

SIEMENS EDA

# Calibre® CMP Model Builder User's and Reference 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Unpublished work.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](http://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](http://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](http://support.sw.siemens.com)

Send Feedback on Documentation: [support.sw.siemens.com/doc\\_feedback\\_form](http://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

<b>Chapter 1</b>		
<b>CMP Modeling Overview.....</b>		<b>13</b>
Benefits of Modeling CMP .....		13
CMP Modeling Workflow .....		14
Key Concepts .....		16
CMP Modeling Requirements .....		17
To Run CMP Model Builder .....		18
Running CMP Model Builder in the GUI .....		18
Running CMP Model Builder from the Command Line .....		18
Syntax Conventions .....		19
<b>Chapter 2</b>		
<b>Getting Started with CMP Modeling .....</b>		<b>21</b>
Process Recipe Files .....		22
Creating a Process Recipe File .....		22
Process Recipe Guidelines .....		29
Process Recipe File Example .....		29
Measured Data (MD) File .....		34
Creating a Measured Data File .....		35
Importing Values to the MD File .....		38
Declaring Optimization Parameters for Calibration Runs (Optimize Tab) .....		42
Analyzing Results to Create the Best Model .....		44
Viewing ThickNT, Erosion, and Dishing Plots .....		47
Finishing the CMP Model .....		48
<b>Chapter 3</b>		
<b>CMP Model Builder in Batch Mode .....</b>		<b>51</b>
cmpoptimize .....		52
<b>Chapter 4</b>		
<b>CMP Modeling Reference .....</b>		<b>57</b>
Symbolic Names .....		57
Numeric Expressions in Parameters .....		58
Measured Data File Format .....		59
Process Recipe File Format .....		65
cmp .....		67
define_mask 1D .....		72
define_mask 2D .....		74
define_model .....		84
deposit .....		92

---

etch .....	102
initialize .....	110
save .....	114
<b>Chapter 5</b>	
<b>Environment Variable Reference .....</b>	<b>117</b>
CALIBRE_MGC_CMP_EXTRACTION_VERSION .....	118
CMP_ENABLE_OAF_LOG_DT .....	119
CMP_EXTR_LARGE_METAL_REGION .....	120
CMP_EXTRACT_POS_EXTRA_SPEEDUP .....	121
CMP_EXTRACTION_VERSION .....	122
CMP_OLD_EROSION .....	123
CMP_PD_MIN_RANGE_FACTOR .....	124
CMPOPTIMIZE_COMPATIBILITY .....	125
CMPOPTIMIZE_THICKNESSNT_MIN .....	126
MGCMP_EROSION .....	127
<b>Chapter 6</b>	
<b>CMP Model Builder GUI Reference.....</b>	<b>129</b>
CMP Model Builder Main Window .....	130
Measured Tab .....	134
Sort Dialog Box .....	136
LineScan Dialog Box .....	138
Recipe Tab .....	142
Initialize Dialog Box .....	144
Etch Dialog Box .....	147
Deposit Dialog Box .....	153
Cmp Dialog Box .....	158
Save Dialog Box .....	162
EcdModel Dialog Box .....	164
CmpModel Dialog Box .....	167
LookUpModel Dialog Box .....	172
Mask1D Dialog Box .....	173
Mask2D Dialog Box .....	175
Expression Dialog Box .....	178
Optimize Tab .....	179
Remote Tab .....	182
Grids Tab .....	183
<b>Appendix A</b>	
<b>Process Recipe File Examples .....</b>	<b>187</b>
File Examples .....	187
Optimizing Parameterized Expressions .....	193
Creating a Surface Plot .....	195
<b>Appendix B</b>	
<b>CMP Model Builder Components.....</b>	<b>197</b>
CMP Models .....	197

---

## Table of Contents

---

Layouts . . . . .	198
Measured Data Files (MD Files) . . . . .	199
Mask 1D and 2D . . . . .	200
Snapshots . . . . .	201
Grid Files . . . . .	202
Grid File Types . . . . .	203
Extracting Grids for Large Test Chips . . . . .	203
Grid Frames (Pixels) . . . . .	205
Optimized CMP Model File . . . . .	209

## Appendix C

Details on Theoretical Models . . . . .	211
---	-----

CMP Model . . . . .	211
Deposit Model . . . . .	215

## Glossary

## Index

## Third-Party Information

## Table of Contents

---

# List of Figures

---

Figure 1-1. Uneven Profile After a Deposition Step .....	14
Figure 1-2. Calibre and CMP Models .....	14
Figure 2-1. Initialize Values for the First Run.....	23
Figure 2-2. Deposition Values for the First Run .....	23
Figure 2-3. Method 1 .....	24
Figure 2-4. Method 1 (continued) .....	25
Figure 2-5. Method 2 .....	26
Figure 2-6. Method 2 (continued) .....	26
Figure 2-7. Electroplating .....	27
Figure 2-8. EcdModel .....	27
Figure 2-9. Cmp.....	28
Figure 2-10. CmpModel .....	28
Figure 2-11. Side By Side Cross Sectional Build Up .....	32
Figure 2-12. Side By Side Cross Sectional Build Up .....	33
Figure 2-13. Measured Data File in CMP Model Builder.....	36
Figure 2-14. Absolute and Relative Thickness .....	37
Figure 2-15. Measuring Erosion and Dishing .....	38
Figure 2-16. Results of Adjusted Line on a Line Scan Plot .....	40
Figure 2-17. Placement of Line Scan Boundaries on a Line Scan Plot.....	41
Figure 2-18. Results of Flip X on a Line Scan Plot.....	41
Figure 2-19. Setting Up an Optimization Run.....	43
Figure 2-20. Selecting a Log File to Analyze .....	45
Figure 2-21. Analyzing Simulated and Measured Plots .....	46
Figure 2-22. Viewing Plots .....	47
Figure 2-23. Plots .....	48
Figure 4-1. Mask1D.....	73
Figure 4-2. Specifications for Block versus Array .....	76
Figure 4-3. LineSpace Example.....	77
Figure 4-4. Example of Block Array Distribution.....	78
Figure 4-5. Example of Staggered Trench Distribution .....	79
Figure 4-6. Mask2D Example .....	81
Figure 4-7. ConvertTo1D Example .....	83
Figure 4-8. Isotropic Deposition .....	93
Figure 4-9. Anisotropic Deposition .....	93
Figure 4-10. Position Choices for Fill Deposition.....	94
Figure 4-11. Anisotropy Factor .....	94
Figure 4-12. Corner Bias .....	95
Figure 4-13. Deposition Rounding in Narrow Trenches .....	96
Figure 4-14. Sidewall Angle .....	98
Figure 4-15. Slope Angle.....	98

Figure 4-16. etch anisotropyFactor . . . . .	103
Figure 4-17. Oscillatory Conditions From Incorrect Extractions . . . . .	104
Figure 4-18. etch sidewallAngle . . . . .	107
Figure 4-19. layer_stack_topography 023 . . . . .	112
Figure 6-1. CMP Model Builder Window . . . . .	130
Figure 6-2. Measured Tab . . . . .	134
Figure 6-3. Sort Dialog Box (Order) . . . . .	136
Figure 6-4. Sort Dialog Box (Priority) . . . . .	137
Figure 6-5. LineScan Dialog Box (Basic) . . . . .	138
Figure 6-6. LineScan Dialog Box (Full) . . . . .	139
Figure 6-7. Recipe Tab . . . . .	142
Figure 6-8. Initialize Dialog Box (Full) . . . . .	144
Figure 6-9. Etch Dialog Box (Basic) . . . . .	147
Figure 6-10. Etch Dialog Box (Full) . . . . .	148
Figure 6-11. Deposit Dialog Box (Initial View) . . . . .	153
Figure 6-12. Cmp Dialog Box (Basic) . . . . .	158
Figure 6-13. Cmp Dialog Box (Full) . . . . .	159
Figure 6-14. Save Dialog Box (Full) . . . . .	162
Figure 6-15. EcdModel Dialog Box (Basic) . . . . .	164
Figure 6-16. EcdModel Dialog Box (Full) . . . . .	165
Figure 6-17. CmpModel Dialog Box (Basic) . . . . .	167
Figure 6-18. CmpModel Dialog Box (Full) . . . . .	168
Figure 6-19. LookUpModel Dialog Box . . . . .	172
Figure 6-20. Mask1D Dialog Box . . . . .	173
Figure 6-21. Mask2D Dialog Box (Basic) . . . . .	175
Figure 6-22. Details of Mask 2D to 1D Conversion . . . . .	176
Figure 6-23. Expression Dialog Box . . . . .	178
Figure 6-24. Optimize Tab . . . . .	179
Figure 6-25. Remote Tab . . . . .	182
Figure 6-26. Grids Tab . . . . .	183
Figure A-1. Expression-Only Recipe . . . . .	194
Figure B-1. Test Chip General Structure . . . . .	198
Figure B-2. Cross Section View of Aluminum RMG Technology . . . . .	199
Figure B-3. Measured Data File in CMP Model Builder . . . . .	200
Figure B-4. Mask1D . . . . .	201
Figure B-5. Mask2D Example . . . . .	201
Figure B-6. Specifying Grid File Outputs From Simulations . . . . .	202
Figure B-7. Viewing Vertical Profiles and 3D Profiles . . . . .	202
Figure B-8. Grid Overlaid On Design . . . . .	206
Figure B-9. Effective Trench Values . . . . .	207
Figure B-10. Exposed Material in Multilayer Stack . . . . .	208
Figure C-1. Parameters for Perimeter Model in CmpModel Dialog Box . . . . .	212
Figure C-2. Same Density, Different Perimeters . . . . .	214

# List of Tables

---

Table 1-1. CMP Components Quick Reference .....	16
Table 1-2. Syntax Conventions .....	19
Table 2-1. Required Initial Column Values for MD File .....	34
Table 2-2. Calibre CMPAnalyzer Settings .....	50
Table 4-1. Functions Supported for Expressions .....	58
Table 4-2. Default Units of Measure .....	65
Table 5-1. Environment Variables for CMP Modeling .....	117
Table 6-1. CMP Model Builder Global Contents .....	131
Table 6-2. Session Menu .....	132
Table 6-3. Measured Menu .....	132
Table 6-4. Remote Menu .....	133
Table 6-5. Measured Tab Contents .....	134
Table 6-6. Sort Dialog Box Contents .....	137
Table 6-7. LineScan Dialog Box (Basic) Contents .....	139
Table 6-8. LineScan Dialog Box (Full) Contents .....	140
Table 6-9. Recipe Tab Buttons .....	142
Table 6-10. Initialize Dialog Box Contents .....	144
Table 6-11. Etch Dialog Box Contents (Basic) .....	148
Table 6-12. Etch Dialog Box Contents (Full) .....	149
Table 6-13. Deposit Dialog Box Contents .....	153
Table 6-14. Cmp Dialog Box Contents (Basic) .....	159
Table 6-15. Cmp Dialog Box Contents (Full) .....	160
Table 6-16. Save Dialog Box Contents .....	162
Table 6-17. EcdModel Dialog Box Contents (Basic) .....	165
Table 6-18. EcdModel Dialog Box Contents (Full) .....	166
Table 6-19. CmpModel Dialog Box Contents (Basic) .....	168
Table 6-20. CmpModel Dialog Box Contents (Full) .....	169
Table 6-21. LookUpModel Dialog Box Contents .....	172
Table 6-22. Mask1D Dialog Box Contents .....	173
Table 6-23. Mask2D Dialog Box Contents .....	175
Table 6-24. Expression Dialog Box Contents .....	178
Table 6-25. Optimize Tab Contents .....	179
Table 6-26. Additional Optimize Tab Contents .....	180
Table 6-27. Remote Tab Contents .....	182
Table 6-28. Grids Tab Contents .....	183
Table 6-29. Additional Grids Tab Contents .....	184
Table B-1. Calibre CMPAnalyzer Settings .....	204
Table B-2. CMPAnalyzer to CMP Model Builder Conversion .....	205
Table B-3. Recipe File Statements and Material Indexes. ....	208
Table C-1. Perimeter Model Fields .....	212



# Chapter 1

## CMP Modeling Overview

---

A chemical mechanical polishing (CMP) model is a model used to predict chip surface topography (heights or positions) for layouts and test structures at various stages in the manufacturing process. The primary use for this model is to predict erosion and dishing effects after CMP. A CMP model is also known as a process recipe file.

<b>Benefits of Modeling CMP .....</b>	<b>13</b>
<b>CMP Modeling Workflow.....</b>	<b>14</b>
<b>Key Concepts .....</b>	<b>16</b>
<b>CMP Modeling Requirements .....</b>	<b>17</b>
<b>To Run CMP Model Builder.....</b>	<b>18</b>
Running CMP Model Builder in the GUI.....	18
Running CMP Model Builder from the Command Line .....	18
<b>Syntax Conventions .....</b>	<b>19</b>

## Benefits of Modeling CMP

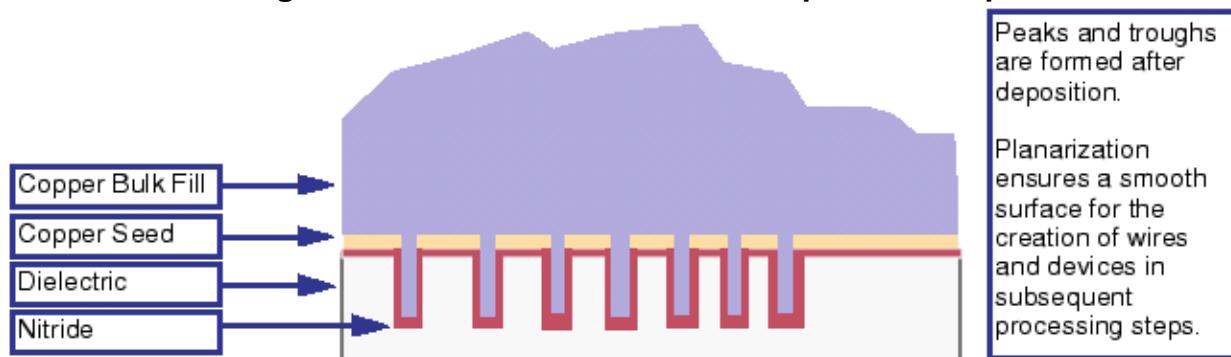
Planarization, or achieving a flat surface, is the physical goal of CMP. CMP modeling allows design and manufacturing teams to find potential planarization issues before manufacturing.

