

```
vsaResGetLayer("R1")   =>  "M1"
```

vsaGetNode

```
vsaGetNode(
     ?x x_coordinate
     ?y y_coordinate
     ?layer layerName
     [?netName netName]
     )
     => s_nodename | nil
```

Description

Returns the name of the node of the specified net, on the specified layer, near the specified x and y coordinates.

Argument

`x`	Specifies the x coordinate of the search point.
`y`	Specifies the y coordinate of the search point.
`layer`	Specifies the name of the layer on which to search for the node.

© 2020 All Rights Reserved.

`netName`	Filters the node based on the specified net name. This is an optional argument.

Values Returned

`s_nodename`	Returns the name of the node as a string.
`nil`	Returns `nil` when the node is not found.

Example

```
vsaGetNode(1.1 2.2 "M1" "VDD")   =>  "VDD:8"
```

vsaNodeGetIR

```
vsaNodeGetIR(
     ?n node_name
     )
     => f_IRdrop | nil
```

Description

Returns the IR drop value for the specified node in volts.

Argument

`n`	Specifies the name of the node for which the IR drop value is to be returned.

Value Returned

`f_IRdrop`	Returns the IR drop value for the specified node in volts.
`nil`	Returns `nil` when no data is available.

© 2020 All Rights Reserved.

Example

```
vsaNodeGetIR("VDD:8")   =>  1.2
```

vsaEMGetQRCDirAndRunName

```
vsaEMGetQRCDirAndRunName()
     => list (s_qrcDirPath s_qrcRunName) | nil
```

Description

This function returns the name of the QRC directory and the corresponding QRC runname as a list.

Argument

None.

Values Returned

`s_qrcDirPath`	Returns the name of the QRC directory.
`s_qrcRunName`	Returns the name of the QRC run.
`nil`	Returns `nil` when no data is available.

Example

```
vsaEMGetQRCDirAndRunName() => ("/anuk/VPSL/testcase/vco_lvs_oa2" "vco")
```

vsaEMGetSimDirAndAnalysisType

```
vsaEMGetSimDirAndAnalysisType()
     => list (s_simDirPath s_simAnalysisType) | nil
```

Description

This function returns the name of the simulation directory and the type of analysis being used for EM analysis as a list.

© 2020 All Rights Reserved.

Argument

None.

Values Returned

`s_simDirPath`	Returns the name of the simulation directory.
`s_simAnalysisType`	Returns the name of the analysis TYPE.
`nil`	Returns `nil` when no data is available.

Example

```
vsaEMGetSimDirAndAnalysisType() => ("/anuk/VPSL/testcase/usim_dy_runs"
     "TRANSIENT")
```

vsaIRGetSimDirAndAnalysisType

```
vsaIRGetSimDirAndAnalysisType()
     => list (s_simDirPath s_simAnalysisType) | nil
```

Description

This function returns the name of the simulation directory and the type of analysis being used for IR drop analysis as a list.

Argument

None.

Values Returned

`s_simDirPath`	Returns the name of the simulation directory.
`s_simAnalysisType`	Returns the name of the analysis TYPE.
`nil`	Returns `nil` when no data is available.

© 2020 All Rights Reserved.

Example

```
vsaIRGetSimDirAndAnalysisType() => ("/anuk/VPSL/testcase/usim_dy_runs"
     "TRANSIENT")
```


© 2020 All Rights Reserved.

Call-Back Functions

These are pre-defined functions that can be set by the user in their code. You can use these functions to call some pre-processing functions within the tool. You can load these functions either through the `.cdsinit` file or through the CIW before invoking Power IR/EM. The following functions are provided:

- vsaEMInitUserCallBack
- vsaEMUserCallBack
- vsaIRUserCallBack

vsaEMInitUserCallBack

```
vsaEMInitUserCallBack()
```

A callback function that is called when the EM Analysis form is invoked for the first time.

Example

```
----- preload.il ------
    procedure( vsaEMInitUserCallBack()
vsaRegisterEMDataFile("/projects/technology1/emDataFile.txt")
vsaRegisterEMDataFile("/projects/technology2/emDataFile.txt")
    )
----- .cdsinit ------
   load("preload.il")
```

Argument

None.

Values Returned

None.

vsaEMUserCallBack

```
vsaEMUserCallBack()
```

A callback function that is called to load the user-specified SKILL functions at the time of clicking the Apply/OK buttons in the EM analysis form.

Argument

© 2020 All Rights Reserved.

None.

Values Returned

None.

Example

```
----- preload.il ------
    procedure(vsaEMUserCallBack()
let(  (a b)
a = vsaEMGetQRCDirAndRunName()
    printf("QRC Dir Name and Run Name: %L\n", a)
b =  vsaEMGetSimDirAndAnalysisType()
    printf("Simulation Dir and Analysis Type: %L \n", b)
    )
)
----- .cdsinit ------
   load("preload.il")
--------CDS.log--------
…….
……..
\a hiFormDone(_vsaElectromigrationForm)
\o  QRC Dir Name and Run Name: ("/VPSL/testcase/vco_lvs_oa2" "vco")
\o  Simulation Dir and Analysis Type: ("/VPSL/testcase/usim_dy_runs" "TRANSIENT")
…….
…….
```

vsaIRUserCallBack

```
vsaIRUserCallBack()
```

A callback function that is called to load the user-specified SKILL functions at the time of clicking the Apply/OK buttons in the IR drop analysis form.

Argument

None.

Values Returned

None.

Example

© 2020 All Rights Reserved.

```
----- preload.il ------
    procedure(vsaIRUserCallBack()
let( ( a)
    a = vsaIRGetSimDirAndAnalysisType()
    printf(" Simulation Dir and Analysis Type: %L\n", a)
    )
)
----- .cdsinit ------
   load("preload.il")
------CDS.log------
……————————
————————————————.
\a hiFormDone(_vsaIRDropForm)
\o  Simulation Dir and Analysis Type: ("/VPSL/testcase/usim_dy_runs" "TRANSIENT")
\o
————————————————.
————————————————
```


© 2020 All Rights Reserved.

Sample emDataFile

```
(
routingLayers = ("poly1" "m1" "m2" "m3" "m4" "m5" "m6" "m7" "m8" "m9" "MD")
viaLayers = ( "cw" "v1" "v2" "v3" "v4" "v5" "v6" "v7" "v8")

xrefLayers = (
( "poly1con" ("cw" "cw"))
( "poly" ("poly1" "poly1"))
)

avgCurrentDensSpecList = (
(nil layer "poly1" minW 0.0 maxW -1.0 currentDensity ((1.100,100) (1.110,110)
(1.125,125)))
(nil layer "m1" minW 0.0 maxW 0.16 currentDensity ((1.434*3.9*(w-0.05)/w,100)
(3.9,110) (0.387*3.9*(w-0.05)/w,125)))
(nil layer "m1" minW 0.16 maxW -1.0 currentDensity ((14*(w-0.05)*deltaT/w,100)
(1.434 *3.9*(w-0.05)/w, 110) (5.3,125)))
(nil layer "m2" minW 0.0 maxW -1.0 currentDensity ((1.9100,100) (1.9110,110)
(1.9125,125)))
(nil layer "m3" minW 0.0 maxW -1.0 currentDensity ((1.9100,100) (1.9110,110)
(1.9125,125)))
(nil layer "m4" minW 0.0 maxW -1.0 currentDensity ((1.9100,100) (1.9110,110)
(1.9125,125)))
(nil layer "m5" minW 0.0 maxW -1.0 currentDensity ((1.9100,100) (1.9110,110)
(1.9125,125)))
(nil layer "m6" minW 0.0 maxW -1.0 currentDensity ((1.9100,100) (1.9110,110)
(1.9125,125)))
(nil layer "m7" minW 0.0 maxW -1.0 currentDensity ((1.9100,100) (1.9110,110)
(1.9125,125)))
(nil layer "m8" minW 0.0 maxW -1.0 currentDensity ((4.9100,100) (4.9110,110)
(4.9125,125)))
(nil layer "cw" minW 0.0 maxW -1 res 11.04 currentDensity ((1.6100,100) (1.119,110)
(1.119,125)))
(nil layer "v1" minW 0.0 maxW -1 res 0.68 currentDensity ((1.2100,100) (1.2110,110)
(1.2125,125)))
(nil layer "v2" minW 0.0 maxW -1 res 0.68 currentDensity ((1.2100,100) (1.2110,110)
(1.2125,125)))
(nil layer "v3" minW 0.0 maxW -1 res 0.68 currentDensity ((1.2100,100) (1.2110,110)
(1.2125,125)))
(nil layer "v4" minW 0.0 maxW -1 res 0.68 currentDensity ((1.2100,100) (1.2110,110)
(1.2125,125)))
(nil layer "v5" minW 0.0 maxW -1 res 0.68 currentDensity ((1.2100,100) (1.2110,110)
(1.2125,125)))
(nil layer "v6" minW 0.0 maxW -1 res 0.68 currentDensity ((1.2100,100) (1.2110,110)
(1.2125,125)))
```


