

SIEMENS EDA

Calibre® CMPAnalyzer User's Manual

Software Version 2021.2
Document Revision 14

SIEMENS

Unpublished work. © 2021 Siemens

This material contains trade secrets or otherwise confidential information owned by Siemens Industry Software, Inc., its subsidiaries or its affiliates (collectively, "Siemens"), or its licensors. Access to and use of this information is strictly limited as set forth in Customer's applicable agreement with Siemens. This material may not be copied, distributed, or otherwise disclosed outside of Customer's facilities without the express written permission of Siemens, and may not be used in any way not expressly authorized by Siemens.

This document is for information and instruction purposes. Siemens reserves the right to make changes in specifications and other information contained in this publication without prior notice, and the reader should, in all cases, consult Siemens to determine whether any changes have been made. Siemens disclaims all warranties with respect to this document including, without limitation, the implied warranties of merchantability, fitness for a particular purpose, and non-infringement of intellectual property.

The terms and conditions governing the sale and licensing of Siemens products are set forth in written agreements between Siemens and its customers. Siemens' **End User License Agreement** may be viewed at: www.plm.automation.siemens.com/global/en/legal/online-terms/index.html.

No representation or other affirmation of fact contained in this publication shall be deemed to be a warranty or give rise to any liability of Siemens whatsoever.

TRADEMARKS: The trademarks, logos, and service marks ("Marks") used herein are the property of Siemens or other parties. No one is permitted to use these Marks without the prior written consent of Siemens or the owner of the Marks, as applicable. The use herein of third party Marks is not an attempt to indicate Siemens as a source of a product, but is intended to indicate a product from, or associated with, a particular third party. A list of Siemens' trademarks may be viewed at: www.plm.automation.siemens.com/global/en/legal/trademarks.html. The registered trademark Linux® is used pursuant to a sublicense from LMI, the exclusive licensee of Linus Torvalds, owner of the mark on a world-wide basis.

Support Center: support.sw.siemens.com

Send Feedback on Documentation: support.sw.siemens.com/doc_feedback_form

Revision History

Revision	Changes	Status/ Date
14	Modifications to improve the readability and comprehension of the content. Approved by Lucille Woo. All technical enhancements, changes, and fixes listed in the <i>Calibre Release Notes</i> for this products are reflected in this document. Approved by Michael Buehler.	Released April 2021
13	Modifications to improve the readability and comprehension of the content. Approved by Lucille Woo. All technical enhancements, changes, and fixes listed in the <i>Calibre Release Notes</i> for this products are reflected in this document. Approved by Michael Buehler.	Released January 2021
12	Modifications to improve the readability and comprehension of the content. Approved by Lucille Woo. All technical enhancements, changes, and fixes listed in the <i>Calibre Release Notes</i> for this products are reflected in this document. Approved by Michael Buehler.	Released October 2020
11	Modifications to improve the readability and comprehension of the content. Approved by Lucille Woo. All technical enhancements, changes, and fixes listed in the <i>Calibre Release Notes</i> for this products are reflected in this document. Approved by Michael Buehler.	Released July 2020

Author: In-house procedures and working practices require multiple authors for documents. All associated authors for each topic within this document are tracked within the Siemens EDA documentation source. For specific topic authors, contact the Siemens Digital Industries Software documentation department.

Revision History: Released documents maintain a revision history of up to four revisions. For earlier revision history, refer to earlier releases of documentation which are available on <https://support.sw.siemens.com/>.

Table of Contents

Revision History

Chapter 1	
Introduction to Calibre CMPAnalyzer.....	11
Workflow with Calibre CMPAnalyzer	11
Requirements for Calibre CMPAnalyzer.....	12
Syntax Conventions	12
Chapter 2	
CMP Analysis	15
CMP Analysis with the Calibre Interactive GUI.....	15
Using the Calibre CMPAnalyzer Batch Flow	21
Chapter 3	
CMP Analysis Results	25
Reviewing Analysis Results	25
Recalculating Hotspots	42
Creating Scatter Plots.....	43
Creating a Simulated Profile Plot	45
Performing Data Analysis in Property View.....	47
Getting Property Values from a Colormap	51
Displaying MaterialNT(T) Properties	52
CMP Hotspots Clustering	55
Running CMP Hotspots Clustering From Calibre RVE.....	55
Running CMP Hotspots Clustering From Calibre YieldServer	60
Exporting Hotspot Reports	62
Exporting Colormap Data	62
Exporting Calibre CMPAnalyzer Thickness Data to File for Extraction Tools	64
Exporting Analysis Results in Batch Mode.....	66
Calculating User-Defined Hotspots	69
Tcl Script for User-Defined Hotspots.....	69
Running Calibre CMPAnalyzer with User-Defined Hotspots	74
Analyzing Calibre CMPAnalyzer User-Defined Hotspot Results	75
Mapping Properties to Hotspot Rule Names	78
Hot Spot Analysis Batch Procedure.....	80
Chapter 4	
CMP Bucketing Flow	81
Running the CMP Bucketing Flow in Calibre Interactive for DFM	81

Chapter 5

Calibre CMPAnalyzer Rerun Flow.....	87
cmp::rerun_extraction	89
Performing the Calibre CMPAnalyzer Rerun Flow	90
Using the Calibre CMPAnalyzer Rerun Flow for New Layers	92
Exporting the DB_EXTENT Layer	94
Setting DB_EXTENT in the Calibre CMPAnalyzer Rerun Flow.....	95

Chapter 6

Calibre CMPAnalyzer Reference	101
Terms and Concepts	101
Calibre CMPAnalyzer Simulator Flow	104
Required Inputs for the Calibre CMPAnalyzer Simulator Flow	104
Thickness Parameters in the Calibre CMPAnalyzer Simulator Flow.....	104
Rule Output in the Calibre CMPAnalyzer Simulator Flow	105
Calibre Interactive GUI Items for Calibre CMPAnalyzer.....	107
CMP Tab on the Inputs Pane	107
Simulator Options Tab on the Inputs Pane.....	113
Area DFM Tab on the DFM Options Pane.....	116
CMP Tab on the Outputs Pane	117
CMP Tab in Calibre RVE for DFM.....	118
CMP Batch Reporting	121
Example Batch Reporting Configuration File	121
CMP YieldServer Commands	124
CMP YieldServer Command Reference.....	124
Calibre YieldServer Scripts for CMPAnalyzer	140
Examples for CMP YieldServer Commands	142
Example Script to Calculate User-Defined Hotspots.....	142
Example Script to Save Reports for Calibre CMPAnalyzer	144
Example to Export Values to a Text File	144
Colormap Stipple Patterns	145
Layer Number Specification Format	146
Pre-Defined Layers File Format	148

Index**Third-Party Information**

List of Figures

Figure 1-1. Calibre CMPAnalyzer GUI Analysis Flow	12
Figure 2-1. Displaying the CMP Tab.....	16
Figure 2-2. Specifying Model Index	17
Figure 2-3. Mapping Layers and Datatypes.....	18
Figure 2-4. Slot Optimization Example 1	18
Figure 2-5. Slot Optimization Example 2	19
Figure 2-6. Calibre Interactive Rules Tab	19
Figure 2-7. Calibre Interactive Inputs Tab.....	20
Figure 2-8. Calibre Interactive Outputs Section	20
Figure 2-9. Calibre Interactive Run Control Tab.....	21
Figure 3-1. Setting the Calibre RVE for DFM Options.....	27
Figure 3-2. Displaying a Colormap	28
Figure 3-3. Colormap Right-Click Menu	29
Figure 3-4. Enable Zoom Synchronization for New Colormaps.....	30
Figure 3-5. Colormap Right-Click Submenu — Zoom Synchronization	31
Figure 3-6. Colormap with Color Range Enabled.....	31
Figure 3-7. Displaying a Second Colormap.....	32
Figure 3-8. Comparing Colormaps	33
Figure 3-9. Displaying a Colormap — Double-Click	34
Figure 3-10. Displaying a Histogram	35
Figure 3-11. Configuring Tooltip Display.....	36
Figure 3-12. Using Tooltip Displays	36
Figure 3-13. Show Synchronized Histogram.....	37
Figure 3-14. Show Profile Plot Options.....	37
Figure 3-15. Profile Plot	38
Figure 3-16. Profile Plots of Colormap Segments (X Direction and Y Direction)	38
Figure 3-17. Profile Plot with User-Defined Settings	39
Figure 3-18. Profile Plot with Updated User-Defined Settings	40
Figure 3-19. Viewing Individual Errors in Calibre RVE.....	42
Figure 3-20. Recalculate Hotspots Dialog Box	42
Figure 3-21. Scatter Plot	43
Figure 3-22. Scatter Plot Options	45
Figure 3-23. Simulated Profile Plot by X Direction	46
Figure 3-24. Selecting Property View by Menu or Button	48
Figure 3-25. Property View GUI Window	49
Figure 3-26. Property View GUI Window Plots	50
Figure 3-27. Show Property by X, Y	51
Figure 3-28. Property Value at Specified X, Y	52
Figure 3-29. MaterialNT Property Colormap	53
Figure 3-30. MaterialT Property Location in Layout Viewer	54

Figure 3-31. Tools — CMP Hotspots Clustering	56
Figure 3-32. Hotspots Clustering Window	57
Figure 3-33. Hotspots Clustering Colormaps	58
Figure 3-34. Hotspots Clustering Colormap Windows	59
Figure 3-35. Hotspots Clustering Layout View Information.....	59
Figure 3-36. Reports for Original Density Histograms	67
Figure 3-37. Calibre CMPAnalyzer Batch Reporting Index Page.....	68
Figure 3-38. User-Defined Hotspot on CMP Tab in Calibre RVE	77
Figure 3-39. Hotspot Tiles Corresponding to Layout View	78
Figure 4-1. Displaying a Colormap	84
Figure 4-2. Colormap with Color Range Enabled.....	84
Figure 5-1. Calibre CMPAnalyzer Rerun Flow.....	87
Figure 5-2. Layer Index Number	91
Figure 5-3. CMP Placeholder Layers	93
Figure 5-4. Last Used Layer Setting	93
Figure 5-5. Runset YieldServer Script Settings.....	95
Figure 5-6. DB_EXTENT LAYER Number.....	97
Figure 5-7. Include Rule File Statement	98
Figure 5-8. Import Extent Layer Path	98
Figure 6-1. Depth of Focus Simulator Parameters	101
Figure 6-2. Dishing Simulator Parameters	102
Figure 6-3. Erosion Simulator Parameters.....	102
Figure 6-4. Simulator Parameters	105
Figure 6-5. Simulator Rules in Calibre RVE for DFM	106
Figure 6-6. Calibre Interactive CMP Tab	107
Figure 6-7. Calibre Interactive Simulator Options Tab.....	113
Figure 6-8. Calibre Interactive Area DFM Tab.....	116
Figure 6-9. Calibre Interactive CMP Tab on the Outputs Pane.....	118
Figure 6-10. CMP Tab in Calibre RVE for DFM	119

List of Tables

Table 1-1. Syntax Conventions	12
Table 3-1. Zoom Shortcut Keys for CMP Analysis	39
Table 3-2. Profile Plot Settings	40
Table 6-1. Thickness Parameters	104
Table 6-2. Contents of the CMP Tab on the Inputs Pane	108
Table 6-3. Layer Table Right-Click Menu	112
Table 6-4. Contents of the Simulator Options Tab on the Inputs Pane	113
Table 6-5. Area DFM Tab — Calibre CMPAnalyzer Run Options	116
Table 6-6. Contents of the CMP Tab on the Outputs Pane	118
Table 6-7. CMP Tab Contents in Calibre RVE for DFM	119
Table 6-8. YieldServer Commands	125
Table 6-9. YieldServer CMP Grid Commands	131
Table 6-10. Stipple Patterns	145
Table 6-11. Layer Number Specification Examples	146

Chapter 1

Introduction to Calibre CMPAnalyzer

Calibre® CMPAnalyzer is a software application that analyzes and improves a design with respect to planarity and thickness requirements.

Chemical mechanical polishing (CMP) is a process that attempts to flatten the surface of a wafer at each layer through both mechanical and chemical methods. It is not a perfect process, and variations in a design lead to variations in surface heights. The Calibre CMPAnalyzer tool reports these variations as hotspots if they are beyond certain threshold values. Metal fill is used to compensate for the design variation to increase the final planarity. It is assumed that the design already has metal fill in it before you run the Calibre CMPAnalyzer tool.

A simulator is necessary to simulate surface heights for a particular manufacturing process. You must supply a model (called a recipe file) to the simulator. Details on the Calibre CMPAnalyzer simulator can be found in the section “[Calibre CMPAnalyzer Simulator Flow](#)” on page 104.

After running your CMP analysis, you can inspect the results using Calibre RVE™ for DFM and a layout viewer such as Calibre DESIGNrev™. You can also output thickness data for parasitic analysis using Calibre parasitic extraction tools or other extraction tools.

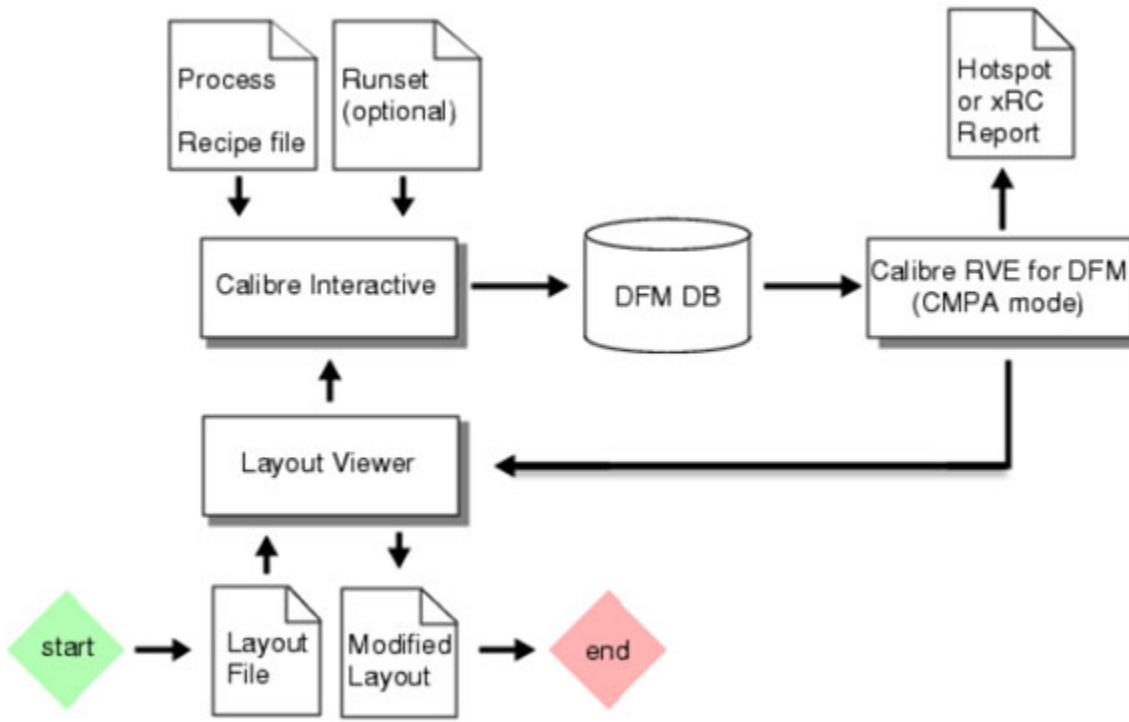
The Calibre CMPAnalyzer tool operates at the die-level — it divides the design into a rectangular grid of tiles and calculates average values of features for each tile.

Workflow with Calibre CMPAnalyzer	11
Requirements for Calibre CMPAnalyzer	12
Syntax Conventions	12

Workflow with Calibre CMPAnalyzer

The workflow for Calibre CMPAnalyzer includes Calibre Interactive, Calibre RVE for DFM, and a layout viewer.

Figure 1-1. Calibre CMPAnalyzer GUI Analysis Flow



Requirements for Calibre CMPAnalyzer

To run the Calibre CMPAnalyzer tool, you must have a process recipe file.

Additionally, you must have the necessary licenses. Refer to the [Calibre Administrator's Guide](#) for more information on the licenses that the Calibre CMPAnalyzer tool uses.

Syntax Conventions

The command descriptions use font properties and several metacharacters to document the command syntax.

You should enter literal text, that which is not in italics, exactly as shown.

Table 1-1. Syntax Conventions

Convention	Description
Bold	Bold fonts indicate a required item.
<i>Italic</i>	Italic fonts indicate a user-supplied argument.
Monospace	Monospace fonts indicate a shell command, line of code, or URL. A bold monospace font identifies text you enter.

Table 1-1. Syntax Conventions (cont.)

Convention	Description
<u>Underline</u>	Underlining indicates either the default argument or the default value of an argument.
UPPercase	For certain case-insensitive commands, uppercase indicates the minimum keyword characters. In most cases, you may omit the lowercase letters and abbreviate the keyword.
[]	Brackets enclose optional arguments. Do not include the brackets when entering the command unless they are quoted.
{ }	Braces enclose arguments to show grouping. Do not include the braces when entering the command unless they are quoted.
‘ ’	Quotes enclose metacharacters that are to be entered literally. Do not include single quotes when entering braces or brackets in a command.
or	Vertical bars indicate a choice between items. Do not include the bars when entering the command.
...	Three dots (an ellipsis) follows an argument or group of arguments that may appear more than once. Do not include the ellipsis when entering the command.

Example:

```
DEvice {element_name [‘(‘model_name‘)’]}
    device_layer {pin_layer [‘(‘pin_name‘)’] ...}
        [‘<auxiliary_layer> ...’]
        [‘(‘swap_list‘) ...’]
    [BY NET | BY SHAPE]
```


Chapter 2

CMP Analysis

You use the Calibre CMPAnalyzer CMP hotspot analysis flow with a physical design. In order to do this, you must run the analysis with Calibre Interactive, which can be run in interactive GUI or batch mode.

CMP Analysis with the Calibre Interactive GUI	15
Using the Calibre CMPAnalyzer Batch Flow	21

CMP Analysis with the Calibre Interactive GUI

The GUI flow is recommended as a first step when doing Calibre CMP analysis.

It allows you to use the Calibre® Interactive™ GUI to set up a run and create a runset file, which saves Calibre Interactive settings and is required in the Calibre CMPAnalyzer batch flow. It also allows you to run the CMP analysis and launch Calibre RVE for DFM to view results.

The default flow for multilayer (two or more layers selected) Calibre CMPAnalyzer runs, performs extraction, simulation, and hotspot calculation for each layer sequentially. As soon as the hotspot calculation for a layer finishes, you can access the dfmdb and analyze the layer data and hotspots with Calibre RVE for DFM while extraction and simulation for the next (upper) layer is in progress. Calibre CMPAnalyzer also supports a multi-processing flow which runs multilayer simulation with extraction and simulation in parallel. See option “Enable Smart Distribution for Multi-Threaded Run” under the Calibre Interactive GUI “[Simulator Options Tab on the Inputs Pane](#)” on page 113 for information on running this flow.

If no threshold values are specified for a default hotspot (erosion, dishing, or depth of focus) in the Calibre Interactive CMP simulator, the hotspot calculation is suppressed in the Calibre RVE for DFM report card. However, the colormaps and histograms are still available in the corresponding menus.

Note

-  If one or more hotspot threshold values are specified, then all of the default hotspots data columns are displayed in Calibre RVE for DFM.
-

See “[Calculating User-Defined Hotspots](#)” on page 69 for more information on user-defined hotspots.

See “[Using the Calibre CMPAnalyzer Batch Flow](#)” on page 21 for information on the batch flow.

Prerequisites

- You have a valid recipe file; see “[Creating a Process Recipe File](#)” in the *Calibre CMP Model Builder User’s and Reference Manual*. Also see “[Calibre CMPAnalyzer Simulator Flow](#)” on page 104.
- You have set CALIBRE_HOME or MGC_HOME to the path to the Calibre software tree. Refer to the [Calibre Administrator’s Guide](#) for information on setting this variable.

Procedure

1. Invoke Calibre DESIGNrev:

```
calibredrv layout
```

where *layout* specifies the path to your layout file. Alternatively, you can open your design by invoking Calibre DESIGNrev without the *layout* argument, choosing **File > Open Layout Files**, browsing to or entering the path to your layout, then clicking **Open**.

2. If the layers are displayed using non-standard colors or fill patterns, choose **Layer > Load Layer Properties**, select your layer properties file, and click **Open**.

Tip

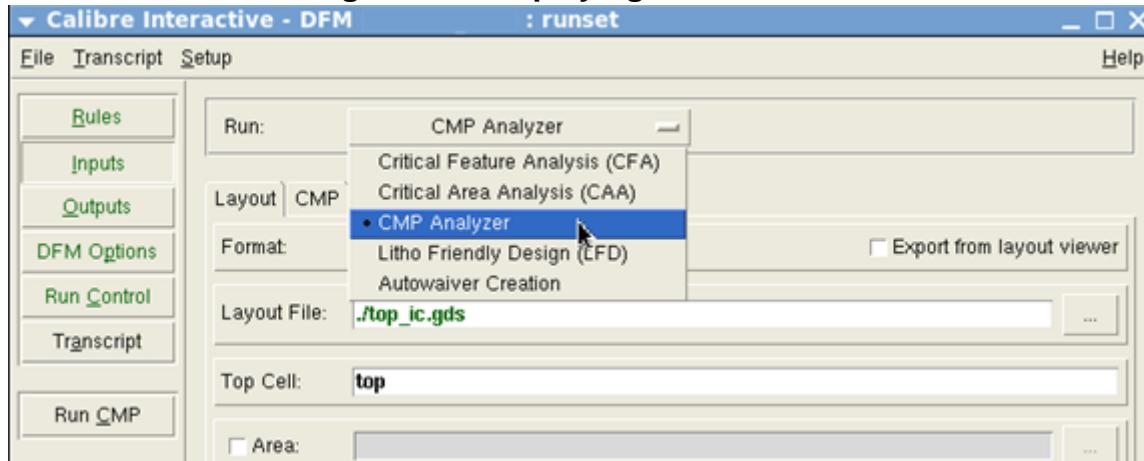
When your layer properties file is named as follows, it is loaded automatically when your layout is opened: *layout_name.layout_extension.layerprops*

3. In Calibre DESIGNrev, choose **Verification > Run DFM**.

This invokes Calibre Interactive in DFM mode. The Load Runset File dialog box also opens, allowing you to specify a Calibre Interactive runset.

4. In the Load Runset File dialog box, click **Cancel**. The Calibre Interactive window will refresh with the **Inputs** page displayed.
5. Click on the dropdown list for **Run** and select **CMP Analyzer**.

Figure 2-1. Displaying the CMP Tab



The **CMP** tab is added to the **Inputs** pane.

6. Click the **CMP** tab to display it. Refer to [Figure 2-2](#) to view the table settings and entry fields.
7. Enter the **Number of Metal Layers** in your design.

The table updates to display one row for each layer in your design and displays additional fields for dishing, erosion, and depth of focus rules.

Note

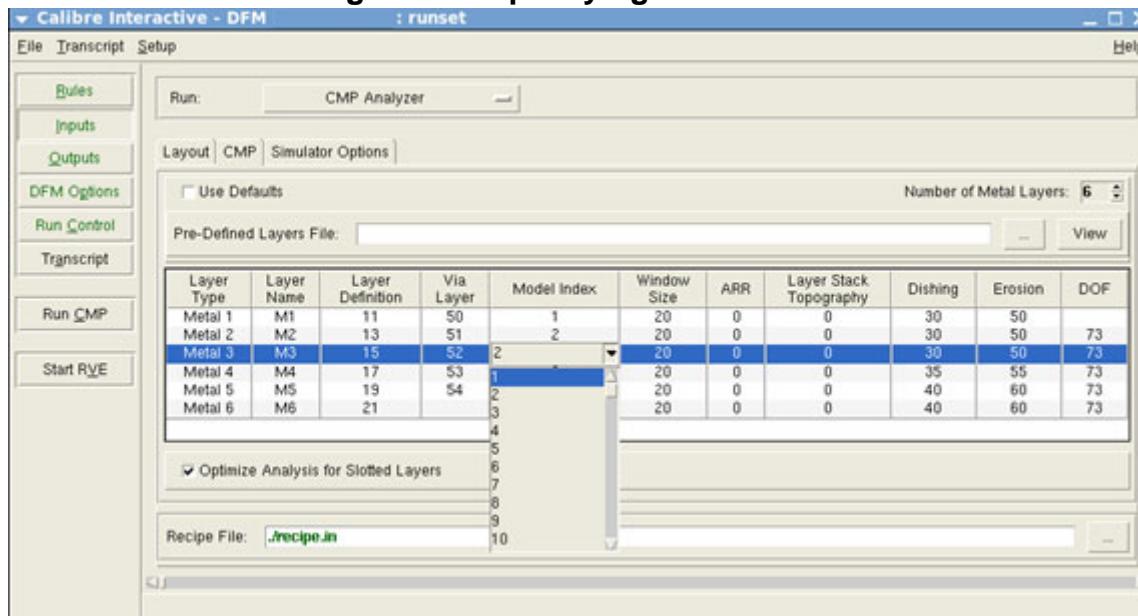
 Do not use the underscore (_) in layer names.

(Optional) You can input an SVRF file in the Pre-Defined Layers File field to specify user-defined layers using Calibre Boolean layer operations. See “[CMP Tab on the Inputs Pane](#)” on page 107 and “[Pre-Defined Layers File Format](#)” on page 148.

8. For each row in the table, modify the Model Index values as needed.

Allowed values for Model Index are 1 through 99, where the index is the sequential number of the model in the recipe file, with the first recipe having a Model Index of 1. Change the values of entries in the table as necessary.

Figure 2-2. Specifying Model Index

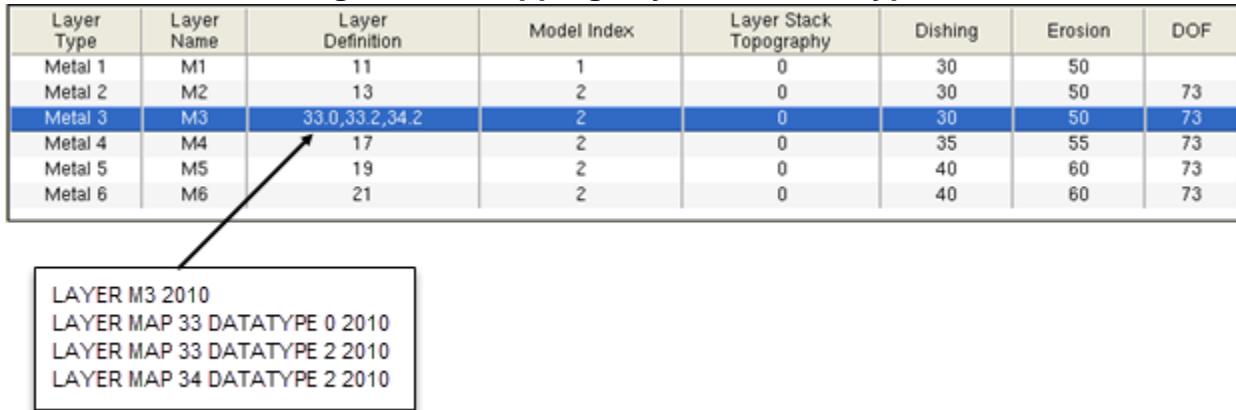


9. Edit other entries in the table as needed. You can use the right-click menu to select the columns to display and to adjust layer parameters.
10. (Optional) You can edit the layer numbers in the table and specify subtractive (-) or negative (~) layers and datatypes. Multiple layers are allowed for some entries.

Figure 2-3 shows how the layer number entry is converted to rule file statements. If you specify a layer_number with no layer_datatype, all data types for that layer are mapped to the same Calibre layer.

See “[Layer Number Specification Format](#)” on page 146 for the complete format and how subtractive and negative layers are handled.

Figure 2-3. Mapping Layers and Datatypes



Layer Type	Layer Name	Layer Definition	Model Index	Layer Stack Topography	Dishing	Erosion	DOF
Metal 1	M1	11	1	0	30	50	
Metal 2	M2	13	2	0	30	50	73
Metal 3	M3	33.0,33.2,34.2	2	0	30	50	73
Metal 4	M4	17	2	0	35	55	73
Metal 5	M5	19	2	0	40	60	73
Metal 6	M6	21	2	0	40	60	73

LAYER M3 2010
 LAYER MAP 33 DATATYPE 0 2010
 LAYER MAP 33 DATATYPE 2 2010
 LAYER MAP 34 DATATYPE 2 2010

11. Choose optimized or standard flow with the **Optimize Analysis for Slotted Layers** checkbox; this setting is checked by default.
 - **Optimized for Slotted (Negative) Layers Flow** — (checkbox checked) In this flow (also referred to as the optimized flow), the negation with DB_EXTENT is done during the CMP simulation. The optimized flow allows you to postpone the negation of negative and subtractive layers until the CMP simulation.

Note

The optimized flow is applied to those layers for which all of the layer numbers for “Layer Definition” are negative (~), subtractive (-), or absent in the GUI (empty field).

- **Standard (Positive) Flow** — (checkbox cleared) In this flow, the negation with DB_EXTENT is done in the rule deck file.

In Figure 2-4, the optimized and standard flows are identical for the M1 layer. The slot optimization does not affect the M1 layer because there are no negative or subtractive layer numbers in the corresponding field.

Figure 2-4. Slot Optimization Example 1

Layer Type	Layer Name	Layer Definition	Model Index	Layer Stack Topography	Dishing	Erosion	DOF
Metal 1	M1	11	1	0	30	50	

In Figure 2-5, the optimized flow affects only layer M2, because the Layer Definition field contains a negative layer.

Figure 2-5. Slot Optimization Example 2

Layer Type	Layer Name	Layer Definition	Model Index	Layer Stack Topography	Dishing	Erosion	DOF
Metal 1	M1	11	1	0	30	50	
Metal 2	M2	~13	2	0	30	50	73

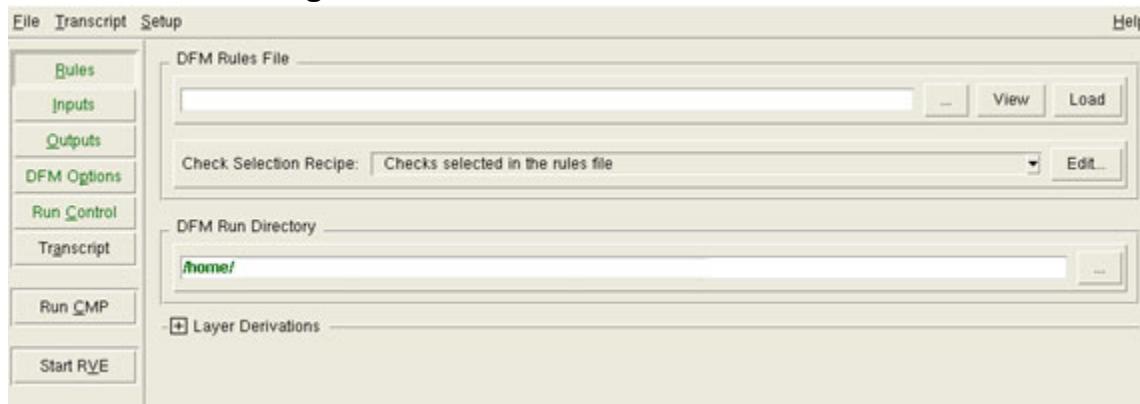
12. Provide the path to your recipe file in the Recipe File field.
13. (Optional) Click the **Simulator Options** tab to specify additional options, such as window size and other processing options. See “[Simulator Options Tab on the Inputs Pane](#)” on page 113.
14. One by one, click the buttons in the left vertical bar of the Calibre Interactive GUI to control other aspects of the CMP run, following the recommendations in the steps below.

Buttons in the left vertical bar become green when Calibre Interactive has enough information to execute a run. Calibre CMPAnalyzer does not use all the data in these fields, so certain settings may not apply. Ensure that none of the buttons appear red.

- a. Click the **Rules** button.

A rule file is optional. If entered, it is included in the control file generated by Calibre Interactive. Sections of the control file are encrypted.

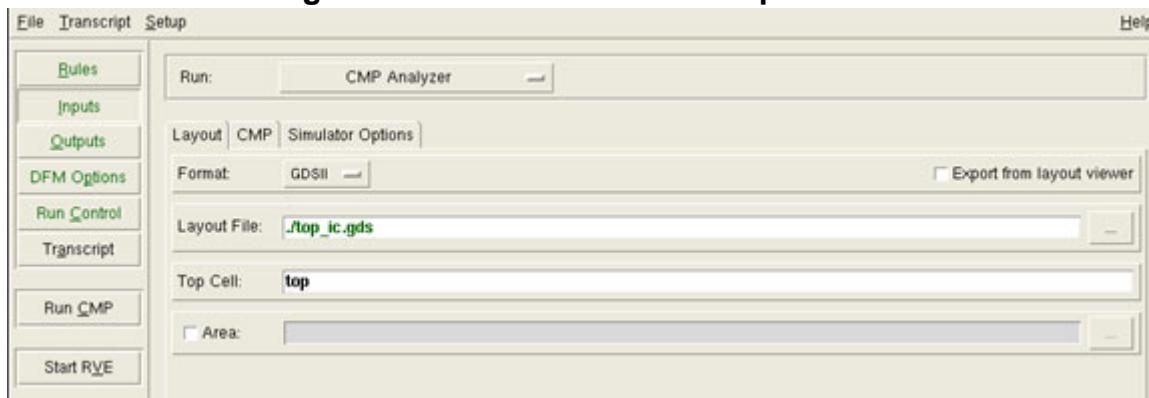
Figure 2-6. Calibre Interactive Rules Tab



- b. Click the **Inputs** button, and select the **Layout** tab.

In the **Layout File** field, enter the path to your layout. Enter the layout top cell next to the **Top Cell** field.

The **Simulator Options** tab contains run specification and process flow options. See “[Simulator Options Tab on the Inputs Pane](#)” on page 113 for a list of these options.

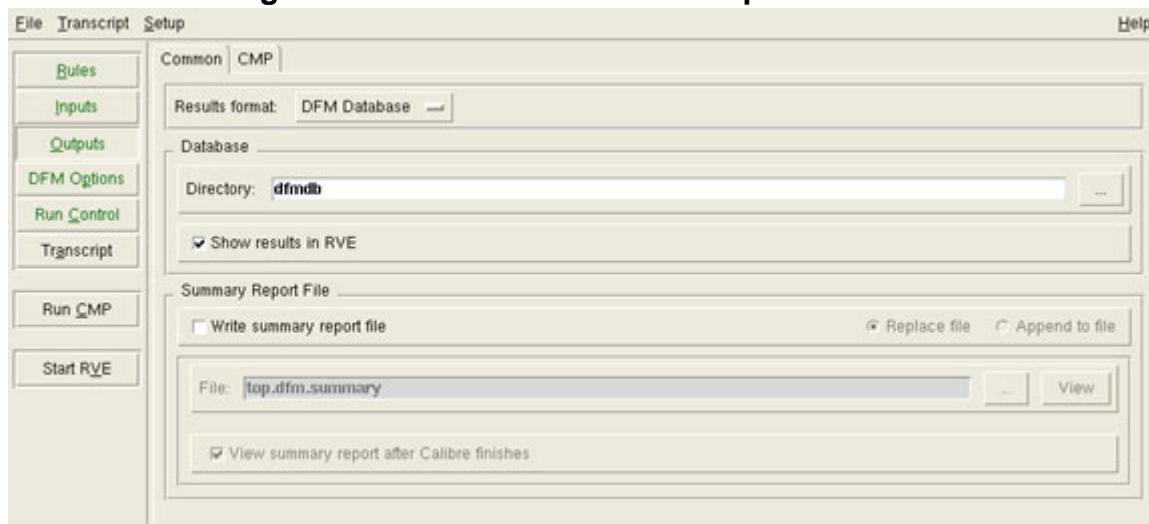
Figure 2-7. Calibre Interactive Inputs Tab

- c. Click the **Outputs** button. If necessary, modify the name of the database in the Directory field.

The database name is the name of a directory containing the results of the CMP Analyzer run, which are output as a DFM database. CMPAnalyzer creates this directory inside your working directory.

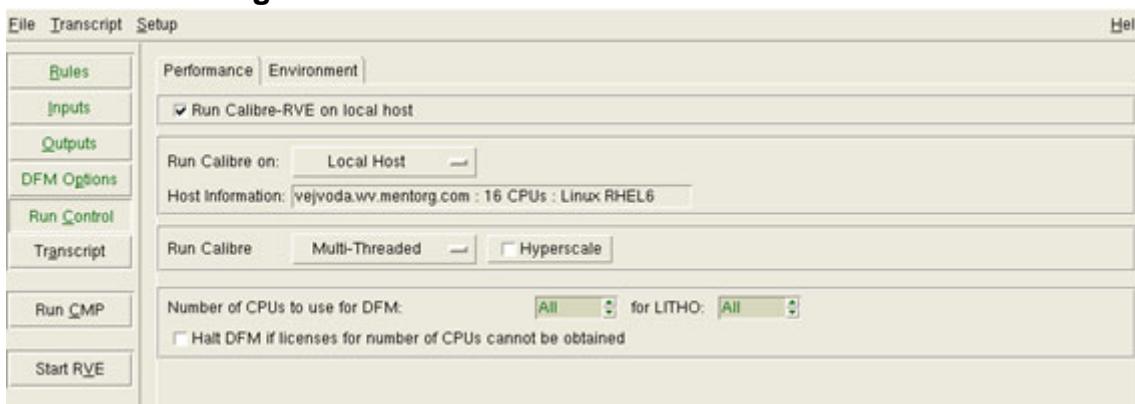
Select the **Show Results in RVE** button.

In the **CMP** tab, you can optionally enable the creation of a CMP summary report file.

Figure 2-8. Calibre Interactive Outputs Section

- d. Click the **Run Control** button. Select any additional items as necessary for your design flow.

Figure 2-9. Calibre Interactive Run Control Tab



15. Choose **File > Save Runset As** to save the runset to a file. Enter “runset” for the file name. If you are planning to run CMPAnalyzer through the command line, exit Calibre Interactive and refer to “[Using the Calibre CMPAnalyzer Batch Flow](#)” on page 21.
16. Invoke CMPAnalyzer by clicking **Run CMP**. This invokes Calibre with the -dfm and -cmp switches.

Run times increase as you increase the number of metal layers or the design complexity. During the run, the density data is extracted from the layout and fed to the simulator for analysis. The simulator calculates thickness data which is then analyzed for hotspots.

17. Calibre RVE for DFM displays automatically when the run is complete. Refer to “[CMP Analysis Results](#)” on page 25 for instructions on how to interpret the CMP analysis results.

Using the Calibre CMPAnalyzer Batch Flow

The Calibre CMPAnalyzer batch flow can be used to run the CMP analysis directly from the command line, and output analysis data to the specified output directory.

This is the recommended method of running a CMP analysis after you have created a runset file.

The following example procedure is a tutorial with steps for running the batch flow. You can perform similar actions using layout data.

Try It!



Calibre CMPAnalyzer Tutorial and Example Kit

Run a batch CMP analysis using the Calibre CMPAnalyzer simulator flow. Also learn how to calculate a user-defined hotspot, run a CMP batch bucketing flow analysis, and export CMP thickness data for parasitic extraction tools.

Go to [this page](#) on Support Center to download the eKit (Documentation tab, Document Types=Getting Started Guide). The link goes to the latest release.