Planarization during manufacturing is critical for uniformity and improvement of depth of focus in the manufacturing process. A rough surface topography affects successive layers that are manufactured. (See [Figure 1-1](#).)

Using a CMP model aids in predicting uniformity and dishing effects for devices at or below 65nm. A CMP model coupled with Calibre® YieldAnalyzer and Calibre® YieldServer provides a platform for improving yield and predicting the effects of manufacturing. CMP modeling is widely used for hotspot detection in shallow trench isolation (STI) and back end of line (BEOL) layers as part of a design for manufacturing flow.

Adding a third dimension to simulations allows designers to more accurately predict and account for effects that impact thickness and thickness variations such as those due to erosion and dishing. Accounting for the variation of the thickness of wires increases the accuracy of circuit timing analysis.

**Figure 1-1. Uneven Profile After a Deposition Step**



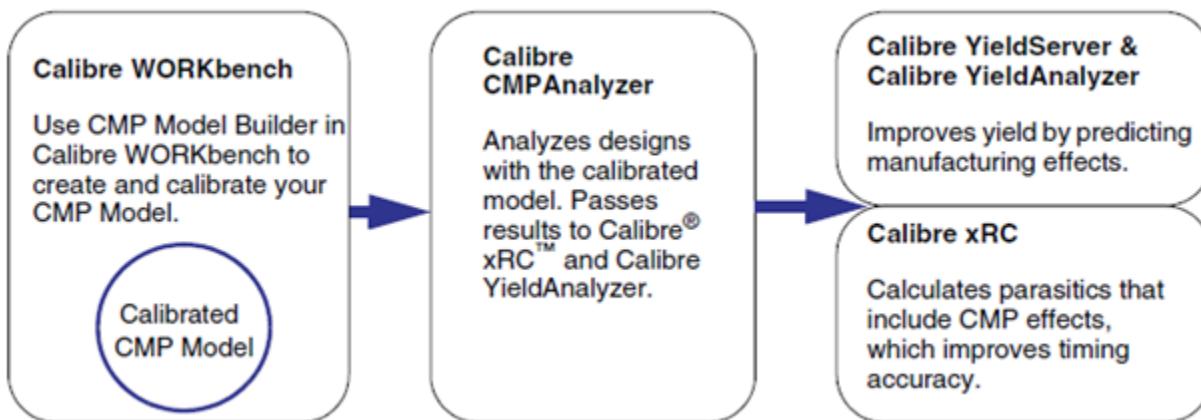
Critical dimensions of integrated circuits continuously shrink for each new technology node, imposing more demanding requirements on wafer planarity and depth of focus (DOF) limits. For example, when gate-last high-k metal gate (HKMG) technology was introduced at the 45 nm technology node, two new CMP steps were introduced in the front end of line (FEOL) process — poly open planarization (POP) and replacement metal gate (RMG). At 20 nm and below, these two CMP steps are especially critical in forming the gates of transistors, because variation in gate height of just a few atomic layers leads to measurable transistor performance variability. The increasingly high cost of lithography due to multi-patterning, combined with more demanding DOF requirements and the increased criticality of the HKMG CMP steps, have emphasized the importance of detecting planarization issues before manufacture.

## CMP Modeling Workflow

CMP models are created by Calibre® CMP Model Builder (a tool available in the Calibre® WORKbench™ platform) as described in this manual. Calibrated CMP models are used along with your design in Calibre® CMPAnalyzer to predict problems such as hot spots.

Figure 1-2 shows how CMP modeling fits into the Calibre tool flow. CMP models are required for running Calibre CMPAnalyzer. They take the form of a calibrated process recipe file and define the process steps used to fabricate a chip.

**Figure 1-2. Calibre and CMP Models**



CMP models are created from a combination of process data and measurements from actual silicon. The process data includes items such as layer stack thicknesses, pressure used in polishing, polishing rates, slurry selectivity, time, and end-point detection (EDP) stop conditions. These are combined with wafer measurements of erosion, dishing, and layer thickness obtained from TEM or SEM images.

Calibrating a CMP model is a five step process:

1. Create a layout for a test chip to be used for both measurement and simulation.

---

**Tip**

 The test chip will be the most accurate for creating models if it is the only chip on the wafer. If this is not possible, the layout should be such that the other chips on the die do not affect the test structures.

---

Test chips play a critical role in developing an accurate CMP model. The test structures should combine a wide range of density, width, and spacing values.

2. Gather thickness measurements from test wafers that use the test structures created in step 1.

Model calibration relies on thickness measurements and profile scans taken at different stages throughout the fabrication process. At a minimum, for BEOL CMP models you must take measurements after ECD and CMP steps. For FEOL CMP models it is helpful to also take measurements after deposition, as CMP models can also include modeling deposition to generate the profile before polishing.

Measurement data must be collected in two types of areas on a test wafer:

- **At Test Structures** — Provides erosion and dishing data
- **In Field Regions** — Provides absolute thickness data

---

**Tip**

 If you see a disagreement between your measured data and the process engineer's expectations, repeat the measurements. Calibration and simulation are not going to correct the difference if the measured data is bad, or if the test structures were biased by other designs on the die.

---

3. Create an initial process recipe file that characterizes your manufacturing flow. The CMP Model Builder tool provides a graphical user interface for creating the initial model and calibration.
4. Remove outliers from the measurement data. Particularly for 28 nm and below, the level of noise in the measurements may be of the same order of magnitude as the erosion and dishing signals. Outliers can lead to large calibration errors and a bad model fit.
5. Optimize the CMP model (process recipe file) to fit test measurements obtained from your process parameters.

CMP Model Builder uses parameterized equations to represent electrochemical deposition (ECD) and CMP. Developing an accurate CMP model involves calibrating the models for both ECD and CMP to measured line scans, copper thicknesses, and oxides for wafers with exposed test structures.

After calibrating the model, use it in Calibre CMPAnalyzer to calculate the surface topography of your designs.

Note that if your manufacturing process changes, you will need to update your CMP models.

## Related Topics

[Getting Started with CMP Modeling](#)

# Key Concepts

Before you begin using CMP Modeling, you need to know some basic types of objects you will be working with.

These are summarized in [Table 1-1](#). See also “[CMP Model Builder Components](#)” on page 197.

**Table 1-1. CMP Components Quick Reference**

Component	Information
Grid	A unit area simulated by CMP Model Builder. The size of a grid is also called a “pixel”. The area of a grid is also called a “frame”.
Layout	Flat GDS or OASIS® <sup>1</sup> file; can be created by CMP Model Builder.
Mask	Automatically-created test structure that contains blocks of trenches (“arrays”) separated by low-density fields. Used to create a test chip.
Measured data file (MD file)	An ASCII file with pre-defined columns that contains information about simulated and measured thickness, dishing, and erosion for your test data.
Optimized model	A process recipe file that has been calibrated. Parameters for each step represent the best-fit data after analyzing simulations.
Process recipe file	Text file that describes your manufacturing process, including polishing steps, by means of CMP Model Builder commands. Can include variables for parameters that you want to calibrate.
Snapshot	A set of files representing the current values of intermediate stages. Snapshots save time when simulating stages beyond the first stage.

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.

## CMP Modeling Requirements

Before using CMP Modeling, you need to have a number of required files and parameters in order to create a fully-functional model.

You need the following items in order to create a CMP model:

- **Calibre WORKbench and CMP Modeling Licenses** — For more information on licensing, refer to the *Calibre Administrator's Guide*.
- **Test Chip** — You need to plan a test structure and physically manufacture it to take measurements of thickness after the stages you are modeling. You need at least one test chip for each metal layer you intend to model.
- **Manufacturing and CMP Process Parameters** — CMP Model Builder runs simulations that reveal the best fit for process parameters. The optimization proceeds more quickly when you supply basic known values such as pad pressure and polish time.
- **Measured Data File** — The measured data file contains coordinate locations and the results of physical measurements on the test chip.
- **Line Scan Data** — The line scan data shows surface height of a particular layer for the same coordinates that are in the measured data file. It is used to extract dishing and erosion values. The line scan data can be generated from measurements taken with an atomic force microscope (AFM).

---

### Tip

 If your test structures include trenches with a high aspect ratio (that is, they are narrow for their depth), check measurements carefully. When an AFM tip is moving too rapidly or its tip is too large, the bottom of the trench is not correctly detected. You may need to measure these structures with SEM equipment instead.

---

## To Run CMP Model Builder

You can run CMP Model Builder either in the Calibre WORKbench GUI (recommended) or as a batch command. The GUI provides fields to produce the initial process recipe file and measured data file. The CMP Model Builder batch method is suitable for running several slightly different simulations after the initial setup.

Running CMP Model Builder in the GUI .....	18
Running CMP Model Builder from the Command Line .....	18

## Running CMP Model Builder in the GUI

The Calibre WORKbench GUI hosts the CMP Model Builder interface. The GUI prompts you for required values when setting up measurements and CMP recipes.

### Prerequisites

- Set the CALIBRE\_HOME environment variable. See “[Setting the CALIBRE\\_HOME Environment Variable](#)” in the *Calibre Administrator’s Guide* for details.
- A license for Calibre WORKbench.

### Procedure

1. Start Calibre WORKbench by typing the following command at the shell prompt:  
`calibrewb`
2. Select **Tools > CMP Model Builder** to start the CMP modeling application.
3. After setting up parameters, click **Run**.

### Related Topics

[Getting Started with CMP Modeling](#)

## Running CMP Model Builder from the Command Line

The CMP Model Builder batch method is suitable for running several slightly different simulations after the initial setup.

### Prerequisites

- Set the CALIBRE\_HOME environment variable. See “[Setting the CALIBRE\\_HOME Environment Variable](#)” in the *Calibre Administrator’s Guide* for details.
- Process recipe file.
- Measured data file.

## Procedure

1. Create a Tcl script that calls the `cmpoptimize` command. See “Examples” in the [cmpoptimize](#) reference.
2. Run the script by calling Calibre WORKbench:

```
calibrewb optimize_script.tcl
```

## Related Topics

[CMP Model Builder in Batch Mode](#)

# Syntax Conventions

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

**Table 1-2. Syntax Conventions**

Convention	Description
<b>Bold</b>	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.
<u>Underline</u>	Underlining indicates either the default argument or the default value of an argument.
[ ]	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.



# Chapter 2

## Getting Started with CMP Modeling

---

A CMP model is also known as a process recipe file. After creating the CMP model, creating a measured data file is required for optimizing the model.

<b>Process Recipe Files</b> .....	<b>22</b>
Creating a Process Recipe File .....	22
Process Recipe Guidelines .....	29
Process Recipe File Example .....	29
<b>Measured Data (MD) File</b> .....	<b>34</b>
<b>Creating a Measured Data File</b> .....	<b>35</b>
<b>Importing Values to the MD File</b> .....	<b>38</b>
<b>Declaring Optimization Parameters for Calibration Runs (Optimize Tab)</b> .....	<b>42</b>
<b>Analyzing Results to Create the Best Model</b> .....	<b>44</b>
<b>Viewing ThickNT, Erosion, and Dishing Plots</b> .....	<b>47</b>
<b>Finishing the CMP Model</b> .....	<b>48</b>

# Process Recipe Files

Process recipe files provide the initial information for optimizing a CMP model. Use the Calibre WORKbench CMP Model Builder tool to create a process recipe file.

<b>Creating a Process Recipe File .....</b>	<b>22</b>
<b>Process Recipe Guidelines.....</b>	<b>29</b>
<b>Process Recipe File Example .....</b>	<b>29</b>

## Creating a Process Recipe File

You can create a process recipe file in Calibre WORKbench based on the information about the manufacturing process. This method is recommended as the GUI checks each command after entry.

### Prerequisites

- Calibre WORKbench and Calibre CMP Model Builder licenses.
- Information about the manufacturing process such as materials, thicknesses, and process times.

### Procedure

1. Start Calibre WORKbench at the command prompt with the following command:

```
calibrewb
```

2. If you have a test chip layout, load it using **File > Open Layout Files**.

Optimization only runs on flat layouts. If the layout is hierarchical, you will be prompted to flatten it before running actual simulation.

3. In Calibre WORKbench, go to **Tools > CMP Model Builder** to display the CMP Model Builder window.
4. Select the **Recipe** tab to start building the process recipe file.
5. Define the layout window to be used for simulations with the initialize command.
  - a. In the **Recipe** tab, click **Add** to show a dropdown menu. Select **initialize**. The Initialize dialog box appears. (See [Figure 2-1](#).)
  - b. In the dialog box, specify the material for analysis. You can type a material or use the scroll arrows to the right of the field; they cycle through typical layer types.
  - c. Specify the initial position of the layer stack. The default is 0 angstroms.
  - d. To use only a portion of the layout for simulation and test measurements, draw a selection rectangle around the region in the Calibre WORKbench main window. In

the Initialize dialog box, click **Use Selection**. The dialog box adds fields showing the X and Y positions. These values are filled into the process recipe file.