© 2020 All Rights Reserved.

```
(nil layer "v7" minW 0.0 maxW -1 res 0.42 currentDensity ((1.8100,100) (1.8110,110)
(1.8125,125)))
)

rmsCurrentDensSpecList = (
(nil layer "poly1" minW 0.0 maxW -1.0 currentDensity ((1.100,100) (1.110,110)
(1.125,125)))
(nil layer "m1" minW 0.0 maxW 0.16 currentDensity ( (1.434 *3.9*(w-0.05)/w,100)
(3.9,110) (0.387*3.9*(w-0.05)/w,125)))
(nil layer "m1" minW 0.16 maxW -1.0 currentDensity ( (14*(w-0.05)*deltaT/w,100)
(1.434 *3.9*(w-0.05)/w,110) (5.3,125)))
(nil layer "m2" minW 0.0 maxW -1.0 currentDensity ((2.9100,100) (2.9110,110)
(2.9125,125)))
(nil layer "m3" minW 0.0 maxW -1.0 currentDensity ((1.9100,100) (1.9110,110)
(1.9125,125)))
(nil layer "m4" minW 0.0 maxW -1.0 currentDensity ((1.9100,100) (1.9110,110)
(1.9125,125)))
(nil layer "m5" minW 0.0 maxW -1.0 currentDensity ((1.9100,100) (1.9110,110)
(1.9125,125)))
(nil layer "m6" minW 0.0 maxW -1.0 currentDensity ((1.9100,100) (1.9110,110)
(1.9125,125)))
(nil layer "m7" minW 0.0 maxW -1.0 currentDensity ((1.9100,100) (1.9110,110)
(1.9125,125)))
(nil layer "m8" minW 0.0 maxW -1.0 currentDensity ((4.9100,100) (4.9110,110)
(4.9125,125)))
(nil layer "cw" minW 0.0 maxW -1 res 11.04 currentDensity ((1.6100,100) (1.119,110)
(1.119,125)))
(nil layer "v1" minW 0.0 maxW -1 res 0.68 currentDensity ((1.2100,100) (1.2110,110)
(1.2125,125)))
(nil layer "v2" minW 0.0 maxW -1 res 0.68 currentDensity ((1.2100,100) (1.2110,110)
(1.2125,125)))
(nil layer "v3" minW 0.0 maxW -1 res 0.68 currentDensity ((1.2100,100) (1.2110,110)
(1.2125,125)))
(nil layer "v4" minW 0.0 maxW -1 res 0.68 currentDensity ((1.2100,100) (1.2110,110)
(1.2125,125)))
(nil layer "v5" minW 0.0 maxW -1 res 0.68 currentDensity ((1.2100,100) (1.2110,110)
(1.2125,125)))
(nil layer "v6" minW 0.0 maxW -1 res 0.68 currentDensity ((1.2100,100) (1.2110,110)
(1.2125,125)))
(nil layer "v7" minW 0.0 maxW -1 res 0.42 currentDensity ((1.8100,100) (1.8110,110)
(1.8125,125)))
)

DCrmsCurrentDensSpecList = (
(nil layer "poly1" minW 0.0 maxW -1.0 currentDensity ((1.100,100) (1.110,110)
(1.125,125)))
(nil layer "m1" minW 0.0 maxW 0.16 currentDensity ( (1.434 *3.9*(w-0.05)/w,100)
(3.9,110) (0.387*3.9*(w-0.05)/w,125)))
```


© 2020 All Rights Reserved.

```
(nil layer "m1" minW 0.16 maxW -1.0 currentDensity ( (14*(w-0.05)*deltaT/w,100)
(1.434 *3.9*(w-0.05)/w,110) (5.3,125)))
(nil layer "m2" minW 0.0 maxW -1.0 currentDensity ((2.9100,100) (2.9110,110)
(2.9125,125)))
(nil layer "m3" minW 0.0 maxW -1.0 currentDensity ((1.9100,100) (1.9110,110)
(1.9125,125)))
(nil layer "m4" minW 0.0 maxW -1.0 currentDensity ((1.9100,100) (1.9110,110)
(1.9125,125)))
(nil layer "m5" minW 0.0 maxW -1.0 currentDensity ((1.9100,100) (1.9110,110)
(1.9125,125)))
(nil layer "m6" minW 0.0 maxW -1.0 currentDensity ((1.9100,100) (1.9110,110)
(1.9125,125)))
)
peakCurrentDensSpecList = (
(nil layer "poly1" minW 0.0 maxW -1.0 currentDensity ((1.100,100) (1.110,110)
(1.125,125)))
(nil layer "m1" minW 0.0 maxW 0.16 currentDensity ( (1.434 *3.9*(w-0.05)/w,100)
(3.9,110) (0.387*3.9*(w-0.05)/w,125)))
(nil layer "m1" minW 0.16 maxW -1.0 currentDensity ( (14*(w-0.05)*deltaT/w,100)
(1.434 *3.9*(w-0.05)/w,110) (5.3,125)))
(nil layer "m2" minW 0.0 maxW -1.0 currentDensity ((3.9100,100) (3.9110,110)
(3.9125,125)))
(nil layer "m3" minW 0.0 maxW -1.0 currentDensity ((1.9100,100) (1.9110,110)
(1.9125,125)))
(nil layer "m4" minW 0.0 maxW -1.0 currentDensity ((1.9100,100) (1.9110,110)
(1.9125,125)))
(nil layer "m5" minW 0.0 maxW -1.0 currentDensity ((1.9100,100) (1.9110,110)
(1.9125,125)))
(nil layer "m6" minW 0.0 maxW -1.0 currentDensity ((1.9100,100) (1.9110,110)
(1.9125,125)))
(nil layer "m7" minW 0.0 maxW -1.0 currentDensity ((1.9100,100) (1.9110,110)
(1.9125,125)))
(nil layer "m8" minW 0.0 maxW -1.0 currentDensity ((4.9100,100) (4.9110,110)
(4.9125,125)))
(nil layer "cw" minW 0.0 maxW -1 res 11.04 currentDensity ((1.6100,100) (1.119,110)
(1.119,125)))
(nil layer "v1" minW 0.0 maxW -1 res 0.68 currentDensity ((1.2100,100) (1.2110,110)
(1.2125,125)))
(nil layer "v2" minW 0.0 maxW -1 res 0.68 currentDensity ((1.2100,100) (1.2110,110)
(1.2125,125)))
(nil layer "v3" minW 0.0 maxW -1 res 0.68 currentDensity ((1.2100,100) (1.2110,110)
(1.2125,125)))
(nil layer "v4" minW 0.0 maxW -1 res 0.68 currentDensity ((1.2100,100) (1.2110,110)
(1.2125,125)))
(nil layer "v5" minW 0.0 maxW -1 res 0.68 currentDensity ((1.2100,100) (1.2110,110)
(1.2125,125)))
(nil layer "v6" minW 0.0 maxW -1 res 0.68 currentDensity ((1.2100,100) (1.2110,110)
(1.2125,125)))
```


© 2020 All Rights Reserved.

```
(nil layer "v7" minW 0.0 maxW -1 res 0.42 currentDensity ((1.8100,100) (1.8110,110)
(1.8125,125)))
)
viaWidthList = ( ( "Cont" 0.2 ) ( "Via1" 0.2 ) )
 //This is optional, the software will try to get this info from the
 // Virtuoso technology file
```


© 2020 All Rights Reserved.

10

Voltage Source Regions

- Overview on page 244
- Creating Voltage Source Regions on page 244
- SKILL Function on page 250

© 2020 All Rights Reserved.

Overview

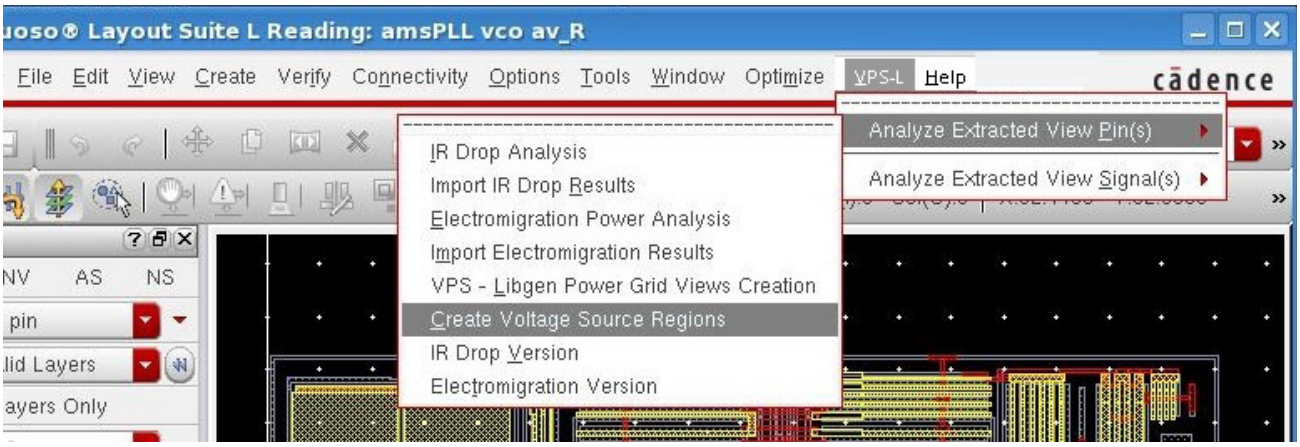
To minimize the EM/IR Drop in the I/O pads, Power IR/EM provides the capability to create voltage source regions in the design before simulation. This helps in avoiding false EM/IR violations by ensuring minimum EM/IR Drop in selected pads in the design.

Creating Voltage Source Regions

Follow these steps to create voltage source regions.