Prerequisites

- You have a valid recipe file; see “[Creating a Process Recipe File](#)” in the *Calibre CMP Model Builder User’s and Reference Manual*. Also see “[Calibre CMPAnalyzer Simulator Flow](#)” on page 104.
- You have a valid runset file in your working directory named “runset.” You can create a runset file by following the procedure in the “[CMP Analysis with the Calibre Interactive GUI](#)” on page 15.
- You have set CALIBRE_HOME or MGC_HOME to the path to the Calibre software tree. Refer to the [Calibre Administrator’s Guide](#) for information on setting this variable.

Procedure

1. Open a text editor and add the following lines to a new file named *batch.sh*:

```
#!/bin/sh
# run CMPAnalyzer in batch mode
calibre -gui -dfm -runset ./runset -batch | tee calibre.log
# run YS script (optional)
calibre -ys -cmp -exec cmp.ys
# run batch reporting (optional)
calibre -rve -dfm dfmdb -report ./rve.cfg -outputdir ./reports -cmp
```

This script runs Calibre Interactive in batch mode using the runset to set the GUI options. The run creates a DFM database with the CMP analysis results. The script then runs a Calibre YieldServer script and runs HTML batch reporting. You can optionally change the default settings of the CMP profile plot in Calibre RVE for DFM by setting the environment variable MGC_CMP_PROFILE_PLOT_SETTINGS in your batch script. For example,

```
export MGC_CMP_PROFILE_PLOT_SETTINGS="{BG red} {LC green}"
```

See [Table 3-2](#) on page 40 for a list of CMP profile plot settings.

2. (Optional) Create a new file named *cmp.ys* in your working directory. This file is executed following a CMP analysis. It can use any of the [CMP YieldServer Commands](#) to export parasitic extraction and hotspot reports, export rule values to a text file, and so on. The following example script opens the DFM database, and saves Calibre xRC and hotspot reports.

```
dfm::open_db dfmdb/top.dfmdb
puts "\nSaving xRC Report..."
cmp::save_xrc_report -dir "xRC"
puts "\nSaving Hotspots Report..."
cmp::save_hotspots_report -file "Hotspots_Report.txt"
```

3. (Optional) Create a batch reporting configuration file named *rve.cfg* in your working directory. Calibre RVE for DFM can read this file to automatically generate colormaps, histograms, and tables in a specified output directory. The syntax of this file is detailed

in “[DFM HTML Reporting](#)” in the *Calibre RVE User’s Manual*. Refer to “[CMP Batch Reporting](#)” on page 121 for examples specific to CMPAnalyzer.

4. Run the batch file:

```
./batch.sh
```

5. If you included batch reporting, open *index.htm* in the *./reports* directory. The HTML report is similar to [Figure 3-37](#).
6. View the results in Calibre RVE, as discussed in “[CMP Analysis Results](#)” on page 25. This section also has information on interpreting the results.

Chapter 3

CMP Analysis Results

You can work with Calibre CMPAnalyzer CMP analysis results using the Calibre RVE for DFM user interface.

Calibre RVE for DFM allows you to view actual scores, display histograms indicating the distribution of scores, and generate colormaps illustrating how scores relate to specific locations within your design. You can also use the Calibre RVE for DFM user interface in conjunction with the Calibre DESIGNrev layout viewer to highlight and view specific problems within your design.

Reviewing Analysis Results.....	25
Recalculating Hotspots	42
Creating Scatter Plots	43
Creating a Simulated Profile Plot.....	45
Performing Data Analysis in Property View.....	47
Getting Property Values from a Colormap.....	51
Displaying MaterialNT(T) Properties	52
CMP Hotspots Clustering.....	55
Exporting Hotspot Reports.....	62
Exporting Colormap Data.....	62
Exporting Calibre CMPAnalyzer Thickness Data to File for Extraction Tools	64
Exporting Analysis Results in Batch Mode.....	66
Calculating User-Defined Hotspots	69
Hot Spot Analysis Batch Procedure.....	80

Reviewing Analysis Results

The easiest way to understand how the raw analysis data relates to your design is to view histograms and colormaps.

The layout is broken up into a number of windows with data values assigned for each window. The default color scheme is such that red represents the highest data values and dark blue represents the lowest data values.

The following example procedure is a tutorial with steps for reviewing analysis results. You can perform similar actions using layout data.

Try It!



Calibre CMPAnalyzer Tutorial and Example Kit

Run a batch CMP analysis using the Calibre CMPAnalyzer simulator flow. Also learn how to calculate a user-defined hotspot, run a CMP batch bucketing flow analysis, and export CMP thickness data for parasitic extraction tools.

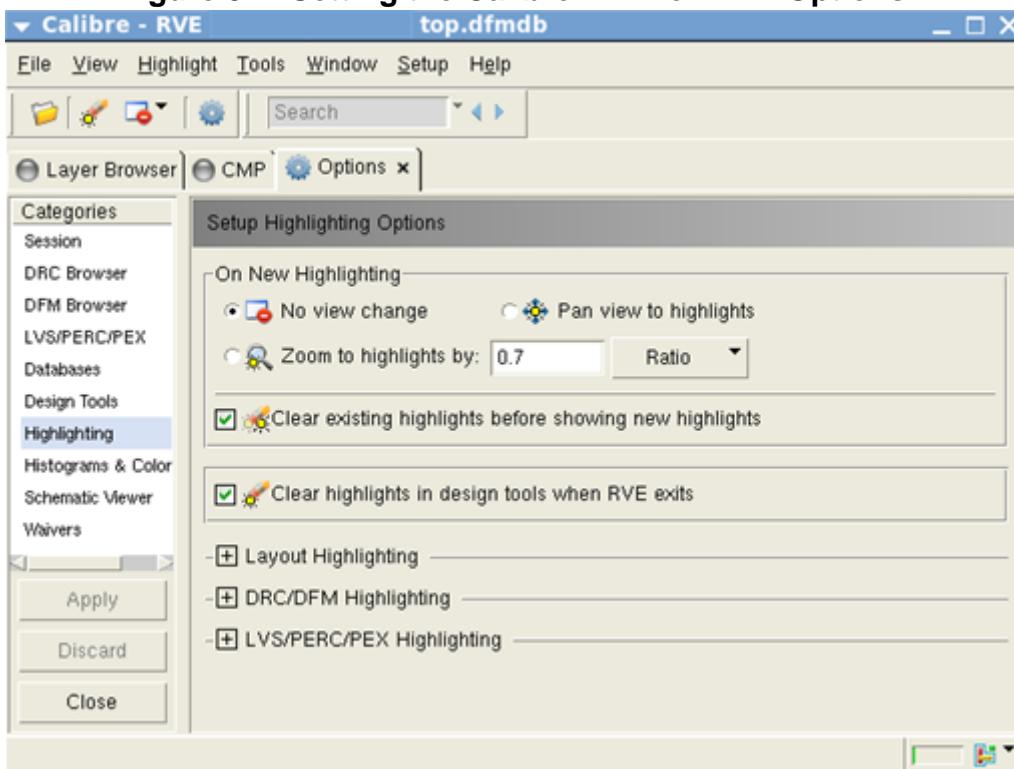
Go to [this page](#) on Support Center to download the eKit (Documentation tab, Document Types=Getting Started Guide). The link goes to the latest release.

Prerequisites

- You have performed a CMP analysis as described in “[CMP Analysis](#)” on page 15, and you have a DFM database containing the analysis results.
- Calibre DESIGNrev or another layout viewer that is supported by Calibre RVE is open with your layout loaded. See “[Communication with the Design Tool](#)” in the *Calibre Interactive User’s Manual* for information on communication between Calibre RVE and the layout viewer.

Procedure

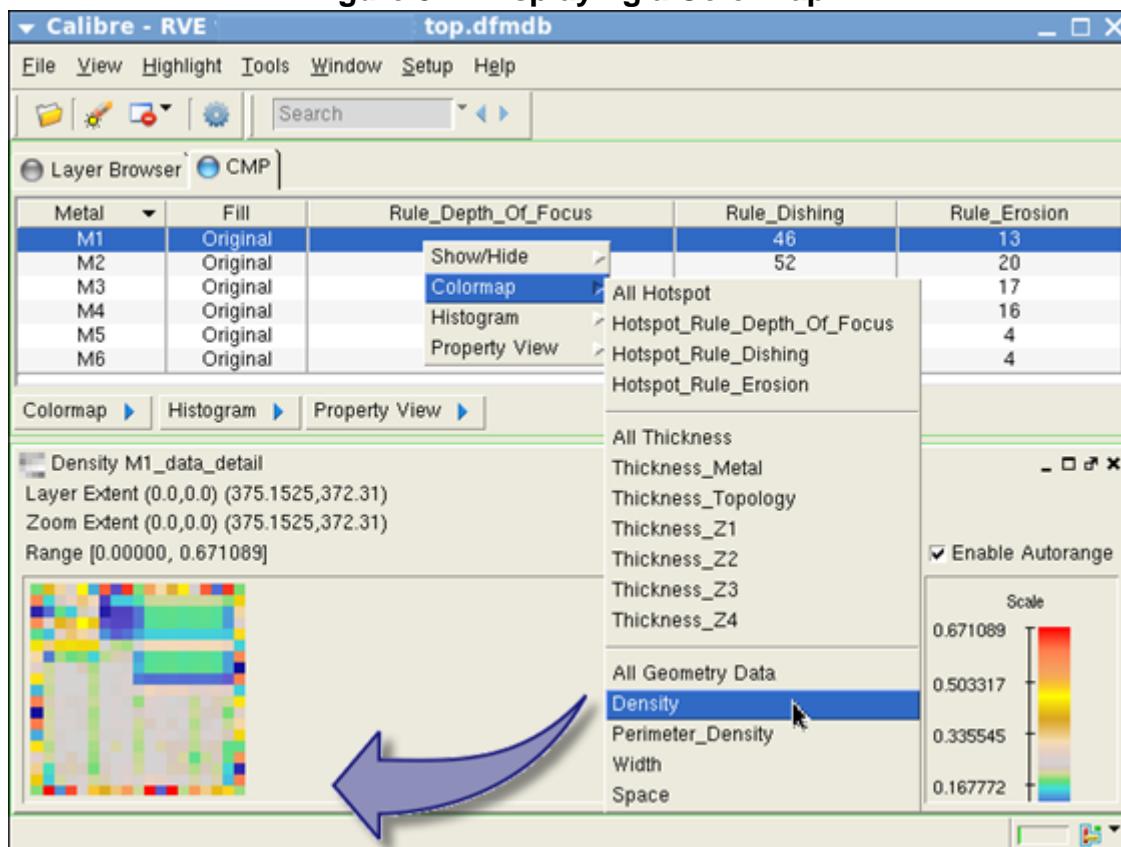
1. Open Calibre RVE for DFM:
 - **From Calibre Interactive** — Click Start RVE. Calibre RVE is opened and the results database on the Outputs pane is loaded.
 - **From a layout viewer** — In Calibre DESIGNrev, select **Verification > Start RVE**. For most other viewers, select **Calibre > Start RVE**.
Specify the DFM database in the “Database” entry and select a “Database Type” of DFM.
2. Set the results viewing highlighting options in the Calibre RVE for DFM user interface.
 - a. Choose **Setup > Options** from the Calibre RVE for DFM menu, then choose the **Highlighting** category.

Figure 3-1. Setting the Calibre RVE for DFM Options

- b. Select **Zoom to highlights by** and set the value to 0.7.
- c. Make sure **Clear existing highlights** is selected.
- d. Switch to the **DFM Browser** category, select **Flat** next to **On double-click, browse layer**.
- e. Click **Apply**, then **Close**.
3. Set the colormap display options.
 - a. Choose **Setup > Options** and click the **Histograms & Colormaps** category.
 - b. In the **Colormaps (DFM Browser only)** section of the pane, uncheck the **Show Borders** checkbox to display colormaps without borders.
 - c. Click **Apply**, then **Close**.
4. Review the data displayed in the **CMP** tab.

When the CMP analysis is complete, a table appears, displaying metal layers (rows) and hotspots (model-based checks) applied to these metal layers (columns). The values in the table represent the number of unique violations for the hotspot and metal layer combination corresponding to that column and row. If there is no value reported, that hotspot is not applicable to that metal layer.
5. Right-click on the M1 row and choose **Colormap > Density**.

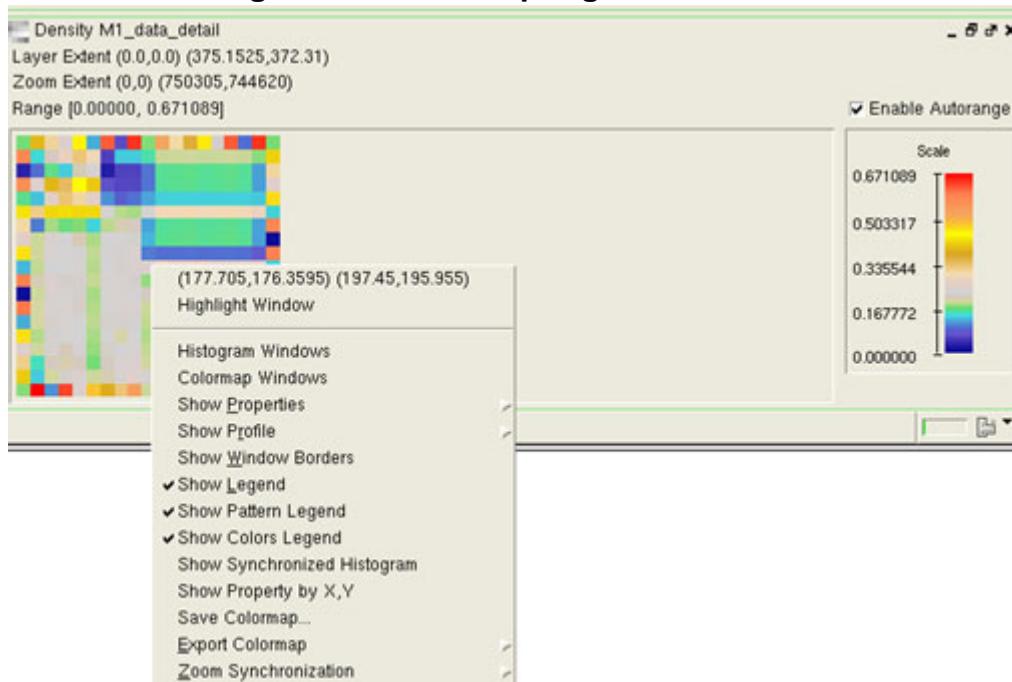
Figure 3-2. Displaying a Colormap



This creates a density colormap for layer M1 inside Calibre RVE for DFM.

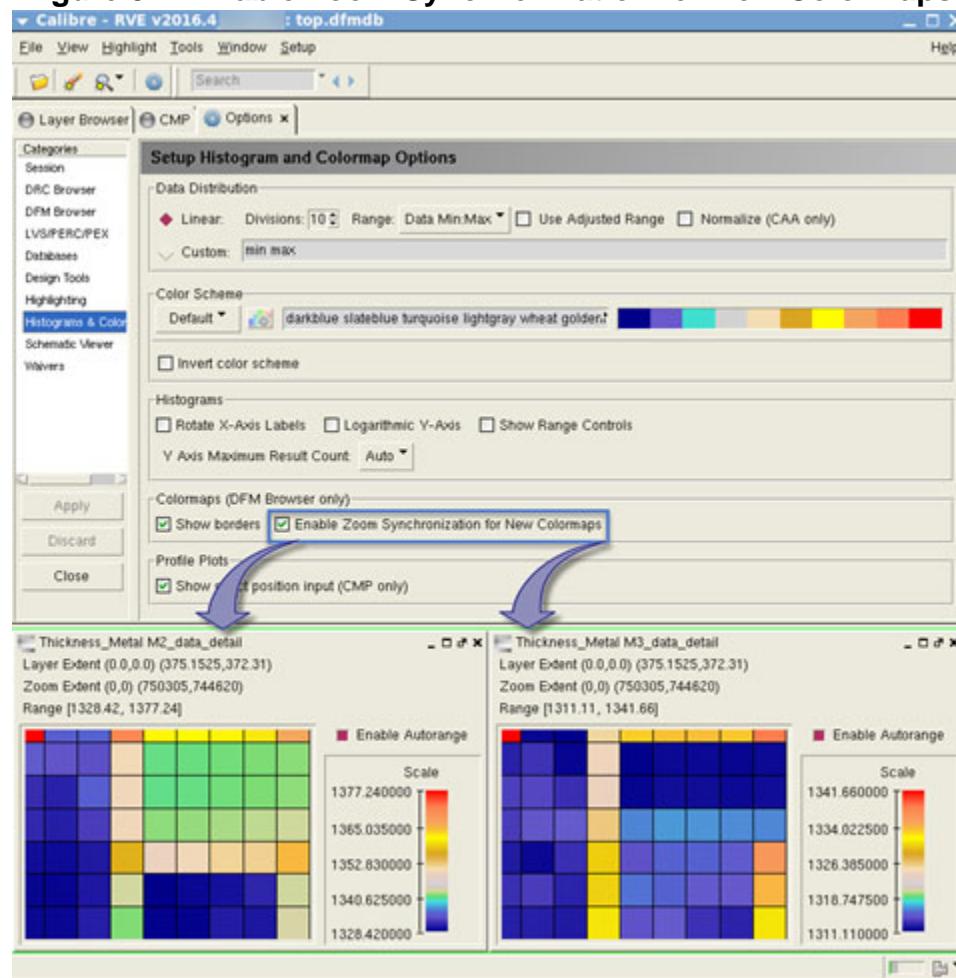
Alternatively, you can choose **Property View > Density** to display a colormap and different plot types within a single window. See “[Performing Data Analysis in Property View](#)” on page 47 for more information.

- a. Right-click on a tile in the colormap and display menu items for property and plot information. You can also control the plot display and save and export colormap data from this menu

Figure 3-3. Colormap Right-Click Menu

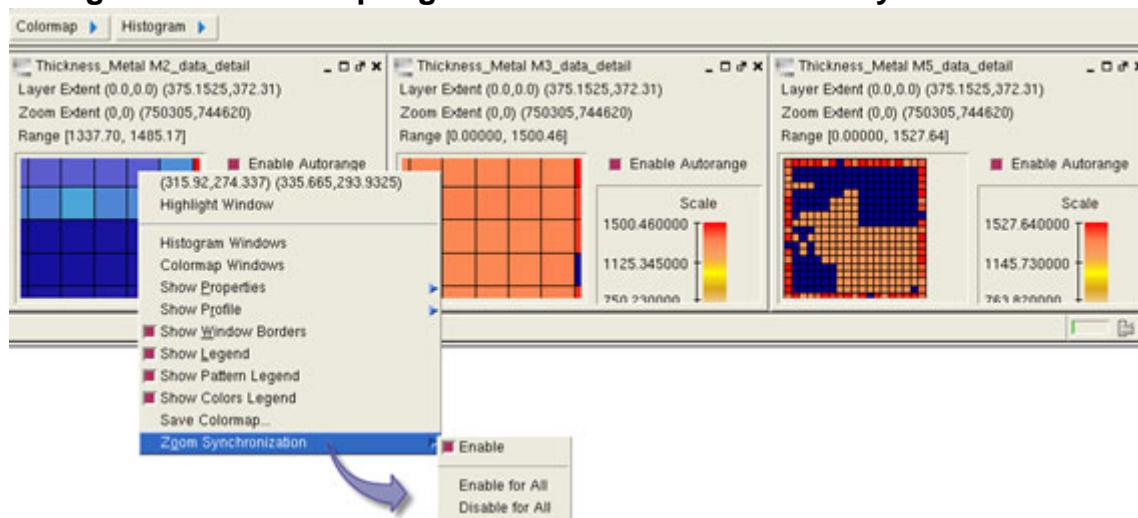
- b. To zoom in on the colormap, press and hold the right mouse button, drag the mouse down and to the right, and release the right mouse button. To zoom out, drag the mouse up and to the left. You can synchronize the zoom regions for two or more colormaps by checking the **Enable Zoom Synchronization for New Colormaps** checkbox under the **Histogram & Colormaps** category of the **Setup > Options** tab and then clicking on **Apply**.

Figure 3-4. Enable Zoom Synchronization for New Colormaps



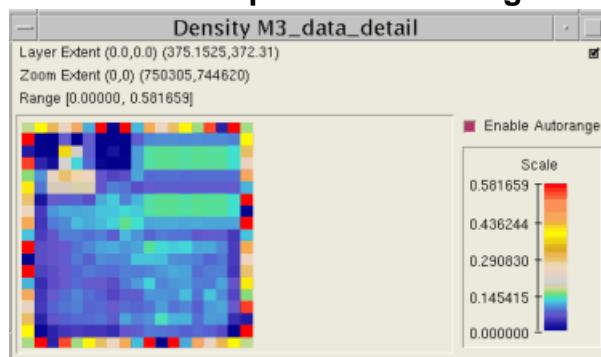
- c. Right-click on a colormap and use the **Zoom Synchronization** submenu to control zoom synchronization between multiple colormaps.
 - o **Enable** — Choose this checkbox to enable or disable zoom synchronization for the current colormap.
 - o **Enable for All** — Choose this menu option to enable zoom synchronization for all current colormaps.
 - o **Disable for All** — Choose this menu option to disable zoom synchronization for all current colormaps.

Figure 3-5. Colormap Right-Click Submenu — Zoom Synchronization



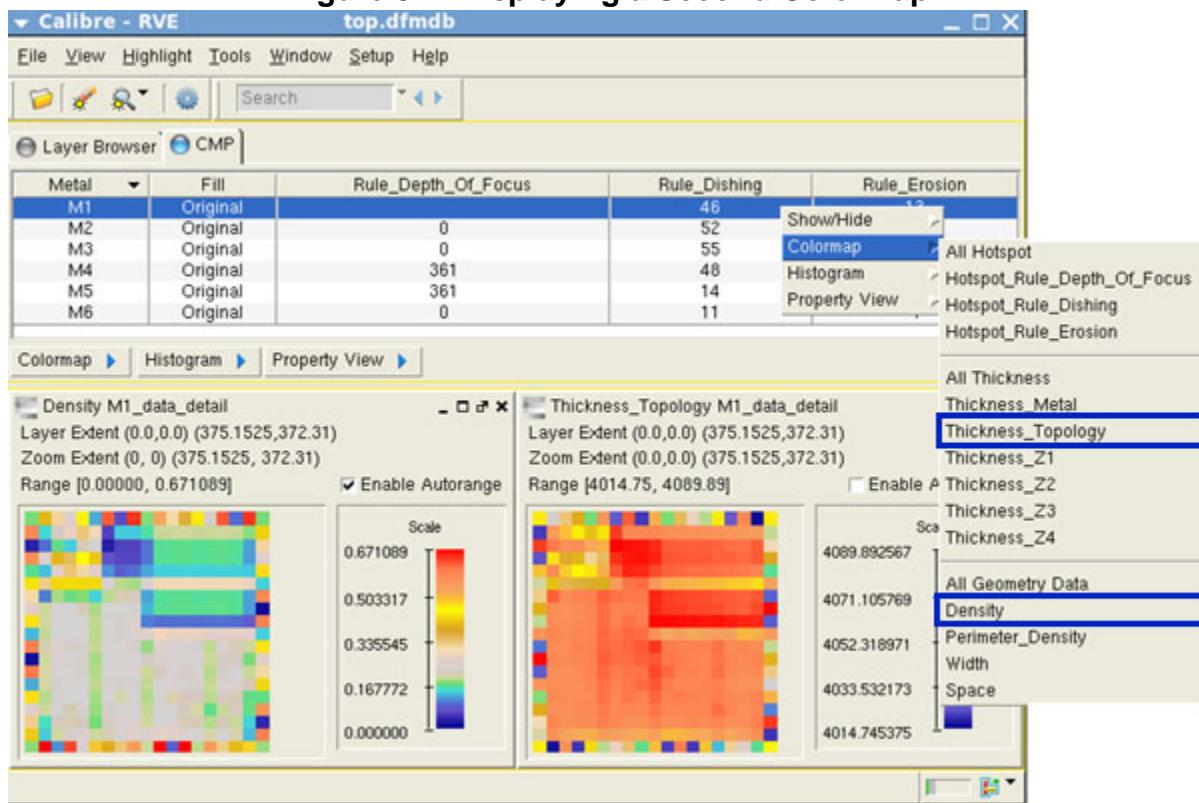
- For colormap color definition, use the **Setup > Options** menu and check the **Enable Autorange** checkbox on the fast colormap to fit colors to the current visible range of values.

Figure 3-6. Colormap with Color Range Enabled



- Right-click on the M1 row again and choose **Colormap > Thickness_Topo** to display a second colormap.

Figure 3-7. Displaying a Second Colormap



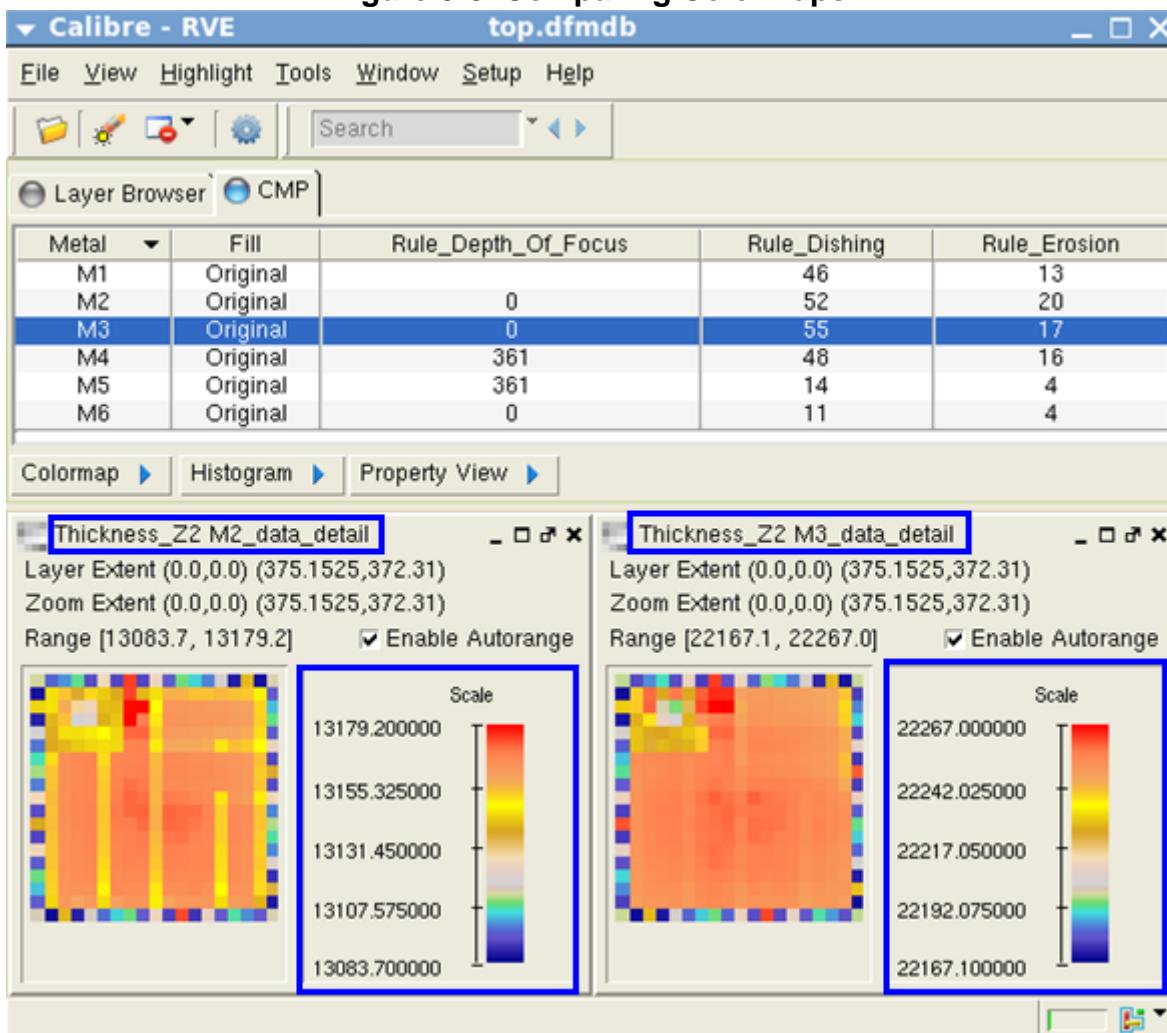
The colormap on the left displays the densities; the one on the right displays the thickness topology. The default color convention is such that the red color indicates the largest values, and the dark blue color indicates the smallest values.



You can change the color scheme by choosing **Setup > Options** and selecting the **Histograms** tab.

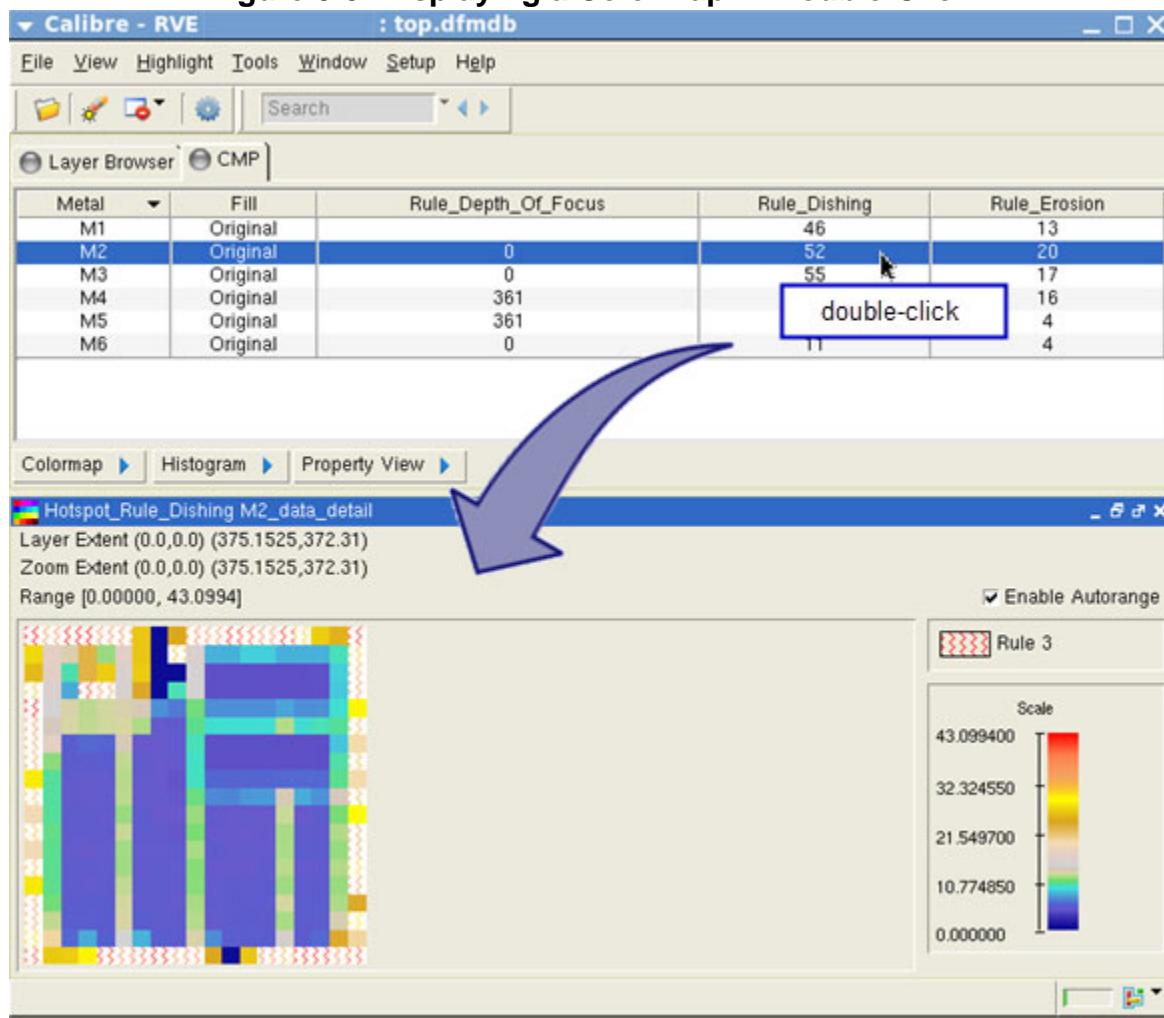
7. In either colormap, hold your mouse over a square of interest to display a tooltip showing the value corresponding to that particular window.
8. Compare metal heights for layer M2 and layer M3.
 - a. Close both existing colormap windows.
 - b. Click on the lower edge of the Calibre RVE for DFM window and drag the edge down to enlarge the window size.
 - c. Right-click the row for M2, and choose **Colormap > Thickness_Z2**.
 - d. Repeat the previous step for the M3 row.
 - e. Choose **Window > Tile Vertical**.

Figure 3-8. Comparing Colormaps



9. Create a colormap and a histogram displaying the results of the Dishing check or any other rule with hotspots.
 - a. From the menu, choose **Window > Close All** to close all existing colormap windows.
 - b. To create a colormap quickly, double-click on the *table cell* that corresponds to the row for M2 and column for Rule_Dishing (or any other rule with hotspots). This creates a colormap of data values color coded throughout the chip.

Figure 3-9. Displaying a Colormap — Double-Click



The colormap for Dishing represents the difference between oxide and metal heights. The color convention is such that the red color indicates the largest check results values, and the dark blue color indicates the smallest check results.

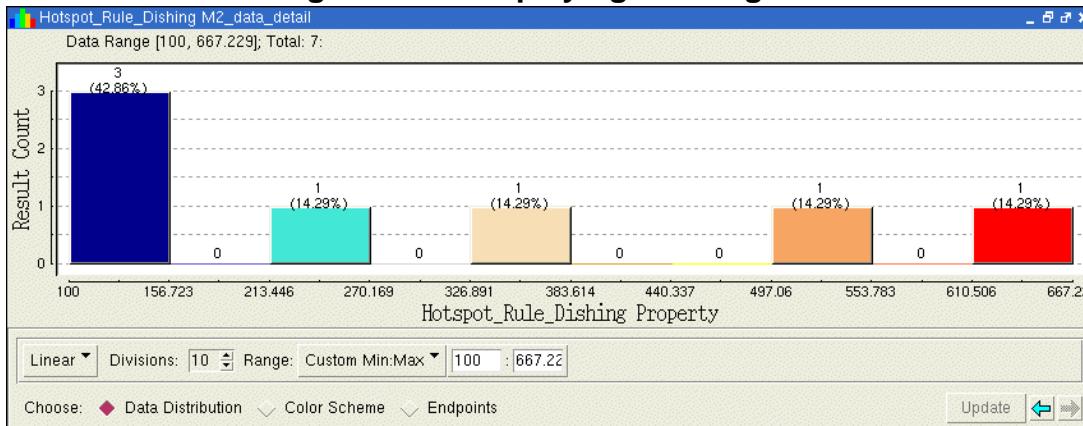
Note the dotted lines near the top and right sides of the colormap. These dotted lines represent the actual extent of the layout relative to the window grid. Extending the colormap beyond the actual extent allows you to visualize hotspots and other data for those portions of the design that might otherwise be represented as a tiny sliver.

You can click on the **Oxide Height Min** stipple pattern in the legend to highlight these tiles on the colormap and in Calibre DESIGNrev. Click the stipple pattern a second time to remove the highlighting.

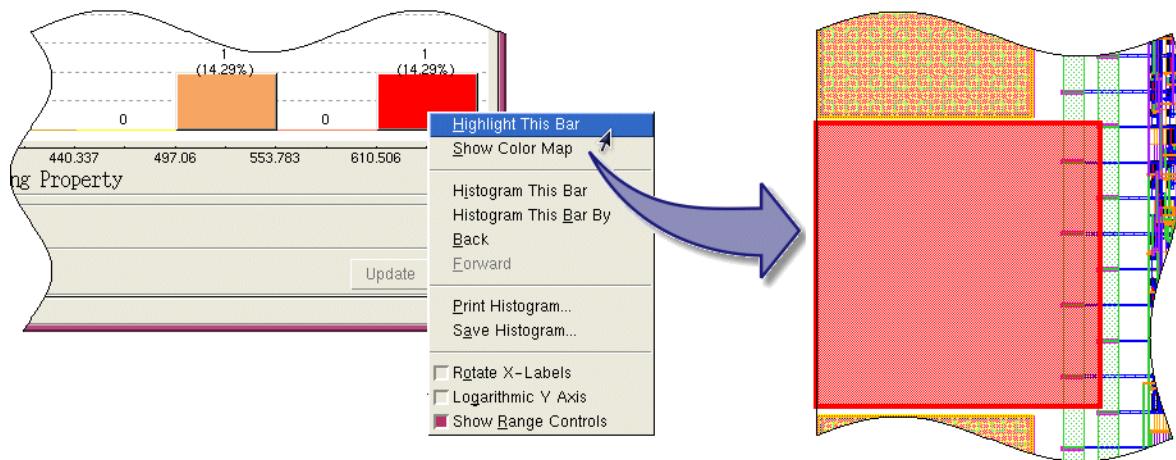
- Close the colormap. Right-click on the row for M2, then select **Histogram > Hotspot_Rule_Dishing**. This creates a histogram of results sorted into 10 bins by default, which you can change. The number above each bar in the histogram represents the number of windows that fall within that data range. To narrow the data range, right-click on the histogram and enable the **Show Range Controls**

option. Next to **Range**, select **Custom Min:Max** and enter 100 for the lower bound. Click **Update**. The histogram is updated to reflect the new data range.

Figure 3-10. Displaying a Histogram



If you have limited time and are interested in only the worst areas of a layout for a particular rule, you can right-click on any histogram bar and choose **Highlight This Bar** to display the corresponding windows in Calibre DESIGNrev.



10. Configure the tooltip display to report other values for that portion of the design.

Each square in a colormap represents a window (or rectangular area) within your design. When you place your cursor over a particular square in the colormap, you display a tooltip that reports the properties and coordinates for that window. By default, only the original property of interest and the coordinates are displayed in the tooltip window. Additional properties can be added from a property list.

- a. Close the histogram from the previous step. Right-click on the M1 row and choose **Colormap > Thickness_Metal**.
- b. Right-click anywhere over the colormap window to display the colormap popup menu.

c. Choose **Show Properties > Density**.