- e. Click **OK** to create the initialize command in your process recipe file.

**Figure 2-1. Initialize Values for the First Run**



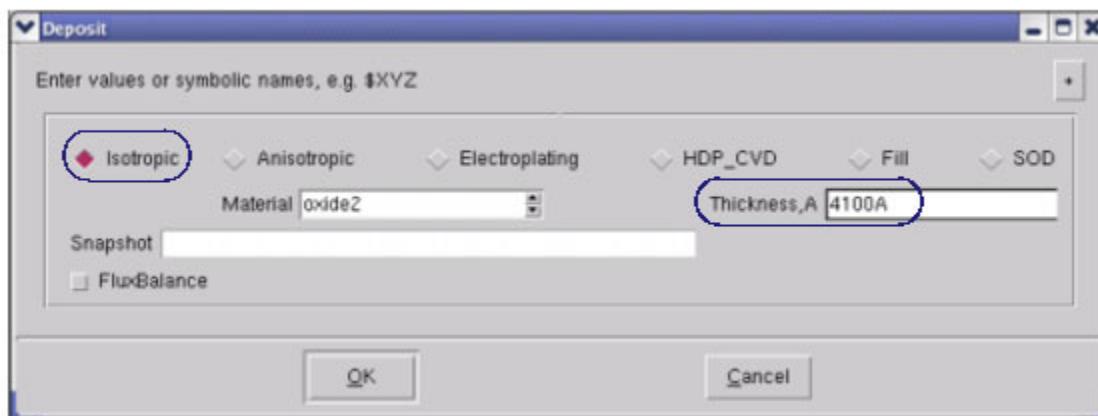
6. Begin building the recipe file with your parameters used in manufacturing.

Click the **Add** button and select **deposit** to add layers of dielectric and nitride. Input the thickness and material names and the deposition method used in your manufacturing flow. Do not use the same material name as was used in the initialize command.

**Tip**

**i** You can specify any name for a deposited or etched layer in your fabrication process. Be sure to use the same material name in subsequent steps that affect the layer or the simulator may return incorrect values. This is one means of protecting your process-related intellectual property.

**Figure 2-2. Deposition Values for the First Run**



7. Define etch steps.

Click the **Add** button and select **etch** to define the etching process step. Define the type of etch (isotropic or anisotropic), material, and the etch depth (Thickness field) used in your flow.

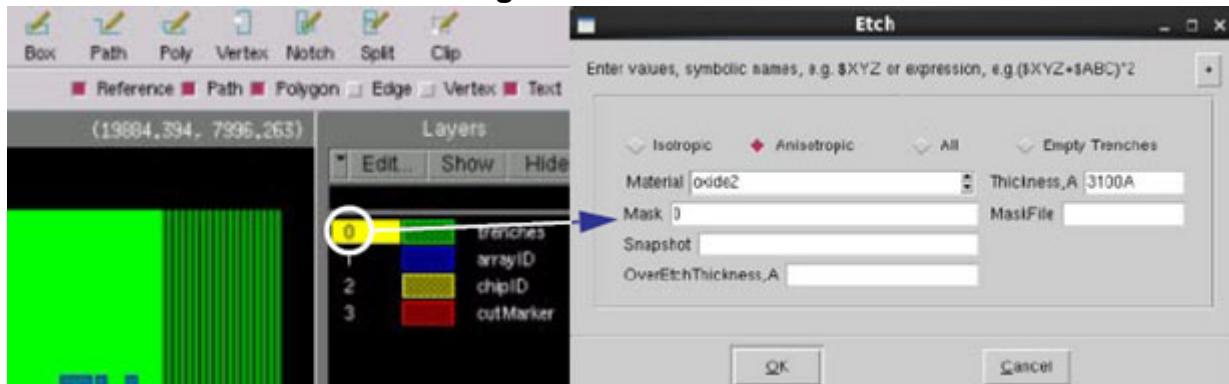
At this point you have two options on how to proceed. Use method 1 if you have your own layout that you want to use for simulation and if you already have thickness measurements for your chip. Use method 2 if you do not have a test chip and want to define one in CMP Model Builder with the `define_mask` command.

- **Method 1:** Using your own layout for simulation and measurements

This method assumes that you have test chip measurement data (thickness, dishing, and other data) for the layout. This allows you to see exactly how CMP and other processes factor into how devices are formed.

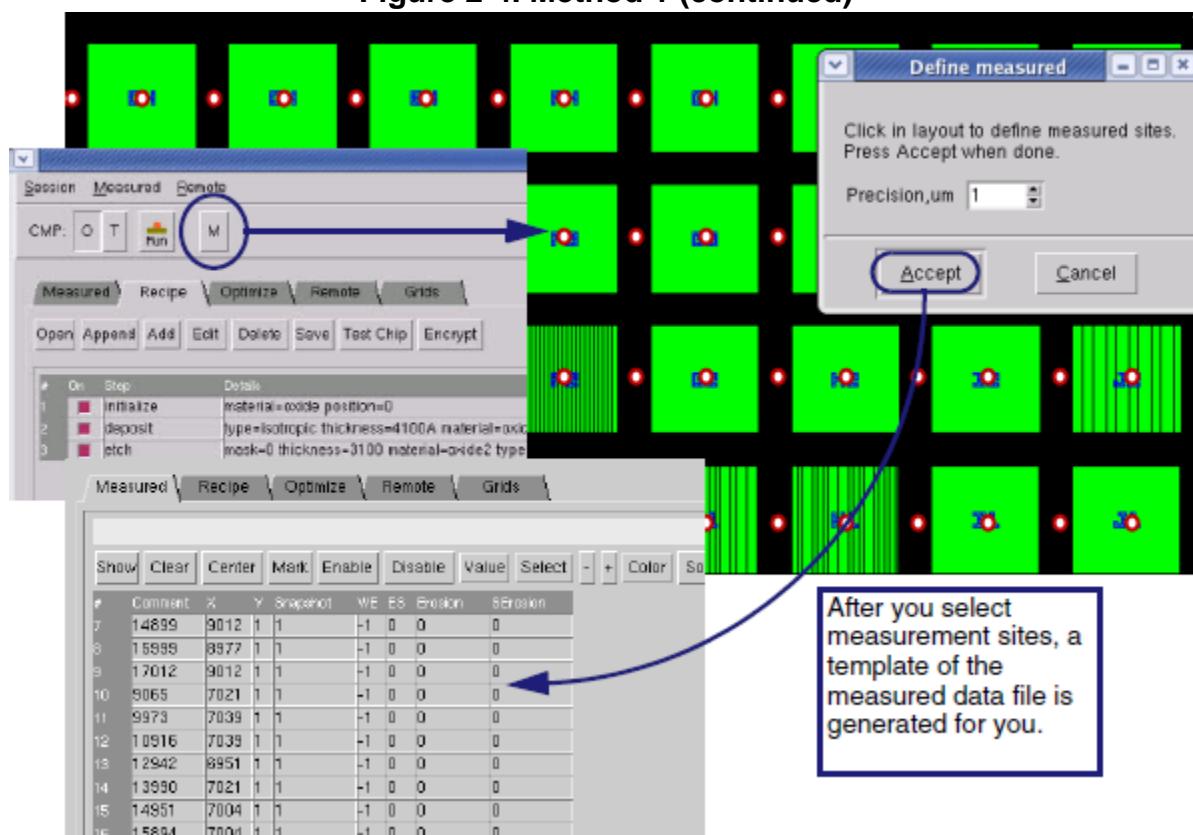
- i. Specify the layer in your own layout to simulate. Enter this layer number in the Mask field.

**Figure 2-3. Method 1**



- ii. In Calibre WORKbench, select the measurement sites to generate the measured data file. Place sites on both areas with shapes and those with no shapes close by. (You can adjust positions later.)

**Figure 2-4. Method 1 (continued)**



A template for your measured values is created in the **Measured** tab.

- **Method 2:** Using the `define_mask` command to generate a mask

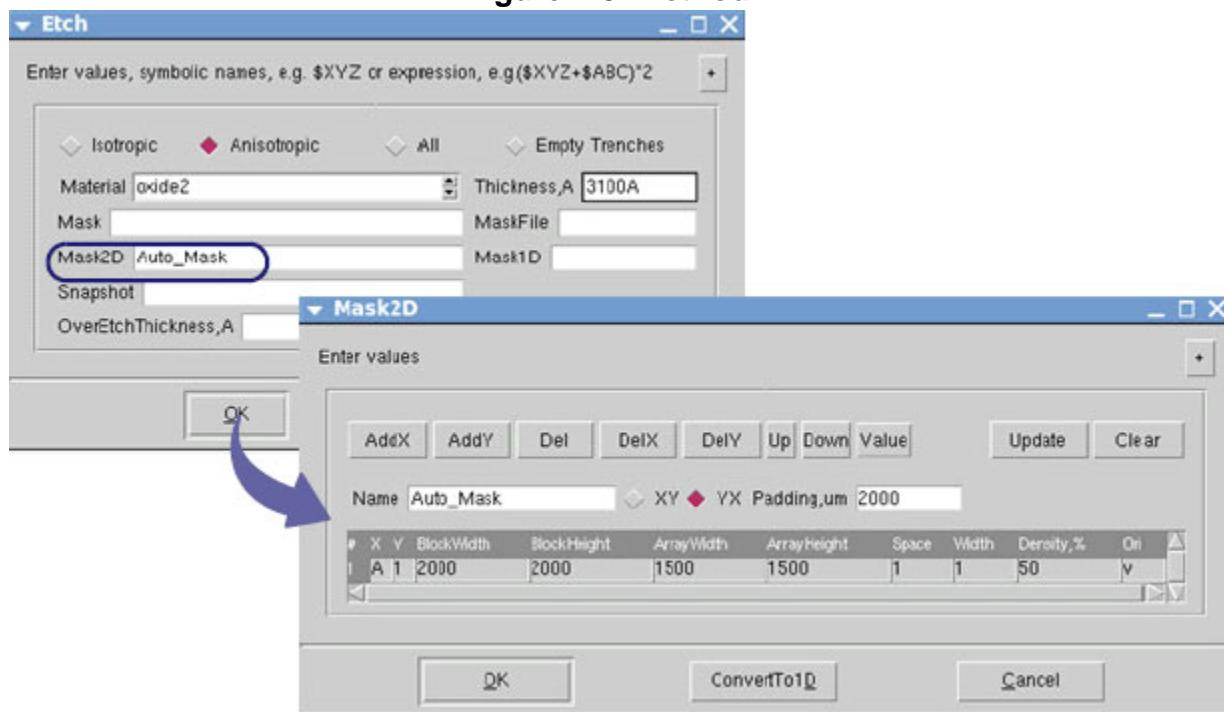
The `define_mask` command is used in CMP Model Builder to create test structures that will be produced in your fabrication facility. Taking measurements for the measured data file is required to complete this flow.

- i. Enter a Mask1D or Mask2D name for your new set of test patterns ([Figure 2-5](#)). This will be created for you in Calibre WORKbench. The Mask1D or Mask2D dialog box automatically appears when you click **OK** in the Etch dialog box.
- ii. Specify your spacing, width, and density values for the test pattern. Click **OK**.

The Block width and height should be significantly larger than the Array width and height to allow for a non-trench, or field, region.

To learn more about the differences between Mask1D and Mask2D, see “[define\\_mask 1D](#)” on page 72 and “[define\\_mask 2D](#)” on page 74.

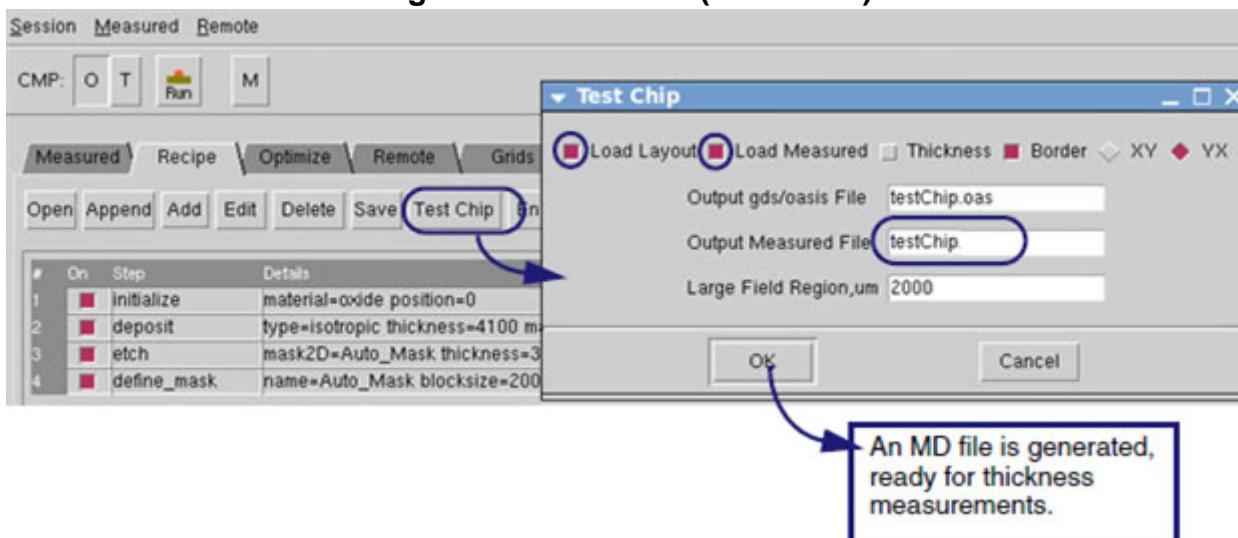
**Figure 2-5. Method 2**



- On the **Recipe** tab in the CMP Model Builder window, click **Test Chip**.