- In the *VPS-L* menu, choose *Analyze Extracted View Pin(s) – Create Voltage Source Regions*. This is shown in the image below.

Figure 10-1 Virtuoso Layout Suite L – Voltage Source Regions



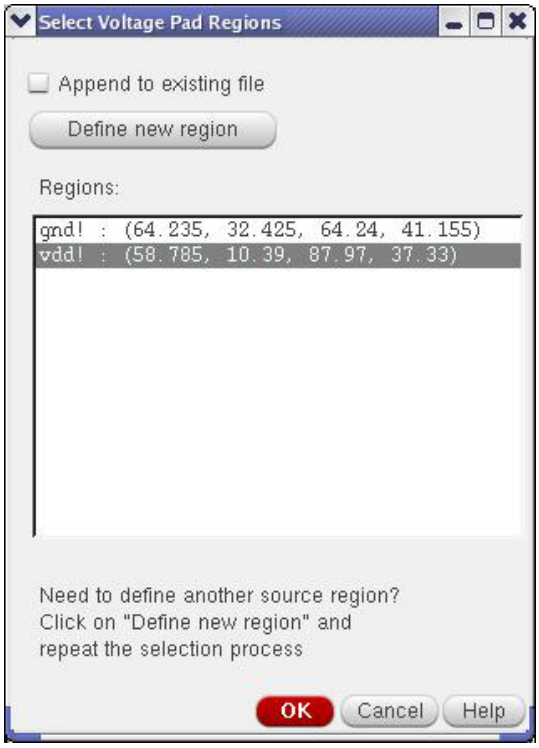
- The Select Voltage Pad Regions form opens.

© 2020 All Rights Reserved.

Figure 10-2 Selecting the Voltage Source Regions

- In this form, select *Append to existing file* to add the source regions created in the previous runs.
- Select *Define new region* to select the region where you want to create a source in the design. Click and drag the cursor to define the region on the layout. When you select the region on the layout, the software selects all instances connected to the power rails and skips all instances connected to signals. The selected source region gets added in the list box provided under the R*egions* field in the Select Voltage Pad Regions form. This is shown below.

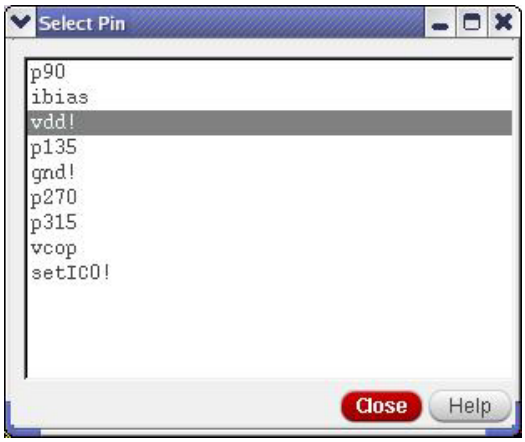
© 2020 All Rights Reserved.



Note: The *Define new region* option is used to select a specific I/O pad that belongs only to a specific power pin.

If the selected region has multiple power rails, the Select Pin form opens. In this form, you can select the pin and click *OK*. This form is shown below.

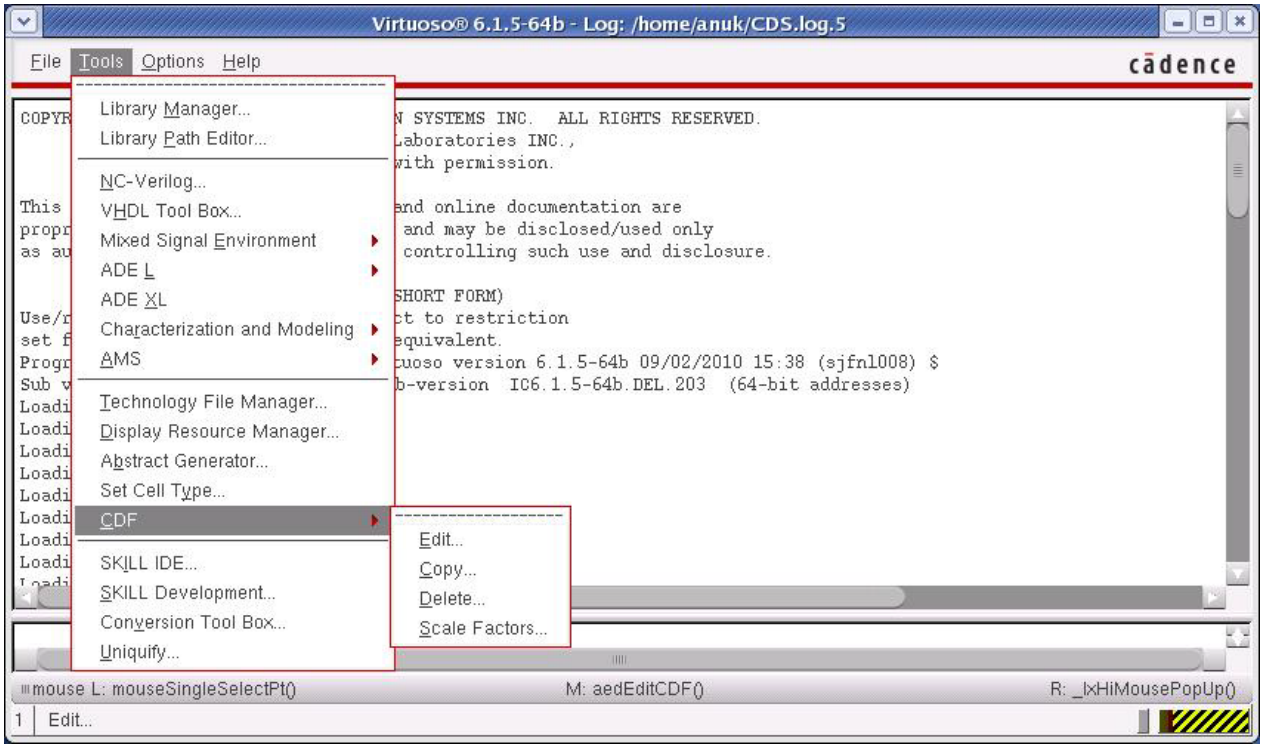
Figure 10-3 Selecting the Pin



© 2020 All Rights Reserved.

- In the Virtuoso Command Interpreter Window (CIW), select *Tools - CDF - Edit*. This is shown below.

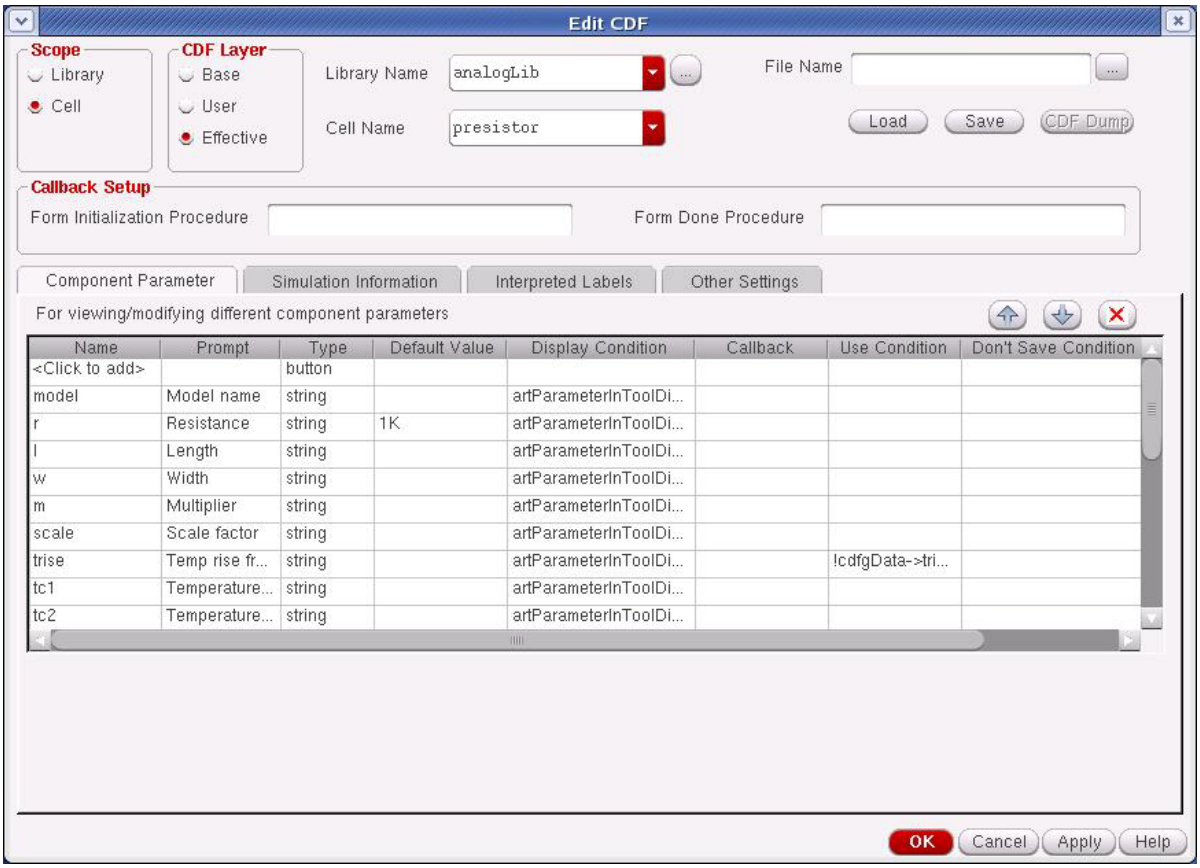
Figure 10-4 Virtuoso Command Interpreter Window - Accessing Edit CDF Form



- The Edit CDF form appears. This form is shown below. Select the Cadence analog *Library Name* (analogLib) and *Cell Name* (presistor).