Figure 3-11. Configuring Tooltip Display

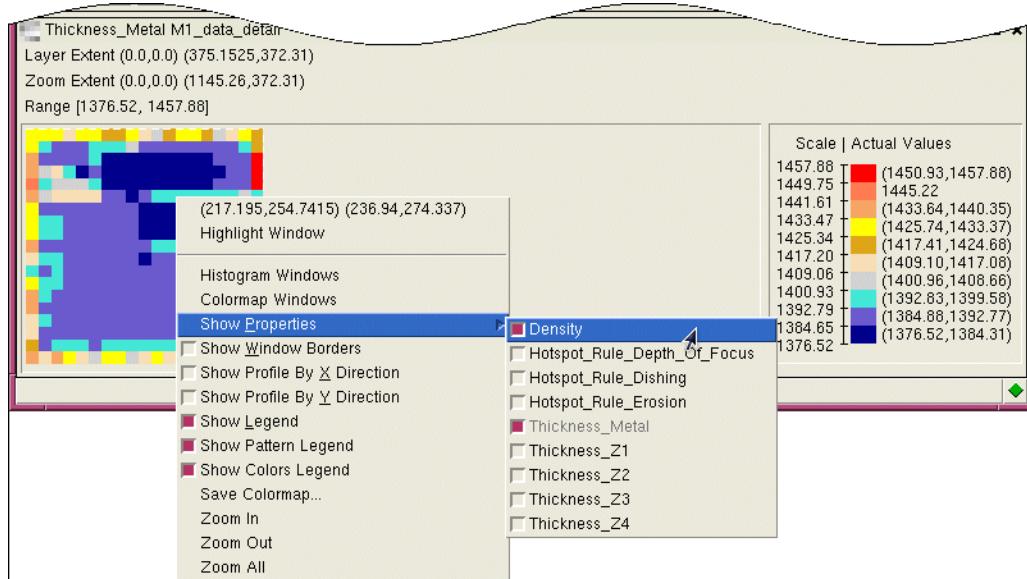
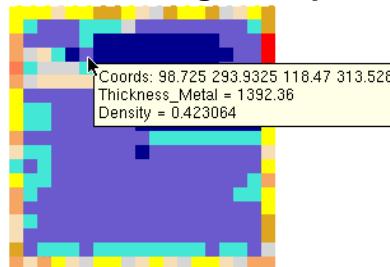
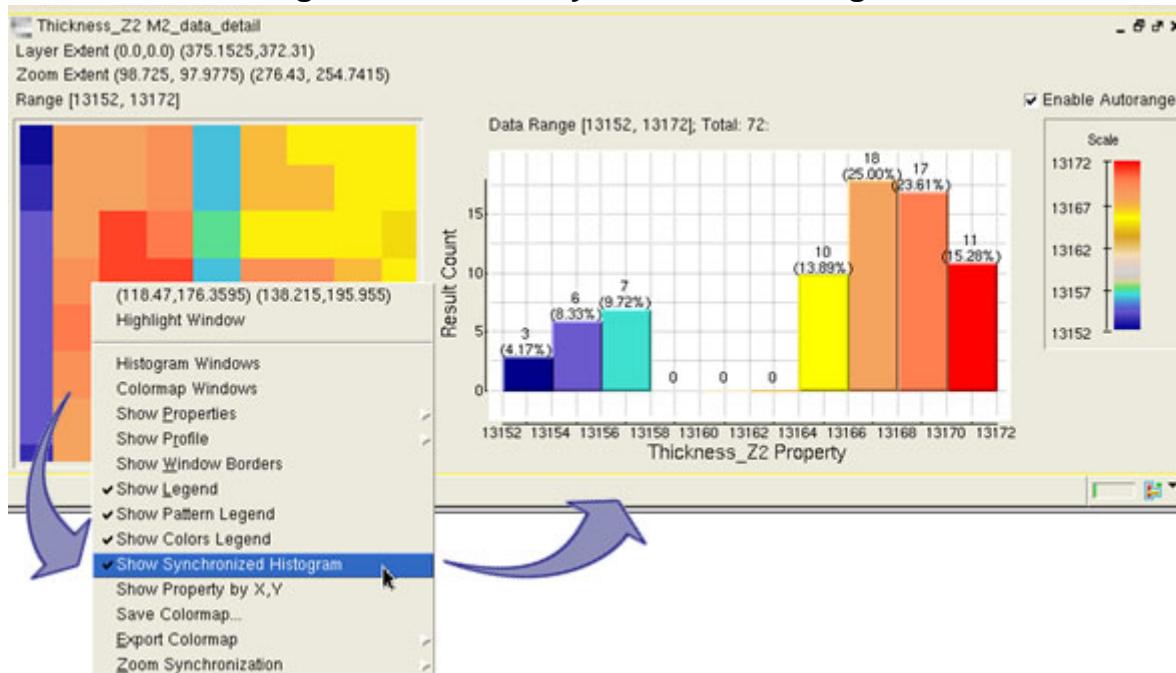


Figure 3-12. Using Tooltip Displays



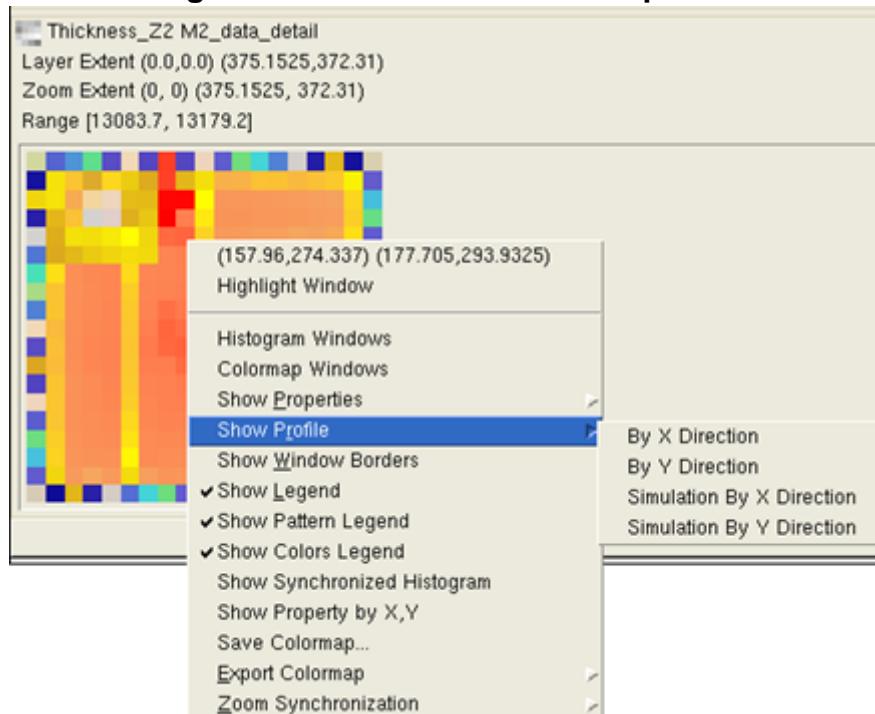
11. Zoom in on a colormap region and choose **Show Synchronized Histogram** from the right-click submenu. This displays a histogram of the colormap zoom region.

Figure 3-13. Show Synchronized Histogram



- Right-click on a colormap window and choose **Show Profile** to view the profile plot options.

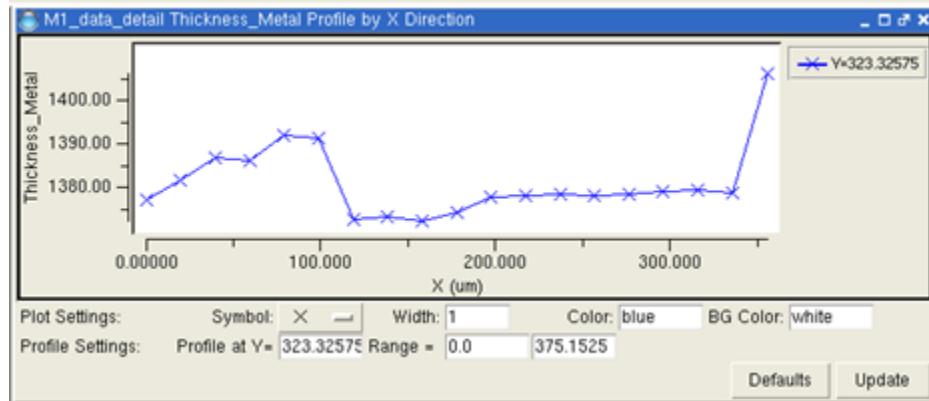
Figure 3-14. Show Profile Plot Options



- Display a profile plot of the design that represents thickness versus the X or Y direction.

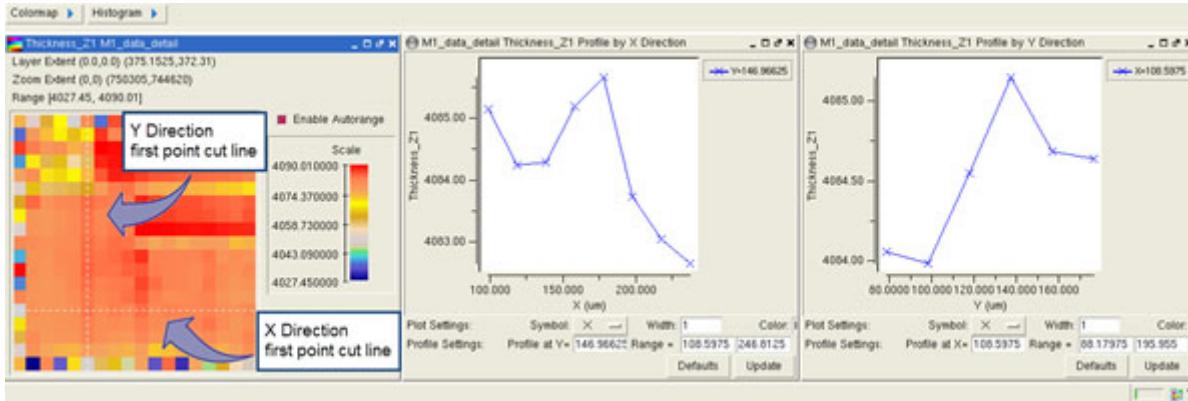
- a. Right-click on the colormap window and choose **Show Profile > By X Direction**.
The profile plot is displayed in a new window as shown in [Figure 3-15](#).

Figure 3-15. Profile Plot



To display a profile plot of a colormap segment, Ctrl+left-click the first and last points of a segment in the colormap window. The profile plot window displays the segment. If the first and last points are not in the same horizontal or vertical cut line, the plot for the cut line starts from first point (first click), and the horizontal or vertical direction is detected automatically as shown in [Figure 3-16](#).

Figure 3-16. Profile Plots of Colormap Segments (X Direction and Y Direction)



To zoom in or zoom out on a profile, do one of the following:

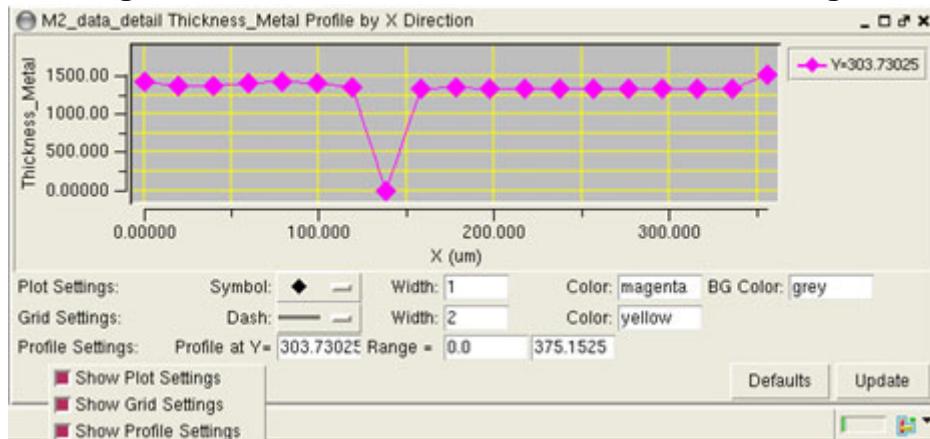
- o To zoom in, hold down the right mouse button and move the cursor left to right.
To zoom out, hold down the right mouse button and move right to left. Zooming out returns the plot to the previous zoom in view.
- o Hold down the right mouse button and choose **Zoom In**, **Zoom Out**, or **Zoom All** from the available menu options.
- o Use the keyboard shortcuts shown in [Table 3-1](#).

Table 3-1. Zoom Shortcut Keys for CMP Analysis

Shortcut	Command
Ctrl + a	Zoom all
Ctrl + z	Zoom in
Z	Zoom out

- b. You can change the profile plot settings using the text fields and dropdown lists. Right-click in the lower-left corner of the profile plot window for a popup menu to show or hide Plot Settings, Grid Settings, and Profile Settings. Adjust the range of the plot by entering new values in the text field and clicking **Update**.

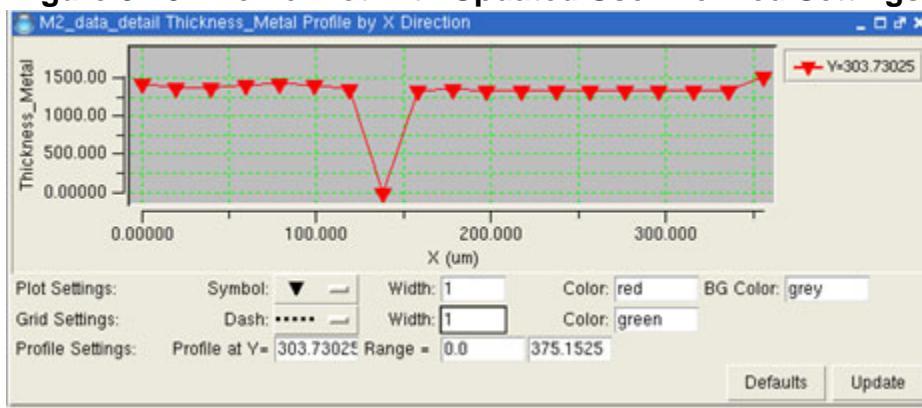
The default configuration of the profile plot settings can be redefined by choosing **Setup > Histograms & Colormaps** and specifying the Plot Settings and Grid Settings.

Figure 3-17. Profile Plot with User-Defined Settings

Plot colors may be specified as Tk symbolic color names or RGB hexadecimal color codes (for example “blue” or “#0000FF”). Some valid color names are red, green, black, grey, blue, yellow, magenta, cyan, darkblue, slateblue, turquoise, lightgray, wheat, goldenrod, sandybrown, and coral.

For example, [Figure 3-18](#) displays an updated profile plot with a red plot line, triangle symbol, green grid lines, and a gray background.

Figure 3-18. Profile Plot with Updated User-Defined Settings



The profile plot settings and values are shown in [Table 3-2](#).

Table 3-2. Profile Plot Settings

Settings	Name	Values	Description
Plot Settings	Symbol	Dropdown list selection.	Set the type of symbol for the plot.
	Width	Number of pixels.	Set the width of the plot line.
	Color	Color name or hexadecimal code.	Set the color of the plot line.
	BG Color	Color name or hexadecimal code.	Set the background color of the plot.
Grid Settings	Dash	Dropdown list selection.	Set the dash style of the grid lines.
	Width	Number of pixels.	Set the width of the grid lines.
	Color	Color name or hexadecimal code.	Set the color of the grid lines.
Profile Settings	Profile at Y=	Coordinate in microns.	Specify profile Y coordinate.
	Range =	Coordinate in microns.	Specify the range of the plot.

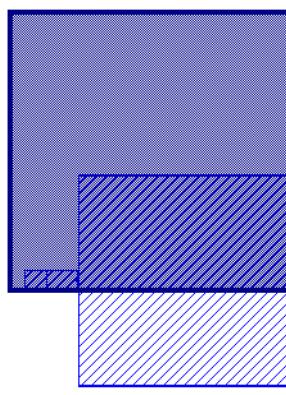
14. Find an area of interest and view the colormap in Calibre DESIGNrev.
 - a. Move your mouse over the red areas in the colormap to view the check results values.
 - b. When you find a window reporting a particularly high value for **Thickness_Metal**, click the right mouse button to display the popup menu.

The first line of data displays the coordinates for that window.

- c. Click on the coordinates in the popup menu.

Notice that the view in the Calibre DESIGNrev window changes. It now displays the area in the design represented by that window.

- d. In the Layers pane in Calibre DESIGNrev, hide all layers except layer M1 and the rve highlight layer. The blue hash mark square highlights the window selection.



- e. When you are done analyzing hotspot rule data, close the colormap in the Calibre RVE for DFM window.

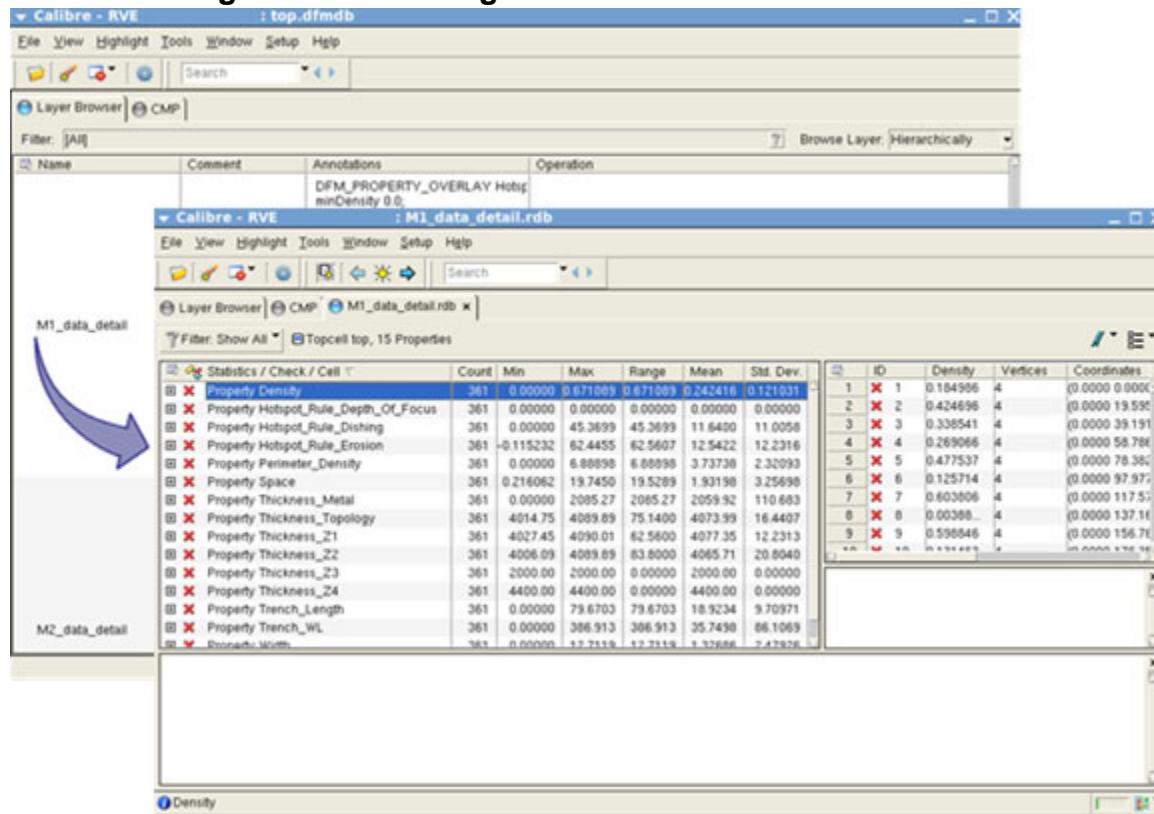
15. View detailed properties:

You can also get a good overview of how well the design is doing against all rules by looking at rule statistics for all rules simultaneously.

- a. In the Calibre RVE for DFM window, click on the **Layer Browser** tab.
- b. Double-click on the row for M1_data_detail.

A Calibre RVE for DRC window appears as a new tab. Inside there is a list of properties with minimum, maximum and mean values. You might be particularly interested in these values when working with your own design. You can display a colormap or a histogram for any particular rule by clicking on the **Histogram** or **Colormap** options from the right-click menu.

Figure 3-19. Viewing Individual Errors in Calibre RVE



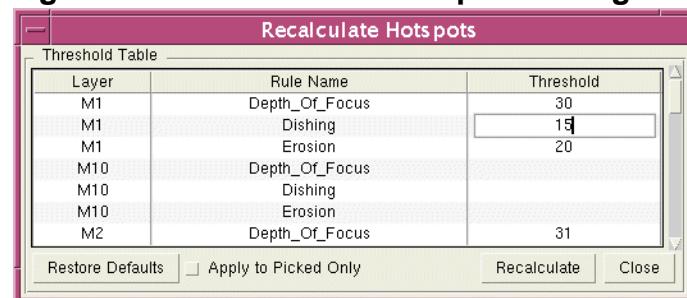
- Once you are done looking at the statistics, close the **M1_data_detail.rdb** tab and switch back to the **CMP** tab inside the Calibre RVE for DFM window.

Recalculating Hotspots

You can change the constraint values on hotspot rules in Calibre RVE for DFM by using the Recalculate Hotspots dialog box (**Tools > CMP > Recalculate Hotspots**).

This dialog box allows you to enter new threshold values for various layer and hotspot rule name combinations. Refer to [Figure 3-20](#).

Figure 3-20. Recalculate Hotspots Dialog Box



Prerequisites

- You have performed a CMP analysis as described in “[CMP Analysis](#)” on page 15, and you have a DFM database containing the analysis results.
- Calibre RVE for DFM is running and the CMP analysis results are displayed in the CMP tab.

Procedure

1. Choose **Tools > CMP > Recalculate Hotspots**. This displays the Recalculate Hotspots dialog box with an empty table.
2. For each row, click in the Layer and Rule Name columns and select a value using the dropdown menus. Enter a new threshold value by clicking on the Threshold column for the row.
3. Click **Recalculate** then **Close** to display the new values in Calibre RVE for DFM.

Tip

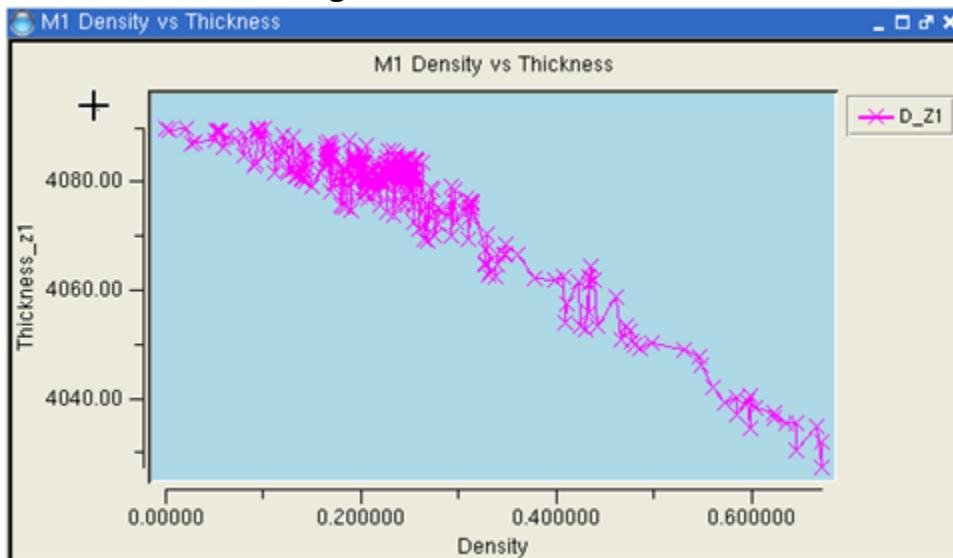
 To apply default threshold values to all layer and rule name combinations in the design, click **Restore Defaults**. To apply defaults to only the **Layer** and **Rule Name** combinations that you selected using the dropdown menus, enable **Apply to Picked Only** before clicking **Restore Defaults**.

Creating Scatter Plots

You can plot the values of one property versus another for a single layer or across two layers using scatter plots.

[Figure 3-21](#) shows an example.

Figure 3-21. Scatter Plot



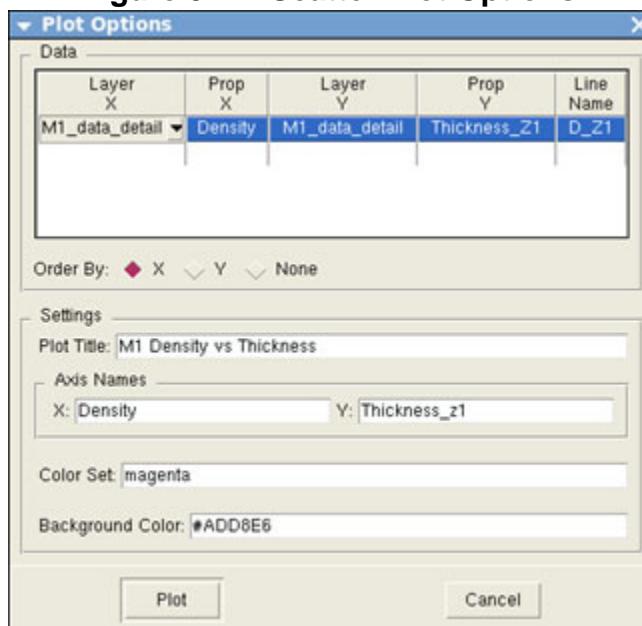
Prerequisites

- You have followed the steps described in “[CMP Analysis](#)” on page 15, and you have a DFM database containing the analysis results.
- Calibre RVE for DFM is running and the CMP analysis results are displayed in the **CMP** tab.

Procedure

1. Choose **Tools > Scatter Plot** to open the Plot Options dialog box.
2. Click in each column in the Data table to enter a value. The columns have the following meaning:
 - **Layer X andProp X** — Specifies the layer and property names to plot on the x-axis. The layer names are normally _detail layers.
 - **Layer Y andProp Y** — Specifies the layer and property names to plot on the y-axis. The layer names are normally _detail layers.
 - **Line Name** — Specifies a name for the plot, which is shown in the legend.
3. Select a setting for **Order By**, which controls how lines are drawn connecting the data points on the scatter plot.
4. In the Settings section, specify the title of the plot, names for the x- and y-axis, and an optional color set. Specify one color for each row in the Data table, separated with spaces. You can optionally specify a background color for the plot (default is black). Colors are specified as Tk symbolic color names (green, blue, red, yellow, magenta, cyan ...) or RGB hexadecimal color code such as #D3D3D3 (light gray).

[Figure 3-22](#) shows an example of the Plot Options dialog box.

Figure 3-22. Scatter Plot Options

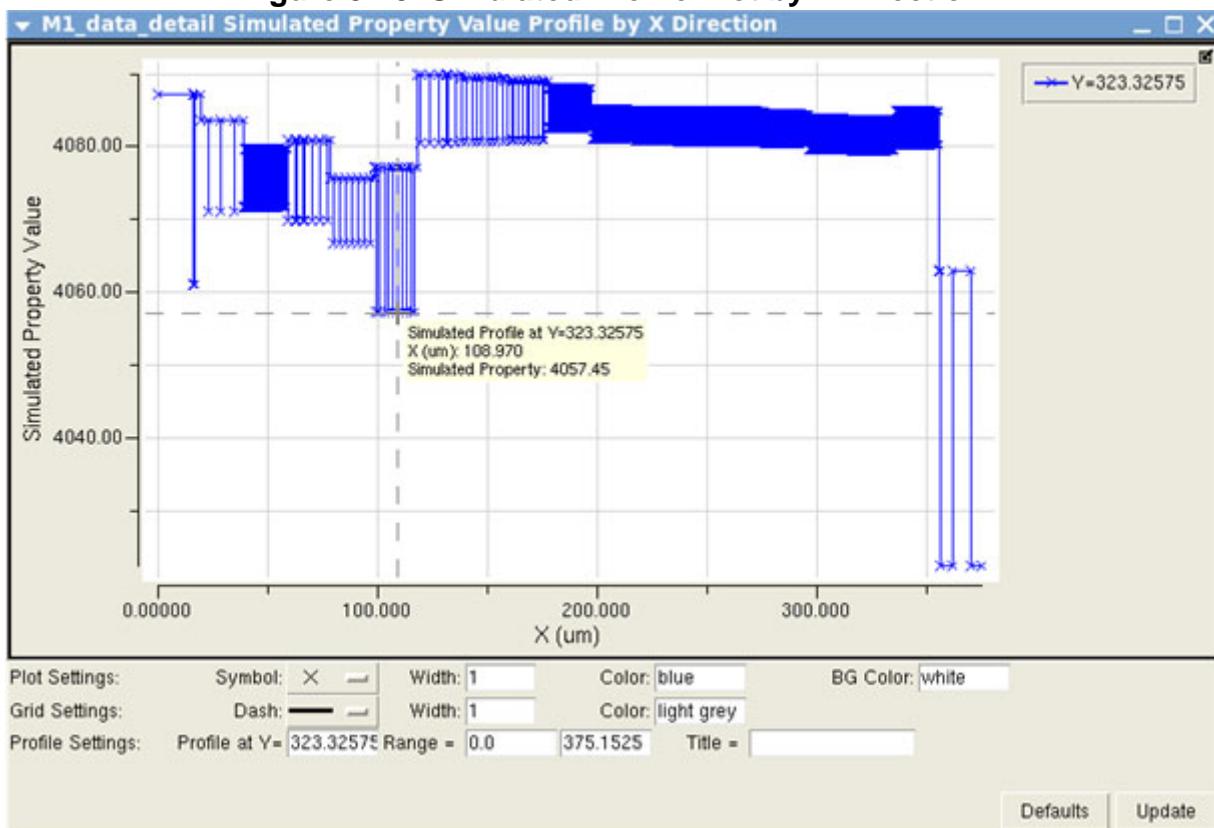
5. Click **Plot** to generate the scatter plot. You can view the values for a particular data point by moving your mouse over it.
6. To save the plot as either a GIF image or a text file, right-click on the plot and choose **Save Plot**.

Creating a Simulated Profile Plot

You can create a profile plot for a property based on simulated data only, using an x- or y-directional scan over the colormap tiles to plot heights.

[Figure 3-23](#) shows an example.

Figure 3-23. Simulated Profile Plot by X Direction



Prerequisites

- You have followed the steps described in “[CMP Analysis](#)” on page 15, and you have a DFM database containing the analysis results.
- Calibre RVE for DFM is running and the CMP analysis results are displayed in the **CMP** tab.

Procedure

1. Right-click on the M1 row and choose **Colormap > Thickness Topology**.

This creates a thickness topology colormap for layer M1 inside Calibre RVE for DFM.

2. Right-click a tile of interest in the colormap to display menu items for property and plot information.

3. Choose **Show Profile > Simulation By X Direction**.

This creates a profile plot of the x-axis cut-line over the selected colormap tile.

- a. Right-click the lower-left corner of the profile plot window to show the plot settings.
- b. Choose **Show Grid Settings** and change the default grid color to “light grey” in the Color field. You can also change the default plot colors, symbols, background, and profile range settings.

- c. Click **Update** to display the profile plot with your updated settings.

Results

You have created a thickness topology profile plot using only simulation data. This type of plot can be used for comparing simulated versus measured atomic force microscopy (AFM) line-scan data.

Performing Data Analysis in Property View

You can use Property View functionality to visualize and analyze colormap, histogram, and profile and simulation plots in a single window. This enables you to easily compare and synchronize the data between different types of plots.

Prerequisites

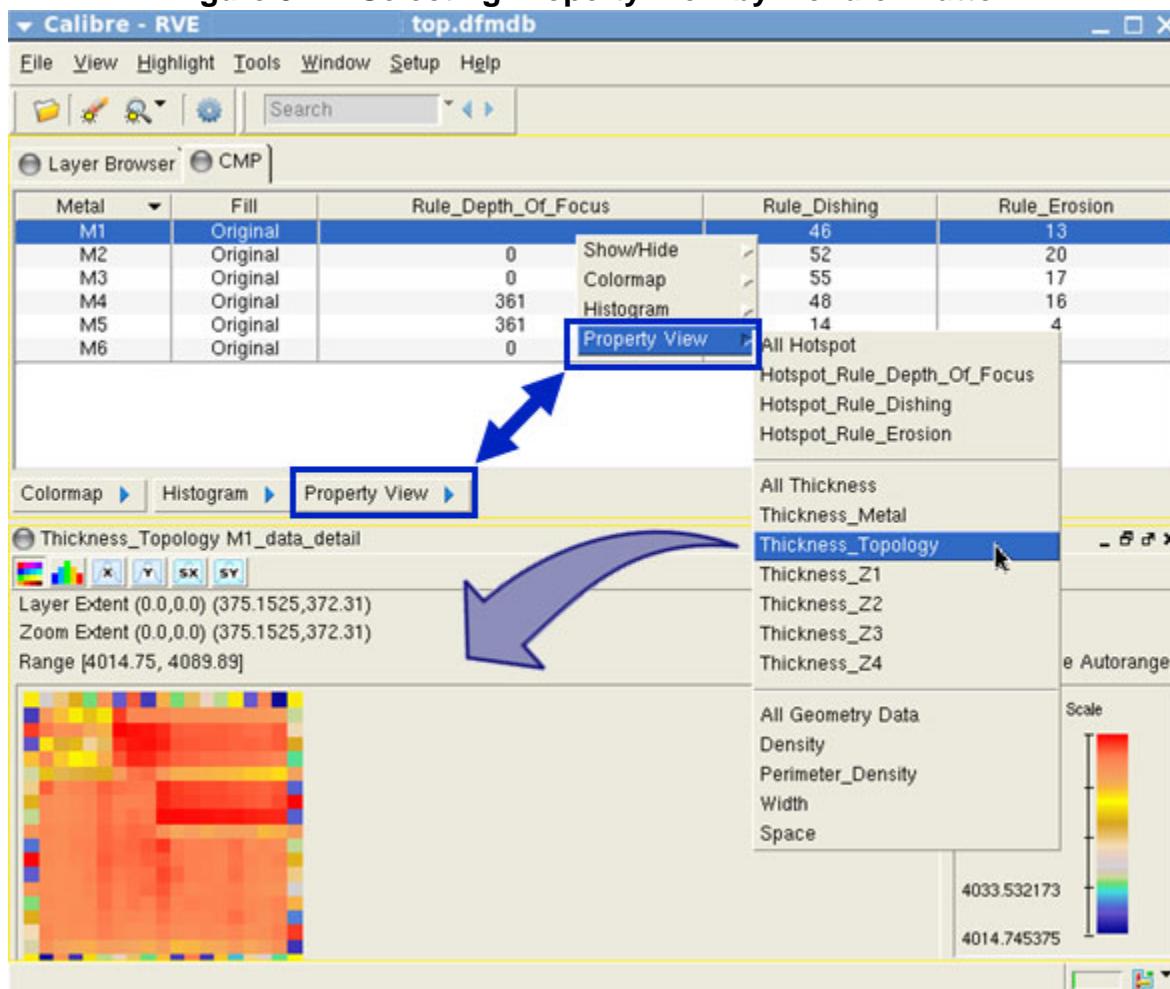
- You have performed a CMP analysis as described in “[CMP Analysis](#)” on page 15, and you have a DFM database containing the analysis results.
- Calibre RVE for DFM is open with data displayed in the **CMP** tab and you have reviewed the **Setup > Options** menu items as described in “[Reviewing Analysis Results](#)” on page 25.

Procedure

1. Right-click on the M1 row and choose **Property View > Thickness Topology**, or alternatively, click the **Property View** button (in the lower-left corner) and choose **Thickness Topology**.

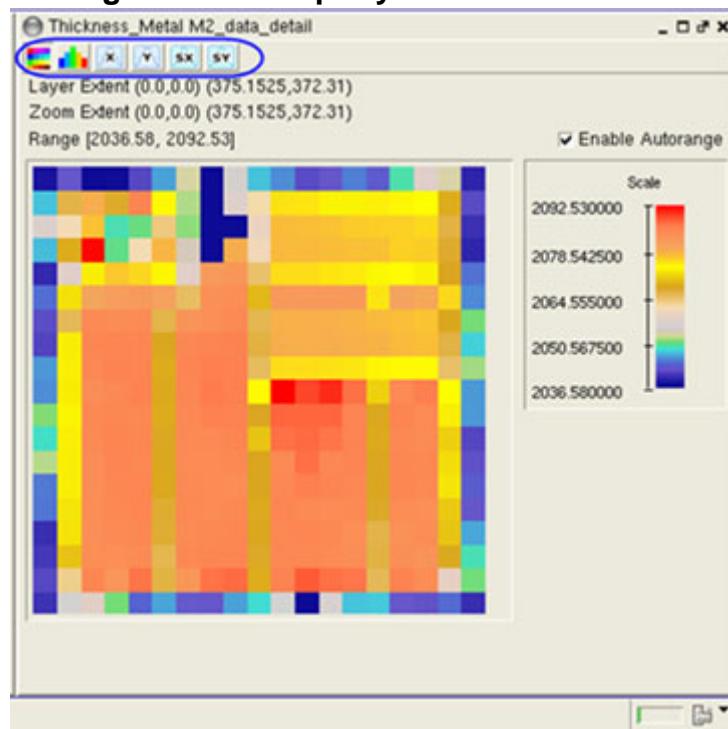
The Property View GUI window opens and displays a colormap of M1 Thickness Topology.

Figure 3-24. Selecting Property View by Menu or Button



2. Use the buttons at the top of the Property View GUI to create different types of plots in the same window.

Figure 3-25. Property View GUI Window



You can toggle the GUI buttons to draw or hide the different plot views. From left-to-right, the GUI buttons control colormap, histogram, profile by X/Y, and simulation by X/Y plot views.

Figure 3-26. Property View GUI Window Plots



3. Zoom in and out of the colormap and observe that the profile by X/Y direction and simulation by X/Y direction plots are synchronized to zoom in and out simultaneously.
The histogram plot can also be synchronized.
4. Perform further analysis of the plot data as needed.
5. Close the Property View GUI window when you are finished with your analysis by choosing **Close** in the dropdown menu in the upper-left corner of the window.

Results

You have used Property View functionality to compare and analyze CMP data using different plot types in a single GUI window.

Getting Property Values from a Colormap

You can show the value for a DFM property at a specific location in a generated Calibre CMPAnalyzer colormap.

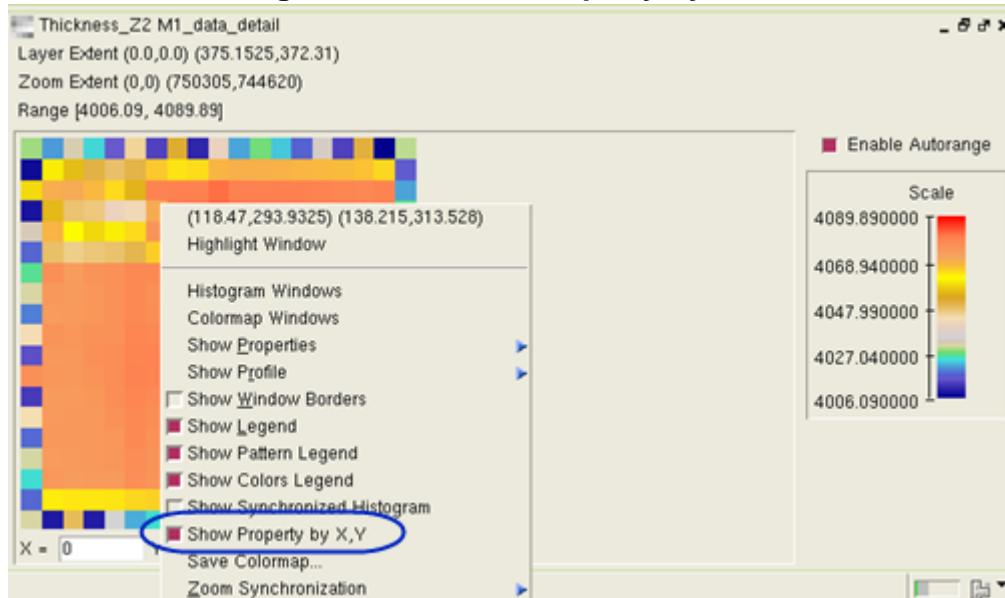
Prerequisites

- You have followed the steps described in “[CMP Analysis](#)” on page 15 to run Calibre CMPAnalyzer in the interactive GUI or batch mode of Calibre Interactive.
- You have opened Calibre RVE for DFM and set the results viewing and highlighting options as described in “[Reviewing Analysis Results](#)” on page 25.