The Test Chip dialog box (Figure 2-6) streamlines the process of creating a measured data file by creating a template for you. Define the output and the name of the new measured data file (Output Measured File field).

**Figure 2-6. Method 2 (continued)**



- Define deposition of barrier layer and copper seed layer.

Click the **Add** button and select **deposit** to define deposition of the barrier layer. Define the thickness, material name, and type of deposition for your manufacturing flow.

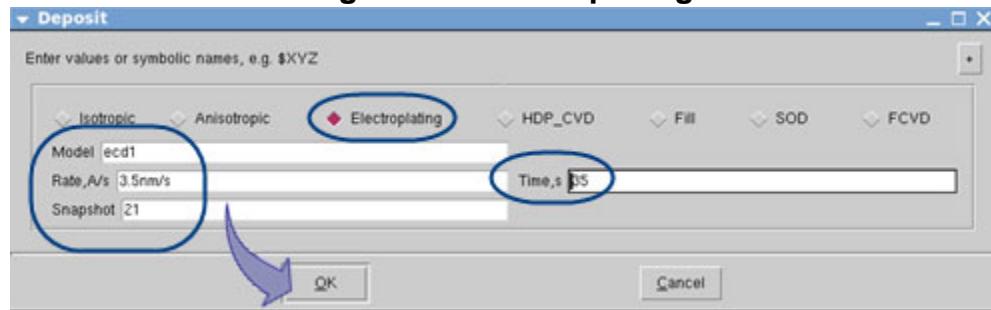
- Define ECD steps for copper bulk layer formation.

Click the **Add** button and select **deposit**. In the Deposit dialog box, select Electroplating. (See [Figure 2-7](#).) Enter a model name. After clicking **OK**, the EcdModel dialog box appears for you to declare the ECD model parameters. (See [Figure 2-8](#).)

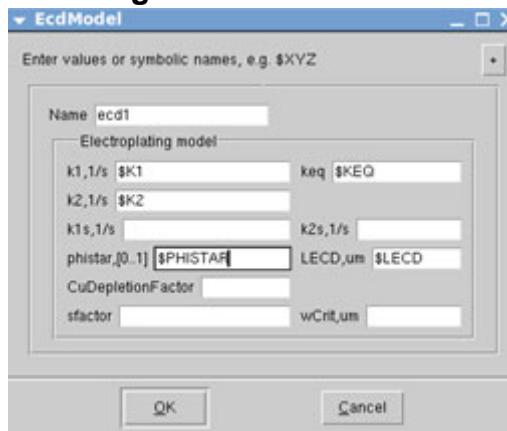
The Snapshot field is filled in automatically if measured data is available at the end of this step. Snapshots are used to identify the process step with the correct data set. If your measured data file has 100 thickness values for electroplating, they are synchronized with this ECD command for tracking purposes.

Clicking **OK** finishes the ECD model. It appears in the process recipe file as `define_model`.

**Figure 2-7. Electroplating**



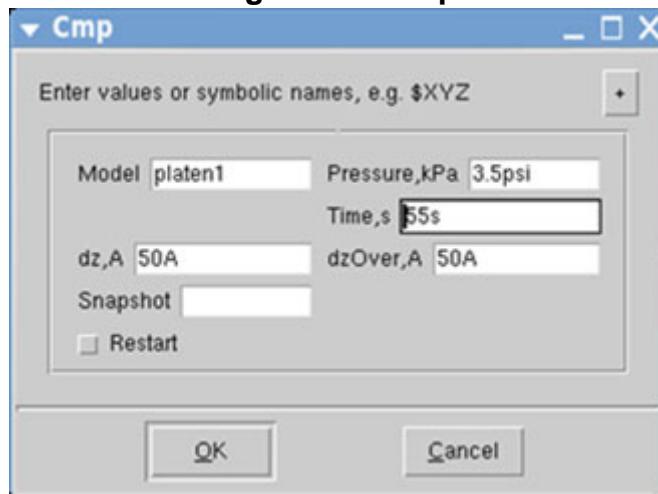
**Figure 2-8. EcdModel**



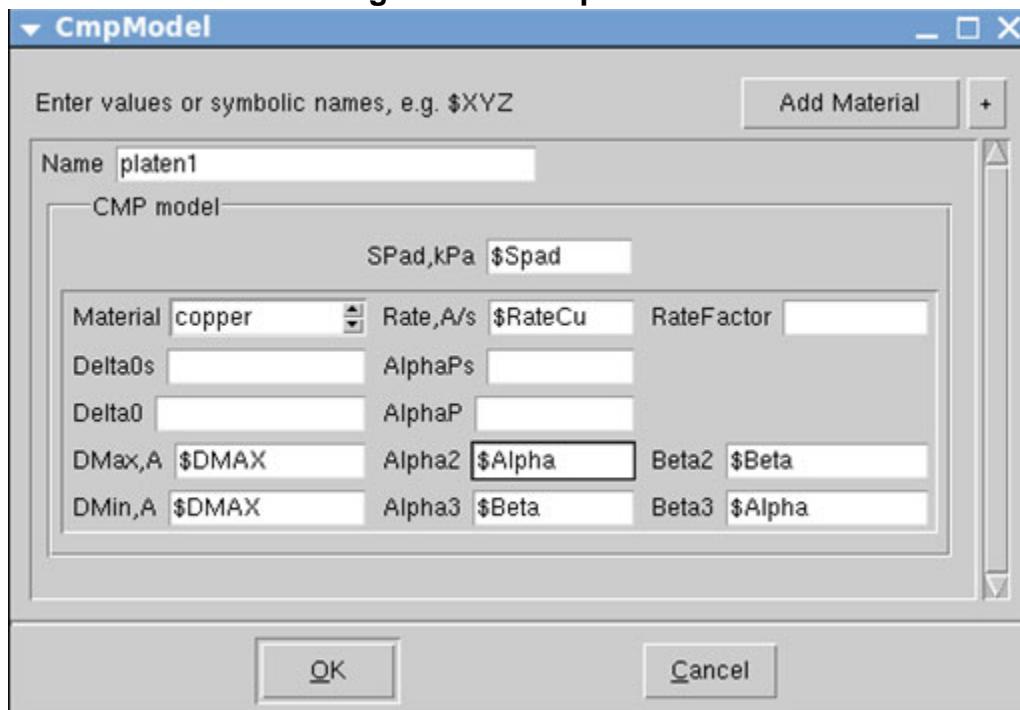
- Now do the same (define models and call them) for CMP. Each polishing stage, or platen, must have its own model definition. For three platens, use three CMP calls and three `define_model` commands. The “[Process Recipe File Example](#)” on page 29 has examples of model declarations for platen 1 (bulk copper removal), platen 2 (copper clear/soft landing), and platen 3 (TaN barrier removal and overpolish).

- a. To define the CMP model, click the **Add** button and select **cmpModel**. You must specify the model name and material. The other fields you plan to simulate should have symbolic names. (For example, SPad as \$SPAD rather than a number.)
- b. To call the CMP model, click the **Add** button and select **cmp**. For the Model field, provide the name given in the CmpModel dialog box.

**Figure 2-9. Cmp**



**Figure 2-10. CmpModel**



11. Save the session file (**Session > Save**) to use it in subsequent steps.

## Process Recipe Guidelines

When creating process recipes you need to be aware of the default units for the values you specify.

### Guidelines

- Thickness specifications are in angstroms by default. You can specify other units in the field.
- Layout coordinate specifications are in microns.
- Polishing rate specifications are in angstroms per second by default.

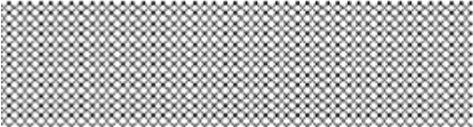
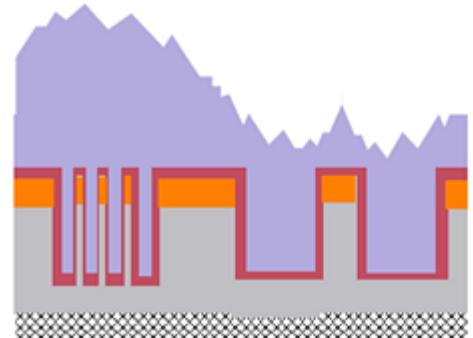
## Process Recipe File Example

This example uses a 2D mask and six measurement sites. There are three CMP models, one for each platen. After the recipe is an illustration showing how the keywords map to actual manufacturing steps.

```
#  
# Process recipe file example with a 2D mask used to  
# define the test chip with 6 sites. The test chip gds file and the  
# measured data file are derived using the 2D mask definition.  
#  
initialize x1=300 y1=300 x2=2200 y2=1700 material=nitride  
deposit thickness=4100A material=oxide type=isotropic  
etch mask2D=mask thickness=3100A material=oxide type=anisotropic  
deposit isotropic material=TaN thickness=100A  
deposit isotropic material=copper thickness=150A  
#  
# initial layer stack:  
# nitride at 0  
# oxide at 4100A ==> trench bottom at 1000A  
# TaN at 4200A  
# Cu seed at 4350A  
#  
# Use different snapshot values to identify data after ECD platen 1  
# platen2 and platen 3 (final result after first layer of metal)  
#  
# electroplating  
deposit model=ecd1 rate=3.5nm/s time=35s dt=0.1s saveDt=5s type=ecd  
deposit model=ecd1 rate=18nm/s time=$TIME dt=0.1s Snapshot=21 type=ecd  
save dishing erosion z1 z2 z3 file=postECD  
#  
# bulk copper removal: platen1  
cmp time=55s dz=5A model=cmp1 pressure=3.5psi Snapshot=22  
save dishing erosion z1 z2 file=postP1  
save state file=optimizedP1  
#  
# platen2: soft landing  
initialize grid=optimizedP1  
cmp time=70s dz=0.5A model=cmp2 pressure=1.5psi Snapshot=23 restart  
save dishing erosion z1 z2 file=postP2  
save state file=optimizedP2  
#  
# platen 3: barrier hard mask and oxide polish  
initialize grid=optimizedP2  
cmp time=60s dz=0.1A model=cmp3 pressure=2.5psi Snapshot=24  
save dishing erosion thickness z1 z2 file=final  
save state file=optimizedP3  
#  
===== Model Definitions =====  
#  
define_model name=ecd1 type=ecd k1=$K1 k2=$K2 k1s=$K1S k2s=$K2S \  
    phistar=$PHISTAR keq=$KEQ rmod=0 smod=0 LECD=$LECD  
#  
# platen 1  
#  
define_model name=cmp1 type=cmp spad=$SPAD material=copper rate=$RATE \  
    Hex=$HEX1 alphal=$ALPHA1 betal=$BETA1 s1=100um  
#  
# platen 2  
#  
define_model name=cmp2 type=cmp spad=$SPAD2 material=copper \  
    rate=$RATE2 material=TaN rate=$RATE2TAN dmax=$DMAX alpha2=$ALPHA2 \  
    beta2=$BETA2 s2=100um dmin=$DMAX alpha3=$ALPHA2 beta3=$BETA2 s3=100um \  
    material=oxide rate=1 dmax=$DMAX alpha2=$ALPHA2 beta2=$BETA2 s2=100 \  
    
```

```
dmin=$DMAX alpha3=$ALPHA2 beta3=$BETA2 s3=100
#
# platen 3
#
define_model name=cmp3 type=cmp spad=$SPAD3 material=copper \
    rate=$RATE3 material=TaN rate=$RATE3TAN dmax=$DMAX31 \
    alpha2=$ALPHA21 beta2=$BETA21 s2=100um dmin=$DMAX31 \
    alpha3=$ALPHA21 beta3=$BETA21 s3=100um material=oxide \
    rate=$RATE3OX dmax=$DMAX32 alpha2=$ALPHA22 beta2=$BETA22 \
    s2=100um dmin=$DMAX32 alpha3=$ALPHA22 beta3=$BETA22 s3=100um
#
# define a 2D mask with 6 blocks
# For this portion of the test layout, the define_mask command makes
# an array of 3x2 blocks with the test structures 400 um long and 400 um
# wide. Space and width values are specified for each set of arrays.
#
#
define_mask name=mask LineUp=YX BlockWidth_A1=500 BlockHeight_A1=500 \
    ArrayWidth_A1=400 ArrayHeight_A1=400 Width_A1=1 Space_A1=0.5 \
    Orientation_A1=v BlockWidth_B1=500 BlockHeight_B1=500 \
    ArrayWidth_B1=400 ArrayHeight_B1=400 Width_B1=1 Space_B1=1 \
    Orientation_B1=v BlockWidth_C1=500 BlockHeight_C1=500 \
    ArrayWidth_C1=400 ArrayHeight_C1=400 Width_C1=2 Space_C1=1 \
    Orientation_C1=v BlockWidth_A2=500 BlockHeight_A2=500 \
    ArrayWidth_A2=400 ArrayHeight_A2=400 Width_A2=0.5 Space_A2=0.5 \
    Orientation_A2=v BlockWidth_B2=500 BlockHeight_B2=500 \
    ArrayWidth_B2=400 ArrayHeight_B2=400 Width_B2=0.5 Space_B2=1 \
    Orientation_B2=v BlockWidth_C2=500 BlockHeight_C2=500 \
    ArrayWidth_C2=400 ArrayHeight_C2=400 Width_C2=0.5 Space_C2=2 \
    Orientation_C2=v
```