© 2020 All Rights Reserved.

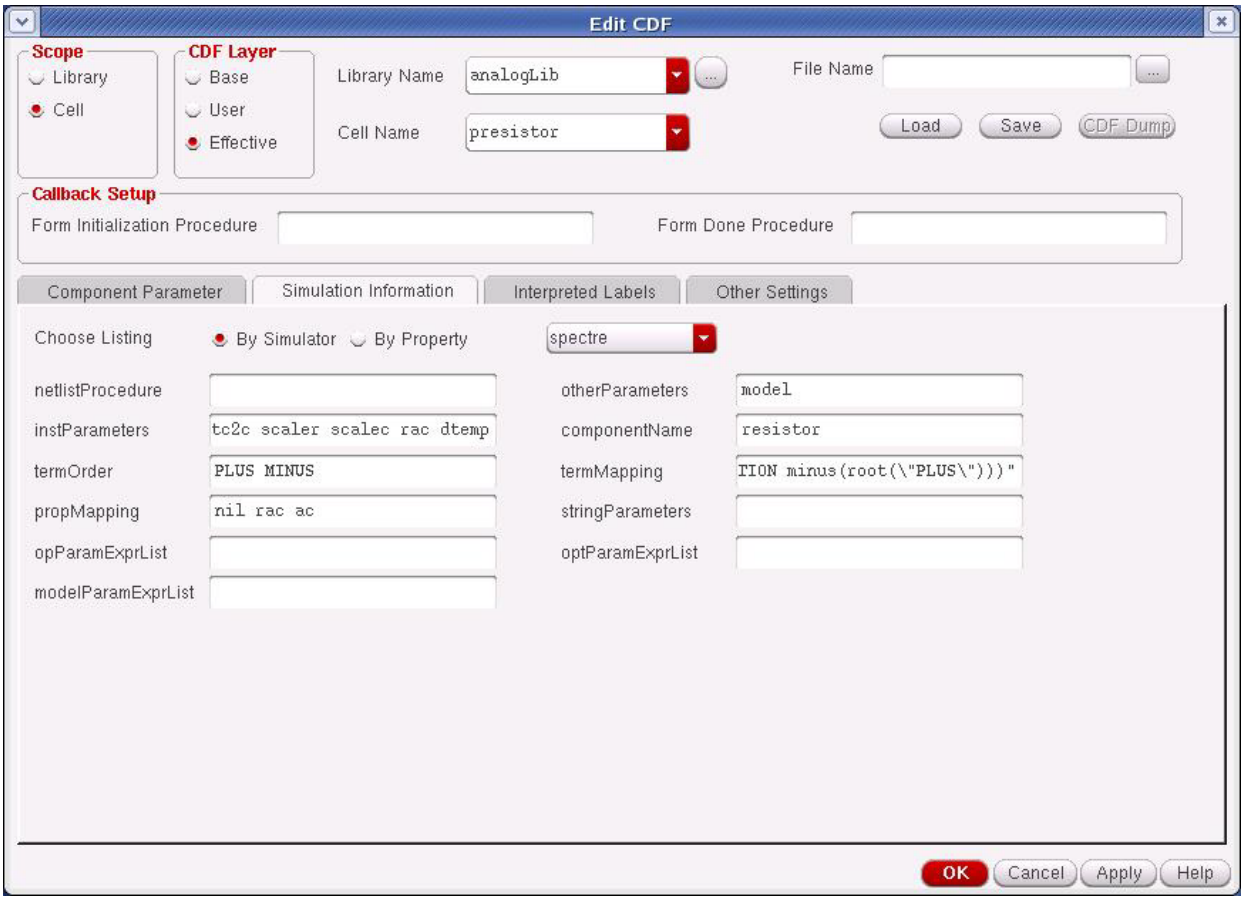
Figure 10-5 Edit CDF Form



- Select *Simulation Information* tab. The form is shown below.

© 2020 All Rights Reserved.

Figure 10-6 Edit CDF Form - Simulation Information



- To select the type of simulator, in the *Choose Listing* field, select *By Simulator.* From the drop-down list, select Spectre as the simulator.
- Select the netlist procedure, `vsaNetlistProc`. For details about this function, see vsaNetlistProc.
- Click *Apply* and then click *OK*.

- Run Spectre simulation. See Running a Spectre Simulation in the “IR/EM Analysis Flow” chapter.
- Perform EM/IR analysis. See EM Analysis of the Power Grid in the “EM Analysis” chapter or IR Drop Analysis in the “IR Drop Analysis” chapter.
- The EM analysis results appear in the Electromigration Analysis Results form. In this form, you can select and highlight a presistor in the design on the layout.

© 2020 All Rights Reserved.

SKILL Function

vsaNetlistProc

```
vsaNetlistProc(
     )
     => t/nil
```

Description

Specifies the pre-defined netlist procedure used for simulation. This function is used to inform the simulator about pad regions, which are regions in the design with zero voltage drop.

After defining the pad regions in the design, you have to set up calls for `vsaNetlistProc` before netlisting. You can also specify your own netlist procedure along with the pre-defined netlist procedure by calling the pre-defined procedure into your netlist procedure destination.

Arguments

None.

Values Returned

`t`	Returns `t` for successful call and return of the netlist procedure.
`nil`	Returns `nil` for unsuccessful call and return of the netlist procedure.

© 2020 All Rights Reserved.

11

Batch Mode Execution

- Overview on page 252
- Batch Command Syntax on page 252
- Command File Syntax on page 253
  - Common Commands on page 253
  - IR Drop Analysis Commands on page 263
  - EM Analysis Commands on page 273
- Sample Command File on page 289
- Sample EM Analysis Output on page 290

© 2020 All Rights Reserved.

Overview

Power IR/EM can be run as a batch command, running from Virtuoso in no-graph mode. Batch mode can be used to perform IR Drop analysis, EM analysis, and for power-grid view (PGV) creation.

Batch Command Syntax

The batch mode is executed from the command line as follows:

```
vsabatch -cmd <cmd_file> -log <log-file>
```

Where:

<cmd_file> is an ASCII file of SKILL commands to define the input data, control the execution of analysis, and specify the output location. For information about the various SKILL commands, see Command File Syntax. To view an example, *cmd_file,* see Sample Command File.

Tip

Starting a line with a semicolon (‘`;`’) indicates a comment.

<log_file> specifies the name and location of a log file of the execution. If `-log` is not specified, the default log file `cds-vsa.log` will be used.

Important

In batch mode, environment variable settings are not used. Therefore, values of environment variables must be specified as commands in the command file. These are `_vsa_em_data_file, _vsa_simulation_directory`, and `_vsa_qrc_run_directory`.

© 2020 All Rights Reserved.

Command File Syntax

The following sections provide detailed description of the command files, their parameters, default values, and examples:

- Common Commands on page 253
- IR Drop Analysis Commands on page 263
- EM Analysis Commands on page 273

Common Commands

- _vsa_extracted_lib_cell_view “library” “cell” “view” on page 254
- _vsa_testbench_lib_cell_view “library” “cell” “view” on page 255
- _vsa_hierarchy “hierarchy_name” on page 256
- _vsa_pinmap "extrPinName" "testBenchPinName" on page 257
- _vsa_simulation_directory “sim_dir_path” on page 258
- _vsa_pres_cellname “cellname” on page 259
- _vsa_analysis_data “TRANSIENT” | “DCOP” on page 260
- _vsa_vavo_db_enable t | nil on page 261
- exit on page 262

© 2020 All Rights Reserved.

_vsa_extracted_lib_cell_view “library” “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"
```


© 2020 All Rights Reserved.

_vsa_testbench_lib_cell_view “library” “cell” “view”

```
_vsa_testbench_lib_cell_view "library" "cell" "view"
```

Specifies the schematic testbench. This is an optional command. However, if a schematic testbench is not specified then the `_vsa_hierarchy` command must be specified.

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"
```


© 2020 All Rights Reserved.

_vsa_hierarchy “hierarchy_name”

```
_vsa_hierarchy "hierarchy_name"
```

Specifies the hierarchy of the design. For a single level of hierarchy, this is an optional command. If the hierarchy is specified, the `_vsa_testbench_lib_cell_view` command must also be specified.

For multiple levels of hierarchy, if you want to run in batch mode, then the testbench lib/cell/view name must be provided in addition to the hierarchy name. In this case, you do not need to specify the pinmap.

Parameter

`hierarchy_name`	Specifies the schematic cell name.

Example

The following command specifies the hierarchy of the design:

```
_vsa_hierarchy "/X1"
```


© 2020 All Rights Reserved.

_vsa_pinmap "extrPinName" "testBenchPinName"

```
_vsa_pinmap "extrPinName" "testBenchPinName"
```

Specifies the pin name mapping between the testbench, through which the simulation was completed, and the specified extracted view hierarchy. This command is required when `_vsa_hierarchy` is used for a single level of hierarchy. It is not required when the `_vsa_testbench_lib_cell_view` command is used.