Procedure

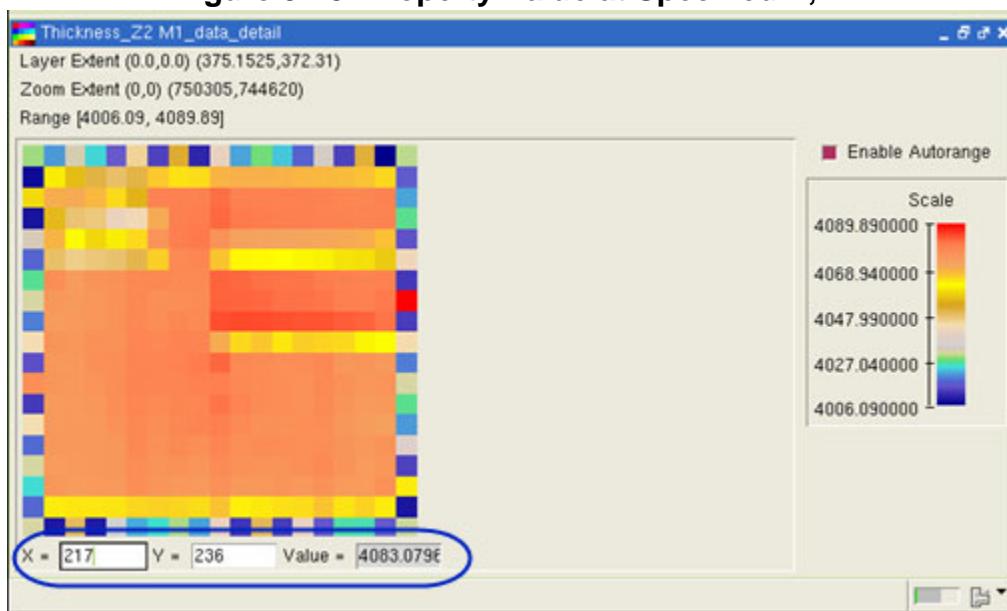
1. Review the results data displayed in the **CMP** tab.
2. Right-click in the M1 row and choose **Colormap > Thickness Z2**.
3. Right-click in the generated colormap and choose **Show Property by X, Y** in the popup menu.

Figure 3-27. Show Property by X, Y



4. Enter the x- and y-coordinates in the fields shown at the bottom of the colormap to display the DFM property value at that location. Coordinates are specified in microns as positive integers.

Figure 3-28. Property Value at Specified X, Y



Displaying MaterialNT(T) Properties

You can display non-trench and trench material properties (MaterialNT and MaterialT) from a DFM database for hotspot analysis.

Prerequisites

- Your process recipe file from Calibre CMP Model Builder has the initialize command syntax that enables MaterialNT and MaterialT properties to be saved to a DFM database. For example, the first initialize command statement in the recipe file contains the following syntax:

```
initialize material=nitride materialNT(T) dfmdb position=0A
```

- You have followed the steps described in “[CMP Analysis](#)” on page 15, and you have a DFM database containing the analysis results.
- Calibre DESIGNrev or another layout viewer that is supported by Calibre RVE is open with your layout loaded. See “[Communication with the Design Tool](#)” in the *Calibre Interactive User’s Manual* for information on communication between Calibre RVE and the layout viewer.
- Calibre RVE for DFM is running and the CMP analysis results are displayed in the **CMP** tab.

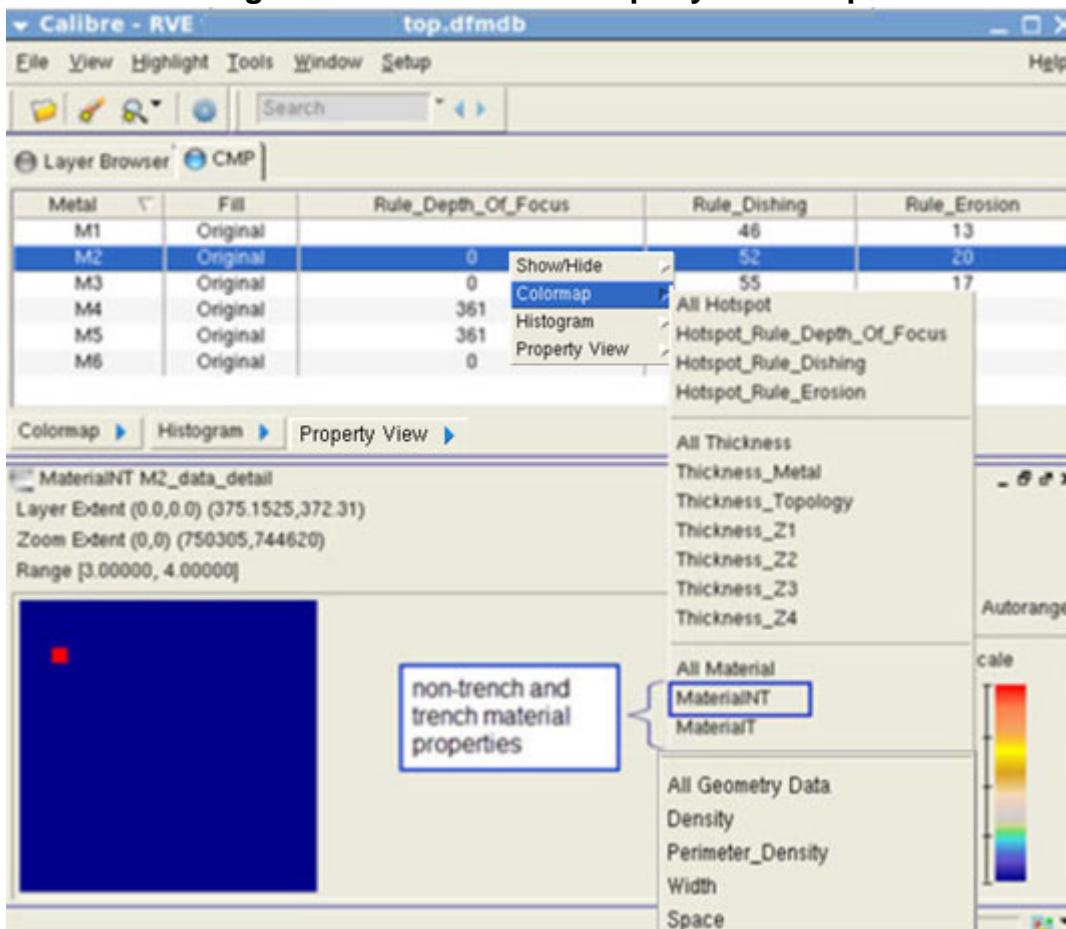
Procedure

1. Set your results viewing preferences and highlighting options in the Calibre RVE for DFM user interface as described in “[Reviewing Analysis Results](#)” on page 25.

2. Review the data displayed in the **CMP** tab.
3. Right-click on the **M2** row (or a metal layer of interest) and choose **Colormap > MaterialNT**.

This creates a colormap of the non-trench material with MaterialNT property.

Figure 3-29. MaterialNT Property Colormap



4. Right-click on the M2 row (or a metal layer of interest) and choose **Colormap > MaterialT**.

This creates a colormap of the trench material with MaterialT property.

5. Right-click on a tile of interest in the colormap and choose **Highlight Window** to display the location of the trench material with MaterialT property in the layout viewer.

Figure 3-30. MaterialT Property Location in Layout Viewer



6. From the menu bar, choose **Window > Close All** to close the colormap windows.
7. Right-click on the **M2** row (or a metal layer of interest) and choose **Colormap > All Material**.
8. This creates colormaps for both the non-trench material with MaterialNT property and the trench material with MaterialT property.

CMP Hotspots Clustering

You can use CMP hotspots clustering to generate data for hotspot analysis. The generated data can be saved to a DFM database for further analysis and applications.

Dynamic hotspot clustering combines single-pixel hotspots into larger clusters of rectangular shapes. The clusters can be passed to other tools for further analysis, redesign, and optimization. Reporting clusters of hotspots by level, instead of single-pixel hotspots, streamlines reporting by significantly reducing the number of reported hotspots.

You run CMP hotspots clustering from the Calibre CMPAnalyzer GUI in Calibre RVE for DFM or from a Calibre YieldServer command line.

CMP hotspots clustering performs the following actions:

- Starts by merging all single-pixel hotspots that share a common edge or vertex into rectangular clusters that cover the hotspots.
- Iteratively merges clusters and remaining single hotspots per the given distance parameters.
- Divides the clustering process into levels.
- Enables control of layer specifications, appropriate levels, and settings through user-selection.
- Automatically stops clustering when no more clusters can be merged for the given distance.
- Ends process by reporting the resulting cluster coordinates at each level of clustering.

Running CMP Hotspots Clustering From Calibre RVE	55
Running CMP Hotspots Clustering From Calibre YieldServer	60

Running CMP Hotspots Clustering From Calibre RVE

You can use Calibre RVE for DFM and the Calibre CMPAnalyzer GUI to set up and run CMP hotspots clustering and visualize the saved results.

Prerequisites

- You have performed a CMP analysis as described in “[CMP Analysis](#)” on page 15, and you have a DFM database containing the analysis results.

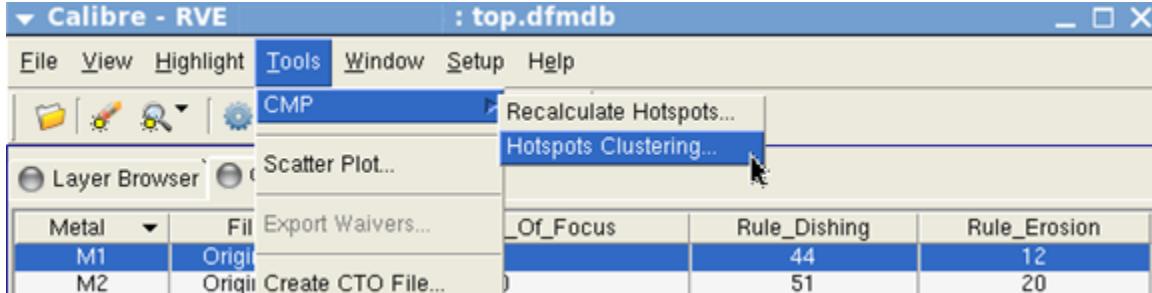
Procedure

1. Start Calibre RVE for DFM and specify your CMP analysis DFM database using one of two methods:
 - From Calibre DESIGNrev, choose **Verification > Start RVE** and perform the following actions in the dialog box:
 - i. Enter the full path and name of your DFM database.
 - ii. Choose Database Type **DFM**.
 - iii. Click **Open**.
 - From a Calibre command line, enter the following:

```
calibre -rve -dfm dfmdb/TOP_CHIP.dfmdb
```

2. Choose **Tools > CMP > Hotspots Clustering** in the **CMP** tab in the Calibre RVE for DFM GUI.

Figure 3-31. Tools — CMP Hotspots Clustering

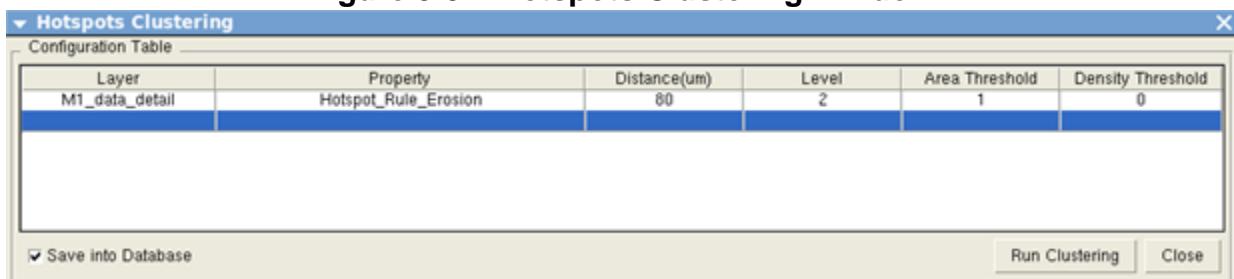


3. In the Hotspots Clustering window, define the Layer and Property fields. Set the clustering distance in microns and the number of cluster levels in the Distance and Level fields.

The clustering distance is the maximum distance between the hotspot clusters for merging to occur.

The number of cluster levels achieved during the run may be less than the number specified if no more clustering can be done within that distance. The default behavior is to build the maximum number of cluster levels and stop when no more clusters fit the specifications.

Figure 3-32. Hotspots Clustering Window

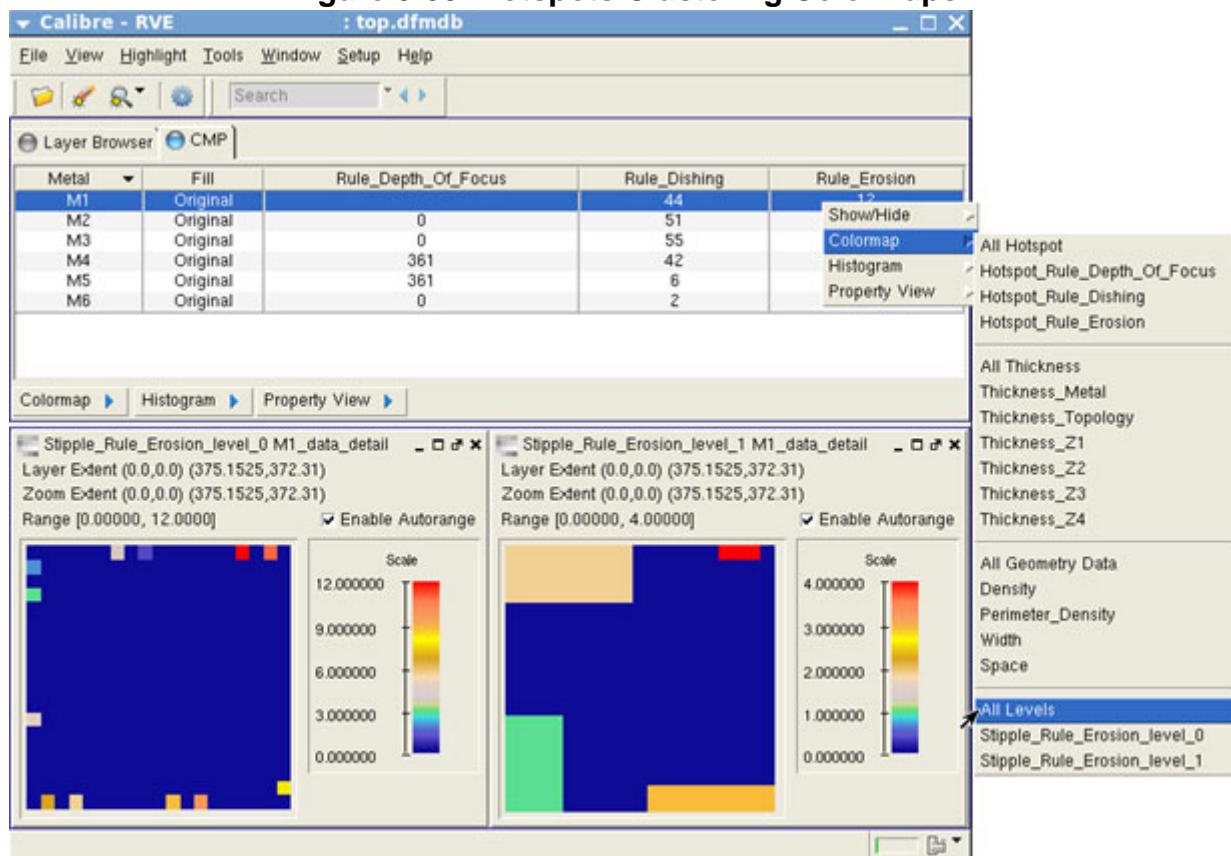


The Area Threshold and Density Threshold settings have default values of 1 and 0, respectively. These range settings can be used to control the coverage of the cluster area on the chip and the single-pixel density of the clusters. See `cmp::run_clusterization` in “[CMP YieldServer Command Reference](#)” on page 124.

The “Save into Database” setting is enabled by default. This saves the clustering data to the DFM database.

4. Click **Run Clustering** to start the run.
5. After the run finishes, access the colormap menu using either of these methods:
 - Click the **Colormap** button on the left side of the GUI.
 - Right-click the layer and select the **Colormap** menu item.
6. Choose “`Stipple_Rule_property_level_number`” to view the hotspot clustering by level or choose **All Levels**.

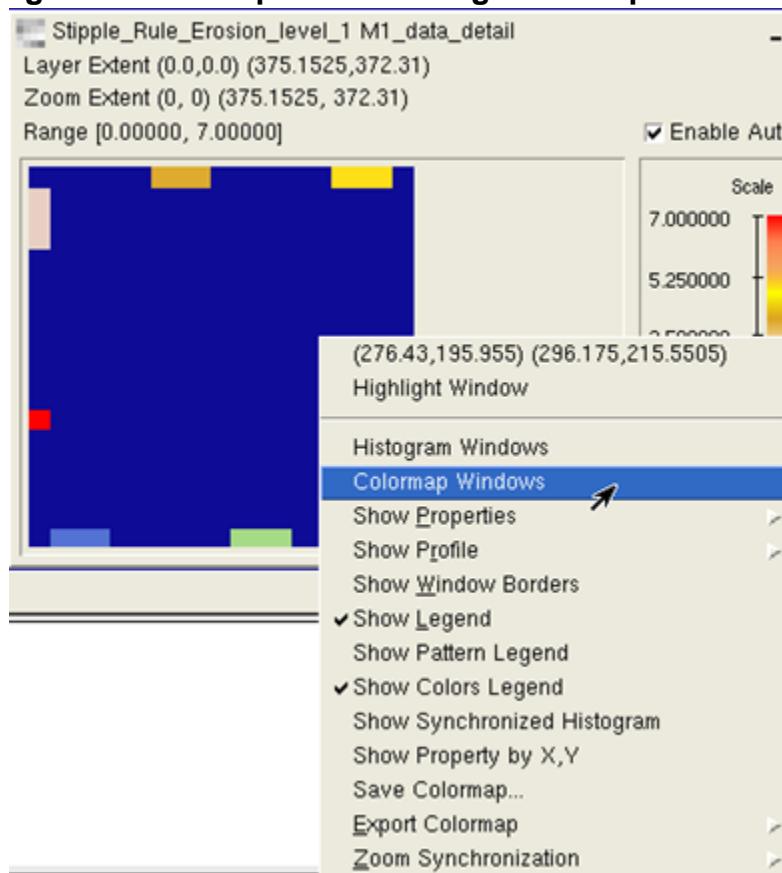
Figure 3-33. Hotspots Clustering Colormaps



Coloring is according to the density of the hotspots in the cluster. For example, the cluster with the largest number of single-pixel hotspot density is colored red, while the cluster with the smallest number of single-pixel hotspot density is colored blue.

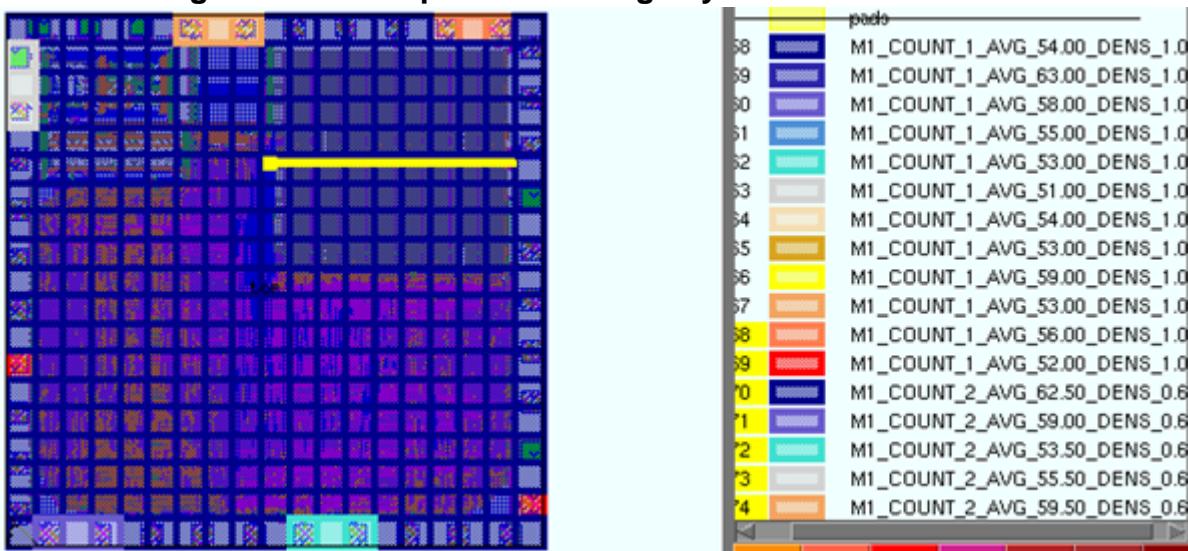
7. (Optional) If you are connected to Calibre RVE for DFM from Calibre DESIGNrev (or other layout viewer), right-click on a cluster colormap and choose **Colormap Windows** to highlight the cluster information in the layout viewer.

Figure 3-34. Hotspots Clustering Colormap Windows



The labels for the highlighted layers contain the layer name, hotspot count, average value, and density for each cluster. The hotspot density for the cluster is defined as the ratio of the number of single-pixel hotspots in the cluster and the number of pixels in the cluster.

Figure 3-35. Hotspots Clustering Layout View Information



For example:

```
M1_COUNT_6_AVG_36.33_DENS_1.00
```

This label represents a cluster that contains six hotspots, with an average value of 36.33, and a hotspot density of 1.00, which means that all pixels in the cluster are single-pixel hotspots.

Results

You have used Calibre RVE for DFM and the Calibre CMPAnalyzer GUI to generate, save, and view the CMP hotspots clustering data. Optionally, you have viewed hotspots clustering information in a connected layout viewer.

Running CMP Hotspots Clustering From Calibre YieldServer

You can use Calibre YieldServer to run CMP hotspots clustering commands to generate, save, and export cluster data for analysis.

Prerequisites

- You have performed a CMP analysis as described in “[CMP Analysis](#)” on page 15, and you have a DFM database containing the analysis results.
- You have access to Calibre YieldServer. Refer to “[CMP YieldServer Command Reference](#)” on page 124 for a list of commands specific to Calibre CMPAnalyzer.

Procedure

1. Open the DFM database from your CMP analysis with Calibre YieldServer at the command line.

```
calibre -ys -cmp -dfmdb dfmdb/TOP_CHIP.dfmdb
```

2. At the Calibre YieldServer command prompt (>), run the cmp::run_clusterization command with your parameters (distance in microns) to generate the hotspot clusters.

```
> set Erosion_clusters [cmp::run_clusterization  
-layer M1_data_detail -property Hotspot_Rule_Erosion -distance 80  
-level 10]
```

See cmp::run_clusterization in “[CMP YieldServer Command Reference](#)” on page 124.

Note

 The cmp::run_clusterization command removes previous clustering data in the current database revision to avoid conflicting data. If the current database revision is frozen, the command creates a new database revision. After the run completes, the command saves the current database revision.

The hotspot clustering data is generated for each level with the cluster coordinates (bottom left and top right coordinates). For example:

```
CLUSTER LEVEL IS 0
cluster 0 extern = (526, 599) (623, 605)
cluster 1 extern = (503, 598) (503, 600)
cluster 2 extern = (472, 598) (472, 600)
cluster 3 extern = (374, 598) (374, 600)
```

3. (Optional) Remove unneeded clustering information for a given layer.

```
> cmp::cleanup_clusters -layer layer_name
```

4. Save the hotspot clustering data to the DFM database.

```
> cmp::save_clusters $Erosion_clusters
```

5. (Optional) Use the cmp::export_clusters command to dump cluster data in .rdb format to a results database file (RDB file) for additional analysis, using one of two command flows in Calibre YieldServer.

- Dump the cluster data from the “ClusterManager” object (generated by the cmp::run_clusterization command). For example:

```
> set clusters [cmp::run_clusterization -layer M1_data_detail
    -property Hotspot_Rule_Erosion -level 2 -distance 80]
```

```
> cmp::export_clusters $clusters -level 1 -file ./clusters.rdb
```

- Dump the cluster data from the DFM database that contains the cluster data (saved by the cmp::save_clusters command). For example:

```
> cmp::export_clusters -layer M1_data_detail -level 1
    -properties Stipple_Rule_Erosion_level_1 -file ./clusters.rdb
```

6. Exit Calibre YieldServer.

```
> exit
```

7. Run Calibre RVE for DFM at the command line on your saved DFM database to plot colormaps of the hotspot clusters by level in the Calibre CMPAnalyzer GUI.

```
calibre -rve -dfm dfmdb/TOP_CHIP.dfmdb
```

Refer to Steps 5 and 7 of “Running CMP Hotspots Clustering From Calibre RVE” for information on plotting colormaps.

Results

You have used Calibre YieldServer with CMP commands to generate, save, and optionally, export CMP hotspots clustering data.

Exporting Hotspot Reports

After investigating the CMP analysis data, you can export the data as a hotspot report.

Note

 The following procedure describes how to save a hotspot report from Calibre RVE. You can also instruct Calibre Interactive to save a summary report with the total hotspot count by enabling the option **Write CMP summary report file** on the **CMP** tab of the Outputs pane in Calibre Interactive.

Prerequisites

- A DFM Database containing the CMP analysis results; see “[CMP Analysis](#)” on page 15.

Procedure

1. Open the DFM Database in Calibre RVE for DFM.
2. Choose **File > Save Hotspots Report** and specify the filename and file type (.txt or .rdb) for the report.

Tip

 You can also use the Calibre YieldServer command “`cmp::save_hotspots_report -file filename -format (“txt” or “rdb”)`” to save a hotspot report.

Results

You can view the hotspot report in a text editor. The report includes rule headings with a list of failing windows (and corresponding hotspot values) for each rule.

Related Topics

[CMP YieldServer Commands](#)

Exporting Colormap Data

You can save and export the colormap data as text files in different grid data formats.

Prerequisites

- You have followed the steps described in “[CMP Analysis](#)” on page 15, and you have a DFM database containing the analysis results.
- You have set your results viewing preferences and highlighting options in the Calibre RVE for DFM user interface as described in “[Reviewing Analysis Results](#)” on page 25.
- Calibre RVE for DFM is running and the CMP analysis results are displayed in the **CMP** tab.

- Optionally, you have access to Calibre WORKbench and Calibre CMP Model Builder. Refer to the [Calibre Administrator's Guide](#) for information on the required licensing.

Procedure

1. In the **CMP** tab of Calibre RVE for DFM, right-click on the M1 row and choose **Colormap > Density**.
2. Right-click on a tile in the colormap to display the menu items and choose **Export Colormap**.

This displays a submenu with the different data formats (**All**, **Grid**, **Coords**, **MatX**, **MatY**, and **MatZ**).

3. Select **All** to export the colormap data in all formats to text files in your current work directory.

This creates a set of files with layer M1 grid data, matrix grid data, and coordinate data (*M1_Density.grid*, *M1_Density.matX*, *M1_Density.matY*, *M1_Density.matZ*, and *M1_Density.crds*).

4. Optionally, you can display the exported colormap data in a Calibre WORKbench main window as a colormap, an XY plot in a graph window, or other plot types.
 - a. Start Calibre WORKbench from a shell command line.
`$MGC_HOME/bin/calibrewb`
 - b. Open the CMP Model Builder interface from the Calibre WORKbench main menubar.
Tools > CMP Model Builder.

- c. Select the **Grids** tab in the CMP Model Builder interface and click the **Open** button.
- d. Select the appropriate **File type** item (All Files (*) or Import matrix (*.matZ, *.matX, *.matY) in the Open Grids window.

Note

 Grid data exported from Calibre CMPAnalyzer RVE colormaps with the file type extension *.grid* have the same format as the CMP Model Builder *.txt* format used for saving geometry and simulation grids. Files with the *.grid* extension are displayed by default when files are opened from the CMP Model Builder **Grids** tab. The selection list of supported formats in the Open Grids window is shown: GP (*.bin, *.txt, *.rdb, *.sta, *.dump, *.grid). See “[Grid File Types](#)” in the *Calibre CMP Model Builder User's and Reference Manual* for more information.

Note

When you open matrix grid files, you must select the entire set of three matched files (`<filename>.matX`, `<filename>.matY`, and `<filename>.matZ`). The file extensions must match this format.

- e. Click the **Show** button to display the colormap in the Calibre WORKbench main window.
- f. See “[Grids Tab](#)” in the *Calibre CMP Model Builder User’s and Reference Manual* to explore other graph, plot options, and functionality.

Results

You have exported colormap data to text files in different grid data formats and optionally, used Calibre WORKbench with the CMP Model Builder interface to display the data. As another application for the matrix format data, you can create 3-D plots in MathWorks® MATLAB™ or other plotting software. See “[Creating a Surface Plot](#)” in the *Calibre CMP Model Builder User’s and Reference Manual*.

Exporting Calibre CMPAnalyzer Thickness Data to File for Extraction Tools

You can create input data for the Calibre parasitic extraction tools by exporting Calibre CMPAnalyzer thickness data to a file.

You can export the file from Calibre RVE for DFM or by using the Calibre YieldServer command `cmp::save_xrc_report`. The format of the output file(s) is described in the “Results” section.

Prerequisites

- You have a DFM database containing the CMP analysis results. See “[CMP Analysis](#)” on page 15.

Procedure

1. Do one of the following, depending on whether you want to do the export from Calibre RVE or with a YieldServer script:
 - From Calibre RVE for DFM
 - i. Choose **File > Save Extraction Data** to open a dialog box for the export.
 - ii. Specify the directory for the output data and click **OK**.
 - With a YieldServer script

- i. Create a YieldServer script with the cmp::save_xrc_report command similar to the following:

```
dfm::open_db dfmdb/top.dfmdb
cmp::save_xrc_report -dir export_dir
dfm::close_db
```

where the output files are written to the *export_dir* directory.

- ii. Execute the YieldServer script from the command line as follows:

```
calibre -ys -cmp -exec export_xrc.ys
```

2. Use the exported files with your extraction tool. See the following topics:

[“Varying Thickness with CMP Files”](#) in the *Calibre xRC User’s Manual*

[“Varying Thickness with CMP Files”](#) in the *Calibre xACT User’s Manual*

[“PEX CMP Mode”](#) in the *SVRF Manual*

Results

The saved extraction data from the CMP analysis is output to the *export_dir* directory.

Output Files

- The output consists of one file per metal layer. The file is named as follows:
 - <layer>.thk.txt — Metal thickness data (Z2-Z3)
 where <layer> is replaced with the layer name. For example, the file for the layer “metal1” is named *metal1.thk.txt*. See [“Thickness Parameters in the Calibre CMPAnalyzer Simulator Flow”](#) on page 104 for additional information on the thickness parameters.

Output File Format

- The output file(s) have one line of data for each window with the following format:

```
x1_ll y1_ll x1_ur y1_ur thickness_value1
x2_ll y2_ll x2_ur y2_ur thickness_value2
...
xN_ll yN_ll xN_ur yN_ur thickness_valueN
```

where the parameters are defined as follows:

- *xN_ll yN_ll* — x- and y-coordinates for the lower left corner of the Nth window.
- *xN_ur yN_ur* — x- and y-coordinates for the upper right corner of the Nth window.

- *thickness_valueN* — The thickness value for the Nth window, where the value is the metal thickness (Z2-Z3).

Related Topics

[CMP YieldServer Commands](#)

Exporting Analysis Results in Batch Mode

If desired, you can export tables, charts, colormaps and histograms that can be viewed in HTML format.

To do this, you must create a configuration file that follows the format described in “[DFM HTML Reporting](#)” in the *Calibre RVE User’s Manual*.

The “[CMP Batch Reporting](#)” on page 121 section contains example batch reporting configuration files that you can modify for use with your design. The following procedure assumes you are creating a new configuration file from scratch.

Prerequisites

- You have followed the steps described in “[CMP Analysis](#)” on page 15, and you have a DFM database containing the analysis results.

Procedure

1. Create a new file in your working directory named *rve.cfg*. This file will be used as the batch reporting configuration file.
2. In the file, create a Common section. This section defines default values for key names that are not specified in other sections. For example:

```
[Common]
GroupsToInclude = *
RulesToInclude = *
BinsToInclude = *
MetricsToInclude = *
Rows = *
Sort = Descending
GroupBy = Metrics
OutputFormat = html, csv, chart
```

3. Create a new section for each report that you want to output.

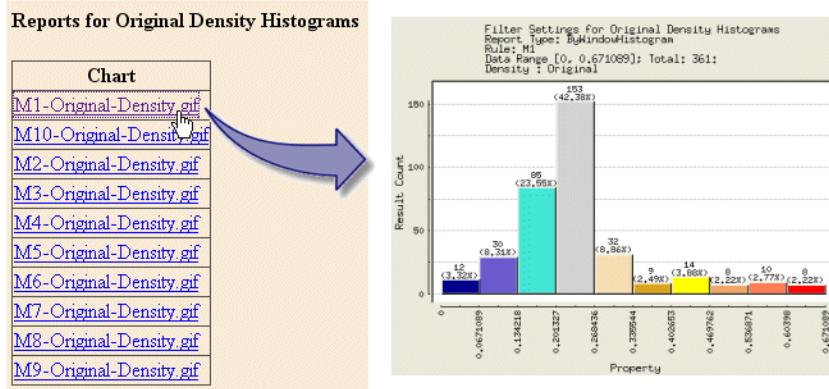
Section names are enclosed in brackets []. The ReportType key name specifies the type of report that is generated. Filtering can also be applied in a number of ways, including using the RulesToInclude, BinsToInclude, and MetricsToInclude key names.

For example, the following section:

```
; Create Density Histograms
[Original Density Histograms]
ReportType = ByWindowHistogram
BinsToInclude = "Original"
MetricsToInclude = "Density"
```

results in the report shown in [Figure 3-36](#), which displays the Density histograms in the Original bin for each metal layer.

Figure 3-36. Reports for Original Density Histograms



4. Create X and Y profile plots.

Creating X and Y profile plots in batch reporting is unique to the CMP analysis flow. You can specify the direction of the plot and location of the cut line, then generate the plot.

To do this, you can include the following in your batch configuration file:

```
//For X direction
[REPORT_NAME]
ReportType = ByWindowColorMap
XProfileInclude = <profile line coordinate>

//For Y direction
[REPORT_NAME]
ReportType = ByWindowColorMap
XProfileInclude = <profile line coordinate>
```

For more information, refer to “[DFM HTML Reporting](#)” in the *Calibre RVE User’s Manual*.

5. Run Calibre as follows:

```
calibre -rve -dfm dfmdb -report config_file \
-outputdir output_directory -cmp
```

where *dfmdb* is the name of the DFM database, *config_file* is the name of the batch reporting configuration file, and *output_directory* is the name of the directory that is created to contain the output files.

6. When the run is complete, open *index.htm* in the *./reports* directory. This file contains links to tables and images of charts, colormaps and histograms. These tables and images are the same ones that can be viewed using Calibre RVE for DFM.

Figure 3-37. Calibre CMPAnalyzer Batch Reporting Index Page

The screenshot displays a web-based interface for Calibre CMPAnalyzer. At the top, there is a terminal window showing the following text:

```
// Calibre v2010.3_21.14 (pre-production) Tue Aug 24 14:44:08 PDT 2010
//
// Copyright Mentor Graphics Corporation 1996-2010
// All Rights Reserved.
//
// THIS WORK CONTAINS TRADE SECRET AND PROPRIETARY INFORMATION
// WHICH IS THE PROPERTY OF MENTOR GRAPHICS CORPORATION
// OR ITS LICENSORS AND IS SUBJECT TO LICENSE TERMS.
//
// Mentor Graphics software executing under x86-64 Linux
//
// This software is in preproduction form and is considered to be
// beta code that is subject to the terms of section 3 of the
// End User License Agreement or your signed agreement with
// Mentor Graphics Corporation, whichever applies.
//
```

To the right of the terminal is the Calibre logo: "CALIBRE® VISUALIZATION". Below the terminal window, the main content area has a header "Calibre Dfm Reports". It includes a "Report Index" table with the following rows:

Report Index
CmpHotspotReport
Original Density Histograms
Original Density Colormaps
Original Depth Of Focus Histograms
Original Depth Of Focus Colormaps
Original Dishing Histograms
Original Dishing Colormaps
Original Erosion Histograms
Original Thickness Metal Histograms
Original Thickness Metal Colormaps
Original Thickness Z1-Z4 Histograms
Original Thickness Z1-Z4 Colormaps
XProfile Plot

Below the report index is a "Filter Settings for CmpHotspotReport" section, which is currently collapsed.

Calculating User-Defined Hotspots

You can calculate user-defined hotspots by running a Tcl script that calls the `cmp::register_hotspot_proc` and `cmp::register_hotspot_threshold_proc` commands. The results can be viewed in Calibre RVE for DFM.

Tcl Script for User-Defined Hotspots.....	69
Running Calibre CMPAnalyzer with User-Defined Hotspots.....	74
Analyzing Calibre CMPAnalyzer User-Defined Hotspot Results.....	75
Mapping Properties to Hotspot Rule Names.....	78

Tcl Script for User-Defined Hotspots

You can calculate user-defined hotspots by running a Tcl script that calls the `cmp::register_hotspot_proc` and `cmp::register_hotspot_threshold_proc` commands. The Tcl script calculates the hotspot metric and compares the value to a threshold.

Tcl scripts for user-defined hotspots are explained in the following sections with a description of the basic contents, a summary of the example code, and two example scripts.

Both of the following example scripts calculate a hotspot based on a perimeter metric using basic contents and Tcl procedures. Additionally, the second example script incorporates CMP grid operations to perform the hotspot calculation.

Note

-  The vector property format is not compatible with iterator-based Tcl scripts (Example 1).
Tcl scripts implemented with grid operations (Example 2) are compatible with the vector property format. See “[Simulator Options Tab on the Inputs Pane](#)” on page 113 for information on vector properties.
-

See also “[Example Script to Calculate User-Defined Hotspots](#)” on page 142 to perform a thickness topology hotspot analysis in Calibre CMPAnalyzer.

Basic Contents

In general, a Tcl script for user-defined hotspots has the following elements:

- Procedure to calculate the hotspot metric for a layer and add the property to the layer. This procedure is called with a detail layer as an argument.
- Procedure to calculate the hotspot threshold property. Typically, the hotspot threshold property indicates the pass/fail status. This procedure is called with the detail layer and a value used in computing the hotspot threshold property. By default, Calibre RVE expects the hotspot threshold property to have the name “Hotspot_<metric>”, where <metric> is the property name of the hotspot metric.

The return value of this procedure should be the number of failing tiles. The return value is displayed in the column for the hotspot rule in the **CMP** tab of Calibre RVE.

- Call to `cmp::register_hotspot_proc` to register the procedure that calculates the hotspot metric. The argument to the `-rule_name` option should be the property name for the hotspot metric. The `-output_properties` argument gives the name of the hotspot threshold property, typically “Hotspot_<metric>”.
- Call to `cmp::register_hotspot_threshold_proc` to register the procedure that calculates the hotspot threshold property. The argument to the `-rule_name` option should be the property name for the hotspot metric. The `-output_properties` argument gives the name of the hotspot threshold property, typically “Hotspot_<metric>”.

Example Code Summary

The example code calculates a user-defined hotspot based on the perimeter metric. The code has this basic outline:

- `proc calc_perimeter(geom_layer)`
Calculates a perimeter value from an input geometry layer. A temporary layer with the property is created using DFM Analyze, then the property “Perim_Density” is added to the `_data_detail` layer for the input geometry layer.
- `proc perimeter_metric(detail_layer)`
Gets the geometry layer from the input detail layer and calls the `proc calc_perimeter()` to calculate the hotspot metric.
- `proc perimeter_fails(detail_layer hs_value)`
Gets the hotspot metric for the layer and compares it to the threshold value. It creates the “Hotspot_Perim_Density” property, which is 0 or 1 to indicate pass/fail. The number of failing windows is totaled and returned; this return value is displayed in Calibre RVE for the hotspot rule.
- `cmp::register_hotspot_proc -rule_name Perim_Density \`
`-proc perimeter_metric`
`cmp::register_hotspot_threshold_proc -rule_name Perim_Density \`
`-proc perimeter_fails values 5 -output_properties "Hotspot_Perim_Density"`

Register the hotspot metric procedure and the hotspot threshold procedure so that they are called during hotspot calculation. The rule name is `Perim_Density`, the same as the property name for the metric. The `-output_properties` value is the hotspot threshold property name, “`Hotspot_Perim_Density`.”