Figure 2-11. Side By Side Cross Sectional Build Up

Cross Sectional Build Up on Silicon	Process Steps	Process Recipe File Entry
	Starting material	initialize x1=300 y1=300 x2=2200 y2=1700 material=substrate pixel=20um
	Dielectric 1 and dielectric 2 deposition	deposit thickness=4100A material=dielectric1 type=isotropic  deposit thickness=500A material=dielectric2 type=isotropic
	After exposure and etch	etch mask2D=mask thickness=3100A material=dielectric type=anisotropic
	After deposition of nitride	deposit isotropic material=TaN thickness=100A
	After copper seed layer and ECD	deposit isotropic material=copper thickness=150A  deposit model=ecd1 rate=3.5nm/s time=35s dt=0.1s saveDt=5s type=ecd  deposit model=ecd1 rate=18nm/s time=\$TIME dt=0.1s Snapshot=21 type=ecd

**Figure 2-12. Side By Side Cross Sectional Build Up**

Cross Sectional Build Up on Silicon	Process Steps	Process Recipe File Entry
	After platen 1	# platen 1: bulk copper removal cmp time=55s dz=5A model=platen1 pressure=3.5psi
	After platen 2	# platen 2: soft landing cmp time=70s dz=0.5A model=platen2 pressure=1.5psi
	After platen 3	# platen 3: barrier hard mask and oxide polish cmp time=60s dz=0.1A model=platen3 pressure=2.5psi

```

#model definitions
define_model name=platen1
type=cmp spad=$SPAD
material=copper rate=$RATE
Hex=$HEX1 alpha1=$ALPHA1
beta1=$BETA1 s1=100um

define_model name=platen2
type=cmp spad=$SPAD2
material=copper rate=$RATE2
material=TaN rate=$RATE2TAN
dmax=$DMAX alpha2=$ALPHA2
beta2=$BETA2

define_model name=platen3
type=cmp spad=$SPAD3
material=copper rate=$RATE3
material=TaN rate=$RATE3TAN
dmax=$DMAX31
alpha2=$ALPHA21
beta2=$BETA21

```

## Measured Data (MD) File

MD files contain thickness information about your test structures and patterns after they have gone through the manufacturing process. The MD file has up to 34 columns that contain simulation and measurement values.

For initial runs, only seven columns are required. Simulation and optimization runs create data for columns that begin with “S” (that is, SErosion, SDishing, SThickT, and SThickNT). Relevant columns for typical simulations are displayed. Other columns that hold optimized values are initially hidden.

S-Columns provide the simulated values calculated during each snapshot in a simulation. For initial runs, these columns can be set to zero. This applies to the following groups:

- **SErosion** — Simulated erosion values.
- **SDishing** — Simulated dishing values.
- **SThickT** — Simulated thickness values for a single layer throughout a process. This column is hidden in the standard view.
- **SThickNT** — Simulated thickness values for a simulation site. ThickNT differs from ThickT in that it tracks the thickness of any exposed layer as others are polished off. This column is hidden in the standard view.

W-Columns provide the weighting confidence of your measurements. These values can be between 0 and 1.

- **WE** — Weights of erosion values.
- **WD** — Weights of dishing values.
- **WTT** — Weights of thickness values of only a single layer.
- **WTNT** — Weights of thickness values of currently exposed layers.

For the initial calibration run, there are ten required columns in the MD file. They are shown in [Table 2-1](#).

**Table 2-1. Required Initial Column Values for MD File**

Values	Description
X Coordinates	Horizontal offset in microns from the lower left corner.
Y Coordinates	Vertical offset in microns from the lower left corner.
Snapshot	Snapshots link the thickness values with their respective commands in the recipe file (deposition, etch, and CMP).
WE	Weighting to give the erosion measurement.
ES	Erosion site values.

**Table 2-1. Required Initial Column Values for MD File (cont.)**

Values	Description
Erosion	Erosion measurement.
WD	Weighting to give the dishing measurement.
Dishing	Dishing measurement.
WTNT	Weighting to give the thickness measurement.
ThickNT	Absolute measurement of thickness.

The complete measured data file has 24 columns, and 10 optional columns for expression calculations. Columns not referenced above should be set to 0 for the initial run.

---

**Tip**

 Simulated measurement sites can be aligned by overriding the X and Y coordinates so that sites are either in the center of a block or in the center of a field region. Link the trench and field regions by entering the row number of the trench in the ES column of the associated field site.

---

## Creating a Measured Data File

Measured data files (MD file, also called an MD table (MDT)) are required for the initial optimization runs. The initial file includes only the measured data and is based on the test layout.

**Figure 2-13. Measured Data File in CMP Model Builder**

#	Comment	X	Y	Snapshot	WE	ES	Erosion	SErosion	WD	Dishing	SDishing	WTNT	ThickT	SThickNT	ThickNT	SThickNT	WZ1	Z1	
252	//	25500.0	23800.0	21	0.0	-1	-3.2	1.9	0.0	202.5	0.0	0.0	0	6748.2	0.0	0	6248.2	0.0	0
253	//	26200.0	23800.0	21	0.0	-1	-4.4	0.4	0.0	213.3	0.0	0.0	0	6749.6	0.0	0	6249.6	0.0	0
254	//	26800.0	23800.0	21	0.0	-1	-9	0.3	0.0	213.8	0.0	0.0	0	6749.7	0.0	0	6249.7	0.0	0
255	//	27350.0	23800.0	21	0.0	-1	-5.3	3.0	0.0	213.8	0.0	0.0	0	6747.1	0.0	0	6247.1	0.0	0
256	//	24500.0	22200.0	21	0.0	-1	-3.1	1.6	0.0	213.8	0.0	0.0	0	6748.4	0.0	0	6248.4	0.0	0
257	//	25500.0	22200.0	21	0.0	-1	-2.5	2.0	0.0	212.6	0.0	0.0	0	6748.0	0.0	0	6248.0	0.0	0
258	//	26200.0	22200.0	21	0.0	-1	-4.4	0.5	0.0	214.9	0.0	0.0	0	6749.6	0.0	0	6249.6	0.0	0
259	//	26800.0	22200.0	21	0.0	-1	-3.9	0.3	0.0	212.6	0.0	0.0	0	6749.7	0.0	0	6249.7	0.0	0
260	//	27350.0	22200.0	21	0.0	-1	-1.7	3.2	0.0	214.4	0.0	0.0	0	6746.9	0.0	0	6246.9	0.0	0
261	//	24500.0	20600.0	21	0.0	-1	-0.8	1.7	0.0	211.6	0.0	0.0	0	6748.3	0.0	0	6248.3	0.0	0
262	//	25500.0	20600.0	21	0.0	-1	-5.4	2.1	0.0	6.2	0.0	0.0	0	6747.9	0.0	0	6247.9	0.0	0
263	//	26200.0	20600.0	21	0.0	-1	-12.21	0.5	0.0	6.4	0.0	0.0	0	6749.5	0.0	0	6249.5	0.0	0

## Prerequisites

- Created initial process recipe file
- Specified sampling sites for thickness measurements
- Measured the thickness at the sampling sites on your test wafer after ECD and CMP

## Procedure

- Add the following heading values to your spreadsheet if they do not already exist.

X Y Snapshot WE ES Erosion WD Dishing WTNT ThickNT

- Input the X Y coordinates of the sampling sites if they do not already exist.
- Input the following values:
  - thickness for field (non-trench) areas (ThickNT) — see [Figure 2-14](#) for where to measure.
  - weighting for the thickness (WTNT) — values should be between 0 and 1. Values above 0.8 indicate a high confidence level. Values below 0.8 should be disabled, or possibly measured again.
  - absolute copper thickness for ECD for platen 1.
  - absolute oxide thickness for platen 3.

For WTNT, values above 0.8 indicate high confidence.

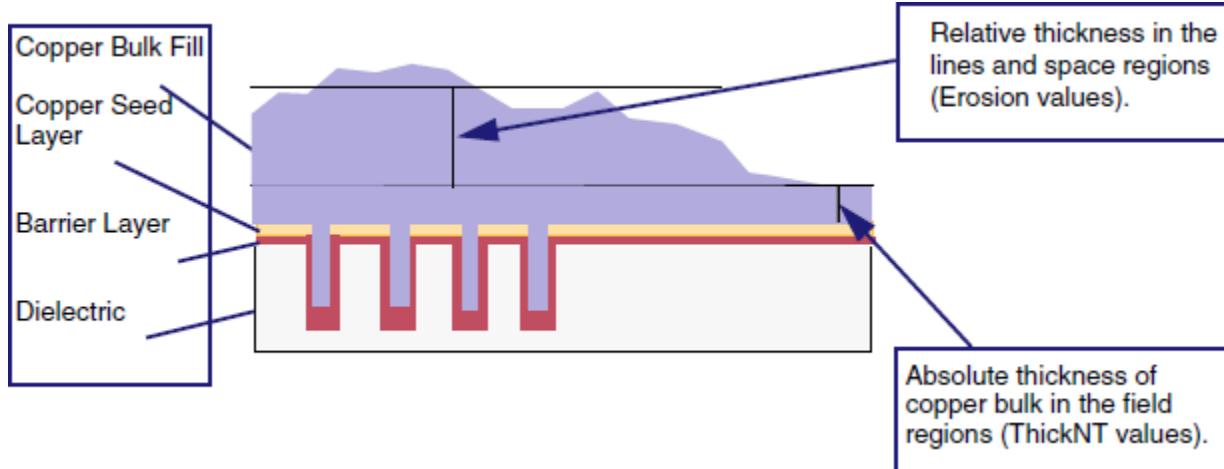
4. Input the erosion (Erosion) and dishing (Dishing) values calculated from line scans and AFM measurements. Erosion values are relative to a particular ThickNT measurement; track the ThickNT value in the ES column.

Erosion and dishing values are calculated from line scans after test wafers have been through the manufacturing and CMP processes that are modeled. These calculated values are relative thicknesses that are input into the erosion and dishing values in the MD file. Relative thickness measurements are shown in [Figure 2-14](#) and [Figure 2-15](#). Line scans also show how these values can be calculated. They are calculated from line scans with the following formulas:

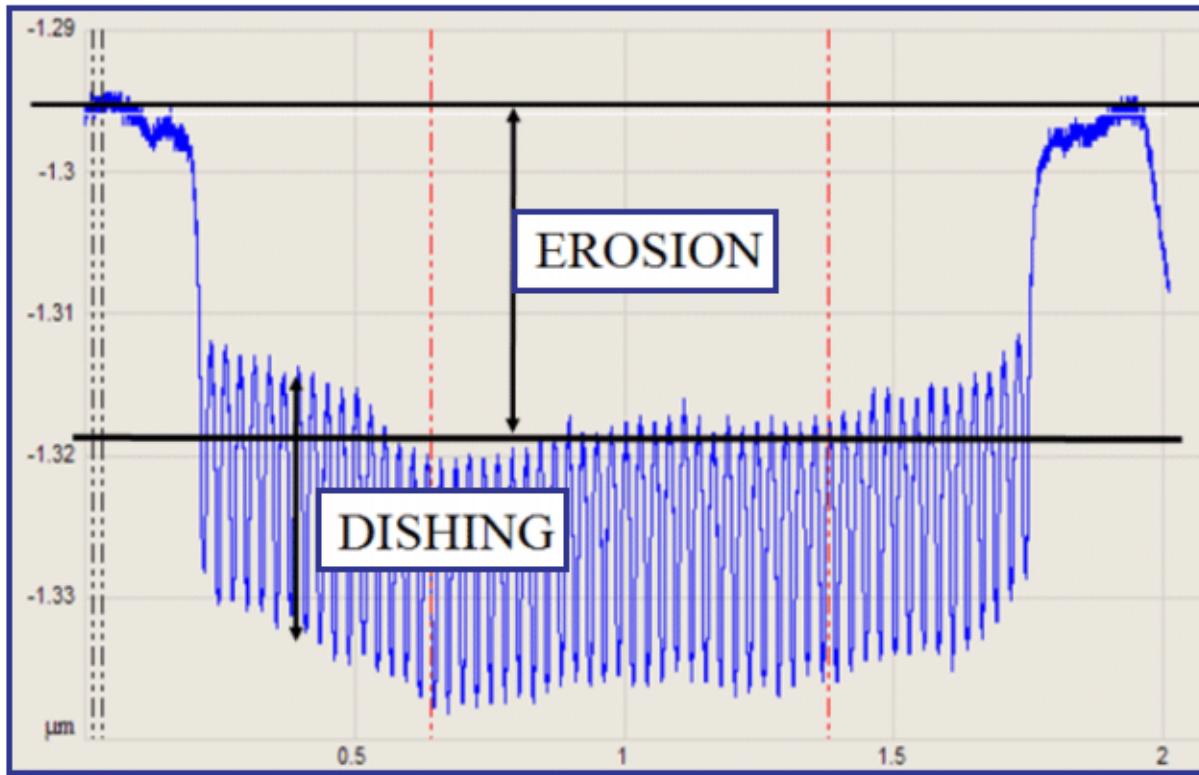
- Erosion = (Measured thickness from nearby field regions) - (Tops of lines and spaces regions)
- Dishing = (Tops of lines and space regions) - (Bottoms of dishing wells)

5. Go to **File > Save Session** to save the process recipe file and the measured data file.

**Figure 2-14. Absolute and Relative Thickness**



**Figure 2-15. Measuring Erosion and Dishing**



## Results

Other columns for the MD file are set to 0 prior to initiating the calibration process.