Parameters

`extrPinName`	Specifies the extracted view pin name.
`testBenchPinName`	Specifies the testbench or schematic pin name.

Example

The following commands specify the pin name mapping between the testbench and the extracted view.

```
_vsa_pinmap  "p0" "/out0"
_vsa_pinmap  "p135" "/out3"
```


© 2020 All Rights Reserved.

_vsa_simulation_directory “sim_dir_path”

```
_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"
```


© 2020 All Rights Reserved.

_vsa_pres_cellname “cellname”

```
_vsa_pres_cellname "cellname"
```

Specifies the cell name of the parasitic resistor used by RCX, and 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"
```


© 2020 All Rights Reserved.

_vsa_analysis_data “TRANSIENT” | “DCOP”

```
_vsa_analysis_data "TRANSIENT" | "DCOP"
```

Specifies the type of analysis to perform. This command is optional.

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.

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"
  ```


© 2020 All Rights Reserved.

_vsa_vavo_db_enable t | nil

```
_vsa_vavo_db_enable t | nil
```

Specifies whether `vavo.db` database will be used for IR drop/EM analysis or not. When set to `nil`, PSF database is used. This command is specified only when you have `vavo.db` database in your simulation directory.

Default: `t`

Parameters

`t`	Specifies that `vavo.db` database will be used for analysis.
`nil`	Specifies that PSF database will be used for analysis.

Example

The following command specifies that the analysis should be performed using the `vavo.db` database:

```
_vsa_vavo_db_enable t
```


© 2020 All Rights Reserved.

exit

```
exit
```

Closes the `virtuoso -nodisplay` application and is used at the end of the command file. If `exit` is not specified, the program will not terminate after the IR/EM analysis is complete. It will wait for further input from you.

This is not desirable in an application running without display.

Example

The following command is used at the end of the command file:

```
exit
```


© 2020 All Rights Reserved.

IR Drop Analysis Commands

- _vsa_use_explicit_resistor “cellname” on page 264
- _vsa_drop_on_allseg “true” | “false” on page 265
- _vsa_ir_pin_name “all” | “pin1 pin2 ...” on page 266
- _vsa_ir_pin_name_file “pin_list.txt” on page 267
- _vsa_ir_sig_name “all” | “sig1 sig2 ...” on page 268
- _vsa_ir_sig_name_file “signal_list.txt” on page 269
- _vsa_output_ir_file “../path/fileName” on page 270
- _vsa_analyze_ir “start_time” “stop_time” on page 271
- _vsa_irdrop_peak “true” | “false” on page 272

© 2020 All Rights Reserved.

_vsa_use_explicit_resistor “cellname”

```
_vsa_use_explicit_resistor "cellname"
```

This command is an advanced directive that causes the IR Drop for a net to be analyzed across design resistors to include more details. This is an optional command and must be expressly stated to make use of this advanced functionality.

For more information, see Analyzing IR Drop Across Design Resistors in the “IR Drop Analysis” chapter.

Parameter

`cellname`	Specifies the cell name of the design resistor.

Example

The following command specifies that IR Drop should be analyzed across design resistors:

```
_vsa_use_explicit_resistor "cellname"
```


© 2020 All Rights Reserved.

_vsa_drop_on_allseg "true" | "false"

```
_vsa_drop_on_allseg "true" | "false"
```

Performs IR Drop analysis on either all the segments of a net or only the segments next to devices. This is an optional command.

Default: `"true"`

Parameters

`true`	Specifies that all segments should be analyzed.
`false`	Specifies that only segments next to the devices should be analyzed.

Example

The following command specifies that IR Drop analysis should be performed on all the segments of a net:

```
_vsa_drop_on_allseg "true"
```


© 2020 All Rights Reserved.

_vsa_ir_pin_name "all" | "pin1 pin2 ..."

```
_vsa_ir_pin_name "all" | "pin1 pin2 ..."
```

Specifies the pin name(s) for which IR Drop analysis is to be performed. These pin names should come from the layout name space. Pin names can be specified either with this command or with the `_vsa_ir_pin_name_file` command.

Parameters

`all`	Specifies that IR Drop analysis should be performed on all pins.
`pin1 pin2...`	Specifies that IR Drop analysis should be performed on selected pins.

Examples

- The following command specifies the pin names for which IR Drop analysis is to be performed:

  ```
  _vsa_ir_pin_name "gnd! ibias p0 p135 p180 p225 p270 p315 p45 p90 setIC0! vcom
  vcop vdd!"
  ```

- The following command specifies that IR Drop analysis is to be performed for all pins:

  ```
  _vsa_ir_pin_name "all"
  ```


© 2020 All Rights Reserved.

_vsa_ir_pin_name_file “pin_list.txt”

```
_vsa_ir_pin_name_file "pin_list.txt"
```

Specifies the path and the name of the file that contains a list of pin names on which IR Drop analysis should be performed. The pin name file should contain a list of pin names separated by spaces (or one pin name per line). Pin names should come from the layout name space.

This is an optional command and should be used instead of the `_vsa_ir_pin_name` command. When both commands are specified, this command is ignored.

Parameter

`pin_list.txt`	Specifies the file that contains names of pins on which analysis is to be performed.

Example

The following command specifies the name of the file that contains the pin names:

```
_vsa_ir_pin_name_file "pin_list.txt"
```


© 2020 All Rights Reserved.

_vsa_ir_sig_name “all” | “sig1 sig2 ...”

```
_vsa_ir_sig_name "all" | "sig1 sig2 ..."
```

Specifies the signal name(s) for which IR Drop analysis should be performed.

Parameters

`all`	Specifies that IR Drop analysis should be performed on all signals.
`sig1 sig2...`	Specifies that IR Drop analysis should be performed on selected signals.

Example

The following command specifies the signal names for which IR Drop analysis should be performed:

```
_vsa_ir_sig_name "I15|n2" "I15|n1" "I15|n5" "I15|n3"
```


© 2020 All Rights Reserved.

_vsa_ir_sig_name_file “signal_list.txt”

```
_vsa_ir_sig_name_file "signal_list.txt"
```

Specifies the path and name of a file that contains a list of signal names on which IR Drop analysis should be performed. The signal name file should contain a list of signal names separated by spaces (or one signal name per line). Signal names should come from the layout name space.

This is an optional command and should be used instead of the `_vsa_ir_sig_name` command. When both commands are specified, this command is ignored.

Parameter

`signal_list.txt`	Specifies the file that contains names of signals on which analysis is to be performed.

Example

The following command specifies the name of the file that contains the signal names:

```
_vsa_ir_sig_name_file "siglist.txt"
```


© 2020 All Rights Reserved.

_vsa_output_ir_file “../path/fileName”

```
_vsa_output_ir_file "../path/fileName"
```

Specifies the location for the results of the IR Drop analysis. These results can be later loaded into the interactive Power IR/EM for review using the *Import IR Drop Results* on the *VPS-L - Analyze Extracted View Pin(s)* menu.

This is an optional command.

Default: `"IRDrop.results"`

Parameters

`../path/fileName`	Specifies the path and file name to write the results of the analysis.

Example

The following command specifies the location of the file to write the results of the IR Drop analysis:

```
_vsa_output_ir_file "ir.res102.batch"
```


© 2020 All Rights Reserved.

_vsa_analyze_ir “start_time” “stop_time”

Specifies a start and stop time for transient analysis. If the start and stop time is 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`.

Default:

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 time for transient analysis:

  ```
  _vsa_analyze_ir "0" "150n"
  ```

- The following command is used for running DCOP analysis or full duration of transient analysis:

  ```
  _vsa_analyze_ir
  ```


© 2020 All Rights Reserved.

_vsa_irdrop_peak “true” | “false”

`_vsa_irdrop_peak "true" | "false"`

Specifies either peak or average IR Drop.

Default: `"true"`

Parameters

`true`	Specifies peak IR Drop analysis.
`false`	Specifies average IR Drop analysis.

Example

The following command specifies average IR Drop analysis:

```
_vsa_irdrop_peak "false"
```


© 2020 All Rights Reserved.

EM Analysis Commands

- _vsa_em_pin_name “all” | “pin1 pin2 ...” on page 274
- _vsa_em_pin_name_file “pin_list.txt” on page 275
- _vsa_em_sig_name “all” | “sig1 sig2 ...” on page 276
- _vsa_em_sig_name_file “signal_list.txt” on page 277
- _vsa_qrc_run_directory “run_dir_name” on page 278
- _vsa_rcx_run_name “run_name” on page 279
- _vsa_em_data_file “../path/fileName” on page 280
- _vsa_output_em_file “../path/fileName” on page 281
- _vsa_em_job_name “jobName” on page 282
- _vsa_em_peak “true” | “false” on page 283
- _vsa_em_avg “true” | “false” on page 283
- _vsa_em_avg_abs “true” | “false” on page 283
- _vsa_em_rms “true” | “false” on page 284
- _vsa_em_dc_rms “true” | “false” on page 284
- _vsa_em_peak_ac “true” | “false” on page 284
- _vsa_em_custom “true” | “false” on page 285
- _vsa_em_save_only_failed_results “true” | “false” on page 286
- _vsa_analyze_em “startTime” “stopTime” on page 287
- _vsa_em_norundir "true" | “false” on page 288

© 2020 All Rights Reserved.