Example 1: Tcl Script for User-Defined Hotspot Calculation

```

# TCL script to generate perimeter property as a "hotspot"

# First use DFM ANALYZE to calculate the perimeter density
proc calc_perimeter { geom_layer X Y } {
    # Calculate perimeter metric
    # Use DFM ANALYZE to create the perimeter density on a temporary layer
    # Note, window size needs to be a variable passed to this function as
    # 20 is only the default
    set svrf "${geom_layer}_PD = DFM ANALYZE $geom_layer \
        \[PERIMETER($geom_layer)/AREA()\] >=0 WINDOW $X $Y \
        INSIDE OF LAYER DB_EXTENT"
    dfm::new_layer -svrf $svrf
    # This generates a _PD layer and a _PD_detail layer

    # Copy the DV property from _PD_detail layer to data_detail layer
    catch {dfm::delete_property ${geom_layer}_data_detail \
        "Perim_Density"} msg
    dfm::add_property ${geom_layer}_data_detail \
        "Perim_Density" 0 -double -default
    set geos_source [dfm::get_geometries ${geom_layer}_PD_detail]
    set geos_destination [dfm::get_geometries ${geom_layer}_data_detail]

    # This works because the layers have the same window geometries and \
    # Calibre sorts them in the same default order
    while { $geos_source ne "" } {
        set s_data [dfm::get_data $geos_source -geometry_property "DV"]
        dfm::add_property $geos_destination "Perim_Density" $s_data -double
        dfm::inc geos_source
        dfm::inc geos_destination
    }
    # delete the temporary layers
    dfm::delete_layer ${geom_layer}_PD
    dfm::delete_layer ${geom_layer}_PD_detail
}

proc perimeter_metric { detail_layer } {
    # Need to get geom data from detail layer name
    # set tokens [split $detail_layer "_data_detail" ]
    # set geom_layer [lindex $tokens 0]
    set nameIdx [string first "_data_detail" $detail_layer]
    set geom_layer [string range $detail_layer 0 [expr $nameIdx - 1]]

    set db_precision [dfm::get_db_precision]
    set geos [dfm::get_geometries $detail_layer ]
    set vertices [dfm::get_data $geos -vertices];
    set tileX [expr ([lindex $vertices 2] - [lindex $vertices 0]) / \
        $db_precision];
    set tileY [expr ([lindex $vertices 3] - [lindex $vertices 1]) / \
        $db_precision];

    # Calculate the perimeter property
    calc_perimeter $geom_layer $tileX $tileY
    # Now have the Perim_Density property on the _data_detail layers
}

proc perimeter_fails { detail_layer hs_value } {

```

```
# Compares the Perim_Density property to a hotspot value and counts
# tiles which fail
# By default, RVE expects a hotspot threshold property name of
# Hotspot_<property>
# Note, property added is "0" if pass and "1" if fail

# Try to delete this property if it already exists might do this in the
# above procs as well
catch {dfm::delete_property $detail_layer "Hotspot_Perim_Density"} msg
dfm::add_property $detail_layer \
    "Hotspot_Perim_Density" 0 -double -default
set geos [dfm::get_geometries $detail_layer ]

# Compare perimeter property to threshold and increment count if fails
set fails 0
while { $geos ne "" } {
    set metric [dfm::get_data $geos -geometry_property \
        "Perim_Density" ]
    if { $metric > $hs_value } {
        dfm::add_property $geos "Hotspot_Perim_Density" 1 -double
        incr fails
    }
    dfm::inc geos
}
puts "Number of $detail_layer Hotspot_Perim_Density $fails"
return $fails
}

# register the calculation and hotspot comparison procs so they get
# calculated with the normal hotspots
cmp::register_hotspot_proc -rule_name Perim_Density \
    -proc perimeter_metric \
    -output_properties "Perim_Density"
cmp::register_hotspot_threshold_proc -rule_name Perim_Density \
    -proc perimeter_fails -values 5 \
    -output_properties "Hotspot_Perim_Density"
```

Example 2: Tcl Script for User-Defined Hotspot Calculation (Grid-Based)

```

# TCL script to generate a perimeter property as a hotspot

# First use DFM ANALYZE to calculate the perimeter density
proc calc_perimeter { geom_layer Xsize Ysize } {
    # Calculate perimeter metric Perim_Density
    # Use DFM ANALYZE to create the perimeter density on a temporary layer
    set svrf "${geom_layer}_PD = DFM ANALYZE $geom_layer \
        \[PERIMETER($geom_layer)/AREA()\] >=0 WINDOW $Xsize $Ysize \
        INSIDE OF LAYER DB_EXTENT"
    dfm::new_layer -svrf $svrf
    # This generates a _PD layer and a _PD_detail layer

    # Always delete a property before adding it back with a default value
    catch {dfm::delete_property ${geom_layer}_data_detail \
        "Perim_Density"} msg
    dfm::add_property ${geom_layer}_data_detail \
        "Perim_Density" 0 -double -default
    # Save the current rev to avoid issues with temp layers
    dfm::save_rev
    # Create a grid from the PD layer, save it to the detail layer
    set geos_source \
        [cmp::create_grid -layer ${geom_layer}_PD_detail -property "DV"]
    cmp::save_grid $geos_source -layer ${geom_layer}_data_detail \
        -property "Perim_Density"
    # Delete the temporary layers
    dfm::delete_layer ${geom_layer}_PD
    dfm::delete_layer ${geom_layer}_PD_detail
    dfm::save_rev
}

proc perimeter_metric { detail_layer } {
    # Need to get geom data from detail layer name
    # Set tokens [split $detail_layer "_data_detail" ]
    # Set geom_layer [lindex $tokens 0]
    set nameIdx [string first "_data_detail" $detail_layer]
    set geom_layer [string range $detail_layer 0 [expr $nameIdx - 1]]

    # Note, X, Y, window sizes are variables extracted from the dfmdb
    set db_precision [dfm::get_db_precision]
    set geos [dfm::get_geometries $detail_layer ]
    set vertices [dfm::get_data $geos -vertices];
    set Xsize [expr ([lindex $vertices 2] - [lindex $vertices 0]) / \
        $db_precision];
    set Ysize [expr ([lindex $vertices 3] - [lindex $vertices 1]) / \
        $db_precision];

    # Calculate the perimeter property
    calc_perimeter $geom_layer $Xsize $Ysize
    # Now have the Perim_Density property on the _data_detail layers
}

proc perimeterfails { detail_layer hs_value } {
    # Compares the Perim_Density property to a hotspot value and counts
    # tiles which fail
    # By default, RVE expects a hotspot threshold property name of
    #Hotspot_<property>
}

```

```
# Note, property added is "0" if pass and "1" if fail

# Always delete a property before adding it back with a default value
catch {dfm::delete_property $detail_layer "Hotspot_Perim_Density"} msg
dfm::add_property $detail_layer \
    "Hotspot_Perim_Density" 0 -double -default

# Compare perimeter property to threshold
set PD_grid [cmp::create_grid -layer $detail_layer \
    -property "Perim_Density"]
set result_grid [cmp::greater_grid $PD_grid -element \
    -limit $hs_value -result_value {1 0}]
cmp::save_grid $result_grid -layer $detail_layer \
    -property "Hotspot_Perim_Density"
# Add up the fails
set fails \
    [expr int([cmp::get_grid_data $result_grid -sum_elements])]

puts "Number of $detail_layer Hotspot_Perim_Density $fails"
# A hotspot threshold proc must always return the number of fails
return $fails
}

# register the calculation and hotspot comparison procs so they get
#calculated with the normal hotspots
cmp::register_hotspot_proc -rule_name Perim_Density \
    -proc perimeter_metric \
    -output_properties "Perim_Density"
cmp::register_hotspot_threshold_proc -rule_name Perim_Density \
    -proc perimeter_fails -values 5 \
    -output_properties "Hotspot_Perim_Density"
```

Related Topics

[CMP YieldServer Commands](#)

Running Calibre CMPAnalyzer with User-Defined Hotspots

You can run Calibre CMPAnalyzer with custom hotspot calculations.

Prerequisites

- You have a runset for CMP analysis. See “[CMP Analysis with the Calibre Interactive GUI](#)” on page 15 for the steps to create a runset.

Note

 If you are using user-defined (custom) hotspots, and you do not want the tool to calculate the default hotspots (Dishing, Erosion, Depth of Focus), make sure your runset does not specify thresholds for those rules on the **CMP** tab of the Inputs pane in Calibre Interactive.

- You have a Tcl script to calculate the user-defined hotspot properties (UDHS file). See “[Tcl Script for User-Defined Hotspots](#)” on page 69.

Procedure

1. You can specify the UDHS file for running Calibre CMPAnalyzer with user-defined hotspots using one of two methods:
 - Set the MGC_CMPA_UD_HOTSPOT_RULES environment variable to the UDHS file path relative to your working directory. For example:

```
setenv MGC_CMPA_UD_HOTSPOT_RULES ./custom_hotspots.tcl
```

- Specify the UDHS file path and filename in the **Simulator Options** tab of the Calibre Interactive DFM GUI (Step 4).

2. Launch Calibre DESIGNrev and load your layout:

```
calibredrv layout
```

3. In Calibre DESIGNrev, choose **Verification > Run DFM**.

This invokes Calibre Interactive in DFM mode. Specify the path to your Calibre CMPAnalyzer runset in the **Load Runset File** dialog box.

4. Click the **Simulator Options** tab on the Inputs pane of the Calibre Interactive GUI and specify an absolute or relative path (./) indicating the filename of your UDHS file in the UDHS File field.

Note

 This step is not needed if you have already set the path and filename of the UDHS file using the environment variable (Step 1, a.).

Specifying the UDHS file in the Calibre Interactive DFM GUI overrides the UDHS file environment variable setting.

5. Click **Run CMP**.
6. Click the **Start RVE** button if Calibre RVE does not start automatically after the run is finished.
7. See “[Analyzing Calibre CMPAnalyzer User-Defined Hotspot Results](#)” on page 75 for instructions on analyzing the results.

Analyzing Calibre CMPAnalyzer User-Defined Hotspot Results

You can view the results of user-defined hotspot rules in Calibre RVE for DFM.

Prerequisites

- Results from a Calibre CMPAnalyzer run with user-defined hotspots. See “[Running Calibre CMPAnalyzer with User-Defined Hotspots](#)” on page 74.
- The hotspot threshold property is named according to the standard practice or specified with the -output_properties argument to `cmp::register_hotspot_threshold_proc` in the Tcl script for user-defined hotspots. The standard practice is to name the hotspot threshold property “Hotspot_<metric>”, where <metric> is the property name of the hotspot metric. See “[Tcl Script for User-Defined Hotspots](#)” on page 69.

If the above statement is not true, see “[Mapping Properties to Hotspot Rule Names](#)” on page 78.

Procedure

1. Open the DFM Database in Calibre RVE for DFM. Do one of the following:
 - From Calibre Interactive, click **Start RVE**.
 - From a layout viewer, choose **Calibre > Start RVE** or **Verification > Start RVE**, depending on your viewer. Specify the DFM Database and click OK.
2. Select the **CMP** tab.

If the rule name for the user-defined hotspot does not appear as a column in the **CMP** tab, right-click in the data table, select **Show/Hide**, and enable the custom metric.

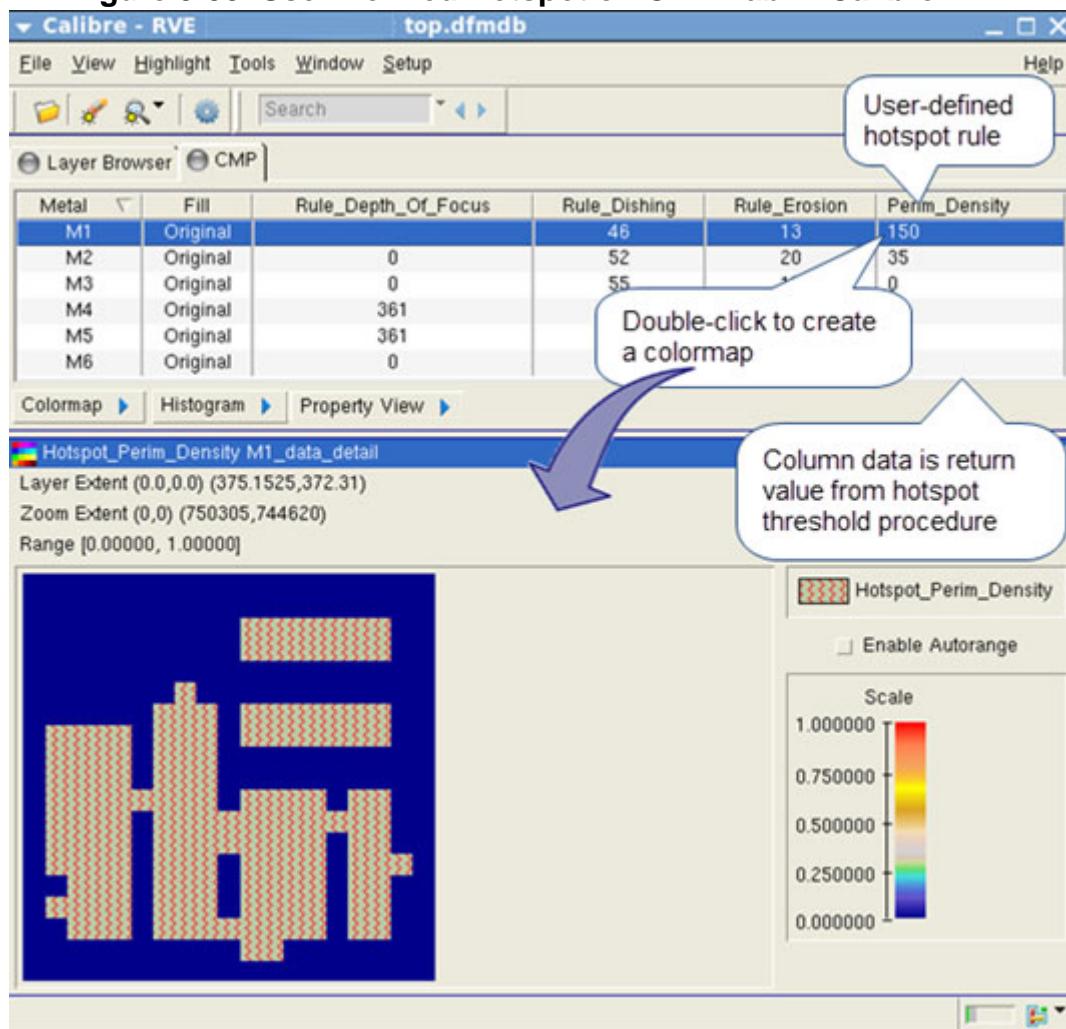
Note

 The **Show/Hide** menu item is not available if no hotspot column data exists.

3. To create a colormap select the desired metal layer on the **CMP** tab and double-click in the column for hotspot rule.

The following figure shows an image of the **CMP** tab in Calibre RVE where the user-defined hotspot metric is “Perim_Density.” See “[Tcl Script for User-Defined Hotspots](#)” on page 69 for example code. The colormap is shown in the lower window.

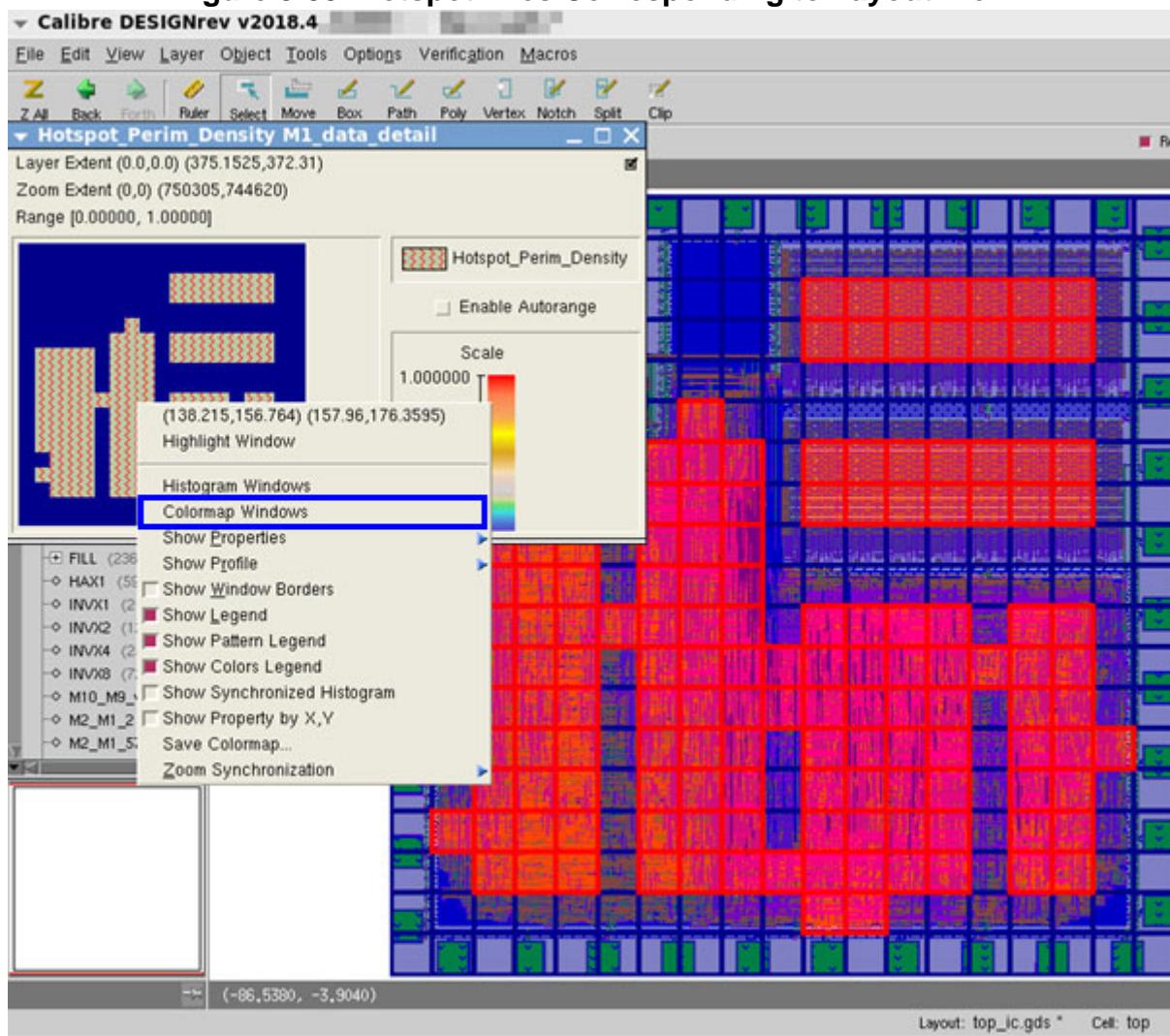
Figure 3-38. User-Defined Hotspot on CMP Tab in Calibre RVE



Results

Right-click in the colormap and select **Colormap Windows** to create a colormap in the layout viewer. This maps the tiles shown in the colormap to corresponding tiles in the layout viewer. [Figure 3-39](#) shows an example of this mapping with the layout shown in Calibre DESIGNrev.

Figure 3-39. Hotspot Tiles Corresponding to Layout View



Mapping Properties to Hotspot Rule Names

You can use Calibre RVE for DFM to map a property name to a user-defined hotspot rule. This is useful if the hotspot threshold property is not named according to the standard practice or is not specified with the `-output_properties` argument to `cmp::register_hotspot_threshold_proc` in your Tcl script for user-defined hotspots.

The standard practice is to name the hotspot threshold property “Hotspot_<metric>”, where <metric> is the property name of the hotspot metric. See “[Tcl Script for User-Defined Hotspots](#)” on page 69.

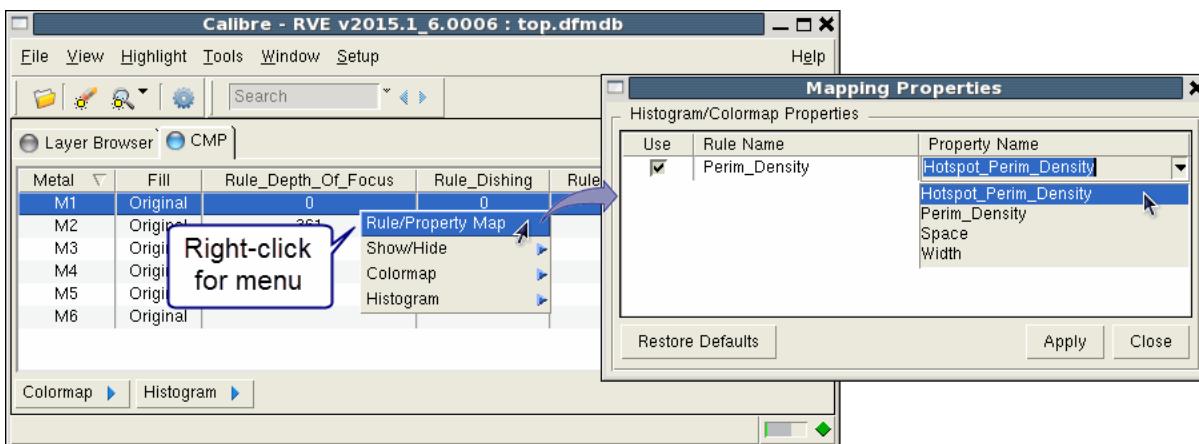
Prerequisites

- The Tcl script for user-defined hotspots does *not* use the `-output_properties` argument to `cmp::register_hotspot_threshold_proc`.

- Results from a Calibre CMPAnalyzer run with user-defined hotspots. See “[Running Calibre CMPAnalyzer with User-Defined Hotspots](#)” on page 74.

Procedure

1. Set the environment variable ALLOW_RULE_MAPPING to 1.
2. Open the DFM Database in Calibre RVE for DFM. Do one of the following:
 - From Calibre Interactive, click **Start RVE**.
 - From a layout viewer, choose **Calibre > Start RVE** or **Verification > Start RVE**, depending on your viewer. Specify the DFM Database and click OK.
3. Select the **CMP** tab.
4. Right-click in the table on the **CMP** tab and choose **Rule/Property Map** to open the Mapping Properties dialog box.
5. Click in the Property Name entry for the rule you want to map and select the correct property name in the dropdown list.



6. Click **Apply** and **Close**.

Results

When you double-click in the hotspot rule column on the **CMP** tab, a colormap is created for the rule using the correct property.

Related Topics

- [Analyzing Calibre CMPAnalyzer User-Defined Hotspot Results](#)
- [Tcl Script for User-Defined Hotspots](#)

Hot Spot Analysis Batch Procedure

You can perform hotspot analysis in batch mode.

Procedure

1. Write a Tcl script that defines a custom hotspot metric. This file must call the `cmp::register_hotspot_proc` and `cmp::register_hotspot_threshold_proc` commands. Refer to “[Tcl Script for User-Defined Hotspots](#)” on page 69 for an example script. Place this file in your working directory.
2. Set the `MGC_CMPA_UD_HOTSPOT_RULES` environment variable to the path (relative to your working directory) to the Tcl script you created in the previous step. For example:

```
setenv MGC_CMPA_UD_HOTSPOT_RULES ./custom_hotspots.tcl
```

3. Run CMPAnalyzer in batch mode:

```
calibre -gui -dfm -runset runset -batch | tee calibre.log
```

4. Launch Calibre DESIGNrev and Calibre RVE for DFM:

```
calibredrv layout -rve -dfm dfmdb
```

5. See “[Analyzing Calibre CMPAnalyzer User-Defined Hotspot Results](#)” on page 75 for instructions on viewing the results in Calibre RVE.

Note that batch reporting does not support custom hotspot metrics.

Related Topics

[CMP Batch Reporting](#)

Chapter 4

CMP Bucketing Flow

The CMP bucketing flow is a pseudo feature-scale simulation option that provides extra information about shapes with different widths for each tile, that is each window, or frame. Based on the simulation results, different thicknesses will be reported for each bucket.

You use Calibre Interactive for DFM to enable the bucketing, and view the results of the bucketing in Calibre RVE for DFM and a layout viewer such as Calibre DESIGNrev or Calibre WORKbench. Note that simulation with bucketing may involve longer run times due to the extra calculations for buckets, especially for large designs.

You must enable bucketing simulation by setting the **Width for Bucketing Range** parameter in Calibre Interactive for DFM using following method:

Running the CMP Bucketing Flow in Calibre Interactive for DFM 81

Running the CMP Bucketing Flow in Calibre Interactive for DFM

You use Calibre Interactive for DFM to enable and run the bucketing simulation. You can view and analyze results using Calibre RVE for DFM and Calibre DESIGNrev or Calibre WORKbench.

Prerequisites

- You have a valid recipe file. Refer to the [Calibre CMP Model Builder User's and Reference Manual](#) for details on creating a recipe file.
- You have set CALIBRE_HOME or MGC_HOME to the path to the Calibre software tree. Refer to the [Calibre Administrator's Guide](#) for information on setting this variable.

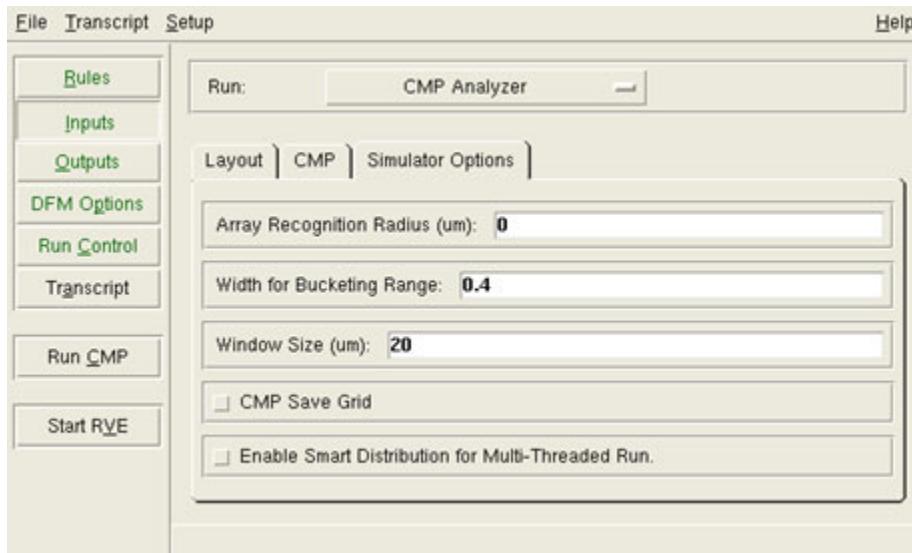
Procedure

1. In a shell, invoke Calibre Interactive for DFM using the following command:

```
calibre -gui -dfm
```

2. Set the simulation design and other settings. Refer to the procedure in “[CMP Analysis with the Calibre Interactive GUI](#)” on page 15.

- Click **Inputs** and then click the **Simulator Options** tab.



By default, Calibre CMPAnalyzer uses a bucketing mechanism that defines the required number of buckets automatically. You enable the bucketing flow by entering a 0 to 1 positive value in the **Width for Bucketing Range** field.

- Width for Bucketing Range** — Specifies a value from 0 to 1 inclusive that is the percentage (limit) used to calculate bucket widths for each tile. Use the following guidance to determine this value:

Width for Bucketing Range	Bucketing Behavior
0	Maximum possible number of buckets. Shapes with different widths are placed in different buckets for each tile. Consequently, the tool generates the maximum possible number of buckets.
small (near 0)	Large number of buckets.
large (near 1)	Small number of buckets.
1	One bucket per tile. All shapes are placed in a single bucket for each tile. This is equivalent to a simple run without bucketing.

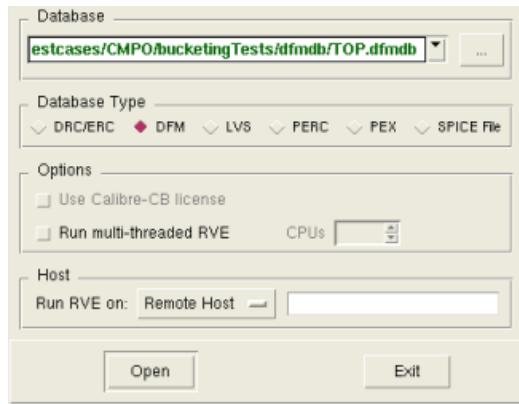
See “[Simulator Options Tab on the Inputs Pane](#)” on page 113 for information about the other options on this tab.

- Clicking **Run CMP** to perform the simulation.

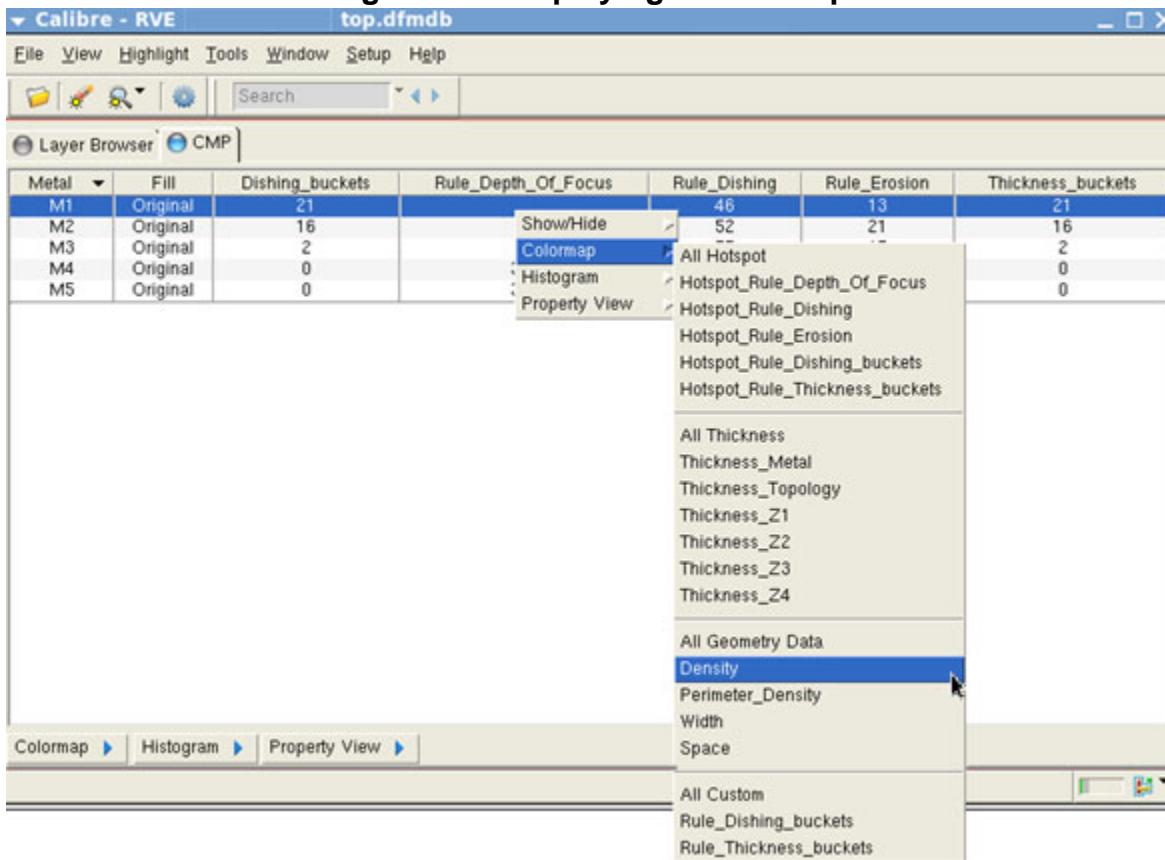
5. When the simulation is completed, invoke Calibre DESIGNrev or Calibre WORKbench with your layout. For example:

```
calibredrv my_design
```

6. From Calibre DESIGNrev or Calibre WORKbench, choose **Verification > Start RVE**.

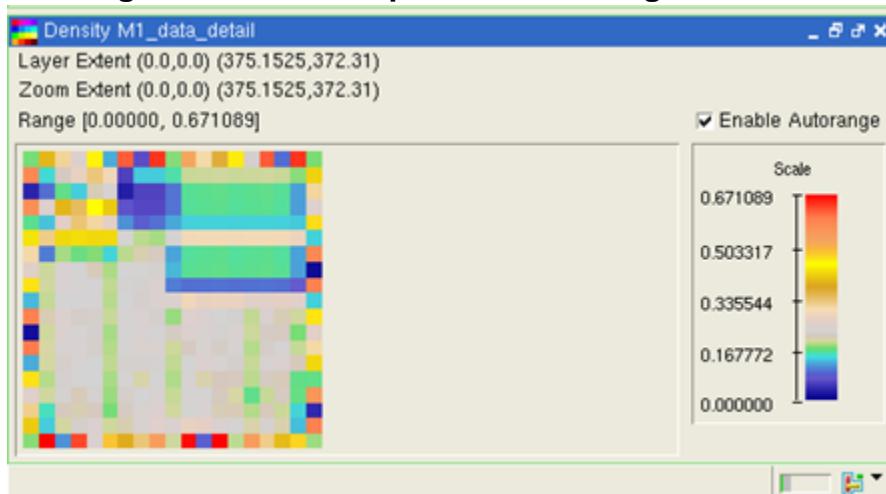


7. In the dialog box, choose the DFM database and click **Open**.
8. Right-click on the M1 row, choose **Colormap**, and choose a property to colormap.

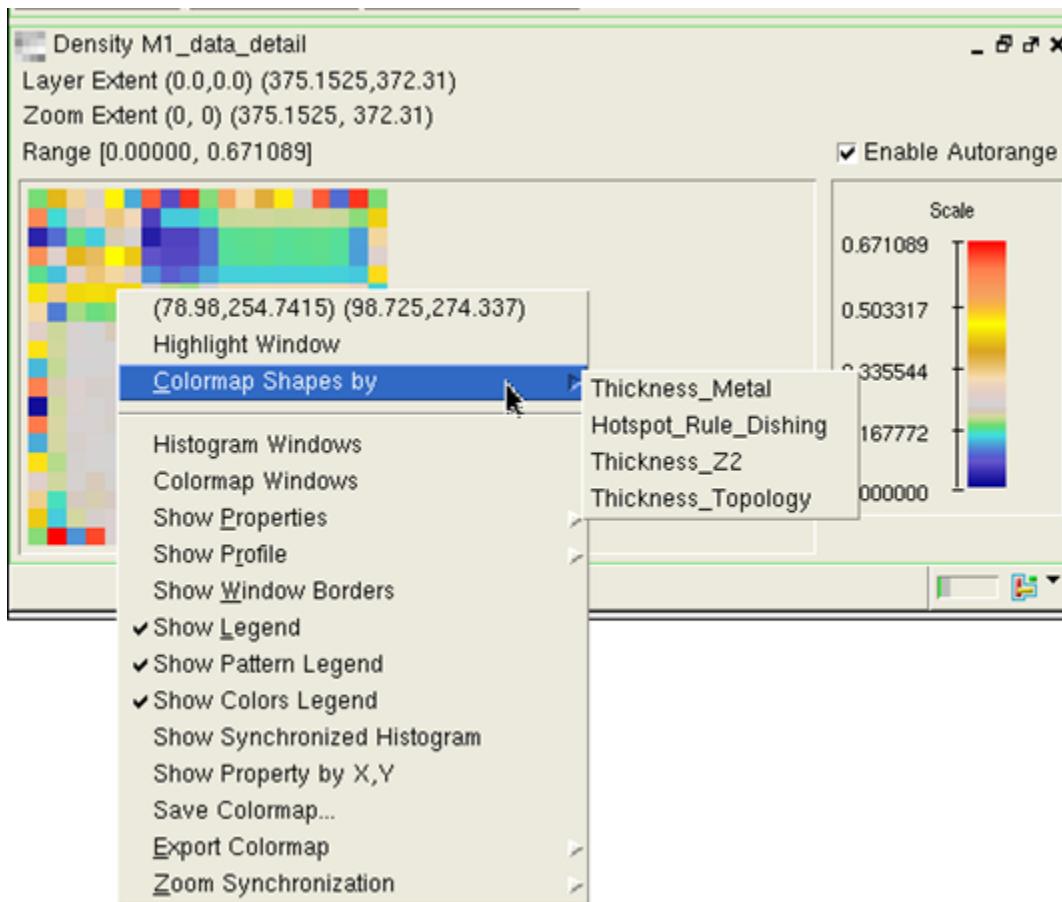
Figure 4-1. Displaying a Colormap

To zoom in on the colormap, click and hold the right mouse button, drag the mouse down and to the right, and release the right mouse button. To zoom out, perform a similar procedure except drag the mouse up and to the left.

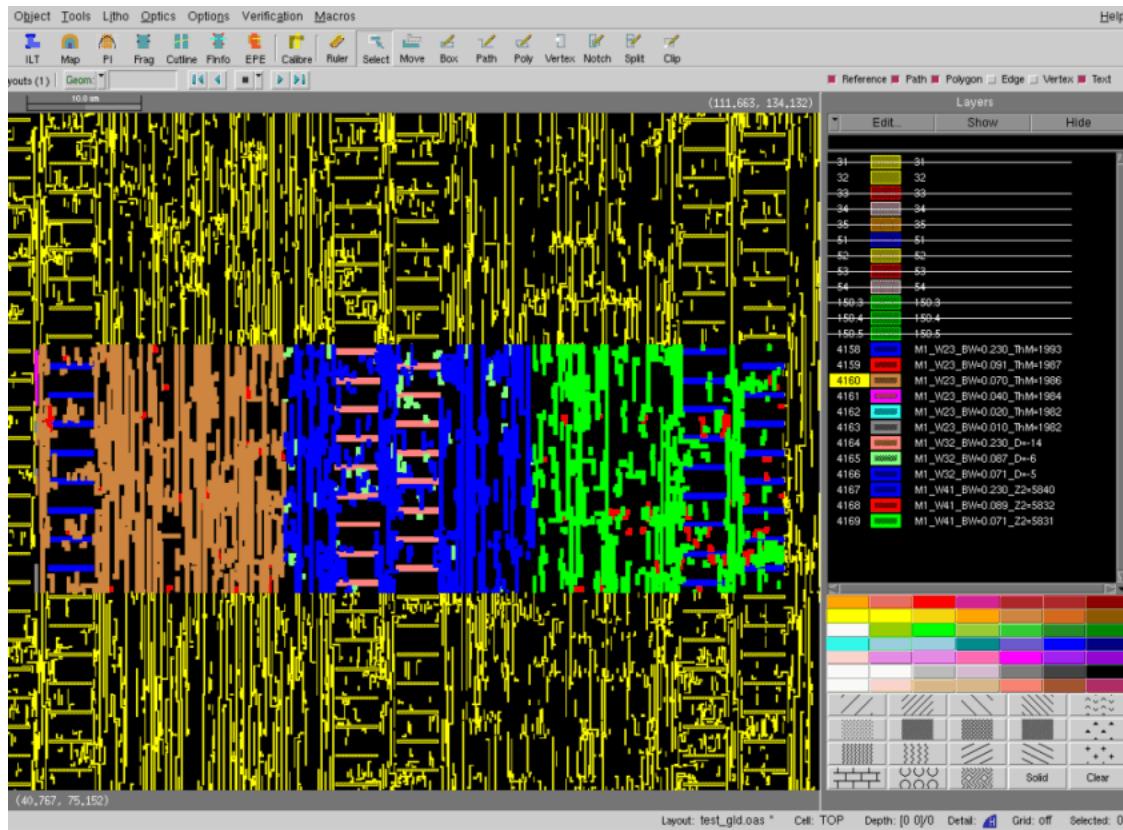
If enabled through the **Setup > Options** menu, you can check the **Enable Autorange** checkbox on the fast colormap to fit colors to the current visible range of values.