These values are adjusted following the simulation run. Press + to expand and - to contract the hidden columns.

The following parameters in your spreadsheet should now have data values.

- X Coordinates
- Y Coordinates
- WTNT (hidden)
- ThickNT
- Erosion
- Dishing
- ES
- WE
- WD

## Importing Values to the MD File

Line scans show erosion and dishing values. They are created by examining test chips with microscopes. The information may be stored as either a spreadsheet of values or as a plot saved

in a graphical format file. You can import the spreadsheet into the CMP Model Builder tool and have the information appear in the MD file.

---

**Tip**

-  For easier correlation, it is best to measure across only one or two blocks of trenches at a time. Measurements should include the field regions on either side of the array.
- 

After the data has been imported, CMP Model Builder can display plots comparing simulated and measured values. This is important for analyzing your CMP models to select the best fit for parameters.

Importing erosion and dishing measurements that are in text format is more accurate than taking measurements from microscope plots. The erosion and dishing measurements should be in an ASCII file that has X and Y coordinates from the test chip layout.

## Prerequisites

- MD file
- Line scan file (measurements from wafers)

## Procedure

1. In the **Measured** tab, select the rows that are to receive erosion and dishing values. Use Ctrl-click to select multiple, non-adjacent rows.

To get the best fit, work on data for only one array (row of the MD file) at a time. If the line scan file has measurements for multiple wafers, you can adjust start and end points in step 7.

2. Click the **LineScan** button. The LineScan dialog box opens. (See “[LineScan Dialog Box](#)” on page 138 for field locations.)
3. In the LineScanFile field, enter the name of the file that contains the measured data.
4. If the file uses a column separator other than whitespace, click the + button to reveal the Separator field and enter the character. Any lines with other non-numeric characters are skipped by CMP Model Builder.
5. Enter the column number for the column with X coordinates and Y coordinates.

Choosing the wrong columns in the line scan results in incorrect erosion and dishing values. In some cases, several attempts are needed to get the correct plots. Line scan data varies from machine to machine, so you must find the right data values to use.

Generally, you want the position of the line scans in the X-axis and the measured values in the Y-axis. If you are using a grid file, the values are “1” and “2”.

6. Click **Show**. This opens the Plot dialog box.
7. Adjust the data as necessary to match the orientation of the MD file.

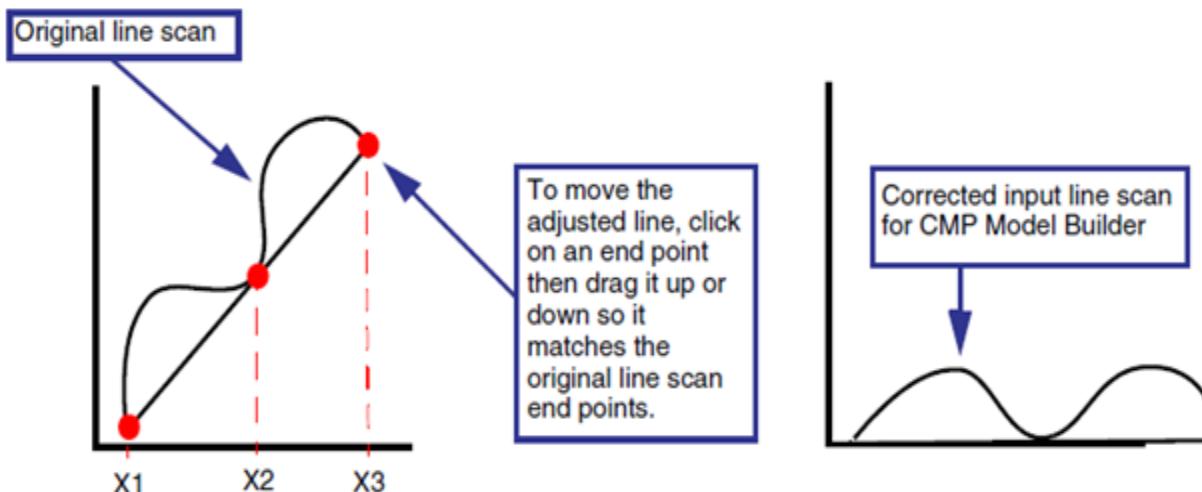
- **Correct angle** (delineation)

If the data is rotated or tilted relative to the measured data, use the adjust line in the plot to reposition the curve ([Figure 2-16](#)). Tilted curves are common because of the difficulty of getting the microscope arm exactly parallel to the wafer.

If the measured data covers more than one array of trenches and the neighboring field regions, you can add additional points to the adjust line. At the bottom of Plot, select the Add Point button and click on the adjust line where you want to move the curve separately.

After the adjust line is positioned along the original line scan data, click the **Update** button.

**Figure 2-16. Results of Adjusted Line on a Line Scan Plot**



- **Initial Offset (Shift)**

Shift is selected by default. The beginning of the line scan is moved to 0.

- i. Reposition the **\*\_adjust\_\* line** so the end points match with the line scan plot as seen in [Figure 2-16](#).
- ii. Select **Add Point** to create more points in the line scan to account for systematic errors in the line scan data.
- iii. After arranging the line scan end points, use **Diff** to redraw the corrected line scan.

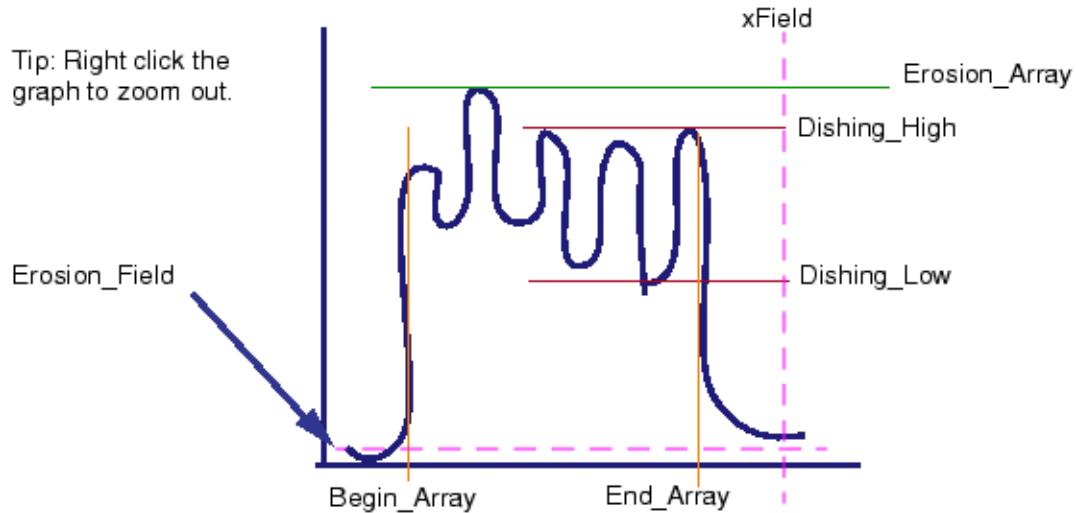
This can also be done using the Adjust Line Scan table if you know the offsets to use.

- **Automatic Region Identification**

If the field and trench areas do not quite match, you can adjust them using either the Begin and End table at the bottom of the Line Scan dialog box, or by adjusting the

markers as shown in [Figure 2-17](#). As you adjust the lines, the dishing and erosion values shown in the LineScan dialog box automatically update.

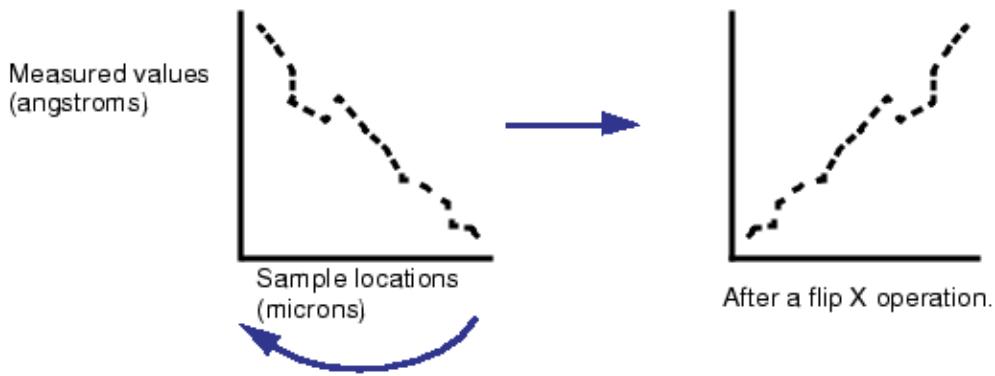
**Figure 2-17. Placement of Line Scan Boundaries on a Line Scan Plot**



- **Orientation**

Use the FlipX option for measurements that are reversed on the X-axis. Give careful consideration as to how the atomic force microscope (AFM) carried out the line scans before flipping the axis.

**Figure 2-18. Results of Flip X on a Line Scan Plot**



8. Click **OK**.

## Results

The CMP Model Builder interface becomes inactive for a period while processing the data. After you dismiss any message dialog boxes, the erosion and dishing values appear in the MD table in the **Measured** tab.

## Related Topics

[LineScan Dialog Box](#)

[Measured Tab](#)

# Declaring Optimization Parameters for Calibration Runs (Optimize Tab)

Calibration involves optimizing the different parameters. You indicate the ones to optimize by using symbolic names. This can be done for the mask, ECD, and CMP model parameters.

## Prerequisites

- A process recipe file
- CMP Model Builder

## Procedure

1. Load the process recipe file.
  - a. In Calibre WORKbench, select **Tools > CMP Model Builder**.
  - b. In the CMP Model Builder window, click the **Recipe** tab.
  - c. Click **Open** and select the process recipe. Click **Open**.
2. Add symbolic names to the Process Recipe file by selecting the step and clicking **Edit** to open a dialog box where you can change parameter values. Symbolic names start with a \$ and have unique names.

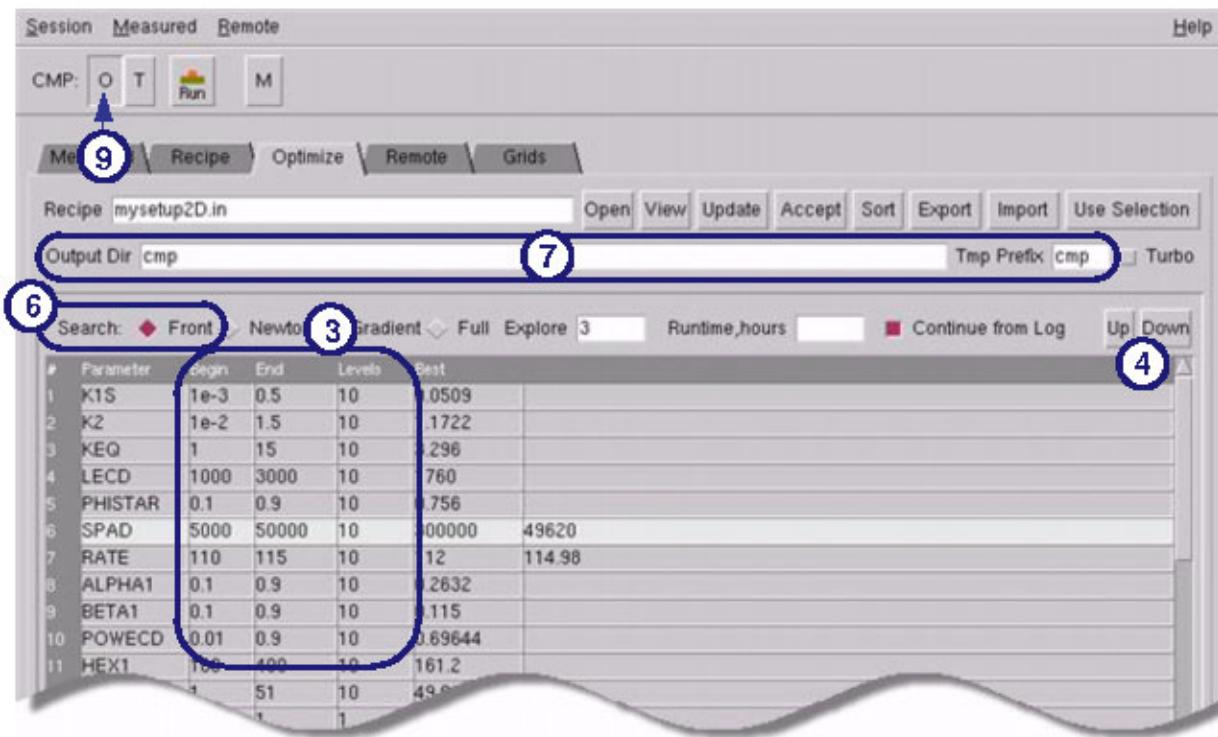
After adding a symbolic value to a model parameter, the parameter becomes available in the **Optimize** tab. For example, in the ECD model, specify the following in the Process Recipe file:

```
define_model name=ecd1 type=ecd k1=$K1 k2=$K2 k1s=$K1S k2s=$K2S \
phistar=$PHISTAR keq=$KEQ rmod=0 smod=0 LECD=$LECD
```

3. Select the **Optimize** tab. Input the Begin and End search values you want to optimize and the interval levels to search in. (Phistar must be between 0 and 1.) See [Figure 2-19](#) on page 43. Optimized values appear in the Best column. If you leave the Levels value set to 1, the parameter remains fixed. It is set to the Best value if one has been found in an earlier optimization or else to the Begin value.
4. Adjust the parameter order using the **Up** and **Down** buttons. Parameters are optimized in the indicated order, so the ones with the most impact should be highest in the list.