_vsa_em_pin_name "all" | "pin1 pin2 ..."

```
_vsa_em_pin_name "all" | "pin1 pin2 ..."
```

Specifies the pin name(s) for which EM analysis is to be performed. These pin names should come from the layout name space. Pin names can be specified with this command or with the `_vsa_em_pin_name_file` command.

Parameters

`all`	Specifies that EM analysis should be performed on all pins.
`pin1 pin2...`	Specifies that EM analysis should be performed on selected pins.

Examples

- The following command specifies that EM analysis should be performed on all pins:

  ```
  _vsa_em_pin_name "all"
  ```

- The following command specifies the pin names for which EM analysis should be performed:

  ```
  _vsa_em_pin_name "gnd! ibias p0 p135 p180 p225 p270 p315 p45 p90 setIC0! vcom
  vcop vdd!"
  ```


© 2020 All Rights Reserved.

_vsa_em_pin_name_file “pin_list.txt”

```
_vsa_em_pin_name_file "pin_list.txt"
```

Specifies the path and name of a file that contains a list of pin names on which EM analysis should be performed. The pin name file should contain a list of pin names separated by spaces (or one pin name per line). Pin names should come from the layout name space.

This is an optional command and should be used instead of the `_vsa_em_pin_name` command. When both commands are specified, this command is ignored.

Parameter

`pin_list.txt`	Specifies the file that contains names of pins on which analysis is to be performed.

Example

The following command specifies the name of the file that contains a list of pin names for which EM analysis is to be performed:

```
_vsa_em_pin_name_file "pinlist.txt"
```


© 2020 All Rights Reserved.

_vsa_em_sig_name “all” | “sig1 sig2 ...”

```
_vsa_em_sig_name "all" | "sig1 sig2 ..."
```

Specifies the signal name(s) for which EM analysis should be performed. These signal names should come from the layout name space. Signal names can be specified with this command or with the `_vsa_em_sig_name_file` command.

Parameters

`all`	Specifies that IR Drop analysis should be performed on all signals.
`sig1 sig2...`	Specifies that IR Drop analysis should be performed on selected signals.

Examples

- The following command specifies that EM analysis should be performed on all signals:

  ```
  _vsa_em_sig_name "all"
  ```

- The following command specifies the signal names for which EM analysis should be performed:

  ```
  _vsa_em_sig_name "I15|n2" "I15|n1" "I15|n5" "I15|n3" "I15|n6" "I15|n4" "inp2"
  "n2" "inm2" "I16|n2"
  ```


© 2020 All Rights Reserved.

_vsa_em_sig_name_file “signal_list.txt”

```
_vsa_em_sig_name_file "signal_list.txt"
```

Specifies the path and name of a file that contains a list of signal names on which EM analysis should be performed. The signal name file should contain a list of signal names separated by spaces (or one signal name per line). Signal names should come from the layout name space.

This is an optional command and should be used instead of the `_vsa_em_sig_name` command. When both commands are specified, this command is ignored.

Parameter

`signal_list.txt`	Specifies the file that contains names of signals on which analysis is to be performed.

Example

The following command specifies the name of the file that contains a list of signal names for which EM analysis is to be performed:

```
_vsa_em_pin_name_file "siglist.txt"
```


© 2020 All Rights Reserved.

_vsa_qrc_run_directory “run_dir_name”

```
_vsa_qrc_run_directory "run_dir_name"
```

Specifies the directory name of the QRC run. This is required for EM analysis.

Parameter

`run_dir_name`	Specifies the directory name of the QRC run.

Example

The following command specifies the directory name of the QRC run:

```
_vsa_qrc_run_directory "../custom/LVS"
```


© 2020 All Rights Reserved.

_vsa_rcx_run_name “run_name”

```
_vsa_rcx_run_name "run_name"
```

Specifies the RCX run name. This is required for EM analysis.

Parameter

`run_name`	Specifies the RCX run name.

Example

The following command specifies the RCX run name:

```
_vsa_rcx_run_name "vco"
```


© 2020 All Rights Reserved.

_vsa_em_data_file "../path/fileName"

```
_vsa_em_data_file "../path/fileName"
```

Specifies the path and file name of the `emDataFile` required for EM analysis. For more information, see the "EM Rules Specification" chapter. 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.

Parameter

`../path/fileName`	Specifies the path and name of the EM rule specification file.

Example

The following command specifies the path and file name of the `emDataFile` required for EM analysis:

```
_vsa_em_data_file "../custom/oa/gpdk/emDataFile.txt"
```


© 2020 All Rights Reserved.

_vsa_output_em_file “../path/fileName”

```
_vsa_output_em_file "../path/fileName"
```

Specifies the path and file name for the EM analysis report output. These results can be later loaded into the interactive Power IR/EM for review using the *Import Electromigration Results* on the *VPS-L* menu. This is an optional command.

Default: `“EM.results”`

Parameter

`../path/fileName`	Specifies the path and file name of the EM analysis output.

Example

The following command specifies the path and file name of the emDataFile required for EM analysis:

```
_vsa_output_em_file "em.res102.batchP"
```


© 2020 All Rights Reserved.

_vsa_em_job_name “jobName”

```
_vsa_em_job_name "jobName"
```

Prepends a job name to output file names.

Parameter

`jobname`	Specifies a job name provided by you for the current run.

During the Power IR/EM execution, the following default files are created internally. These files are removed when the job is completed.

```
vsa2CmdFile.cmd
vsa2Results.txt
vsa2Input.txt
```

To run multiple jobs simultaneously, use the `_vsa_em_job_name` command so that the output files for different runs are prepended with different job names. For example:

```
_vsa_em_job_name "myjob3"
```

The following output files are created. These are removed when the job is completed.

```
myjob3_vsa2CmdFile.cmd
myjob3_vsa2Results.txt
myjob3_vsa2Input.txt
```

The following file is output for viewing:

```
myjob3.EM.results
```


© 2020 All Rights Reserved.

_vsa_em_peak “true” | “false”

```
_vsa_em_peak "true" | "false"
```

Specifies peak EM analysis. This is required for EM analysis.

Default: `false`

Example

The following command is used to specify peak EM analysis.

```
_vsa_em_peak "true"
```

_vsa_em_avg “true” | “false”

```
_vsa_em_avg "true" | "false"
```

Specifies average EM analysis. This is required for EM analysis.

Default: `false`

Example

The following command is used to specify average EM analysis.

```
_vsa_em_avg "true"
```

_vsa_em_avg_abs “true” | “false”

```
_vsa_em_avg_abs "true" | "false"
```

Specifies average of absolute EM analysis. This is required for EM analysis.

Default: `false`

Example

The following command is used to specify the average of absolute EM analysis.

```
_vsa_em_avg_abs "true"
```


© 2020 All Rights Reserved.

_vsa_em_rms “true” | “false”

```
_vsa_em_rms "true" | "false"
```

Specifies root mean square (RMS) EM analysis. This is required for EM analysis.

Default: `false`

Example

The following command is used to specify the RMS EM analysis.

```
_vsa_em_rms "true"
```

_vsa_em_dc_rms “true” | “false”

```
_vsa_em_dc_rms "true" | "false"
```

Specifies root mean square (RMS) of DC EM analysis. This is required for EM analysis.

Default: `false`

Example

The following command is used to specify the RMS DC EM analysis.

```
_vsa_em_dc_rms "true"
```

Example

The following command is used to specify the RMS DC EM analysis.

```
_vsa_em_dc_rms "true"
```

_vsa_em_peak_ac “true” | “false”

```
_vsa_em_peak_ac "true" | "false"
```

Specifies AC-Peak EM analysis. This is required for EM analysis.

Default: `false`

© 2020 All Rights Reserved.

Example

The following command is used to specify AC-Peak EM analysis.

```
_vsa_em_peak_ac "true"
```

_vsa_em_custom “true” | “false”

```
_vsa_em_custom "true" | "false"
```

Specifies Custom EM analysis. This is required for EM analysis.

Default: `false`

Example

The following command is used to specify custom EM analysis.

```
_vsa_em_custom "true"
```


© 2020 All Rights Reserved.

_vsa_em_save_only_failed_results “true” | “false”

```
_vsa_em_save_only_failed_results "true" | "false"
```

Specifies that only EM analysis results of failed nets are saved. This is required for EM analysis.

Default: `false`

Note: Use this command before using the `_vsa_analyze_em` command.

Example

The following command is used to specify that only EM analysis results of failed nets are saved.

```
_vsa_em_save_only_failed_results "true"
```


© 2020 All Rights Reserved.

_vsa_analyze_em “startTime” “stopTime”

`_vsa_analyze_em`

Specifies a start and stop time for transient EM analysis. If the start and stop time is 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.`

Parameters

`startTime`	Specifies the start time for the transient analysis.
`stopTime`	Specifies the stop time for the transient analysis.

Example

The following command is used to specify start and stop time for transient analysis:

```
_vsa_analyze_em _vsa_analyze_em "0" "1n"
```


© 2020 All Rights Reserved.

_vsa_em_norundir "true" | “false”

```
_vsa_em_norundir "true" | "false"
```

Determines whether Power IR/EM will read the `RCXSpiceINIT` file or the extracted view. If `_vsa_em_norundir` flag is set to `true`, then the QRC run directory name in the batch file should be the path where `RCXspiceINIT` file is stored.

Default: `"false"`

Parameters

`true`	Specifies the path where `RCXspiceINIT` file is stored.
`false`	Specifies the path of the extracted view.