Figure 4-2. Colormap with Color Range Enabled

9. Right-click on a tile in the colormap, select **Colormap Shapes by** and choose one of the following:**Thickness_Metal**, **Hotspot_Rule_Dishing**, **Thickness_Z2**, or **Thickness_Topo**logy, as shown in the following figure.



The corresponding buckets are displayed in the layout viewer as layers with different colors.



Results

In a Calibre DESIGNrev or Calibre WORKbench layout viewer, the bucketing information is displayed as layer names using the following format:

{metal layer}_W{window index}_BW={number}_{Property}={number}

where Property abbreviations are as follows:

- **ThM** — Thickness_Metal
- **D** — Hotspot_Rule_Dishing
- **Z2** — Thickness_Z2
- **ThT** — Thickness_Topology

BW stands for bucket width.

The values of bucket properties are given in Angstroms.

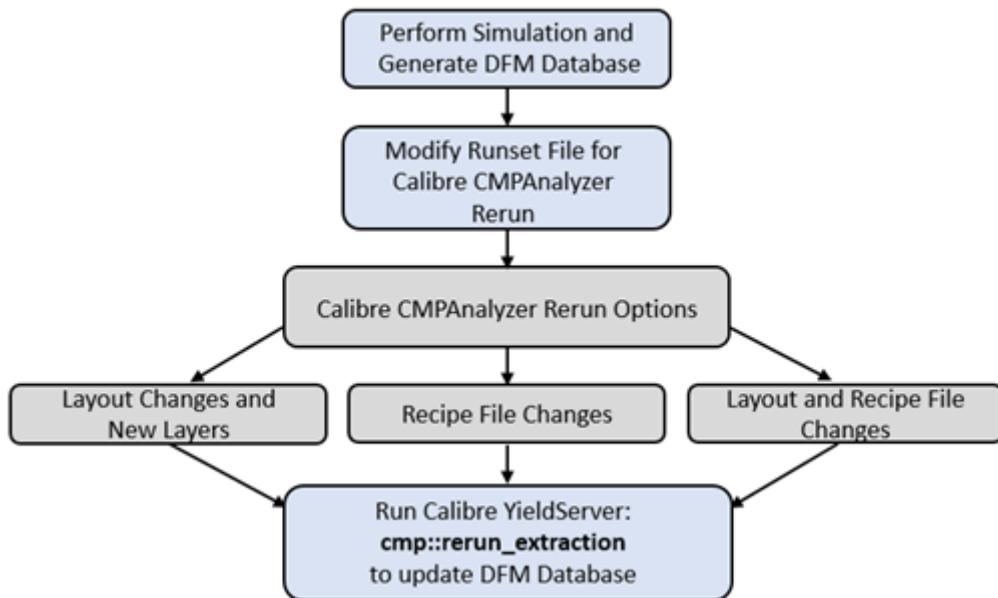
Chapter 5

Calibre CMPAnalyzer Rerun Flow

The Calibre CMPAnalyzer rerun flow enables you to selectively re-extract and perform simulation on designs with newly added layers, layout changes, and process recipe changes.

The Calibre CMPAnalyzer rerun flow takes a DFM database (*dfmdb*) from a previous Calibre CMP analysis as input along with a modified runset containing select layers and settings. The rerun flow uses the modified runset and Calibre YieldServer commands to rerun the CMP analysis. During the analysis, the rerun flow only re-extracts and simulates the selected layers in the modified runset and reuses the data for any unchanged layers when regenerating the DFM database. This reduces the time spent regenerating the data for the entire DFM database and enables you to quickly assess intermediate design and process changes.

Figure 5-1. Calibre CMPAnalyzer Rerun Flow



Note

- For layout changes, the DB_EXTENT layer and top cell name of the design must remain the same for the original run and the rerun.

For data consistency within the DFM databases, a final and full CMP analysis should be run after the Calibre CMPAnalyzer rerun flow to generate a new DFM database when all the modifications in the layout and recipe file are done.

cmp::rerun_extraction 89

Performing the Calibre CMPAnalyzer Rerun Flow.....	90
Using the Calibre CMPAnalyzer Rerun Flow for New Layers.....	92
Exporting the DB_EXTENT Layer	94
Setting DB_EXTENT in the Calibre CMPAnalyzer Rerun Flow.....	95

cmp::rerun_extraction

Calibre YieldServer command

Reruns extraction and multilayer simulation in the Calibre CMPAnalyzer rerun flow.

Usage

```
cmp::rerun_extraction -layers sorted_list_of_layer_indexes -runset modified_runset  
[-resimulate]
```

Arguments

- **-layers sorted_list_of_layer_indexes**

A required argument that specifies a list of sorted (from lowest to highest) layer-index numbers. The list is space-separated and enclosed in brackets ([]) as shown:

```
[list first_layer_index_number ... last_layer_index_number]
```

For example:

```
[list 2 4]
```

- **-runset modified_runset**

A required argument that specifies the path to the Calibre Interactive runset used for the CMP analysis and modified for the re-extraction.

Note

 The DB_EXTENT layer and top cell name in the modified runset for the re-extraction run must be the same as in the runset used for the original CMP analysis. See “[Exporting the DB_EXTENT Layer](#)” on page 94 and “[Setting DB_EXTENT in the Calibre CMPAnalyzer Rerun Flow](#)” on page 95 for more information.

- **[-resimulate]**

An optional argument that resimulates all layers starting from the lowest index specified in the -layers option. For example, if [list 2 4] corresponds to layers M2 and M4, respectively, all layers are resimulated starting from M2.

Description

This command is used in the Calibre CMPAnalyzer rerun flow. It creates a temporary DFM database from a modified runset and an original DFM database. Then it runs the extraction only in the temporary DFM database. When the extraction finishes, the command moves the extraction data from the temporary DFM database to the original DFM database and optionally runs resimulation.

As part of this flow, the `cmp::rerun_extraction` command requires an environment variable specification for the original DFM database creation as follows:

```
setenv CMP_ENABLE_RERUN_SIMULATION 1
```

The environment variable can be specified from the command line according to your shell environment or within the Calibre Interactive runset (in both original and modified runsets) as follows:

```
*dfmEnvVars: {CMP_ENABLE_RERUN_SIMULATION 1 Runset}
```

Examples

This example re-extracts the layers in the runset that correspond to layer index 2 and layer index 4. It then resimulates all the layers starting from layer index 2 to the last layer specified in the modified runset.

```
cmp::rerun_extraction -layers [list 2 4] -runset ./modified_runset  
-resimulate
```

Performing the Calibre CMPAnalyzer Rerun Flow

Perform the basic steps of the Calibre CMPAnalyzer rerun flow using Calibre Interactive and Calibre YieldServer commands.

Note

 If you are adding new layers to your design, see “[Using the Calibre CMPAnalyzer Rerun Flow for New Layers](#)” on page 92.

Prerequisites

- You have set `CALIBRE_HOME` or `MGC_HOME` to the path in the Calibre software tree. Refer to the [Calibre Administrator’s Guide](#) for information on setting this variable.
- You have a valid recipe file. Refer to the [Calibre CMP Model Builder User’s and Reference Manual](#) for details on creating a recipe file.
- You have a runset for the original CMP analysis. See “[CMP Analysis with the Calibre Interactive GUI](#)” on page 15 for more information on creating a runset.
- You have access to Calibre YieldServer. Refer to “[CMP YieldServer Command Reference](#)” on page 124 for a list of commands specific to Calibre CMPAnalyzer.

Procedure

- From a command line, set the following environment variable and value according to your shell environment:

```
setenv CMP_ENABLE_RERUN_SIMULATION 1
```

This keeps the necessary data during the original DFM database creation for further re-extraction and simulation.

- Create the original DFM database using Calibre Interactive and the original runset for the CMP analysis. For example:

```
calibre -gui -dfm -batch original_runset
```

The original database may be constructed in batch or GUI modes.

The required DB_EXTENT layer file is exported during this step or exported separately. See “[Exporting the DB_EXTENT Layer](#)” on page 94.

- Add the DB_EXTENT layer import settings to the modified runset. See “[Setting DB_EXTENT in the Calibre CMPAnalyzer Rerun Flow](#)” on page 95.

The runset keywords or equivalent GUI settings for exporting the extent layer are not needed in the modified runset used to import the extent layer and rerun the extraction.

- Open the original DFM database using Calibre YieldServer.

```
calibre -ys -cmp -dfmdb original_dfmdb
```

- Specify the cmp::rerun_extraction command with arguments at the Calibre YieldServer command prompt (>). For example (specify in one line):

```
> cmp::rerun_extraction -layers [list 2 4] -runset ./modified_runset
-resimulate
```

This re-extracts and resimulates the layers in the modified runset as specified by the layer index list, for example, [list 2 4]. The layer index number is based on the order of the layers in the CMP layers table in the Calibre Interactive Inputs pane. The first layer (top-left entry) in the table is layer index 1. For example:

Figure 5-2. Layer Index Number

Layer Type	Layer Name
#1	Metal 1
#2	Metal 2
#3	Metal 3
Metal 4	M4
Metal 5	M5
Metal 6	M6

Note

 If you are using an updated recipe file, you must have a valid recipe with process changes corresponding to the layer specifications in the Calibre CMPAnalyzer rerun flow.

6. (Optional) Rerun the hotspots calculation at the Calibre YieldServer command prompt and examine the results from layer index 4 and up in Calibre RVE for DFM.

```
> cmp::calculate_hotspots
```

See “[Reviewing Analysis Results](#)” on page 25 and “[cmp::rerun_extraction](#)” on page 89 for more information.

Using the Calibre CMPAnalyzer Rerun Flow for New Layers

Create a modified runset in the Calibre Interactive GUI for the Calibre CMPAnalyzer rerun flow when new layers are added to your design after CMP analysis.

Prerequisites

- You have set CALIBRE_HOME or MGC_HOME to the path in the Calibre software tree. Refer to the [Calibre Administrator’s Guide](#) for information on setting this variable.
- You have a valid recipe file. Refer to the [Calibre CMP Model Builder User’s and Reference Manual](#) for details on creating a recipe file.
- You have a runset for the CMP analysis. See “[CMP Analysis with the Calibre Interactive GUI](#)” on page 15 for the steps to create a runset.

Procedure

1. From a command line, set the following environment variable according to your shell environment:

```
setenv CMP_ENABLE_RERUN_SIMULATION 1
```

This keeps the necessary data during the original DFM database creation for further re-extraction and simulation.

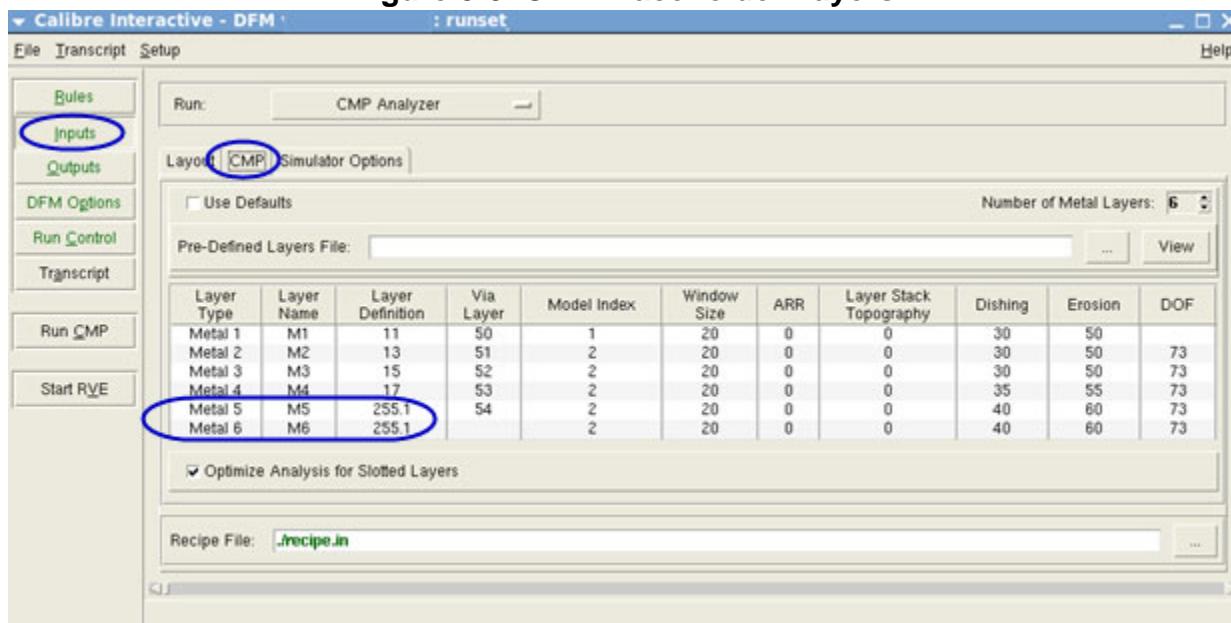
2. Open the runset file in the Calibre Interactive for DFM GUI.

```
calibre -gui -dfm original_runset
```

3. In the Inputs pane, select **CMP Analyze** in the **Run** menu.
4. Click the **CMP** tab to display the layer table.

Enter placeholder layer definitions in the table for the layers that you plan to add (any existing layer in the design can be a placeholder).

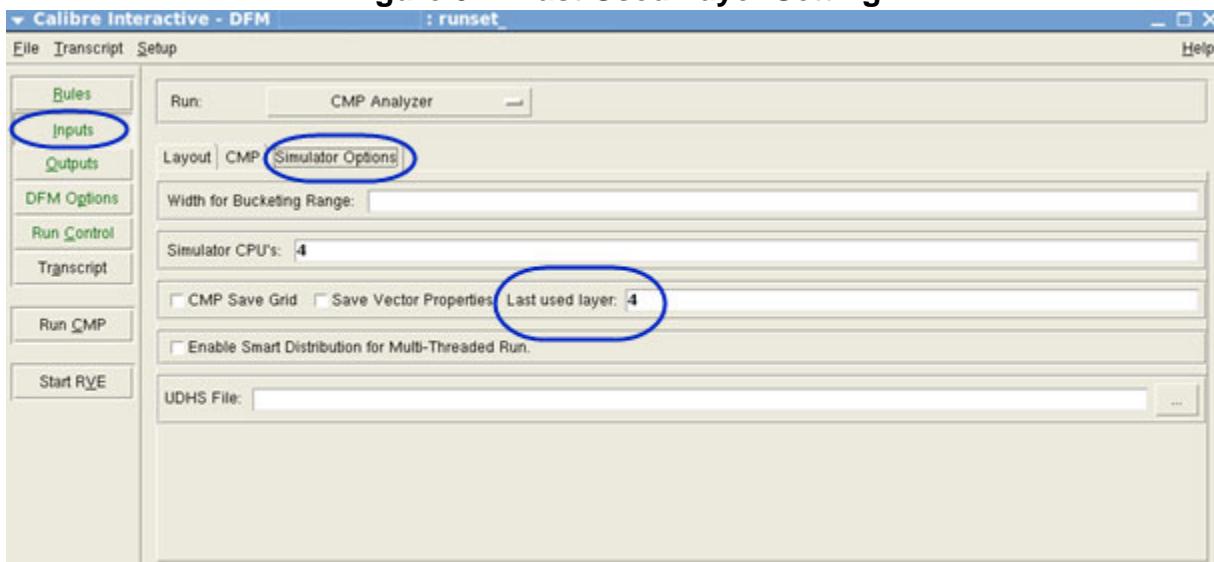
Figure 5-3. CMP Placeholder Layers



- Click the **Inputs** button on the left toolbar.
- Click the **CMP** tab in the top navigation bar.
- Click the **Simulator Options** tab.
- Set the **Last used layer** field to the number of the last used metal layer.

This is the last layer number that the original extraction and simulation should run.

Figure 5-4. Last Used Layer Setting



- Click the **Run CMP** button to run the analysis.

See “[CMP Analysis with the Calibre Interactive GUI](#)” on page 15 for more information.

- Add the new layers to your design and update the layer table in the **CMP** tab with the new layer definitions (replacing the placeholders).

8. Update the **Last used layer** number according to the added layers and save the modified runset.
9. Perform the Calibre CMPAnalyzer rerun flow using your modified runset to re-extract and simulate the new layers. See “[Performing the Calibre CMPAnalyzer Rerun Flow](#)” on page 90.

Exporting the DB_EXTENT Layer

Export the DB_EXTENT layer in the original CMP analysis DFM database for import to the Calibre CMPAnalyzer rerun flow. This ensures data consistency between the DFM databases.

Note

 The DB_EXTENT layer can be exported independently from the rerun flow, or it can be generated during the creation of the original DFM database (see Step 2 in “[Performing the Calibre CMPAnalyzer Rerun Flow](#)” on page 90).

Prerequisites

- You have set CALIBRE_HOME or MGC_HOME to the path in the Calibre software tree. Refer to the [Calibre Administrator’s Guide](#) for information on setting this variable.
- You have a valid recipe file. Refer to the [Calibre CMP Model Builder User’s and Reference Manual](#) for details on creating a recipe file.
- You have a runset for the CMP analysis. See “[CMP Analysis with the Calibre Interactive GUI](#)” on page 15 for the steps to create a runset.

Procedure

1. From a command line, set the following environment variable according to your shell environment:

```
setenv CMP_ENABLE_RERUN_SIMULATION 1
```

This keeps the necessary data during the original DFM database creation for further re-extraction and simulation.

2. Create a one-line Tcl script (*export_db_extent.tcl*) to export the DB_EXTENT layer using Calibre YieldServer commands. For example:

```
dfm:::write_oas -layer_info [list DB_EXTENT 1000 0] \
-file db_extent.oas
```

The script saves the exported DB_EXTENT layer from the original DFM database as an OASIS layer file. Alternatively, the script can save the DB_EXTENT layer as a GDSII layer file. For example:

```
dfm:::write_gds -layer_info [list DB_EXTENT 1000 0] \
-file db_extent.gds
```

Note

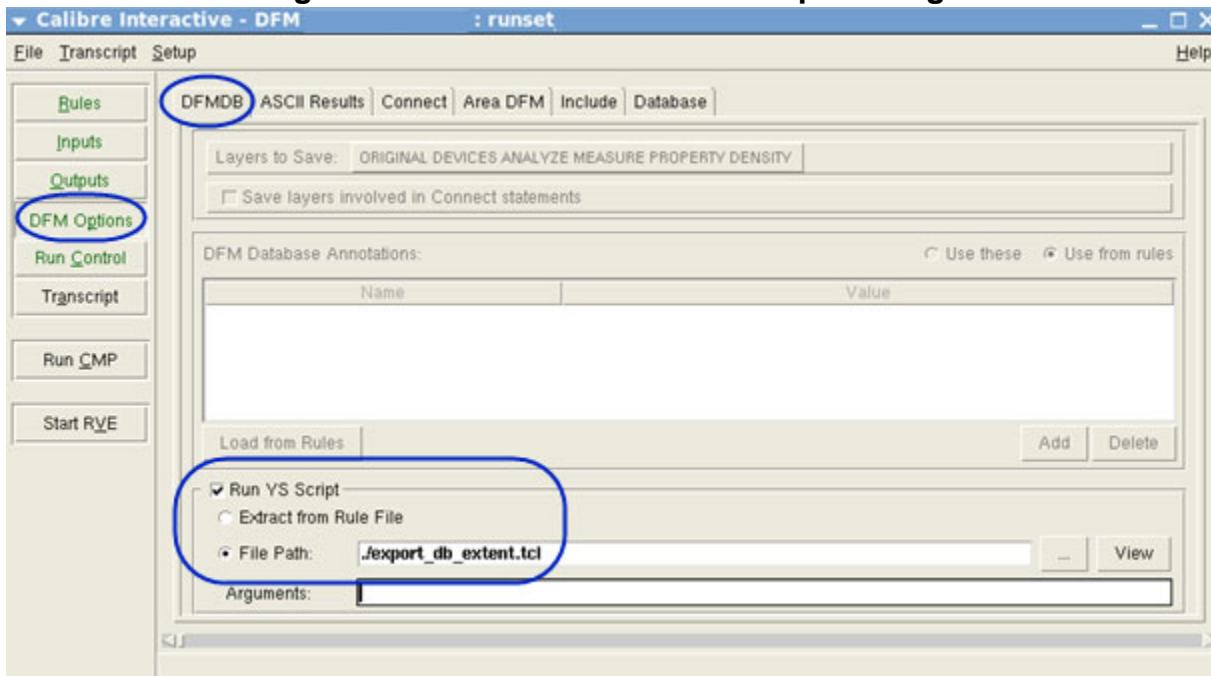
 The exported DB_EXTENT layer file must have the same layout system (OASIS or GDSII) as the original layout system.

3. Add keywords to the original Calibre Interactive runset (manually or use the GUI) to run the DB_EXTENT Tcl script using Calibre YieldServer and save the runset.

```
*dfmDFMUseYSScript: 1  
*dfmDFMYSScript: ./export_db_extent.tcl
```

This runset can be used for creating the original DFM database and exporting the DB_EXTENT layer file. See “[Performing the Calibre CMPAnalyzer Rerun Flow](#)” on page 90.

Figure 5-5. Runset YieldServer Script Settings



Results

The exported DB_EXTENT layer file can be imported to the subsequent Calibre CMPAnalyzer rerun flow. See “[Setting DB_EXTENT in the Calibre CMPAnalyzer Rerun Flow](#)” on page 95.

Setting DB_EXTENT in the Calibre CMPAnalyzer Rerun Flow

Enter the DB_EXTENT settings in the Calibre Interactive GUI and import the DB_EXTENT layer file to the Calibre CMPAnalyzer rerun flow. This ensures data consistency between the DFM databases.

Note

-  The DB_EXTENT layer and top cell name in the modified runset for the Calibre CMPAnalyzer rerun flow must be the same as in the runset used for the original CMP analysis.
-

Prerequisites

- You have set CALIBRE_HOME or MGC_HOME to the path in the Calibre software tree. Refer to the *Calibre Administrator's Guide* for information on setting this variable.
- You have a valid recipe file. Refer to the *Calibre CMP Model Builder User's and Reference Manual* for details on creating a recipe file.
- You have a runset for the CMP analysis. See “[CMP Analysis with the Calibre Interactive GUI](#)” on page 15 for the steps to create a runset.
- You have completed “[Exporting the DB_EXTENT Layer](#)” on page 94.

Procedure

1. From a command line, set the following environment variable according to your shell environment:

```
setenv CMP_ENABLE_RERUN_SIMULATION 1
```

This keeps the necessary data during the original DFM database creation for further re-extraction and simulation.

Alternatively, you can set the environment variable in the Calibre Interactive GUI by selecting **Setup > Set Environment** and clicking the **Add** button, or you can add the following keyword to your runset files:

```
*dfmEnvVars: {CMP_ENABLE_RERUN_SIMULATION 1 Runset}
```

2. Open the runset file in the Calibre Interactive for DFM GUI.

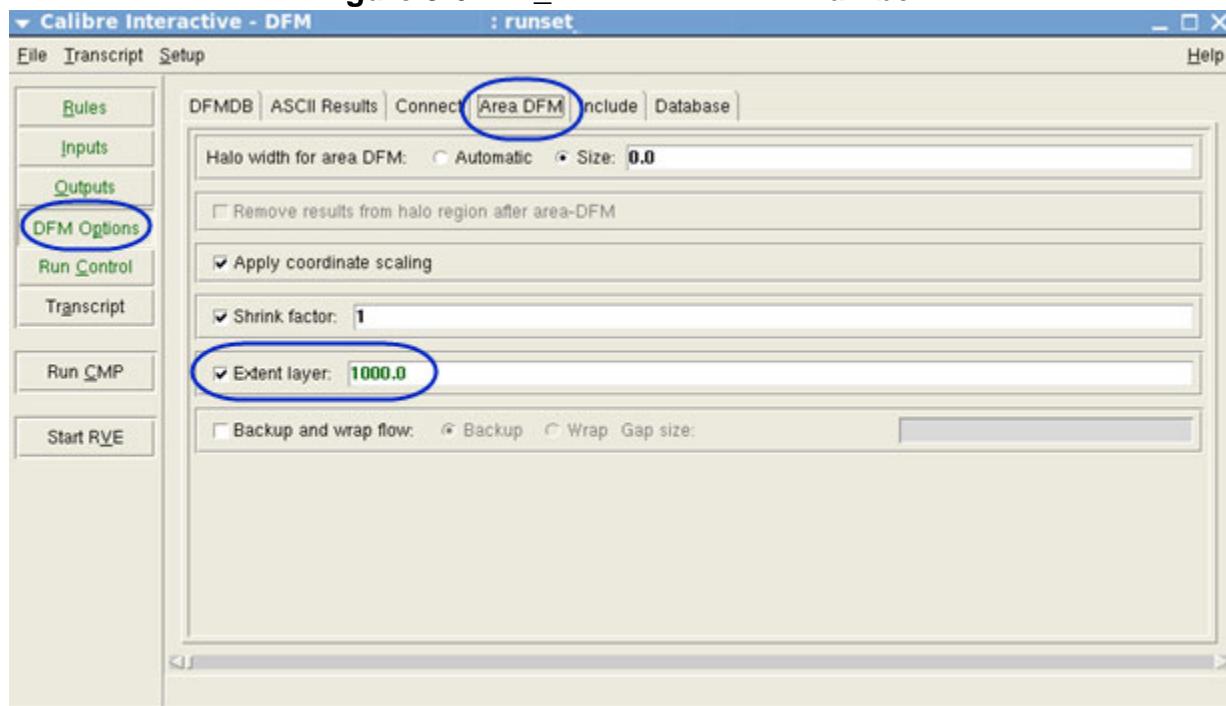
```
calibre -gui -dfm original_runset
```

3. Select **DFM Options** in the left pane of the GUI.

4. Click the **Area DFM** tab.

5. Select the **Extent layer** checkbox and enter the layer number and datatype (optional) for the DB_EXTENT layer in the original DFM database.

Figure 5-6. DB_EXTENT LAYER Number

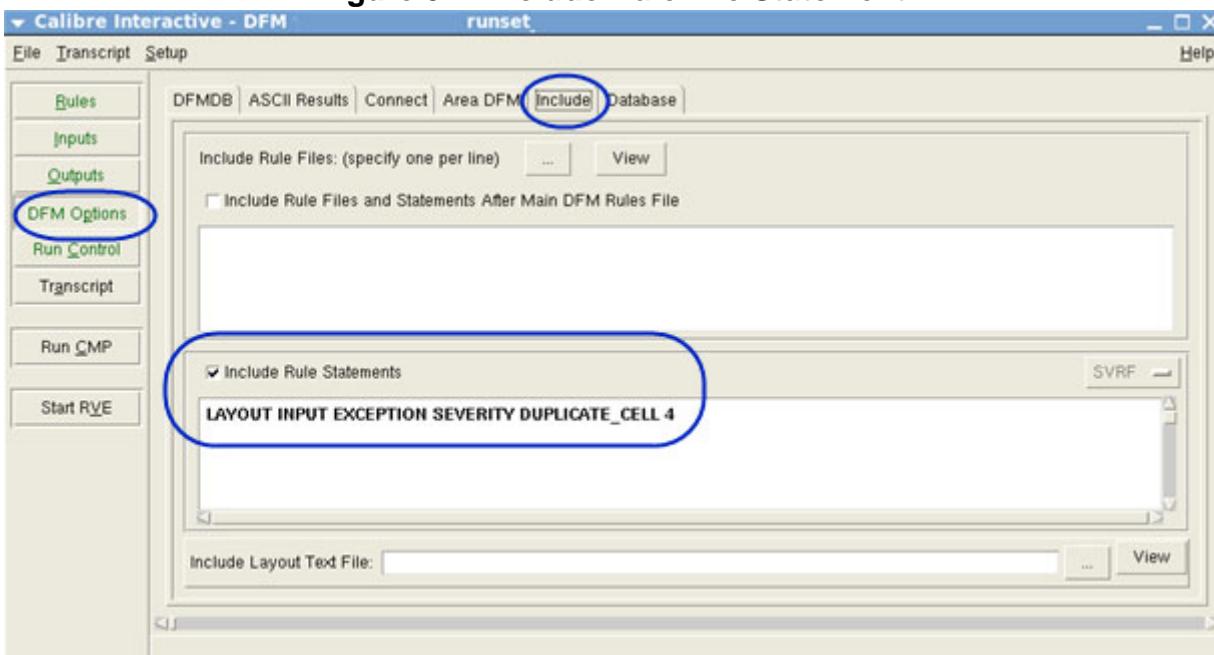


6. Click the **Include** tab.
7. Select the **Include Rule File Statements** checkbox and enter the following rule statement in the text field:

LAYOUT INPUT EXCEPTION SEVERITY DUPLICATE_CELL 4

This statement suppresses the top-cell error for name conflicts due to multiple layout records. See “[Layout Input Exception Severity](#)” in the *Standard Verification Rule Format (SVRF) Manual*.

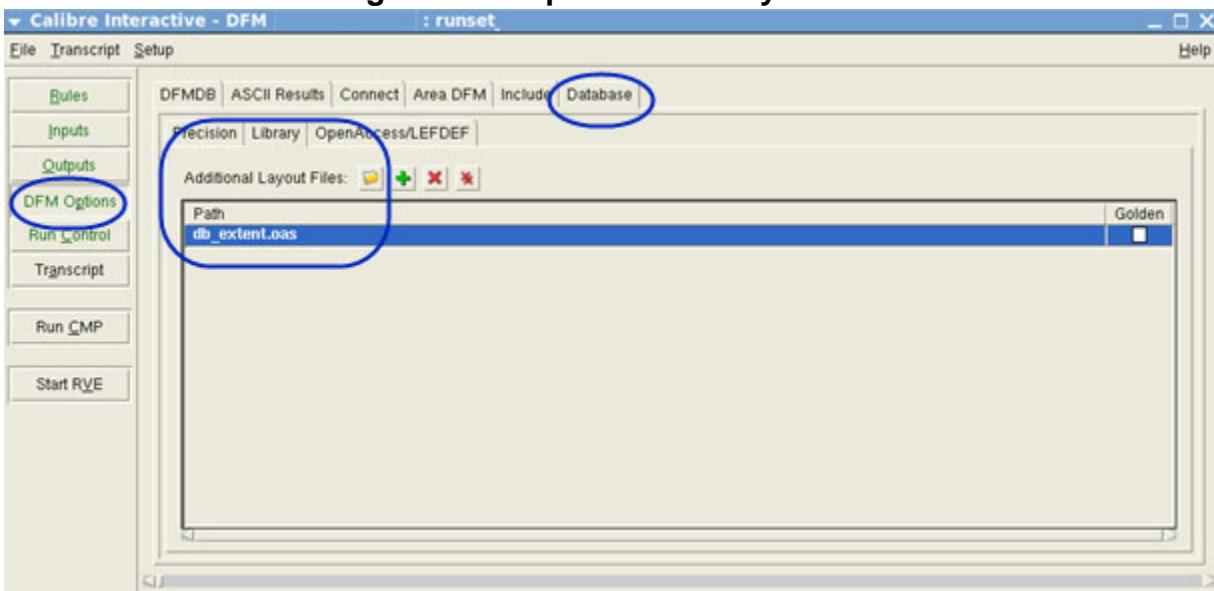
Figure 5-7. Include Rule File Statement



8. Click the **Database** tab.
9. Click the **Library** tab and open the DB_EXTENT layer file for import to the Calibre CMPAnalyzer rerun flow.

This is the OASIS or GDSII DB_EXTENT layer file saved in “[Exporting the DB_EXTENT Layer](#)” on page 94. The import occurs during the run and does not require any actions for merging.

Figure 5-8. Import Extent Layer Path



10. Save the settings for the DB_EXTENT layer in the modified runset.

11. Perform the Calibre CMPAnalyzer rerun flow using the modified runset (with the DB_EXTENT layer settings) along with the other settings used by the Calibre CMPAnalyzer rerun flow. See “[Performing the Calibre CMPAnalyzer Rerun Flow](#)” on page 90.

Chapter 6

Calibre CMPAnalyzer Reference

Reference material for the Calibre CMPAnalyzer tool is provided.

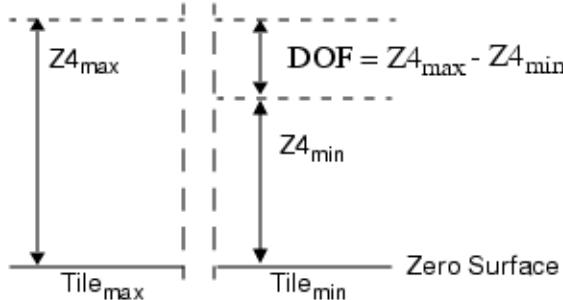
Terms and Concepts	101
Calibre CMPAnalyzer Simulator Flow	104
Calibre Interactive GUI Items for Calibre CMPAnalyzer	107
CMP Tab in Calibre RVE for DFM.....	118
CMP Batch Reporting.....	121
CMP YieldServer Commands	124
Colormap Stipple Patterns	145
Layer Number Specification Format.....	146
Pre-Defined Layers File Format.....	148

Terms and Concepts

Definitions and explanations for a number of terms and concepts related to the CMP process and Calibre CMPAnalyzer.

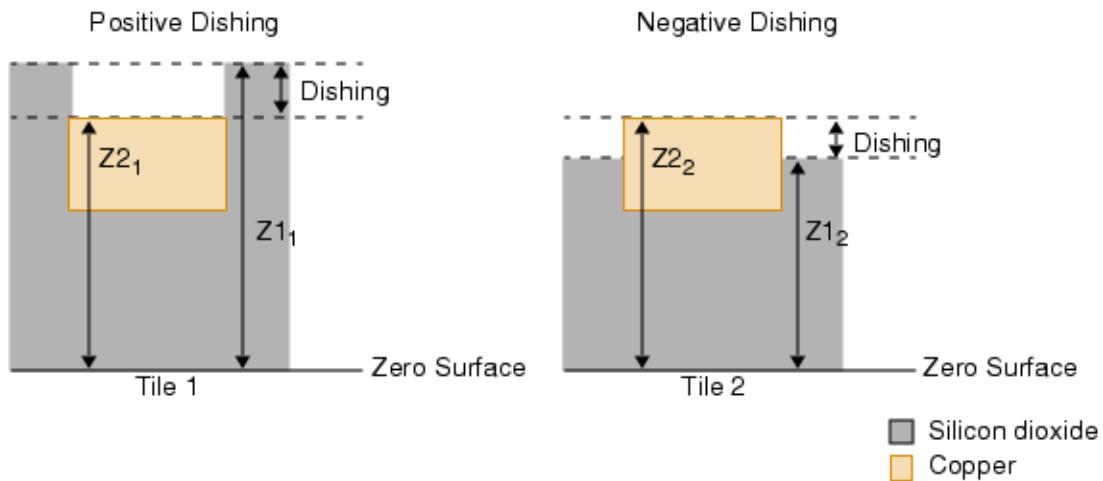
- **Depth of Focus** — Refers to the maximum distance from a focal plane over which an acceptable pattern can be imaged. At each patterning step, the quality of the printed image depends on the planarity of the underlying oxide. If the oxide height differences are too large, images may fall out of the depth of focus range, which results in a loss of pattern fidelity during resist exposure. This ultimately leads to undesirable variation in wire width after etch. For the [Calibre CMPAnalyzer Simulator Flow](#), depth of focus is defined as the global maximum tile height minus the global minimum tile height, or $Z4_{\max} - Z4_{\min}$, and is measured before etch. Refer to the following figure.

Figure 6-1. Depth of Focus Simulator Parameters



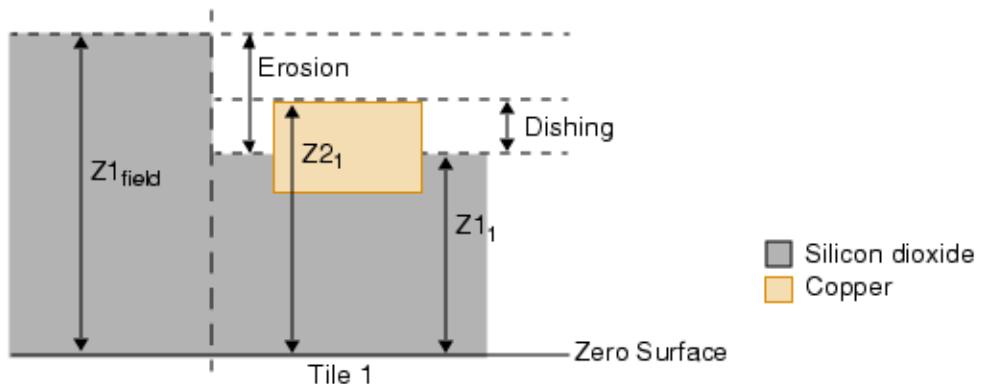
- **Dishing** — Refers to the result of a copper removal rate that is greater than the oxide removal rate (positive dishing), or to the result of a copper removal rate that is less than the oxide removal rate (negative dishing). In the [Calibre CMPAnalyzer Simulator Flow](#), dishing is defined as the difference between the oxide and metal heights, or $Z1 - Z2$, for a given tile as shown in the following figure.

Figure 6-2. Dishing Simulator Parameters



- **Erosion** — Occurs when the CMP process removes more oxide in one area compared with another area due to pattern variation across the chip. This can lead to a reduction in oxide height, which can contribute to systematic issues in the layers above. Values for erosion are reported for the [Calibre CMPAnalyzer Simulator Flow](#). In this flow, erosion is defined as the difference between a global reference oxide height and the maximum local tile height, or $Z1_{field} - Z1_{tile}$. The value of $Z1_{field}$ is calculated from the average topography heights of a range of tiles that contain minimal pattern density. Refer to the following figure.

Figure 6-3. Erosion Simulator Parameters



- **Hotspot** — An area that exceeds a specified threshold value of a metric calculated from simulation results. Hotspots can be thought of as simulation-based DRC violations.

- **Tile** — A square section of a design. Calibre CMPAnalyzer divides designs into approximately 20 um by 20 um rectangular tiles (windows), and the analysis is performed based on average values for each tile—individual features within a tile are not simulated.
- **Grid** — Calibre CMPAnalyzer calculates a number of parameters for each tile and outputs a collection of grids which contain all the values for a given property with the geometry information (coordinates, window number) associated with the tile. Certain grid operations can be used in CMP post-processing and user-defined hotspot Tcl scripts in the Calibre® YieldServer interface. For more information see “[Calculating User-Defined Hotspots](#)” on page 69 and “[CMP YieldServer Commands](#)” on page 124.
- **Trench Bottom** — The distance from the zero surface to the bottom of the metal. The trench bottom value corresponds to parameter Z3 for the [Calibre CMPAnalyzer Simulator Flow](#).

Calibre CMPAnalyzer Simulator Flow

Running the Calibre CMPAnalyzer simulator flow requires a process recipe file. The tool reports certain thickness parameters and rules.

Required Inputs for the Calibre CMPAnalyzer Simulator Flow	104
Thickness Parameters in the Calibre CMPAnalyzer Simulator Flow	104
Rule Output in the Calibre CMPAnalyzer Simulator Flow	105

Required Inputs for the Calibre CMPAnalyzer Simulator Flow

To run the Calibre CMPAnalyzer simulator flow, you must provide a process recipe file that defines a surface topography model.

For details on creating a recipe file, see “[Creating a Process Recipe File](#)” in the *Calibre CMP Model Builder User’s and Reference Manual*.