Generally the parameters with the most impact are tied to removal rates, such as dmax and dmin. Alpha and beta parameters have less effect and should be lower in the list.

**Figure 2-19. Setting Up an Optimization Run**



5. Repeat steps 2 through 4 for your CMP models so the Process Recipe file looks similar to the following example:

```

define_model name=cmp1 type=cmp spad=$SPAD material=copper \
rate=$RATE Hex=$HEX1 alpha1=$ALPHA1 beta1=$BETA1 s1=100um \

define_model name=cmp2 type=cmp spad=$SPAD2 \
material=copper rate=$RATE2 material=TaN rate=$RATE2TAN \
dmax=$DMAX alpha2=$ALPHA2 beta2=$BETA2 s2=100um dmin=$DMAX \
alpha3=$ALPHA2 beta3=$BETA2 s3=100um material=oxide rate=1 \
dmax=$DMAX alpha2=$ALPHA2 beta2=$BETA2 s2=100 dmin=$DMAX \
alpha3=$ALPHA2 beta3=$BETA2 s3=100 \

define_model name=cmp3 type=cmp spad=$SPAD3 material=copper \
rate=$RATE3 material=TaN rate=$RATE3TAN dmax=$DMAX31 \
alpha2=$ALPHA21 beta2=$BETA21 s2=100um dmin=$DMAX31 \
alpha3=$ALPHA21 beta3=$BETA21 s3=100um material=oxide \
rate=$RATE3OX dmax=$DMAX32 alpha2=$ALPHA22 beta2=$BETA22 \
s2=100um dmin=$DMAX32 alpha3=$ALPHA22 beta3=$BETA22 s3=100um \

```

6. Select the search method to optimize your model parameters. For this walk-through, choose Front and set the Explore field to 3. Clear any values in the runtime field.
7. Choose an output directory to store your grid files. The files stored in the output directory follow the naming convention specified in the Tmp Prefix field.

8. If you have a large test chip, the next step runs more quickly if you extract the grid files first using Calibre CMPAnalyzer. The following conditions may indicate the test chip will extract faster with Calibre CMPAnalyzer:
  - The test chip has billions of shapes.
  - The test chip is larger than a centimeter on a side.
  - The process step being modeled is an oxide layer using a negative mask with dummy fill, and the other two test chip conditions are close to being met.

If this applies to your test chip, see “[Extracting Grids for Large Test Chips](#)” on page 203.

9. Choose Optimize mode (**O** button), then click **Run** to find the optimized parameters. Calibrated values appear in the Best column.

If the recipe refers to a hierarchical GDS or OASIS file, the Clip Layout dialog box appears. You must create a flat version of all or part of the layout for optimization to run.

- To create a flat version of the entire layout, specify a name for the new file in Output Oasis File and click **OK**.
- To create a flat version of a portion of the layout, you need to supply a region and name.
  - i. Supply a region by either entering coordinates of the selection corners in the X and Y fields or clicking **Use Selection**. **Use Selection** sets the X and Y fields to the corner coordinates of the currently displayed portion of the layout.
  - ii. To include a halo region for any potentially truncated shapes, set the halo size in the Extra field. This is optional.
  - iii. Specify a name for the new file in Output Oasis File and click **OK**.

## Results

CMP Model Builder attempts to run the process recipe, as shown by the status message in the lower right corner of the CMP Model Builder window.

Any problems with the recipe or input are reported to the WORKbench transcript in the terminal window, and also in a message box. After fixing these errors, save the modified recipe or layout and click **Run** again.

# Analyzing Results to Create the Best Model

After a successful optimization attempt, you analyze data plots to isolate outlier data points that might skew the model accuracy. Group measured and simulated values together for analysis.

The lines in the plots are labeled with two prefixes and the property in the form `r#_type_property`, such as “`r1_S_dishing`.” The parts are as follows:

- The `r#_` prefix denotes which run a line is from.
- The *type* is one of `M_`, `S_`, and `E_`. These denote Measured, Simulated, and Error values, respectively.
- The *property* refers to properties seen in the MDT such as “dishing”, “erosion”, or “ThickNT.”

Each symbol on the data plot represents an absolute thickness measurement in the MD file.

Generally, it is best to start by comparing measured and simulated thickness (`r1_S_Thickness` and `r1_M_Thickness`). If thickness looks good, analyze erosion (`r1_S_erosion` and `r1_M_erosion`) and dishing (`r1_S_dishing` and `r1_M_dishing`). For each plot, adjust the process recipe file so that it results in simulated data (`r1_S*`) that look similar to measured data (`r1_M*`). When these three plots meet your specifications you can start using them for analysis, simulations, and DFM applications.

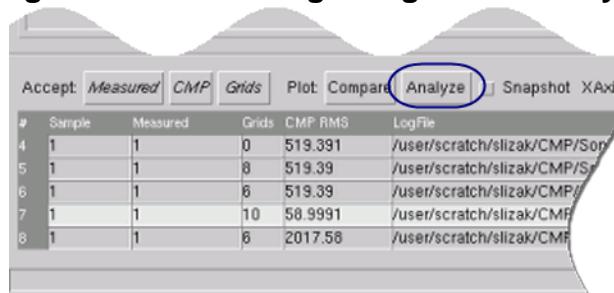
## Prerequisites

- Optimized CMP model

## Procedure

1. Run a CMP model trial run to obtain a CMP RMS value if you have not already done so.
2. Select the model trial run in the table of log files and click **Analyze**.

**Figure 2-20. Selecting a Log File to Analyze**



A screenshot of the Calibre® CMP Model Builder software interface. At the top, there is a menu bar with options like Accept, Measured, CMP, Grids, Plot, Compare, Analyze (which is highlighted with a blue oval), Snapshot, and XAxis. Below the menu is a table with columns: #, Sample, Measured, Grids, CMP RMS, and LogFile. The table contains eight rows of data. Row 4 has a LogFile path of /user/scratch/slizak/CMP/Sop. Row 5 has a LogFile path of /user/scratch/slizak/CMP/Sop. Row 6 has a LogFile path of /user/scratch/slizak/CMP/Sop. Row 7 has a LogFile path of /user/scratch/slizak/CMP/Sop. Row 8 has a LogFile path of /user/scratch/slizak/CMP/Sop.

#	Sample	Measured	Grids	CMP RMS	LogFile
4	1	1	0	519.391	/user/scratch/slizak/CMP/Sop
5	1	1	8	519.39	/user/scratch/slizak/CMP/Sop
6	1	1	8	519.39	/user/scratch/slizak/CMP/Sop
7	1	1	10	58.9991	/user/scratch/slizak/CMP/Sop
8	1	1	8	2017.58	/user/scratch/slizak/CMP/Sop

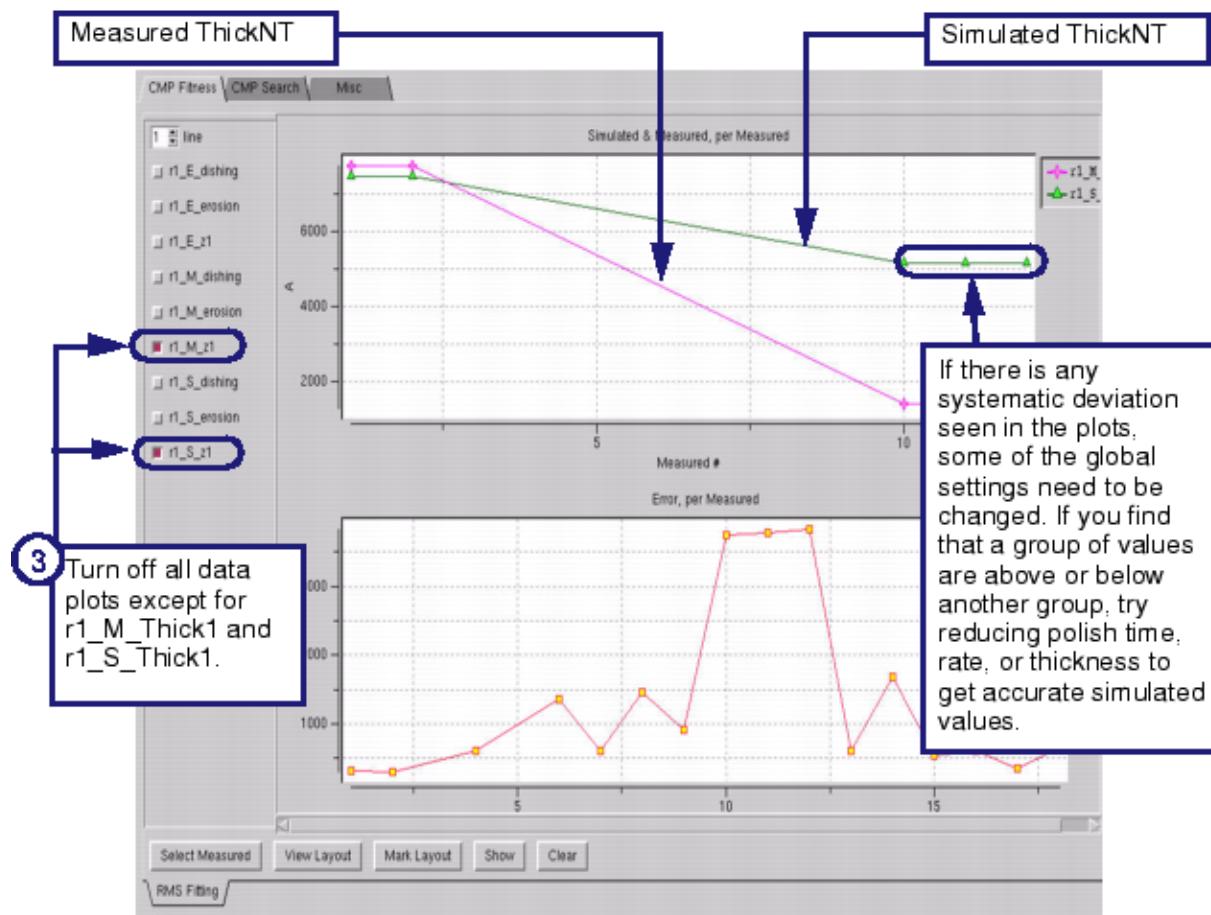
3. View measured and simulated data in pairs. The first pair to analyze is `r1_M_Thickness` and `r1_S_Thickness` as shown in [Figure 2-21](#). Turn off the other data sets to inspect only this set of values.

For plots that do not match well because of systematic deviations, adjust the parameters in the process recipe file then run another model in (T)ry mode to see if the CMP RMS value is lower.

For simulated values that deviate by a large amount, or for spikes in plots that should be steadily decreasing, try disabling a data point:

- a. Select the data point in the plot.
- b. Click the **Select Measured** button. The **Measured** tab in the CMP Model Builder window highlights the data point.
- c. Click **Disable** to isolate the data point from the rest of the file.
- d. Run another model in (T)ry mode. If this is the outlier, a lower CMP RMS value should result. Continue changing the model parameters until the measured values and the simulated values are alike.

**Figure 2-21. Analyzing Simulated and Measured Plots**



4. If the absolute measurements (ThickNT's in the first plot) match, look at the erosion data (`r1_M_erosion` and `r1_S_erosion`), then the dishing data (`r1_M_dishing` and `r1_S_dishing`).
5. After trial runs have met your specifications, the optimized model and the best values for your parameters are located in the directory specified in **Outdir**. The optimized model has the `.in` file extension. This is the final optimized CMP model.

**Tip**

**i** In order to check the exact measured site, select the data plot values of interest and click **Mark Layout** to see the measurement site in the Calibre WORKbench main display window.

## Viewing ThickNT, Erosion, and Dishing Plots

You should check the optimization results to tune your model. The results are in the grid files. The CMP Model Builder GUI allows you to view the results and compare simulated versus measured values in plots.