Example

The following command is used to specify that Power IR/EM should read the `RCXSpiceINIT` file:

```
_vsa_em_norundir "true"
```


© 2020 All Rights Reserved.

Sample Command File

```
;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_use_explicit_resistor "cellname"
_vsa_drop_on_allseg  "true"
_vsa_ir_pin_name  "all"
;OR "pin1 pin2 pin3 ..."
;_vsa_ir_pin_name_file "pinlist.txt"
_vsa_ir_sig_name  "all"
;Or "sig1 sig2 sig3  ..."
;_vsa_ir_sig_name_file  "siglist.txt"
_vsa_output_ir_file "ir.res101.batch"
;_vsa_analyze_IR "0" "170n"
_vsa_analyze_ir

;following command are for EM analysis
_vsa_em_pin_name  "all"
;_vsa_em_pin_name_file "pinlist.txt"
;_vsa_em_sig_name  "all"
;_vsa_em_sig_name_file "siglist.txt"
_vsa_qrc_run_directory  "/custom/WORK/AV/LVS_emir"
_vsa_rcx_run_name "vco"
_vsa_em_data_file "emDataFile.txt"
_vsa_output_em_file "em.res101.batchP"
_vsa_em_peak "true"
_vsa_em_avg "true"
_vsa_em_rms "true"
;_vsa_analyze_em "startTime" "stopTime"
_vsa_analyze_em

exit
```


© 2020 All Rights Reserved.

Sample EM Analysis Output

```
ELECTROMIGRATION ANALYSIS RESULTS
CURRENT DENSITY MPV = false
RESULTS FILE CREATED    = "May 15 18:38:53 2015"
PROGRAM VERSION         = IC6.1.7-64b.main.284 (10.6)
DFII VERSION            = IC6.1.7-64b.main.284
QRC VERSION=3.1.7_USR2 SPECTRE VERSION=10.1.1.412.isr23   ULTRASIM
VERSION=10.1.1.412.isr23
SIMULATOR               = "spectre"
USER SUPPLIED VALUES:
   TOP LIBRARY NAME     = "amsPLL"
   TOP CELL NAME        = "TB1_vco_single"
   TOP VIEW NAME        = "schematic"
   LIBRARY NAME         = "amsPLL"
   CELL NAME            = "vco"
   VIEW NAME            = "av_R"
   TOOL                 = "power"
  QRC DIRECTORY    = "/grid/cic/user1/IC617/test/VPS_L/testcases/AV/LVS_emir/vco"
   QRC RUNNAME = "."
   SIMULATION RESULTS   = "/grid/cic/user1/IC617/test/VPS_L/testcases/simulation/
TB1_vco_single/spectre/schematic"
   SIMULATION DIR       = "/grid/cic/user1/IC617/test/VPS_L/testcases/simulation/
TB1_vco_single/spectre/schematic/psf"
   SIMULATION TRAN      = "tran-tran"
   RESULTS TYPE         = "TRANSIENT"
   TRANSIENT START      = "0"
   TRANSIENT STOP       = "17n"
   AVERAGE              = t
   DC-RMS               = nil
   RMS                  = nil
   PEAK                 = t
   AC-PEAK              = nil
   AVG-ABS              = nil
   CUSTOM               = nil
   SIM TEMP             = "27.0C"
   AVERAGE SPEC TEMP    = 100.0C
   PEAK SPEC TEMP       = 100.0C
   JUNCTION TEMP        = 110.0C
   PIN NAME             = ("gnd!" "vdd!")
   SIGNAL NAME        = <none specified>
  VIA WIDTH LIST      = ((nil nil) ("Cont" 0.2) ("Via1" 0.2) ("Via2" 0.2) ("Via3"
0.2) ("Via4" 0.2) ("Via5" 0.2))
```


© 2020 All Rights Reserved.

```
   CADGRID              = 0.9
   ROUTING LAYERS       = ("Poly" "Metal1" "Metal2" "Metal3" "Metal4" "Metal5"
"Metal6")
   VIA LAYERS           = ("Cont" "Via1" "Via2" "Via3" "Via4" "Via5")
   CROSS REF LAYERS     = (("poly" ("Poly" "Poly")) ("mt1" ("Metal1" "Metal1"))
("mt2" ("Metal2" "Metal2")) ("mt3" ("Metal3" "Metal3")) ("mt4" ("Metal4" "Metal4"))
("mt5" ("Metal5" "Metal5")) ("mt6" ("Metal6" "Metal6")))
  AVG CURRENT DENSITY  = ((nil layer "Via5" minW 0.0 maxW -1 res 0.68 currentDensity
"1.21")
(nil layer "Via4" minW 0.0 maxW -1 res 0.68 currentDensity "1.21")
(nil layer "Via3" minW 0.0 maxW -1 res 0.68 currentDensity "1.21")
(nil layer "Via2" minW 0.0 maxW -1 res 0.68 currentDensity "1.21")
(nil layer "Via1" minW 0.0 maxW -1 res 0.68 currentDensity "1.21")
(nil layer "Cont" minW 0.0 maxW -1 res 11.04 currentDensity "1.61")
(nil layer "Metal6" minW 0.0 maxW -1.0 currentDensity "1.91")
(nil layer "Metal5" minW 0.0 maxW -1.0 currentDensity "1.91")
(nil layer "Metal4" minW 0.0 maxW -1.0 currentDensity "1.91")
(nil layer "Metal3" minW 0.0 maxW -1.0 currentDensity "1.91")
(nil layer "Metal2" minW 0.0 maxW -1.0 currentDensity "1.91")
(nil layer "Metal1" minW 0.16 maxW -1.0 currentDensity "(14 * w * 5)")
(nil layer "Metal1" minW 0.0 maxW 0.16 currentDensity "(1.434 * 3.9 * w)")
(nil layer "Poly" minW 0.0 maxW -1.0 currentDensity "1.1"))
   PEAK CURRENT DENSITY  = ((nil layer "Via5" minW 0.0 maxW -1 res 0.68
currentDensity "1.21")
(nil layer "Via4" minW 0.0 maxW -1 res 0.68 currentDensity "1.21")
(nil layer "Via3" minW 0.0 maxW -1 res 0.68 currentDensity "1.21")
(nil layer "Via2" minW 0.0 maxW -1 res 0.68 currentDensity "1.21")
(nil layer "Via1" minW 0.0 maxW -1 res 0.68 currentDensity "1.21")
(nil layer "Cont" minW 0.0 maxW -1 res 11.04 currentDensity "1.61")
(nil layer "Metal6" minW 0.0 maxW -1.0 currentDensity "1.91")
(nil layer "Metal5" minW 0.0 maxW -1.0 currentDensity "1.91")
(nil layer "Metal4" minW 0.0 maxW -1.0 currentDensity "1.91")
(nil layer "Metal3" minW 0.0 maxW -1.0 currentDensity "1.91")
(nil layer "Metal2" minW 0.0 maxW -1.0 currentDensity "3.91")
(nil layer "Metal1" minW 0.16 maxW -1.0 currentDensity "(14 * w * 5)")
(nil layer "Metal1" minW 0.0 maxW 0.16 currentDensity "(1.434 * 3.9 * w)")
(nil layer "Poly" minW 0.0 maxW -1.0 currentDensity "1.1"))
--------------------------------------------------------------------------------
RESULTS AT "PEAK"
         Instance                        Measured No of  Minimum  Current Density
% Failed   Name   Layer   Current   Width  Vias  Width  Density    Area    Limit
32.2%     rh458  Via1      1.4398m   0.9   3    1.190  1.600(mA/um)   0.120   1.210
```


© 2020 All Rights Reserved.

```
pass-21.0% rh392  Via1     860.08u   0.9   1    0.711  0.956(mA/um)   0.040   1.210
pass-21.2% rh385  Via1     858.04u   0.9   1    0.709  0.953(mA/um)   0.040   1.210
pass-21.9% rh377  Via1     850.23u   0.9   1    0.703  0.945(mA/um)   0.040   1.210
pass-24.7% rh399  Via1     819.71u   0.9   1    0.677  0.911(mA/um)   0.040   1.210
pass-38.9% rh405  Via1     665.78u   0.9   1    0.550  0.740(mA/um)   0.040   1.210
pass-40.7% rh440  Via1     645.56u   0.9   1    0.534  0.717(mA/um)   0.040   1.210
pass-41.3% rh418  Via1     639.16u   0.9   1    0.528  0.710(mA/um)   0.040   1.210
pass-42.6% rh451  Via1     625.58u   0.9   1    0.517  0.695(mA/um)   0.040   1.210
pass-42.6% rh429  Via1     625.25u   0.9   1    0.517  0.695(mA/um)   0.040   1.210
pass-43.7% rh414  Via1     613.64u   0.9   1    0.507  0.682(mA/um)   0.040   1.210
pass-46.4% rh380  Via1     583.52u   0.9   1    0.482  0.648(mA/um)   0.040   1.210
-------------------------------------------------------------------------------
RESULTS AT "AVERAGE"
         Instance                        Measured No of  Minimum  Current Density
% Failed   Name   Layer  Current  Width  Vias  Width  Density     Area     Limit
pass-31.2% rh458  Via1    749.33u   0.9   3   0.619  0.833(mA/um) 0.120   1.210
pass-70.0% rh405  Via1    326.75u   0.9   1   0.270 0.363(mA/um)  0.040   1.210
pass-77.6% re347  Metal2  1.2859m    3   NA   0.673 0.429(mA/um)  NA      1.910
pass-77.6% rh457  Via1    244.04u   0.9   3   0.202 0.271(mA/um)  0.120   1.210
pass-77.7% re295  Metal2  1.2771m    3   NA    0.669 0.426(mA/um) NA     1.910
pass-77.8% re402  Via1    242.1u    0.9  1    0.200 0.269(mA/um) 0.040     1.210

*******************************************************************************
*******************************************************************************
Instance  Simulation    Simulation    Resistance  pin/signal x           y
Name      Netname1      Netname2
rh382   X1._85\:vdd!   X1._132\:vdd!   0.666700  vdd!      71.725000    21.090000
re117   X1._65\:gnd!   X1._66\:gnd!    1.193300  gnd!      50.522000    41.155000
re336   X1._188\:gnd!  X1._189\:gnd!   0.110000  gnd!      22.292000    1.880000
rh406   X1._146\:gnd!  X1._147\:gnd!   0.500000  gnd!      90.265000    27.600000
re287   X1._147\:vdd!  X1._148\:vdd!   0.096670  vdd!      26.132000    21.500000
rh380   X1._104\:vdd!  X1._130\:vdd!   0.500000  vdd!      80.810000    20.830000
rf106   X1._21\:vdd!   X1._23\:vdd!    1.569300  vdd!      12.700000    28.800000
rf307   X1._74\:gnd!   X1._75\:gnd!    0.172000  gnd!      58.880000    2.885000
rf427   X1._94\:vdd!   X1._95\:vdd!    0.637500  vdd!      77.385000    30.890000
rf329   X1._84\:gnd!   X1._2\:gnd!     1.271300  gnd!      43.772000    0.923000
rh183   X1._68\:gnd!   X1._69\:gnd!    1.000000  gnd!      53.415000    32.405000
```