Thickness Parameters in the Calibre CMPAnalyzer Simulator Flow

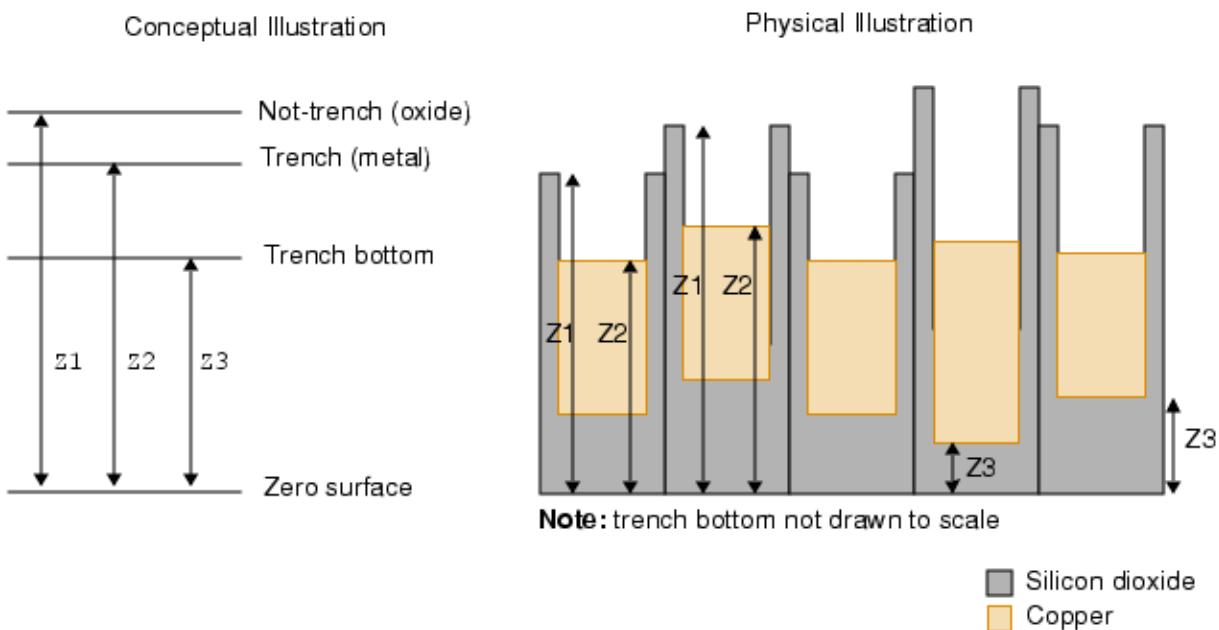
The Calibre CMPAnalyzer simulator flow reports results on certain thickness parameters. All thickness data are reported in whole angstroms.

Table 6-1. Thickness Parameters

Parameter	Name	Description
Z1	Thickness_Z1	The oxide height used for physical hotspot calculations.
Z2	Thickness_Z2	The metal height used for physical hotspot calculations.
Z3	Thickness_Z3	The height of the trench bottom. This is the distance from the zero surface to the bottom of the metal.
Z4	Thickness_Z4	The height of the oxide surface before trench patterning.
Z2 - Z3	Thickness_Metal	The metal thickness.
topology	Thickness_Topo	The pattern-density averaged surface height defined as follows: $Z1 * (1-D) + Z2 * D$, where D is the pattern density.

The following figure shows how these thickness parameters are represented in a design.

Figure 6-4. Simulator Parameters



Rule Output in the Calibre CMPAnalyzer Simulator Flow

Calibre CMPAnalyzer flow reports results in Calibre RVE for DFM for a number of rules.

The rules in the simulator flow are defined as follows.

- **Rule_Depth_Of_Focus** — $Z4_{\max} - Z4_{\min}$ (across the entire die).
- **Rule_Dishing** — $Z1 - Z2$ (for the local tile).
- **Rule_Erosion** — $Z1_{\text{field}} - Z1_{\text{tile}}$, where $Z1_{\text{field}}$ is defined as $\text{AVG}(Z1 * (1-D) + Z2 * D)$ for all tiles in the interval $[D_{\min}, D_{\min} * (1 + x)]$, where D is the pattern density and $x \leq 0.1$.

Note

If the minimum pattern density (D_{\min}) is zero, then $Z1_{\text{field}} = \text{AVG}(Z1)$ over all zero density frames.

Depth of focus, dishing, and erosion data are reported in whole angstroms.

The following figure shows how these rules are displayed in Calibre RVE for DFM.

Figure 6-5. Simulator Rules in Calibre RVE for DFM

The screenshot shows the Calibre RVE software interface with the title bar "Calibre - RVE top.dfmdb". The menu bar includes File, View, Highlight, Tools, Window, Setup, and Help. Below the menu is a toolbar with icons for folder, pencil, magnifying glass, and gear, followed by a search bar. The main window contains a table titled "Layer Browser" with a "CMP" tab selected. The table has columns: Metal, Fill, Rule_Depth_Of_Focus, Rule_Dishing, and Rule_Erosion. The data is as follows:

Metal	Fill	Rule_Depth_Of_Focus	Rule_Dishing	Rule_Erosion
M1	Original	0	46	13
M2	Original	0	52	20
M3	Original	0	55	17
M4	Original	361	48	16
M5	Original	361	14	4
M6	Original	0	11	4

At the bottom of the interface are buttons for Colormap, Histogram, and Property View, along with a zoom control.

Calibre Interactive GUI Items for Calibre CMPAnalyzer

Calibre Interactive has tabs and options that are specific to Calibre CMPAnalyzer runs.

CMP Tab on the Inputs Pane	107
Simulator Options Tab on the Inputs Pane.....	113
Area DFM Tab on the DFM Options Pane	116
CMP Tab on the Outputs Pane	117

CMP Tab on the Inputs Pane

To access: Click the **Inputs** button, choose the **CMP** tab.

The **CMP** tab on the Inputs pane of Calibre Interactive sets options for the Calibre CMPAnalyzer run such as the number of layers and the recipe file for the CMP simulator flow.

Tab contents are described in [Table 6-2](#). A right-click menu makes it easy to select columns to display and to adjust layer parameters; the right-click menu is described in [Table 6-3](#).

Figure 6-6. Calibre Interactive CMP Tab

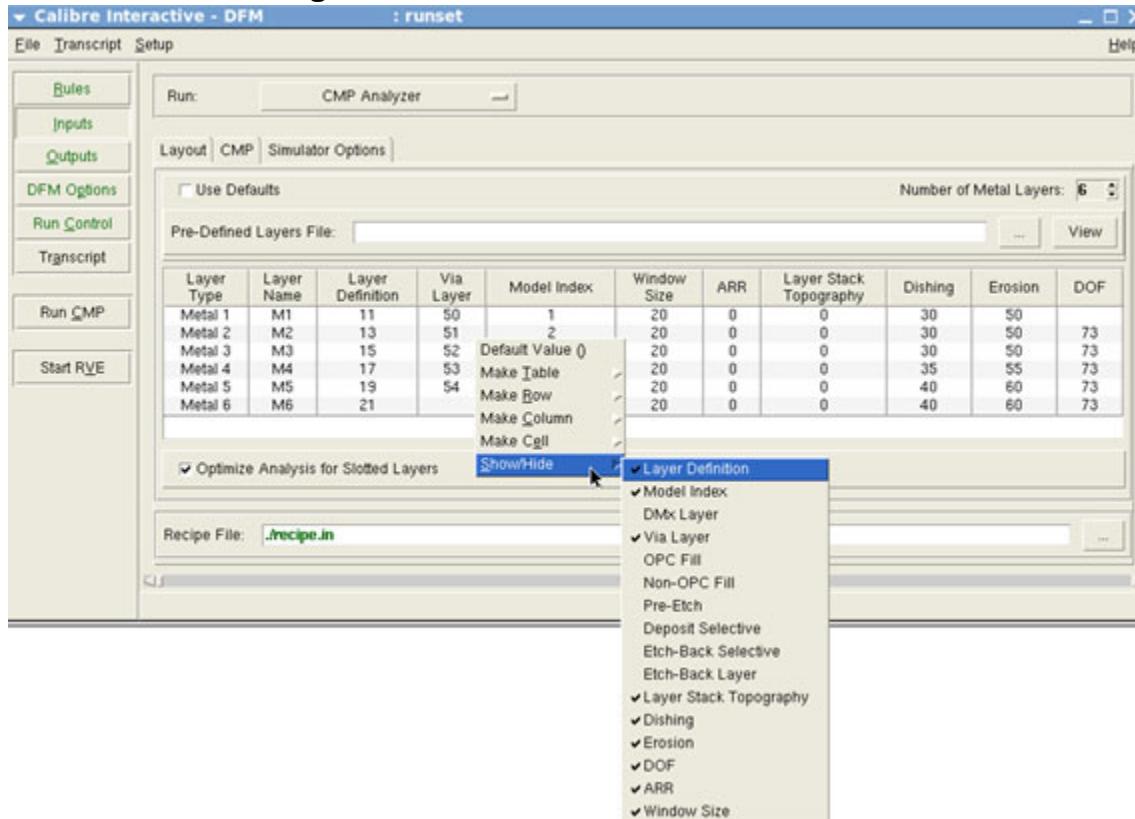


Table 6-2. Contents of the CMP Tab on the Inputs Pane

Element	Description
Use Defaults	Uses the default layer numbers.
Number of Metal Layers	The number of metal layers (or layers for analysis) in your design. The value you supply in this field controls the number of layers (rows) displayed.
Pre-Defined Layers File	Optional entry field for an SVRF input file containing predefined layers and layer operations. Use the “View” button to review or edit the input file. Refer to “ Pre-Defined Layers File Format ” on page 148 for file content information and conventions.
Layer Type Column	Displays the type of each layer for analysis. This field may be edited.
Layer Name Column	The “name” of the layer. This is a customizable name used to assign a readable string to the layer to be simulated. The layer numbers represent the layer position within the layer stack. Do not use the underscore (_) in layer names. Predefined layer names can be entered if a Pre-Defined Layers File is specified. Refer to “ Pre-Defined Layers File Format ” on page 148.
Layer Definition Column	Displays the GDS or OASIS® ¹ layer number and datatype associated with each layer in the rule file. The layer numbers represent the layer position within the layer stack. Refer to “ Multiple-Step Layer Number Specifications ” on page 147 for information on specifying multiple etch steps within a layer definition cell. Multiple predefined layers, each separated by a semicolon (;), can be specified per cell for multiple-etch flows. Do not mix predefined layer names with GDS or OASIS layer number and datatype names in the same cell. Predefined layers can be specified with a positive (default) or negative (~) option. Refer to “ Pre-Defined Layers File Format ” on page 148.
Model Index Column	Specifies the model used for the layer. Allowed values for Model Index are 1 through 99, where the index is the sequential number of the model in the recipe file, with the first recipe having a Model Index of 1.
DMx Layer Column	(Not displayed.) Dummy metal exclusion layers. By default, this column is empty.
Via Layer Column	Metal interconnect layers that connect a metal layer upward to the next metal layer in the stack.

Table 6-2. Contents of the CMP Tab on the Inputs Pane (cont.)

Element	Description
OPC Fill Column	(Not displayed.) Layers used to specify any pre-existing dummy fill that is processed by OPC. By default, this column is empty.
Non-OPC Fill Column	(Not displayed.) Layers used to specify any pre-existing dummy fill that is not processed by OPC. These layers normally obey larger design rule values than OPC fill layers (which obey the same design rules as the original metal). By default, this column is empty.
Pre-Etch Column	(Not displayed.) Displays a layer number and datatype when pre-etching is defined in the recipe file. Pre-etch layers can be specified with a positive (default) or negative (~) option. Multiple pre-etch steps can be specified within a row (layer) if they are separated by a semicolon (;). For example, specify <i>layer1;layer1</i> and <i>layer1.mask1;layer2</i> , where <i>layern.maskn</i> are the respective layer and mask numbers for the step. By default, this column is empty.
Deposit Selective Column	(Not displayed.) Optional mask layer that can be used to selectively define the areas for isotropic deposit. Deposit Selective mask layers can be specified with a positive (default) or negative (~) option. Multiple selective deposit steps can be specified within a row (layer) if they are separated by a semicolon (;). Merged layers are indicated by a comma (,). For example, if <i>layer1.mask1,layer2.mask1;layer3.mask1</i> is specified, where <i>layern.maskn</i> are the respective layer and mask numbers for the step, the analysis runs the first step on the merged layer (<i>layer1.mask1,layer2.mask1</i>) and the second step on <i>layer3.mask1</i> . By default, this column is empty.

Table 6-2. Contents of the CMP Tab on the Inputs Pane (cont.)

Element	Description
Etch-Back Selective Column	<p>(Not displayed.) Optional mask layer that can be used with or without Etch-Back Layer to selectively define the areas for etch-back. If not specified with Etch-Back Layer, the entire selected area is etched. Etch-Back Selective mask layers can be specified with a positive (default) or negative (~) option.</p> <p>Multiple etch-back selective steps can be specified within a row (layer) if they are separated by a semicolon (;) for example, <i>layer1;layer1</i> and <i>layer1.mask1;layer2</i>, where <i>layern.maskn</i> are the respective layer and mask numbers for the step. An extra or a leading (;) is used to specify an empty step (no layer) for example, <i>layern; ;layern</i> or <i>;layern;layern</i>. If specified with Etch-Back Layer, the number of specifications (step count) must match in both columns. Multiple etch-back selective steps must be either all positive or all negative.</p> <p>Predefined layer names can be entered if a Pre-Defined Layers File is specified. Refer to “Pre-Defined Layers File Format” on page 148.</p> <p>By default, this column is empty.</p>
Etch-Back Layer Column	<p>(Not displayed.) Optional mask layer that can be used with or without Etch-Back Selective to define features used for etch-back. If not specified with Etch-Back Selective, etch-back is performed over the entire design layer.</p> <p>Multiple etch-back mask layers can be specified within a row (layer) if they are separated by a semicolon (;) for example, <i>layer1;layer1</i> and <i>layer1.mask1;layer2</i>, where <i>layern.maskn</i> are the respective layer and mask numbers. An extra or leading (;) is used to specify an empty layer (no layer) for example, <i>layern; ;layern</i> and <i>;layern;layern</i>. If specified with Etch-Back Selective, the number of specifications (step count) must match in both columns.</p> <p>Predefined layer names can be entered if a Pre-Defined Layers File is specified. Refer to “Pre-Defined Layers File Format” on page 148.</p> <p>By default, this column is empty.</p>

Table 6-2. Contents of the CMP Tab on the Inputs Pane (cont.)

Element	Description
Layer Stack Topography Column	<p>Specifies the initial topography profile to use at the bottom of a layer when simulating a stack of layers. Possible single integer values per layer with thickness parameters are as follows:</p> <ul style="list-style-type: none"> • 0 — Use Z1 (field, or non-trench height). This is the default. • 1 — Use an average of Z1(1-density) + Z2(density), for each frame. • 2 — Use the Z2 (trench height). • 3 — Use max {Z1, Z2} for each frame. • 4 — Use min {Z1, Z2} for each frame. • 5 — Use a planar surface with max {Z1, Z2} over all frames. <p>Refer to “Thickness Parameters in the Calibre CMPAnalyzer Simulator Flow” on page 104 to see how thickness parameters are represented in a design.</p>
Dishing Column	Threshold value for the Dishing rule.
Erosion Column	Threshold value for the Erosion rule.
DOF Column	Threshold value for the Depth of Focus rule.
Array Recognition Radius (ARR) Column	<p>Specifies an optional parameter that is the radius of a search area that results in the recognition of array regions and the correct extraction of their geometry grid values (width, space, pattern density, and others).</p> <p>Specify a positive value in um or 0 (default) to disable array recognition.</p>
Window Size Column	<p>Specifies the window size for the simulator. Enter a single dimension or two space-separated numbers for the x- and y-dimensions.</p> <ul style="list-style-type: none"> • Positive numbers — The window dimensions are adjusted so that the analysis region is covered by an odd number of equal-sized windows. If a single dimension is specified, the window may not be exactly square. • Negative numbers — The exact window size is used. If necessary, the simulator adds empty area to the windows in the final row and column in order to use the exact window size. If this occurs, the windows in the top row and far right column of Calibre RVE colormaps appear smaller than the other windows.
Recipe File	The path to the recipe file.

1. OASIS® is a registered trademark of Thomas Grebinski and licensed for use to SEMI®, San Jose.
SEMI® is a registered trademark of Semiconductor Equipment and Materials International.

Table 6-3 describes the right-click menu elements available in the Layer Table. Use this menu and its submenus to change the sign (negative, subtractive, or positive) for the Layer Definition, DMx Layer, Via Layer, OPC Fill, and Non-OPC Fill table entries.

Table 6-3. Layer Table Right-Click Menu

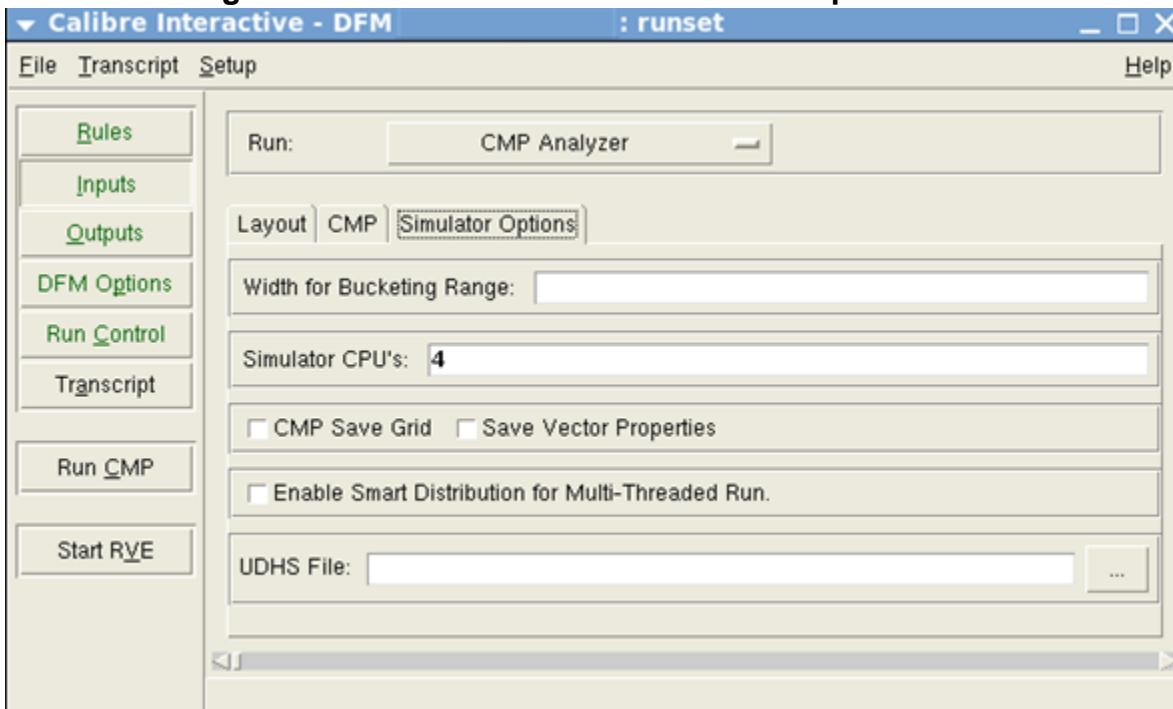
Element	Description
Default Value	Shows the default parameter value for the selected Layer Type and column cell.
Make Table	Changes the sign for all layers in the Layer Table.
Make Row	Changes the sign for layers in the selected row.
Make Column	Changes the sign for layers in a selected column.
Make Cell	Changes the sign for one specific layer.
Show/Hide	Select which columns are shown or hidden.

Simulator Options Tab on the Inputs Pane

To access: Select the Inputs pane and click the **Simulator Options** tab. The run type must be CMPAnalyzer.

The **Simulator Options** tab in Calibre Interactive has options for the bucketing flow, the number of CPUs for simulation, saving grid files and vector properties, the Calibre CMPAnalyzer rerun flow, distributed processing, and the user-defined (custom) hotspot flow.

Figure 6-7. Calibre Interactive Simulator Options Tab



Objects

Table 6-4. Contents of the Simulator Options Tab on the Inputs Pane

Option	Description
Width for Bucketing Range	Specifies the percentage (limit) used to calculate bucket widths for each tile. Enter a value from 0 and 1, inclusive, to enable the bucketing flow. A value of 0 results in the maximum possible number of buckets; a value of 1 results in one bucket for each tile. See " Running the CMP Bucketing Flow in Calibre Interactive for DFM " on page 81 for complete details on setting this value.
Simulator CPUs	Specifies the number of CPUs to use for the simulation. The default number is 4.
CMP Save Grid	Specifies to save grid files during the simulation.

Table 6-4. Contents of the Simulator Options Tab on the Inputs Pane (cont.)

Option	Description
Save Vector Properties	<p>Specifies to save the data for each property as a vector on a single-geometry CMP layer. The vector format collects all data for each property, while requiring only one access to the DFM results database.</p> <p> Note: The vector property format is compatible with Tcl scripts for user-defined hotspot (UDHS) implemented with Calibre YieldServer grid commands. Iterator-based Tcl scripts are not compatible with the vector property format.</p>
Last used layer	<p>Only visible when the following environment variable is set:</p> <pre>setenv CMP_ENABLE_RERUN_SIMULATION 1</pre> <p>Specifies the last layer number that should be run in the original extraction for the Calibre CMPAnalyzer Rerun Flow. See “Performing the Calibre CMPAnalyzer Rerun Flow” on page 90.</p>
Enable Smart Distribution for Multi-Threaded Run	<p>Specifies to enable Calibre CMPAnalyzer multi-processing flow (runs extraction mode and simulation mode in parallel). This option applies only to runs with multilayer (two or more layers selected) simulation and when the “Run Calibre” “Multi-Threaded” execution mode is selected in the Run Control pane Performance tab of Calibre Interactive.</p> <p>The following limitations apply to the “Number of CPUs to use for DFM:” specification in the Performance tab when enabling the multi-processing flow:</p> <ul style="list-style-type: none"> • If the specified number of CPUs is “All”, the multi-processing flow is enabled only if there are ≥ 8 CPUs available. Additional licenses are required for CPU “All” in the multi-processing flow (parallel flow) compared to the default flow (sequential flow). • If the specified number of CPUs is N, the multi-processing flow is enabled only if there are N number of CPUs > 8 available. In the case where the user specifies N CPUs, no additional licenses are required, but Calibre uses less threads for extraction than if the user specifies CPUs “All”. <p> Note: For license information, refer to the Calibre Administrator’s Guide.</p>

Table 6-4. Contents of the Simulator Options Tab on the Inputs Pane (cont.)

Option	Description
UDHS File	Specifies a Tcl script to calculate the user-defined hotspot properties (UDHS file). The path to this file can be an absolute or relative path (./) indicating the filename. The UDHS file specification in this field overrides the UDHS environment variable setting. See “ Running Calibre CMPAnalyzer with User-Defined Hotspots ” on page 74 for complete details on this flow.

Related Topics

[Running the CMP Bucketing Flow in Calibre Interactive for DFM](#)

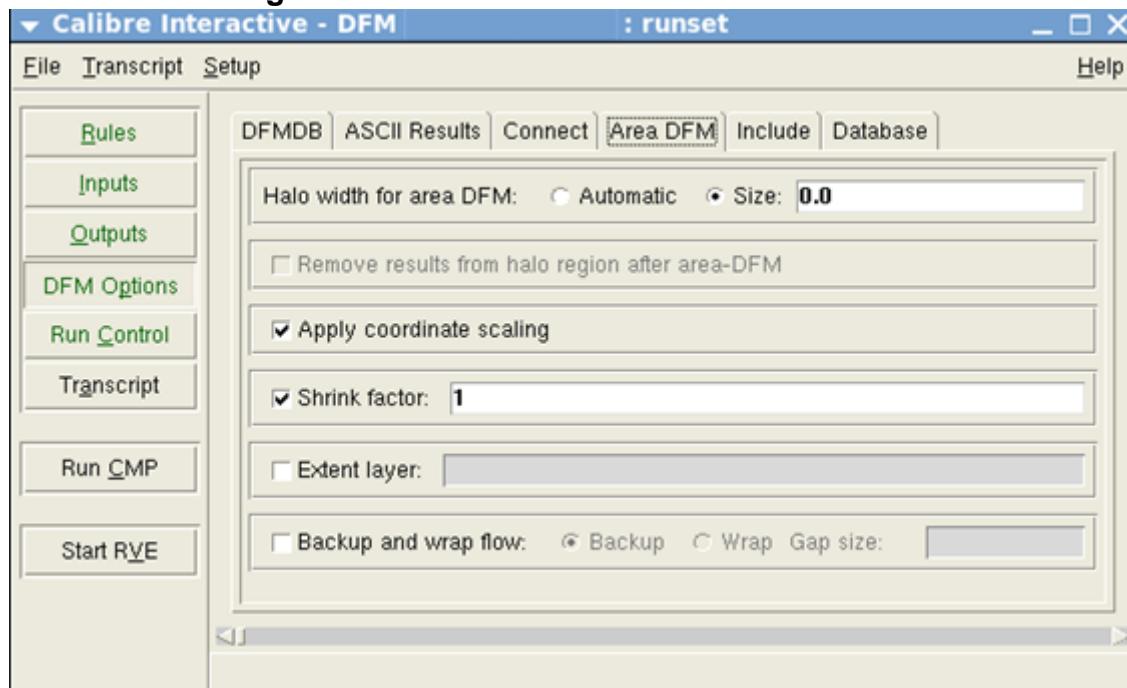
[CMP Analysis with the Calibre Interactive GUI](#)

Area DFM Tab on the DFM Options Pane

To access: Select the DFM Options pane and click the **Area DFM** tab. The run type must be CMPAnalyzer.

The **Area DFM** tab in Calibre Interactive has options for DFM area analysis and options specific to Calibre CMPAnalyzer.

Figure 6-8. Calibre Interactive Area DFM Tab



Objects

See “[Performing Area DFM](#)” in the *Calibre Interactive User’s Manual* for the additional options in the **Area DFM** tab that are non-specific to Calibre CMPAnalyzer.

Table 6-5. Area DFM Tab — Calibre CMPAnalyzer Run Options

Option	Description
Shrink factor	Specifies to apply a shrink factor to the layout. This simulates the effect of an optical shrink factor on the layout. Adds Layout Magnify and DRC Magnify Results statements to the Calibre Interactive control file.

Table 6-5. Area DFM Tab — Calibre CMPAnalyzer Run Options (cont.)

Option	Description
Extent layer	<p>Specifies the extent layer with a layer number or with <i>layer_number.datatype</i> format. If only the layer number is specified, then the datatype is assumed to be 0 (zero).</p> <p>When enabled, the Extent layer option changes the definition of the DB_EXTENT marker layer, which defines the extent layers required for the analysis.</p> <ul style="list-style-type: none"> • If the layer field is empty (default), $DB_EXTENT = MERGE(EXTENT DRAWN ORIGINAL)$. • If the layer field has a specified layer number, $DB_EXTENT = MERGE(EXTENT layer)$. <p>When used with the Backup and wrap flow option, the Extent layer option determines the original extent layer before the window size is adjusted.</p> <p>When used with the Calibre CMPAnalyzer Rerun Flow, this option specifies the required extent layer to export in the original DFM database. See “Exporting the DB_EXTENT Layer” on page 94.</p>
Backup and wrap flow	<p>Specifies to enable the backup and wrap flow in Calibre CMPAnalyzer. The flow derives an extent layer that is the next larger multiple of the tile (window) size, ensuring that tiles at design edges are filled. When enabled, the flow supports either of two modes:</p> <ul style="list-style-type: none"> • Backup — Selects shapes starting at the original design edges and shifts them outward to fill the empty areas of the new extent layer. This is the default mode. • Wrap — Emulates the die in the wafer context by selecting and copying shapes from edges and corners to fill empty spaces in the new extent layer. The “Gap size” field specifies the space between existing and copied shapes (default is 0, no space). <p>The Backup and wrap flow option may be used with or without the Extent layer option. Only fixed window size and same x- and y-dimensions are supported.</p>

CMP Tab on the Outputs Pane

To access: Select the Outputs pane in Calibre Interactive and click the **CMP** tab. The run type must be CMPAnalyzer.

The **CMP** tab on the Outputs pane of Calibre Interactive has options related to the CMP Summary Report.

Figure 6-9. Calibre Interactive CMP Tab on the Outputs Pane

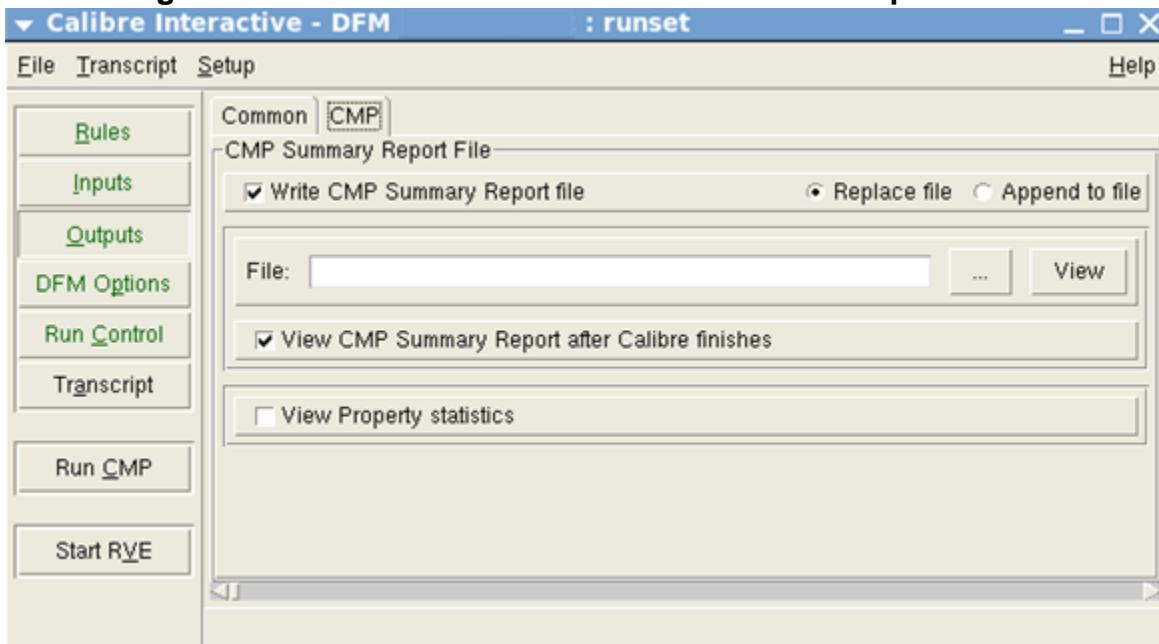


Table 6-6. Contents of the CMP Tab on the Outputs Pane

Element	Description
Write CMP Summary Report file	Writes a summary report to the specified file.
Replace file Append to file	Selects whether to overwrites the summary report file or append to it.
View CMP Summary Report after Calibre finishes	If enabled, opens the summary report in a text window when the CMP run completes.
View Property statistics	Includes, for each layer, a summary of the attached properties. This summary includes the minimum, maximum, range, average, and standard deviation of the property values.

CMP Tab in Calibre RVE for DFM

To access: Click the **CMP** tab in Calibre RVE for DFM with CMPAnalyzer results loaded. See “[Reviewing Analysis Results](#)” on page 25.

The **CMP** tab displays the results of a Calibre CMPAnalyzer run. You can create histograms and colormaps from the tab.

Figure 6-10. CMP Tab in Calibre RVE for DFM

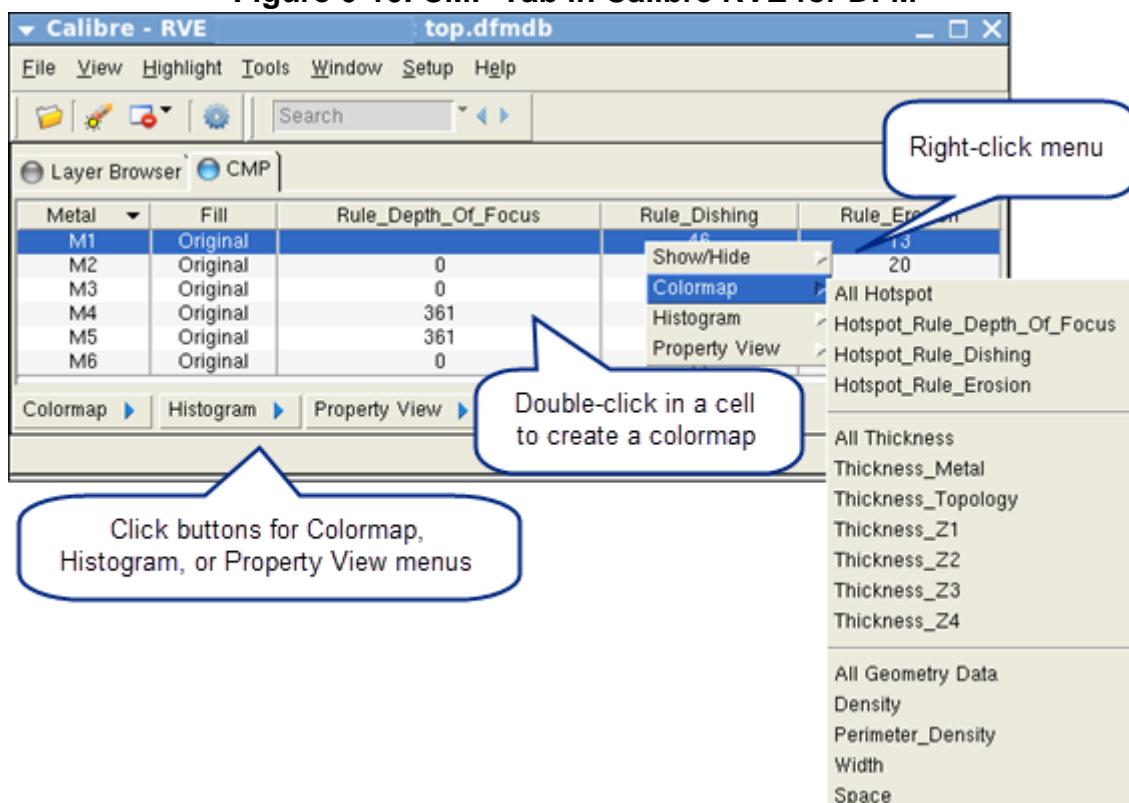


Table 6-7. CMP Tab Contents in Calibre RVE for DFM

Element	Description
Table	Displays one row for each metal layer analyzed. Double-click in a result cell to create a colormap. Right-click for a context-sensitive menu.
Colormap Button	Displays the Colormap menu for the selected layer.
Histogram Button	Displays the Histogram menu for the selected layer.
Property View Button	Displays the Property View menu for the selected layer.

Usage Notes

To set colormap and histogram options, do one of the following:

- Right-click in the histogram or colormap display for an option menu.
- Choose **Setup > Options** to open the **Options** tab and select the **Histograms and Colormaps** category.

To display the results in a connected layout viewer:

- For colormaps, right-click in the region of interest in the colormap and select **Highlight Window** or **Colormap Windows**.
- For histograms, right-click in a histogram bar and select **Highlight This Bar** or **Show Color Map**.

To display different plot types in a single window for comparison:

- Click the **Property View** button and select a property to colormap in the window.
- Select different plot types to display in the same window by clicking the GUI buttons at the top of the Property View plot window.

For more information, refer to “[Reviewing Analysis Results](#)” on page 25, “[Performing Data Analysis in Property View](#)” on page 47, and “[Using Calibre RVE for DFM](#)” in the *Calibre RVE User’s Manual*.

Related Topics

[CMP Analysis Results](#)

CMP Batch Reporting

Batch reporting can be used to output charts, colormaps, histograms and tables from Calibre RVE for DFM to a specified directory.

It is used with the Calibre CMPAnalyzer batch flow. Refer to “[Using the Calibre CMPAnalyzer Batch Flow](#)” on page 21.

The configuration file follows the format described in “[DFM HTML Reporting](#)” in the *Calibre RVE User’s Manual*.

The following is a sample configuration file that is designed to output a CMP hotspot report, and histograms and colormaps for all metrics.

To generate the reports, run Calibre as follows:

```
calibre -rve [-turbo] -dfm dfmdb -report config_file  
-outputdir output_dir -cmp
```

where:

- *dfmdb* — The name of the DFM database directory.
- *config_file* — The name of the configuration file.
- *output_dir* — The desired output directory for the reports.

Example Batch Reporting Configuration File **121**

Example Batch Reporting Configuration File

The batch reporting configuration file is used to create an HTML formatted report of CMPAnalyzer results for the flow.

The following provides an example batch reporting configuration file:

```
; Example Batch Reporting Configuration File (MGC Flow)

[Common]
; Common Parameters

GroupsToInclude = *
RulesToInclude = *
BinsToInclude = *
MetricsToInclude = *
Rows = *
Sort = Descending
GroupBy = Metrics
OutputFormat = html, csv, chart
;FilterBin = "Original"
;FilterMetric = ""
;FilterOperation = >=
;FilterValue = 0

; Create Hotspot Report Card
[CmpHotspotReport]
ReportType = CmpHotspotReport

; Create Density Histograms
[Original Density Histograms]
ReportType = ByWindowHistogram
BinsToInclude = "Original"
MetricsToInclude = "Density"

; Create Density Colormaps
[Original Density Colormaps]
ReportType = ByWindowColormap
BinsToInclude = "Original"
MetricsToInclude = "Density"

; Create Depth_Of_Focus Histograms
[Original Depth_Of_Focus Histograms]
ReportType = ByWindowHistogram
BinsToInclude = "Original"
MetricsToInclude = "Hotspot_Rule_Depth_Of_Focus"

; Create Depth_Of_Focus Colormaps
[Original Depth_Of_Focus Colormaps]
ReportType = ByWindowColormap
BinsToInclude = "Original"
MetricsToInclude = "Hotspot_Rule_Depth_Of_Focus"

; Create Dishing Histograms
[Original Dishing Histograms]
ReportType = ByWindowHistogram
BinsToInclude = "Original"
MetricsToInclude = "Hotspot_Rule_Dishing"

; Create Dishing Colormaps
[Original Dishing Colormaps]
ReportType = ByWindowColormap
```

```
BinsToInclude = "Original"
MetricsToInclude = "Hotspot_Rule_Dishing"

; Create Erosion Histograms
[Original Erosion Histograms]
ReportType = ByWindowHistogram
BinsToInclude = "Original"
MetricsToInclude = "Hotspot_Rule_Erosion"

; Create Erosion Histograms
[Original Erosion Histograms]
ReportType = ByWindowHistogram
BinsToInclude = "Original"
MetricsToInclude = "Hotspot_Rule_Erosion"

; Create Thickness_Metal Histograms
[Original Thickness_Metal Histograms]
ReportType = ByWindowHistogram
BinsToInclude = "Original"
MetricsToInclude = "Thickness_Metal"

; Create Thickness_Metal Colormaps
[Original Thickness_Metal Colormaps]
ReportType = ByWindowColormap
BinsToInclude = "Original"
MetricsToInclude = "Thickness_Metal"

; Create Thickness_Z1-Z4 Histograms
[Original Thickness_Z1-Z4 Histograms]
ReportType = ByWindowHistogram
BinsToInclude = "Original"
MetricsToInclude = "Thickness_Z*"

; Create Thickness_Z1-Z4 Colormaps
[Original Thickness_Z1-Z4 Colormaps]
ReportType = ByWindowColormap
BinsToInclude = "Original"
MetricsToInclude = "Thickness_Z*"

; Profile Plot
[XProfile Plot]
ReportType = ByWindowColormap
BinsToInclude = "Original"
MetricsToInclude = "Thickness_Metal"
XProfileInclude = {130,285,305}
```

CMP YieldServer Commands

Certain commands can be used to run Calibre CMPAnalyzer directly from Calibre YieldServer.

To use these commands, you must invoke Calibre YieldServer with the -cmp switch to gain access to the cmp:: namespace.

When writing Calibre YieldServer scripts and user-defined hotspot scripts, all procedure names, rule names, and property names must use only alphanumeric characters, that is letters and numbers, with the underscore (_) being the only allowable non-alphanumeric character.

Annotation and property string values may be any legal Tcl string.

CMP YieldServer Command Reference	124
Calibre YieldServer Scripts for CMPAnalyzer	140
Examples for CMP YieldServer Commands.....	142

CMP YieldServer Command Reference

Calibre YieldServer includes commands specific to Calibre CMPAnalyzer.

See “[Examples for CMP YieldServer Commands](#)” on page 142 and “[Tcl Script for User-Defined Hotspots](#)” on page 69 for example scripts using these commands.

Refer to “[Syntax Conventions](#)” on page 12 for the command syntax conventions used in the following commands and descriptions.

The following table contains CMP YieldServer commands. These commands can be used in Tcl procedures to calculate hotspots and write report data.

Table 6-8. YieldServer Commands

Command	Description
<pre>cmp::calculate_hotspots [-threshold_values '{' {'layer_name hotspot_name hotspot_value [hotspot_name hotspot_value ...] '}' ...'} [-no_defaults]</pre>	<p>Runs hotspots calculation for the specified input layers and hotspot name and threshold value pairs.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • -threshold_values '{' '{'layer_name hotspot_name hotspot_value [hotspot_name hotspot_value ...] '}' ...'}' <p>Specifies one or more input layers and a list of hotspot name and threshold value pairs for each one. Supported hotspot rule names are: Dishing, Erosion, and Depth_Of_Focus.</p> <p>If this argument is not specified, all predefined rules are calculated using default threshold values, and all registered rules are calculated using threshold values specified in the user-defined hotspots rule file.</p> <ul style="list-style-type: none"> • -no_defaults <p>Specifies when to omit default hotspot calculations. Use this option when user-defined hotspot rules have been specified. Do not use this option with -threshold_values.</p>
<pre>cmp::cleanup_clusters -layer <i>layer_name</i></pre>	<p>Removes hotspot cluster information from the DFM database for the specified layer after a cmp::run_clusterization command.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • -layer <i>layer_name</i> <p>Specifies the name of the layer that the clustering data is removed from in the DFM database.</p>

Table 6-8. YieldServer Commands (cont.)

Command	Description
<pre>cmp::export_clusters -file <i>filename</i> -layer <i>layer_name</i> -level <i>level_number</i> [<i>cluster_collection</i>] [-format {<u>ldb</u> oasis gds}] [-properties <i>property_list</i>] [-label <i>label_name</i>]</pre>	<p>Exports hotspot cluster data from a cmp::run_clusterization command to the specified results database (RDB) file.</p> <p>The command has two flows (See Step 7 in “Running CMP Hotspots Clustering From Calibre YieldServer” on page 60 for flow examples):</p> <ul style="list-style-type: none"> • Flow 1 — Dumps hotspot clusters data generated by the cmp::run_clusterization command (.ldb format is default). • Flow 2 — Dumps hotspot clusters from a DFM database that contains clusters saved by the cmp::save_clusters command. <p>Arguments:</p> <ul style="list-style-type: none"> • -file <i>filename</i> File path to the results database file. • -layer <i>layer_name</i> Specifies the layer, from which the cluster data is exported. (required for Flow 2). • -level <i>level_number</i> Specifies the cluster level (required for both flows). • [<i>cluster_collection</i>] Specifies an object that is a collection of all hotspot cluster data from a cmp::run_clusterization command and saved by a cmp::save_clusters command. (Flow 1 and Flow 2) For example: <pre>cmp::export_clusters \$clusterManager -layer M1 -level 1 -file ./clusters.ldb</pre> <ul style="list-style-type: none"> • [-format {<u>ldb</u> oasis gds}] Specifies format options for the output database (RDB format is the default), OASIS, or GDSII. • [-properties <i>property_list</i>] Specifies the cluster property name in the DFM database, from which the data are exported. (required for Flow 2) • [-label <i>label_name</i>] Defines a label that can be used in the RDB instead of the original layer name. For GDS and OASIS output, do not use existing layer names in the DFM database as a label.

Table 6-8. YieldServer Commands (cont.)

Command	Description
<pre>cmp::register_hotspot_proc -rule_name <i>hotspot_rule_name</i> -proc <i>proc_name</i> [-output_properties <i>list_of_property_names</i>]</pre>	<p>Registers the procedure as a hotspot metric to be calculated for the specified rule during the hotspot calculation phase.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • -rule_name <i>hotspot_rule_name</i> Specifies the hotspot rule name. The rule name should be the same as the property name for the hotspot metric. • -proc <i>proc_name</i> Specifies the procedure name. • -output_properties <i>list_of_property_names</i> Specifies list of property names generated by the hotspot procedure. <p>Example:</p> <pre>cmp::register_hotspot_proc -rule_name hs1 -proc hs1_metric</pre> <p>See “Tcl Script for User-Defined Hotspots” on page 69 for a complete example.</p>
<pre>cmp::register_hotspot_rules -file <i>hotspot_rule_file</i></pre>	<p>Registers hotspot rules defined in the specified <i>hotspot_rule_file</i>. The file must contain definitions of hotspot metric calculation and hotspot threshold checking Tcl procedures. You must specify this command before you specify cmp::run_cmp_analysis and cmp::calculate_hotspots.</p> <p>The hotspot metric calculation procedure should accept a layer as its only input argument. For example:</p> <pre>proc hs1_metric { layer } { ... }</pre> <p>The threshold procedure should take two arguments — the layer name and the threshold value — and should return the number of failed tiles. For example:</p> <pre>proc hs1_fails { layer value } { # calculate the number of failing tiles return \$failCount }</pre>

Table 6-8. YieldServer Commands (cont.)

Command	Description
<pre>cmp::register_hotspot_threshold_p roc -rule_name hotspot_rule_name -proc proc_name -values threshold_value [-output_properties list_of_property_names]</pre>	<p>Registers a procedure that checks metric values against specified threshold values.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • -rule_name hotspot_rule_name <p>Specifies the hotspot rule name. The rule name should be the same as the property name for the hotspot metric.</p> <ul style="list-style-type: none"> • -proc proc_name <p>Specifies the procedure name.</p> <ul style="list-style-type: none"> • -values threshold_value <p>Specifies a threshold value.</p> <ul style="list-style-type: none"> • -output_properties list_of_property_names <p>Specifies list of property names generated by the hotspot procedure.</p> <p>Example:</p> <pre>cmp::register_hotspot_threshold_proc -rule_name hs1 -proc hs1_fails -values 100</pre> <p>See “Tcl Script for User-Defined Hotspots” on page 69 for important information on writing the Tcl procedure.</p>
<pre>cmp::rerun_extraction -layers sorted_list_of_layer_indexes -runset modified_runset [-resimulate]</pre>	<p>Reruns extraction and multilayer simulation in the “Calibre CMPAnalyzer Rerun Flow” on page 87.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • -layers sorted_list_of_layer_indexes <p>Specifies a sorted list of layer index numbers.</p> <ul style="list-style-type: none"> • -runset modified_runset <p>Specifies the name of the updated runset used for the Calibre CMPAnalyzer rerun flow.</p> <ul style="list-style-type: none"> • -resimulate <p>Resimulates all the layers starting from the lowest layer index number specified in the -layers option.</p>

Table 6-8. YieldServer Commands (cont.)

Command	Description
<pre>cmp::run_clusterization -layer <i>layer_name</i> -property <i>property_name</i> -distance <i>max_distance</i> [-level <i>level_number</i>] [-area_threshold [1,0]] [-density_threshold [0,1]]</pre>	<p>Runs hotspot clustering and generates cluster data. See “Running CMP Hotspots Clustering From Calibre YieldServer” on page 60</p> <p>Arguments:</p> <ul style="list-style-type: none"> • -layer <i>layer_name</i> Specifies the layer for clustering. • -property <i>property_name</i> Specifies the property name for clustering. If no hotspots are defined for the given property, an error message is generated. • -distance <i>max_distance</i> Specifies the maximum distance between clusters for merging to occur (microns). • [-level <i>level_number</i>] Specifies the maximum level of clustering. The clusterization stops at this level, even if merging is still possible. By default, the clusterization goes to the maximum possible clustering level and stops. • [-area_threshold [1,0]] Prevents cluster generation in areas greater than (Chip Area * area_threshold). This prevents the cases where a cluster can cover all or a large part of the chip. The default is 1. • [-density_threshold [0,1]] Prevents cluster generation if the density of single-pixel hotspots is less than the density_threshold. The density of single-pixel hotspots in the cluster is defined as the ratio of the number of single-pixel hotspots to the number of pixels. The default is 0.

Table 6-8. YieldServer Commands (cont.)

Command	Description
<pre>cmp::run_cmp_analysis [-input_layers <i>list</i>] [-input_layers_types <i>types</i>] [-no_save] [-step {extraction simulation hotspot}] [-stop <i>stop_layer_number</i>] [-recipefile <i>file</i>]</pre>	<p>Runs CMP analysis (the current revision must be the master revision).</p> <p>Arguments:</p> <ul style="list-style-type: none"> • -input_layers <i>list</i> The list of layers to use (the default is all layers). • -input_layers_types <i>types</i> A list of the types of the input layers to use (corresponding to Model Index values in the GUI table for the CMP simulator). • -no_save When specified, does not save and freeze the database after the analysis is complete. The default is to save and freeze the database. • -step {extraction simulation hotspots} Run only the specified step of the CMP analysis flow (extraction, simulation, or hotspot calculation). Note that the extraction step can only be performed once. If the simulation step is skipped (the extraction and hotspot calculation steps are performed, in that order), only user-defined hotspots are calculated after extraction. Extraction is done automatically if you specify the simulation step and extraction has not yet been performed. • -stop <i>stop_layer_number</i> Stop hotspot simulation at the specified number of metal layers in the stack. This option enables partial stack simulation. For example, if -stop 3 is specified, hotspot simulation will occur for the first three metal layers. • -recipefile <i>file</i> Runs the simulation step with the specified recipe <i>file</i>.
<pre>cmp::save_clusters <i>cluster_collection</i></pre>	<p>Saves a collection of all hotspot cluster data to an object generated by the cmp::run_clusterization command.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • <i>cluster_collection</i> Specifies an object that is a collection of all cluster data from a cmp::run_clusterization command. It can be referenced by a cmp::export_clusters command. <pre>cmp::save_clusters \$clusterManager</pre>

Table 6-8. YieldServer Commands (cont.)

Command	Description
cmp::save_hotspots_report -file <i>filename</i> [-format { <u>txt</u> rdb}]	Writes a file that for all hotspot rules, contains coordinates of windows that violate thresholds for the rules. Arguments: <ul style="list-style-type: none">• -file <i>filename</i> Specifies the path to the output file.• -format {<u>txt</u> rdb} Specifies format options for the output file as plain text format (the default) or RDB file format.
cmp::save_xrc_report -dir <i>path</i>	Saves thickness data to a set of files. Arguments: <ul style="list-style-type: none">• -dir <i>path</i> Path to the output directory.

The following table contains CMP YieldServer commands for grid operations. These grid commands can be used in Tcl procedures to calculate hotspots based on the properties available on the simulation result layers.

Table 6-9. YieldServer CMP Grid Commands

Command	Description
cmp::abs_grid <i>grid</i>	Returns the absolute grid for the specified grid. Each element of the result grid is an absolute (value) of the corresponding element of the input grid. Arguments: <i>grid</i> Specifies the grid object.
cmp::add_grid <i>grid1</i> <i>grid2</i>	Returns the sum grid of two specified grids. Each element of the result grid is a sum of the corresponding elements of the input grids. Arguments: <ul style="list-style-type: none">• <i>grid1</i> Specifies the first grid object.• <i>grid2</i> Specifies the second grid object. Grid objects must have the same dimensions. One of the grid arguments can be a double-precision floating-point number.

Table 6-9. YieldServer CMP Grid Commands (cont.)

Command	Description
<p><code>cmp::and_grid</code> <i>grid1 grid2</i></p>	<p>Returns the AND grid of two specified grids. Each element of the result grid is a result of a logical AND operation on the corresponding elements of the input grids.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • <i>grid1</i> Specifies the first grid object. • <i>grid2</i> Specifies the second grid object. <p>Grid objects must have the same dimensions.</p> <p>One of the grid arguments can be a double-precision floating-point number.</p>
<p><code>cmp::average_area_grid</code> <i>grid</i> -radius {R '{Rx Ry}'}</p>	<p>Returns a grid where each element is calculated as an average value of the given neighboring area in the input grid.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • <i>grid</i> Specifies the grid object. • -radius {R '{Rx Ry}'} Defines the radius of area around the element for which the average value is calculated. For border elements, the average value is calculated based on the existing neighboring elements. <p>For example, the average value for the first element of a grid (a00) with radius 1 is calculated as follows:</p> $\text{average(a00)} = (\text{a00} + \text{a01} + \text{a10} + \text{a11})/4$

Table 6-9. YieldServer CMP Grid Commands (cont.)

Command	Description
<pre>cmp::create_grid { -layer <i>layer_name</i> -property <i>property_name</i> -column <i>column_count</i> -row <i>row_count</i>} [-value <i>initial_value</i>]</pre>	<p>Creates a new grid. The grid can be created either based on values of the specified property of the layer or based on the specified row and column dimensions and initial value.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • -layer <i>layer_name</i> Specifies the layer name. • -property <i>property_name</i> Specifies the name of a property of the layer. • -column <i>column_count</i> Specifies the column count of the created grid. • -row <i>row_count</i> Specifies the row count for the created grid. • -value <i>initial_value</i> Specifies an initial value to be assigned to all elements of the grid (double-precision floating-point number). The default value is 0.
<pre>cmp::divide_grid <i>grid1</i> <i>grid2</i></pre>	<p>Returns a quotient grid of two specified input grids. Each element of the result grid is a quotient of the corresponding elements of the input grids.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • <i>grid1</i> Specifies the first grid object. • <i>grid2</i> Specifies the second grid object. <p>Grid objects must have the same dimensions.</p> <p>One of the grid arguments can be a double-precision floating-point number.</p>

Table 6-9. YieldServer CMP Grid Commands (cont.)

Command	Description
<pre>cmp::equal_grid grid {-row -column -element} -limit limit_value -result_value {t_value '{t_value f_value}'}</pre>	<p>Returns a grid matrix with values <i>t_value</i> if the selected parameter for the corresponding element is equal to the limit and <i>f_value</i> otherwise. By default, <i>f_value</i> is the corresponding value of the input grid.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • grid Specifies the grid object. • -row -column -element Defines the parameter of the grid used in the comparison operations. • -limit limit_value Defines the comparison value (double-precision floating-point number). • -result_value {t_value '{t_value f_value}'} Defines the elements in the result grid matrix based on the outcome of the comparison operation. The values can be either a double-precision floating-point number or a pair of double-precision floating-point numbers.

Table 6-9. YieldServer CMP Grid Commands (cont.)

Command	Description
<pre>cmp::export_grid grid -file <i>filename</i> [-standard] [-matx -maty -matz] [-rdb] [-coords] [-layer <i>layer_name</i>] [-property <i>property_name</i>] [{-less -greater -equal -less_equal -greater_equal} <i>threshold_value </i></pre>	<p>Dumps grid data into a specified file.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • <i>grid</i> Specifies the grid object. • -file <i>filename</i> File used for saving grid data. • -standard Specifies to dump the grid sizes, coordinates, and values. This is the default. • -matx Specifies to dump the x-coordinates of the grid. • -maty Specifies to dump the y-coordinates of the grid. • -matz Specifies to dump a 2D matrix with property values. • -rdb Specifies to output a file with property values in RDB file format (.rdb). The default format is text. • -coords Specifies to dump a text file similar to -standard but with tile coordinates and property information. • -layer <i>layer_name</i> Specifies the name of the layer, mandatory for RDB (.rdb) format. • -property <i>property_name</i> Specifies the name of the property (can be a string), mandatory for RDB (.rdb) format. • {-less -greater -equal -less_equal -greater_equal} <i>threshold_value Specifies the condition type and threshold value. For example, “-greater 0” means that the RDB file contains tiles with threshold value > 0. If none of these options are specified, the RDB file contains all tiles.</i> • -append Specifies to append additional grid data to an existing RDB output file.

Table 6-9. YieldServer CMP Grid Commands (cont.)

Command	Description
<pre>cmp::get_grid_data grid {-sum_elements -product_elements -average -standard_deviation -minimum -maximum -row_count -column_count -elements_count {-row row_number -column column_number}}}</pre>	<p>Returns information for the specified grid.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • <i>grid</i> Specifies the grid object. • -sum_elements Returns summary of elements of the specified grid. • -product_elements Returns product of elements of the specified grid. • -average Returns average value of elements of the specified grid. • -standard_deviation Returns standard deviation of elements of the specified grid. • -minimum Returns the minimum value of elements of the specified grid. • -maximum Returns the maximum value of elements of the specified grid. • -row_count Returns row count of the specified grid. • -column_count Returns column count of the specified grid. • -elements_count Returns elements count of the specified grid. • -row <i>row_number</i> -column <i>column_number</i> Returns value of elements for the row and column number of the specified grid.

Table 6-9. YieldServer CMP Grid Commands (cont.)

Command	Description
<pre>cmp::gradient_grid grid -radius {R '{'Rx Ry'}'} [-max -min -average]</pre>	<p>Returns a grid where each element is a calculated gradient for a corresponding element of the input grid in the given neighboring area.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • grid Specifies the grid object. • -radius {R '{'Rx Ry'}'} Defines the radius from the element for which the gradient is calculated. • -max -min -average Defines which difference from the element in the neighboring area should be used. By default, the command calculates the maximum difference. <p>Example:</p> <pre>cmp::gradient_grid grid1 -radius 1 -average</pre> <p>The average value for the first element (a00) of <i>grid1</i> with radius 1 is calculated as follows:</p> <pre>average(a00)=(a00 + a01 + a10 + a11)/4</pre>
<pre>cmp::greater_equal_grid grid {-row -column -element} -limit limit_value -result_value {t_value '{'t_value f_value'')}</pre>	<p>Returns a grid matrix with values <i>t_value</i> if the selected parameter for the corresponding element is greater than or equal to the limit and <i>f_value</i> otherwise. By default, <i>f_value</i> is the corresponding value of the input grid.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • grid Specifies the grid object. • -row -column -element Defines the parameter of the grid used in the comparison operation. • -limit limit_value Defines the comparison value. • -result_value {t_value '{'t_value f_value'')} Defines the elements in the result grid matrix based on the outcome of the comparison operation. The values can be either a double-precision floating-point number or a pair of double-precision floating-point numbers.

Table 6-9. YieldServer CMP Grid Commands (cont.)

Command	Description
<pre>cmp::greater_grid grid {-row -column -element} -limit limit_value -result_value {t_value '{t_value f_value}'}</pre>	<p>Returns a grid matrix with values <i>t_value</i> if the selected parameter for the corresponding element is greater than the limit and <i>f_value</i> otherwise. By default, <i>f_value</i> is the corresponding value of the input grid.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • grid Specifies the grid object. • -row -column -element Defines the parameter of the grid used in the comparison operation. • -limit <i>limit_value</i> Defines the comparison value. • -result_value {<i>t_value</i> '{<i>t_value f_value</i>}'} Defines the elements in the result grid matrix based on the outcome of the comparison operation. The values can be either a double-precision floating-point number or a pair of double-precision floating-point numbers.
<pre>cmp::less_equal_grid grid {-row -column -element} -limit limit_value -result_value {t_value '{t_value f_value}'}</pre>	<p>Returns a grid matrix with values <i>t_value</i> if the selected parameter for the corresponding element is less than or equal to the limit and <i>f_value</i> otherwise. By default, <i>f_value</i> is the corresponding value of the input grid.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • grid Specifies the grid object. • -row -column -element Defines the parameter of the grid used in the comparison operation. • -limit <i>limit_value</i> Defines the comparison value. • -result_value {<i>t_value</i> '{<i>t_value f_value</i>}'} Defines the elements in the result grid matrix based on the outcome of the comparison operation. The values can be either a double-precision floating-point number or a pair of double-precision floating-point numbers.

Table 6-9. YieldServer CMP Grid Commands (cont.)

Command	Description
<pre>cmp::less_grid grid {-row -column -element} -limit limit_value -result_value {t_value '{t_value f_value}'}</pre>	<p>Returns a grid matrix with values <i>t_value</i> if the selected parameter for the corresponding element is less than the limit and <i>f_value</i> otherwise. By default, <i>f_value</i> is the corresponding value of the input grid.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • <i>grid</i> Specifies the grid object. • -row -column -element Defines the parameter of the grid used in the comparison operation. • -limit <i>limit_value</i> Defines the comparison value. • -result_value {<i>t_value</i> '{<i>t_value f_value</i>}'} Defines the elements in the result grid matrix based on the outcome of the comparison operation. The values can be either a double-precision floating-point number or a pair of double-precision floating-point numbers.
<pre>cmp::multiply_grid grid1 grid2</pre>	<p>Returns product grid of two grids. Each element of the result grid is a product of the corresponding elements of the input grids.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • <i>grid1</i> Specifies the first grid object. • <i>grid2</i> Specifies the second grid object. <p>Grid objects must have the same dimensions.</p> <p>One of the grid arguments can be a double-precision floating-point number.</p>
<pre>cmp::not_grid grid</pre>	<p>Returns the NOT grid of the specified grid. Each element of the result grid is the result of a logical NOT operation on the corresponding element of the input grid.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • <i>grid</i> Specifies the grid object.

Table 6-9. YieldServer CMP Grid Commands (cont.)

Command	Description
cmp::or_grid <i>grid1 grid2</i>	<p>Returns the OR grid of two grids. Each element of the result grid is the result of a logical OR operation on the corresponding elements of the input grids.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • <i>grid1</i> Specifies the first grid object. • <i>grid2</i> Specifies the second grid object. <p>Grid objects must have the same dimensions.</p> <p>One of the grid arguments can be a double-precision floating-point number.</p>
cmp::save_grid <i>grid</i> -layer <i>layer_name</i> -property <i>property_name</i>	<p>Saves grid elements to layer geometries as a property.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • <i>grid</i> Specifies the grid object. • -layer <i>layer_name</i> Specifies a layer name. • -property <i>property_name</i> Specifies a property name.
cmp::subtract_grid <i>grid1 grid2</i>	<p>Returns the difference grid of two grids. Each element of the result grid is a difference of the corresponding elements of the input grids.</p> <p>Arguments:</p> <ul style="list-style-type: none"> • <i>grid1</i> Specifies the first grid object. • <i>grid2</i> Specifies the second grid object. <p>Grid objects must have the same dimensions.</p> <p>One of the grid arguments can be a double-precision floating-point number.</p>

Calibre YieldServer Scripts for CMPAnalyzer

You can create a Calibre YieldServer script with CMP commands and commands in the dfm:: namespace.

A Calibre YieldServer script that operates on a DFM database created with Calibre CMPAnalyzer can include, in addition to the commands shown in [Table 6-8](#), any of the commands in the dfm:: namespace (refer to the [Calibre YieldServer Reference Manual](#) for a complete list of these commands).

When writing Calibre YieldServer scripts and user-defined hotspot scripts, all procedure names, rule names, and property names must use only alphanumeric characters, that is letters and numbers, with the underscore (_) being the only allowable non-alphanumeric character. Annotation and property string values may be any legal Tcl string.

The script can be executed following a CMPAnalyzer run by entering the following at the command line:

```
calibre -ys -cmp -exec script_file_name
```

Note that for this approach, you must include a dfm::open_db command in the script to open the DFM database.

Alternatively, you can run a Calibre YieldServer script automatically as part of a CMPAnalyzer run by adding the following lines to your runset file:

```
*dfmDFMUseYSScript: 1  
*dfmDFMYSScript: script_file_name
```

where *script_file_name* is the name of the Calibre YieldServer script file. It is also possible to run scripts by using “source *filename*” in the Calibre RVE for DFM terminal window.

Examples for CMP YieldServer Commands

Examples for use with Calibre CMPAnalyzer.

Also see “[Tcl Script for User-Defined Hotspots](#)” on page 69.

Example Script to Calculate User-Defined Hotspots	142
Example Script to Save Reports for Calibre CMPAnalyzer	144
Example to Export Values to a Text File.....	144

Example Script to Calculate User-Defined Hotspots

You can use CMP YieldServer commands in a Tcl procedures to calculate user-defined hotspot information.

The following example script uses CMP grid operations and certain properties to perform a thickness topology hotspot analysis in Calibre CMPAnalyzer. Adding the user-defined hotspot property as a DFM_METRIC annotation enables batch reporting for that property.

Note

 Calibre CMPAnalyzer analysis with **Save Vector Properties** enabled in the [Simulator Options Tab on the Inputs Pane](#) of Calibre Interactive is compatible with Tcl scripts for user-defined hotspots implemented with CMP grid operations. Iterator-based Tcl scripts are not compatible with the vector property format.

See also “[Tcl Script for User-Defined Hotspots](#)” on page 69 to calculate a hotspot based on a perimeter metric using basic contents and Tcl procedures.

```

proc thickness_topo { detail_layer } {

    ## Grid results can be stored in Tcl global variables for reuse
    global topo_grid
    set hotspot_rule "Thickness_Topo"

    ## Add detail layer annotation
    if {![regexp "Rule_Thickness_Topo" [dfm::list_annotation_values \
        -layer $detail_layer -annotation DFM_METRIC]]} {
        dfm::add_annotation -layer $detail_layer -annotation DFM_METRIC \
            -value "Rule_Thickness_Topo"
    }

    ## Always delete a property before attempting to add it
    catch {dfm::delete_property $detail_layer $hotspot_rule} msg
    dfm::add_property $detail_layer $hotspot_rule 0 -double -default

    ## Create a grid for each property of interest from the detail layer
    set cuHeight_grid [cmp::create_grid -layer $detail_layer \
        -property "Thickness_Z2"]
    set ildHeight_grid [cmp::create_grid -layer $detail_layer \
        -property "Thickness_Z1"]
    set density_grid [cmp::create_grid -layer $detail_layer \
        -property "Density"]

    ## Math operations on grids
    ## This example computes the same values as the default
    ## "Thickness_Topology" property
    set invDensity_grid [cmp::subtract_grid 1 $density_grid]
    set density_cuHeight_grid [cmp::multiply_grid $density_grid \
        $cuHeight_grid]
    set invDensity_ildHeight_grid [cmp::multiply_grid $invDensity_grid \
        $ildHeight_grid]

    ## Save the final result
    set topo_grid($detail_layer) [cmp::add_grid $density_cuHeight_grid \
        $invDensity_ildHeight_grid]
    cmp::save_grid $topo_grid($detail_layer) -layer $detail_layer \
        -property $hotspot_rule
}

proc topo_gradient { detail_layer threshold } {

    ## Grid results can be stored in Tcl global variables for reuse
    global topo_grid
    set hotspot_rule_name "Hotspot_Thickness_Topo"

    ## Always delete a property before attempting to add it
    catch {dfm::delete_property $detail_layer $hotspot_rule_name} msg
    dfm::add_property $detail_layer $hotspot_rule_name 0 -double -default

    ## Grid operations to compute a gradient and compare it to some
    ## threshold
    set diff_grid [cmp::gradient_grid $topo_grid($detail_layer) \
        -radius 1 -max]
    set result_grid [cmp::less_grid [cmp::subtract_grid $diff_grid \
        $threshold] -element -limit 0 -result_value 0]
}

```

```
## Save the result as a property with a value
cmp::save_grid $result_grid -layer $detail_layer \
    -property $hotspot_rule_name

## Hotspot grids can only have values 0, 1
## 0 means no hotspot, 1 is a hotspot
set hotspot_grid [cmp::greater_grid $result_grid -element \
    -limit 0 -result_value {1 0}]

## Save the result as a hotspot property
cmp::save_grid $hotspot_grid -layer $detail_layer \
    -property $hotspot_rule_name

## Calculate and return the number of failing tiles
set fail [expr int([cmp::get_grid_data $hotspot_grid -sum_elements])]
return $fail
}

## Register both procs for use in Calibre CMPAnalyzer
cmp::register_hotspot_proc -rule_name "Thickness_Topo" \
    -proc thickness_topo -output_properties "Thickness_Topo"
cmp::register_hotspot_threshold_proc -rule_name "Thickness_Topo" \
    -proc topo_gradient -values 50 -output_properties \
    "Hotspot_Thickness_Topo"
```

Example Script to Save Reports for Calibre CMPAnalyzer

A script can be used to save hotspot and Calibre xRC reports.

```
puts "\nSaving xRC Report..."
cmp::save_xrc_report -dir "xRC"
puts "\nSaving Hotspots Report..."
cmp::save_hotspots_report -file "Hotspots_Report.txt"
```

Example to Export Values to a Text File

You can use CMP YieldServer commands in a procedure to export Calibre CMPAnalyzer hotspot information to a text file.

The following procedure can be used to export **Dishing** values to a text file.

```

proc export_prop {prop_layer prop_name filename} {
    # write geometry vertices and property value to file
    set outfh [open $filename w+]
    set geos [dfm::get_geometries $prop_layer]
    set precision [dfm::get_db_precision]
    while { $geos ne "" } {
        set verts [dfm::get_data $geos -vertices]
        set bbx0 [expr [lindex $verts 0] / $precision]
        set bby0 [expr [lindex $verts 1] / $precision]
        set bbx1 [expr [lindex $verts 2] / $precision]
        set bby1 [expr [lindex $verts 3] / $precision]
        set value [dfm::get_data $geos -geometry_property $prop_name]
        puts "$bbx0 $bby0 $bbx1 $bby1 $value"
        puts $outfh "$bbx0 $bby0 $bbx1 $bby1 $value"
        dfm::inc geos
    }
    close $outfh
}

export_prop M1_data_detail Hotspot_Dishing metaladb.txt

```

This procedure first opens a file for writing, then writes the coordinates of the bounding boxes and property value for each geometry. Note that the procedure assumes that all geometries on the layer are rectangular.

Colormap Stipple Patterns

Explanation of the various stipple patterns that can appear on colormaps.

You can click on any stipple pattern in a colormap legend to toggle highlighting the stipple pattern tiles on the colormap and in the layout viewer.

Table 6-10. Stipple Patterns

Pattern	Name	Description
	Oxide Height Min	Tiles of minimum oxide height.
	Oxide Height Max	Tiles of maximum oxide height.
	Metal Height Min	Tiles of minimum metal height.
	Metal Height Max	Tiles of maximum metal height.
	Oxide and Metal Min	Tiles of minimum oxide and metal height.
	Oxide and Metal Max	Tiles of maximum oxide and metal height.

Layer Number Specification Format

Use the GUI **CMP** tab in the Calibre Interactive Inputs pane to enter layer information for your design. The layer numbers and datatypes must follow certain format specifications.

Note

 Optionally, you can specify user-defined layers in a Pre-Defined Layers File, which supports Calibre Boolean layer operations. See “[Pre-Defined Layers File Format](#)” on page 148 for the content and format of this file.

Layer Number and Datatype Format Specifications

Layer numbers and datatypes follow this format:

```
[ '-' | '~' ] layer_number.layer_datatype  
[, [ '-' | '~' ] layer_number.layer_datatype] ...
```

where:

- *layer_number* — Specifies the layer number.
- *layer_datatype* — Specifies the layer datatype.
- - | ~ — Optional argument set that indicates if the layer is a subtractive layer (-) or a negative layer (~). By default, the layer is a positive layer.

The following logic is applied to the final layer that is simulated:

```
result_layer = (OR_positive_layers OR (DB_EXTENT NOT AND_negative_layers))  
NOT OR_subtractive_layers
```

where:

<i>OR_positive_layers</i>	Result of OR operation between positive layers.
<i>AND_negative_layers</i>	Result of AND operation between negative layers.
<i>OR_subtractive_layers</i>	Result of OR operation between subtractive layers.

[Table 6-11](#) lists examples of values and how they translate to the final layer.

Table 6-11. Layer Number Specification Examples

Layer Number Value	Final Layer
31.0, 32.0	OR 31.0 32.0
31.0, -117.0	31.0 NOT 117.0
-32.0	DB_EXTENT NOT 32.0
-31.0, -32.0	DB_EXTENT NOT (OR 31.0 32.0)
10.1, ~11.2, -12.3	(10.1 OR (DB_EXTENT NOT 11.2)) NOT 12.3

Table 6-11. Layer Number Specification Examples (cont.)

Layer Number Value	Final Layer
10.1, ~11.2, ~11.3	10.1 OR (DB_EXTENT NOT (AND 11.2 11.3))

Also see “[CMP Analysis with the Calibre Interactive GUI](#)” on page 15.

Multiple-Step Layer Number Specifications

Multiple steps can be specified for a layer (including predefined layers) if the layer numbers are separated by a semicolon (;). For example, multiple etch steps can be specified with layers defined as *layer1;layer2; ...layern*. The number of layers defined must be the same as the number of etch steps in the recipe file and must appear in the same sequence. Similarly, you can define multiple-step layer specifications for pre-etching, selective deposit, etch-back selective, and etch-back layers. See “[CMP Tab on the Inputs Pane](#)” on page 107 in the Calibre Interactive GUI for more specification details.

Pre-Defined Layers File Format

Input for: Calibre Interactive **CMP** tab for a Calibre CMPAnalyzer run.

You can use a valid SVRF file in the Pre-Defined Layers File field to specify user-defined layers and Calibre layer operations for a CMP simulation and hotspot analysis.

Note

 See “CMP Analysis with the Calibre Interactive GUI” step 7 for file input field information.

Format

SVRF and Conventions

Layer definitions and operations in the Pre-Defined Layers File must conform to SVRF and the following conventions:

- Layers defined by the Calibre CMPAnalyzer Layer Name should not match any layer name defined in the Pre-Defined Layers File. If a match exists, an error message is generated.
- When all layers for analysis are specified in the Pre-Defined Layers File, the layout base layer (lowest simulation layer) must be defined with the LAYOUT BASE LAYER statement. If a layout base layer is not defined, a warning message is issued. You can specify this statement in the Pre-Defined Layers File, or alternatively, in Calibre Interactive as follows:
 - In the Calibre Interactive GUI, select **Setup > DFM Options**, select the **Include** tab, enable the checkbox “Include Rule Statements”, and enter the statement in the text field.
 - In the Calibre Interactive runset, use the “dfmSVRFCmds” parameter to specify the statement.
- User-defined layer numbers in the Pre-Defined Layers File must start from 3001 and be within the allowed range of CMP layer numbers from 3001 to 9999.
- Calibre supported Boolean layer operations can be used for defining layers for CMP.
- Calibre encryption formats created with `calsvrfencrypt` and `caltvfencrypt` commands are supported for SVRF and TVF rules, respectively.

Layer definition and operation syntax in the Pre-Defined Layers File is as follows:

```
LAYER name original_layer [original_layer ...]  
LAYER MAP source_layer DATATYPE source_type target_layer  
LAYOUT BASE LAYER name  
<layer derivation>
```

Parameters

- **LAYER *name* *original_layer* [*original_layer* ...]**
Specifies the input layer names and numbers.
 - ***name*** — Specifies the required name of an original layer or layer set.
 - ***original_layer*** — Specifies the required original layer or layer set.
where possible choices are as follows:
 - The layer number of an original layer within the allowed range of 3001 to 9999.
 - The name of a layer or layer set defined by another Layer operation.
- **LAYER MAP *source_layer* DATATYPE *source_type* *target_layer***
Specifies datatype maps from GDS or OASIS input to Calibre layer numbers. Multiple LAYER MAP statements can be used to map several layer and datatype combinations to a single layer number between 3001 and 9999.
 - ***source_layer*** — Specifies the required positive integer that is a layer in the layout database.
 - **DATATYPE** — Specifies the required keyword that instructs the tool to create a datatype map.
 - ***source_type*** — Specifies the required positive integer that is a particular datatype in the layout database.
 - ***target_layer*** — Specifies the required positive integer that is the layer number to be used by Calibre.
- **LAYOUT BASE LAYER *name***
Specifies the required base layer when all layers for analysis are in the Pre-Defined Layers File.
 - ***name*** — Specifies the required name of an original layer. For example,

```
LAYOUT BASE LAYER M1x
```

The requirement for the base layer is as follows:
 - The layer must be an original drawn layer, not a derived layer.
 - The layer must be at the lowest level of the hierarchy.Typically only one layer is needed for a LAYOUT BASE LAYER. For BEOL simulations, M1 is typically used. For FEOL simulations, the original Diffusion layer may be used.
- **<*layer derivation*>**
Specifies a valid SVRF Boolean layer operation that derives a layer for CMP.

Examples

This example uses the following Boolean layer operation to define a layer AXL for CMP:

```
((2.0 AND 2.2) NOT 2.4) OR 2.3
```

This is the SVRF file content for a sample Pre-Defined Layers File (*cmpa.svrf*) that defines the layer AXL:

```
LAYER AX      3001
LAYER AXFIN   3002
LAYER AXFILL  3003
LAYER AC      3004
LAYER MAP 2   DATATYPE 0  3001
LAYER MAP 2   DATATYPE 2  3002
LAYER MAP 2   DATATYPE 3  3003
LAYER MAP 2   DATATYPE 4  3004
AXL = ((AX AND AXFIN) NOT AC) OR AXFILL
```

Index

— Symbols —

[]¹³
{}¹³
|¹³

— B —

Batch (HTML) reporting, [121](#)
Bold words, [12](#)
Bucketing flow
 enabling, [82](#)
 overview, [81](#)
 results, [84](#)

— C —

Calibre Interactive, CMP tab, [107](#)
Calibre RVE, CMP tab, [118](#)
CMP analysis, batch mode, [21](#)
CMP analysis, running, [15](#)
CMP Tab
 in Calibre Interactive, [107](#)
 in Calibre RVE for DFM, [119](#)
CMPAnalyzer results, [25](#)
Colormaps
 creating, [28](#), [119](#)
 stipple patterns, [145](#)
 tooltips, [35](#)
Courier font, [13](#)
Custom hotspots, [69](#)

— D —

Depth of focus, defined, [101](#)
DFM Database, and CMPAnalyzer, [20](#)
Dishing, defined, [102](#)
Displaying colormaps, [119](#)
Displaying histograms, [119](#)
Displaying property views, [119](#)
Double pipes, [13](#)

— E —

Erosion, defined, [102](#)

Extraction tools, exporting CMP data to, [64](#)

— H —

Heavy font, [12](#)
Histograms, creating, [35](#)
Histograms, displaying, [119](#)
Hotspots
 defined, [102](#)
 recalculating, [42](#)
 user-defined, [69](#)
HTML reporting, [121](#)

— I —

Italic font, [12](#)

— L —

Layer number, specification in Calibre
 Interactive, [146](#)
Layer properties file in Calibre DESIGNrev, [16](#)

— M —

Minimum keyword, [13](#)

— P —

Parentheses, [13](#)
Pipes, [13](#)
Profile plot, [38](#)
Properties, viewing in Calibre RVE, [41](#)
Property views, displaying, [119](#)

— Q —

Quotation marks, [13](#)

— R —

Rerun flow
 enabling, [90](#)
 export extent, [94](#)
 extent settings, [95](#)
 new layers, [92](#)
 overview, [87](#)

— S —

- Scatter plots, [43](#)
- Simulated profile plots, [45](#)
- Simulator flow reference, [104](#)
- Slanted words, [12](#)
- Square parentheses, [13](#)
- Stipple patterns in colormaps, [145](#)

— T —

- Tile, defined, [103](#)
- Tooltip display, [37](#)
- Trench bottom, defined, [104](#)

— U —

- Underlined words, [13](#)

— Y —

- YieldServer commands, [124](#)

Third-Party Information

Details on open source and third-party software that may be included with this product are available in the `<your_software_installation_location>/legal` directory.