© 2020 All Rights Reserved.

Glossary

This glossary defines the terms and concepts that you need to understand to use Power IR/EM effectively.

APS

APS is a next-generation SPICE simulator that provides high-performance, high-capacity circuit simulation with full Spectre accuracy.

current density

Current density is the amount of current per area that a material can reliably handle, usually expressed in milliamps per micron.

DC

DC is an abbreviation for “direct current,” which is the current that flows in only one direction. The current in a power grid is generally DC. DC is also a standard representation of a waveform that is typically used as a voltage source in circuit simulation. It displays a flat waveform.

DC operating-point (DCOP) simulation

DC simulation is a full vector simulation that calculates only DC operation points.

dynamic analysis

Dynamic analysis is a method of analyzing a circuit to obtain operating currents and voltages. It is a time-based analysis of the circuit netlist over a specified period, such as a clock cycle.

Electromigration

Electromigration is the mass transport of metal atoms resulting from the transfer of momentum between conducting electrons and diffusing metal atoms. It occurs in metal wires that conduct high current densities. Conducting electrons collide with diffusing metal atoms and impel the atoms in the direction of electron flow. This collision produces a mass flux opposite to that of the current. Divergences in this mass flux can result in damage to the conductor in the form of voids or hillocks. This problem can be minimized by limiting the current density in metal conductor lines, vias, and contacts.

extracted view

An extracted view is a cell view generated by Pegasus that stores layout extraction and connectivity information from the layout.

© 2020 All Rights Reserved.

IR Drop

IR Drop is a reduction in voltage that occurs on the VDD network of an integrated circuit because of the current flowing through the resistance of the VDD network itself (V = I x R).

layout name space

Defines the net names as those that occur in the layout database as opposed to those in the schematic. The layout name space is generally used when the analysis is performed on the physical layout data rather than the logical schematic data.

LVS

LVS, or layout as compared to schematic, is a methodology that extracts the connectivity from the layout and compares it to the connectivity of the netlist.

presistor

The parasitic resistor extracted by RCX and added into the extracted view.

Spectre

Spectre is a SPICE-like network simulator from Cadence.

static analysis

Static analysis is a method of analyzing a circuit without using time-based simulation.

steady-state analysis

Steady-state analysis is an analysis of a system that assumes that the components have steady-state values.

transient analysis

Transient analysis is a time-based analysis of a system.

UltraSim

A fast-MOS based simulator from Cadence.

© 2020 All Rights Reserved.

Index

Numerics

64-bit mode, starting 22

A

Absolute EM analysis 283
AC-Peak EM analysis 284
Add Explicit Vias option 51
allpub 57
allpub option 55
Analog Design Environment 11
Analyze Extracted View Pin(s) 73, 107, 128, 244, 270
Analyze Extracted View Signal(s) 73, 74, 107, 128
Analyze Results 76
Analyze Results option 117
Append to existing file 245
APS 52, 75, 116
Average EM analysis 283

B

Batch mode 252, 256

C

Cadence Help, launching 37
Calculate IR Drop 75
CDS_AUTO_64BIT environment variable, setting 22
Common Commands 253
config views 54, 57
Create Voltage Source Regions 244
current density 52, 77, 133, 136, 206, 207, 215, 293
CurrentDensSpecList parameter 207
Custom EM analysis 285
Customized results display 142

D

DC 293
DC operating point simulation 49, 53, 76, 117
DC operating-point (DCOP) simulation 293
DCOP 76, 117, 271, 287
Define region 245
DFII layers 205
DFII PDK library 206
Display Netnames 135
documentation, accessing 37
dynamic analysis
  definition 293

E

Edit CDF 247
Electromigration 106, 293
Electromigration analysis 53, 180
Electromigration Analysis Results 131, 142
Electromigration Analysis Results form 249
Electromigration Power Analysis 107
Electromigration Power Analysis form 80, 125
Electromigration Power Grid Analysis 82, 125, 127
Electromigration Power-Grid Analysis 64
EM - IR 126
EM analysis 106, 249, 252, 260, 275, 283
EM Analysis Commands 273
EM analysis of signal nets 126
EM Analysis Output 290
EM Analysis Results 145
EM violations 135
EM/IR violations 244
emDataFile 64, 215
environment variable
  _vaUseNewLayers 65
  _vsaApplyRThreshold 67
  _vsaBulkTerminal 69
  _vsaDBRefPinThreshold 68
  _vsaDisableAllDesignResistorAnalysis

© 2020 All Rights Reserved.

66
_vsaDrfFile 65
_vsaDumpResultDBPath 69
_vsaEMColumnViewConfigFile 65
_vsaEMDataFile 64
_vsaEMDeltaT 64
_vsaEMMapFile 64
_vsaEMPreExecuteFile 64
_vsaEMQRCDir 64
_vsaEMSimDir 64
_vsaEMTechFile 64
_vsaEnablePowerGridRules 66
_vsaEnablePWL 66
_vsaICTConductorUnit 66
_vsaIgnoreLayers 65
_vsaIgnoreResistors 65
_vsaIRDropSimDir 64
_vsaLinearInterpolation 66
_vsaLSFMachineString 66
_vsaMicronUnit 69
_vsaP2Lvs 65
_vsaRMSMultiplierPin 67
_vsaSelectLayers 65
_vsaSelectResistors 65
_vsaSupport28nmRules 66
_vsaTechFile 66
_vsaTechnologyNode 66
extracted view 44, 53, 75, 82, 293

G

Get Layout Window Number 86, 133

H

help, launching Cadence Help 37

I

Import Electromigration Results 128, 281
Import IR Drop Results 270
IR (Voltage) Drop Analysis 79, 80, 82
IR (Voltage) Drop Analysis form 80
IR Drop 72, 264, 272
IR drop 294
IR Drop analysis 53, 64, 72, 73, 74, 75, 79, 82, 252, 260, 265, 266
IR Specific Commands 263

L

layout name space 267, 275
Layout Window 86, 133
LibGen Power Grid Views Creation 182
LVS 294

M

Mark and Unmark 87, 133
measure statements 55, 57

N

net resistance 51
Netlisting 50

O

Outputs - Save All 55

P

Parasitic Resistance Width 52
Parasitic Resistor Netnames 135
Peak EM analysis 283
Plot waveforms 137
Power IR/EM 73, 107, 126, 182, 244, 270, 281
power-grid view 180, 252
PowerGridView Creation 182
presistor 216
PVS LVS 42

Q

QRC Results Directory 109

R

README file, where to find 22
Related documents
Voltus-Fi documentation 12
related documents

© 2020 All Rights Reserved.

other documentation 12
RMS EM analysis 284
routing layers 205

S

Sample Command File 289
saSelectLayers 65
Select and Zoom 134
Select Presistors Inside a Window 137
Select Voltage Pad Regions 244
Setup 45
Simulated Temp 133
Simulation Directory 116
Simulation Directory option 75
Simulation Information 248
Spectre 52, 54, 56, 75, 116, 133, 249, 294
Spectre simulation directory 64
static analysis 294
steady-state analysis 294

T

test bench 53
transient analysis 53, 294
Transient simulation 76, 118
Type of IR Drop 77
Type of Results 133, 136

U

UltraSim 52, 75, 116
Use RCXspiceINIT for layer info 108
Use RCXspiceINIT for layer info option 108

V

via layers 205
via resistance 51
via width list 212
View PDK Current Density Limits 139
Virtuoso 73, 107, 252
Virtuoso Command Interpreter
Window 247
Virtuoso schematic editor 53
voltage source regions 244
VPS
license string 19
license strings 19

Z

Zoom To Net 88

© 2020 All Rights Reserved.