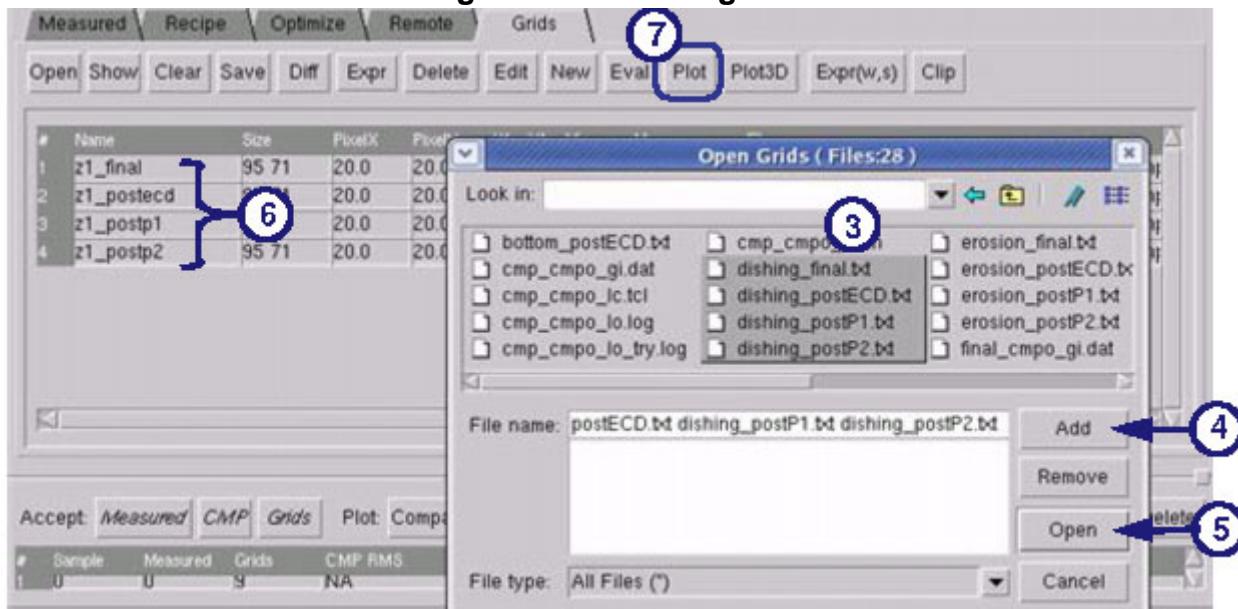
### Prerequisites

You must have an optimized CMP model.

### Procedure

1. Go to the **Grids** tab.
2. Click **Open** to select the grid files to view.
3. Highlight the group of plots you want to view together (for example, dishing plots, erosion plots or ThickNT plots).

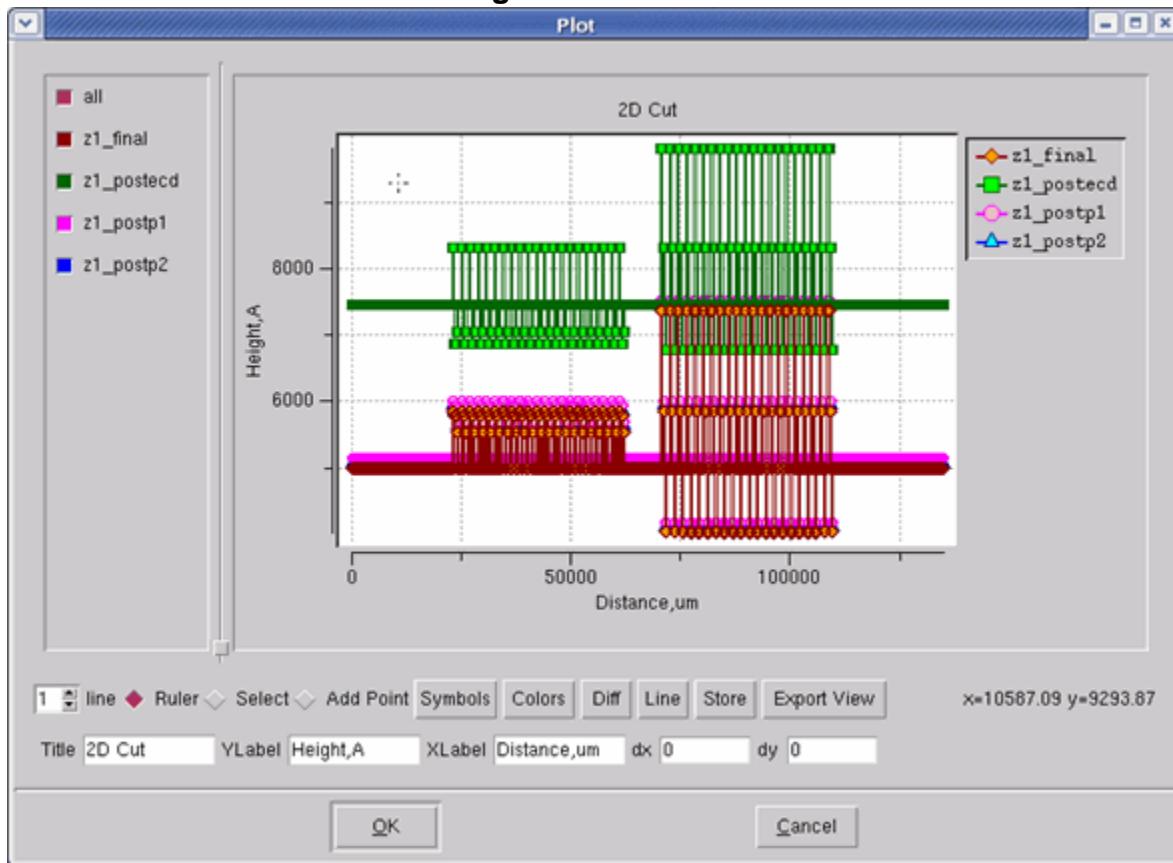
**Figure 2-22. Viewing Plots**



4. Click **Add**.
5. Next, click **Open**. This populates the list of grid files to view.
6. Highlight the grids you want to plot.

7. Click **Plot**. This plots of all the selected grids as seen in [Figure 2-23](#).

**Figure 2-23. Plots**



**Tip**

**i** You can also export grids to plot in other tools. See “[Creating a Surface Plot](#)” on page 195.

---

## Finishing the CMP Model

After optimizing your model, validate it against a production design by comparing the simulation results to measurements on the production chip. To validate, package the model and use it in Calibre CMPAnalyzer.

Depending on the needs of your users, there may be several criteria for validation. The most widely used criteria are:

- topography trend dependence on pattern density
- match between simulations and measured layer thickness

For full instructions, see the [Calibre CMPAnalyzer User’s Manual](#).

## Prerequisites

- An optimized CMP model.
- Appropriate licenses for running Calibre CMP Model Builder.

## Procedure

1. If you are satisfied with the parameter values, create the final recipe file.
  - a. Select the **Optimize** tab.
  - b. Click the **(T)ry** button. This indicates that Calibre should use the values in the Best column and compute RMS.
  - c. Click **Run**. This starts the trial run, and also causes the final recipe (in readable format) to be written out to *<Output\_Dir>/<Tmp\_Prefix>.cmpo\_so.in*.
  - d. To create an encrypted recipe for use with Calibre CMPAnalyzer, select the **Recipe** tab and click **Encrypt**. You are prompted to name the encrypted recipe file. A suffix of *.in\_encrypt* is appended automatically.

---

### Note



Encrypted recipes cannot be decrypted. Be sure to save either the session file or the *.in\_encrypt\_tmp* file in case you ever need to refer back to the values chosen.

---

2. Save the session file (**Save > Session**). This preserves the optimization information and process recipe in case you want to make changes later.
3. Look in the output directory for a file ending with *.in* or *.in\_encrypt*. This is the output recipe.

The output directory is shown in the Output Dir field of the **Optimize** tab.

The output recipe is a process recipe file. It contains the results of the optimization run instead of symbolic names for parameters. Depending on the simulation settings, it may also use different final targets. For instance, you may run an ECD simulation using `targetThicknessNT`. The optimized model (output recipe) specifies a time and rate instead that produced the target thickness in simulation.

4. Try the recipe out with the Calibre CMPAnalyzer interface to look at results from a production design.
  - a. Verify that the output recipe uses “`mask=layer_number`” instead of defining a mask array.
  - b. Select **Verification > Run DFM** from the WORKbench menu.

- c. In the Calibre® Interactive™ window, provide the inputs shown in [Table 2-2](#):

**Table 2-2. Calibre CMPAnalyzer Settings**

Pane	Field: Setting
Inputs	Run: CMP Analyzer CMP tab > Layer Number: Layout's equivalent metal layer CMP tab > Recipe File: Output recipe file
Rules	DFM Run Directory: An empty directory
Outputs	Results Format: DFM Database Directory: new directory to store DFM results

- d. Click **Run CMP**.

## Results

Calibre Interactive displays the transcript as it runs. If you selected Show Results in RVE in the Outputs pane, when the run is complete Calibre® RVE™ displays the metal layers. You can use the menus in RVE to look for hotspots and overall layer thickness.

## Related Topics

[Calibre CMPAnalyzer Users Manual](#)

# Chapter 3

## CMP Model Builder in Batch Mode

---

CMP Model Builder can be invoked in batch mode or through the GUI in Calibre WORKbench.

This section covers the syntax and how to generate output for CMP Model Builder in batch mode runs. For distributed runs, a remote file or a remote string is also required. More information on distributed runs can be found in the [Calibre Administrator's Guide](#).

The `cmpoptimize` command requires a process recipe file and measured data file as inputs.

---

### Note

---

 For distributed runs, use the same platform for the primary host and the remotes.

---

CMP Model Builder in batch mode is executed through Calibre WORKbench at the command prompt along with a Tcl script. For example:

```
calibrewb optimize.tcl  
cmpoptimize ..... 52
```

## cmpoptimize

Runs simulations and returns the RMS error of the simulated versus measured data.

### Usage

```
cmpoptimize -recipe filename -mf filename \
[-bbox x1 y1 x2 y2] [-l layout_file] [remote_settings] \
[-outdir directory] [-log filename | -nostdout] \
[-recipeOut filename] \
[-mfOut filename] \
[-search {front | gradient | newton | full}] \
[-explore integer | -runtime hours] \
[-epsilon floating_pt_value] \
[-frontlogfile filename [-optimization_parameter begin_val end_val integer]]
```

---

#### Note

 When a script is exported from the GUI, optimization parameters are also included in the Tcl script.

---

### Arguments

- **-recipe *filename***  
Name of parameterized process recipe file (required).
- **-mf *filename***  
Name of spread sheet file with measured data (required).
- **-bbox *x1 y1 x2 y2***  
An optional argument that defines the coordinates of the bounding box for optimization. All four coordinates are required. They define the lower left corner and the upper right corner of the simulation domain.
- **-l *layout\_file***  
An optional argument that specifies the input layout. The layout must be loaded into Calibre WORKbench before running the cmpoptimize command.
- ***remote\_settings***  
An optional argument that specifies a method for multithreaded execution or to use a computer other than the one running Calibre WORKbench. The default execution method is single-threaded with no distributed processing.

Only one of the following choices can be used per cmpoptimize command:

**{-remotefile | -remote} *filename***

Uses the machines specified in the file in parallel. The total number of CPUs used cannot be greater than 128.

**-remotestring *string***

Works the same as -remotefile, except the file contents are passed in *string* instead of a file.

**-turbo**

Enables multithreaded calibration. The number of threads to be used is determined by the number of CPUs available on your machine and the MT scalability of the search method chosen. By default, this number cannot be greater than 32. Additionally, if you do not want to use all available CPUs, you can limit the number of threads by calling Calibre WORKbench with the -threads option specified. For example:

```
calibrewb -threads 2 cmp.tcl
```

- **-outdir *directory***

An optional argument specifying an output directory. The log file, recipeOut, mfOut files, and all grid files saved during the execution of the process recipe are written to *directory*. If *directory* does not exist, cmpoptimize creates it.

- **-log *filename* | -nostdout | -log *filename* -stdout**

An optional argument to name the log file and restrict where output is sent. By default, the output of the cmpoptimize command is sent to both standard output and a log file. To send output only to stdout, specify “-log stdout”. To send output only to a named log file, specify “-log *my\_log* -nostdout” where *my\_log.log* is the name you want to use.

If -log and -mfOut have the same *filename*, only the *filename.log* is written because the output contains the data that would also be in the -mfOut file.

- **-recipeOut *filename***

An optional argument that causes a final recipe to be written to *filename* at the end of the optimization run. The final recipe replaces all variables with the optimum values. By default, a final recipe is not written.

- **-mfOut *filename***

An optional argument that causes a spreadsheet of values to be written to *filename.dat* at the end of the optimization run. The spreadsheet is similar to the measured data file. The -mfOut file contains a copy of the input **-mf** file together with the simulated values calculated using the best values obtained in the optimization for the variables in the recipe.

If -log and -mfOut have the same *filename*, only the *filename.log* is written because the output contains the data that would also be in the -mfOut file.

- **-search {front | gradient | newton | full}**

An optional argument specifying the type of optimization method to use. The front method is the default and is strongly recommended. When using the front method, the tool issues

messages that specify how many remote hosts and CPUs can be used simultaneously at a particular point in the front search. For example:

```
OPTIMIZER: 3-way parallel functor eval
OPTIMIZER: 2-way parallel functor eval
OPTIMIZER: 4-way parallel functor eval
OPTIMIZER: 6-way parallel functor eval
OPTIMIZER: 4-way parallel functor eval
```

You can adjust the number of remote hosts and CPUS used by the front method based on this information.

See the search argument to “[modelflow\\_v2](#)” in the *Calibre WORKbench User’s and Reference Manual* for complete information. The search method is ignored if there are no optimization parameters.

- **-explore integer**

An optional argument that sets the maximum number of iterations for the search. The default is [80000](#), the upper limit for Calibre WORKbench for front searches. This cannot be set with -runtime. If used, it also requires -search be set.

- **-runtime hours**

An optional argument that specifies the maximum run time in hours for the optimization. This cannot be set with -explore. If used, it also requires “-search front” be set.

- **-epsilon floating\_pt\_value**

An optional argument specifying the convergence criteria when “-search newton” is used. See the table “[Norm Values for Different Optimization Values](#)” in the *Calibre WORKbench User’s and Reference Manual*.

- **-frontlogfile filename**

An optional argument specifying the name of a log file from a previous optimization run. This log file must use the same parameters (same parameter names, same order, and same range for all parameters) as the **-recipe**. This can be used to continue a front simulation run without repeating all the simulations that have already been made. The specified log file is scanned and the front method is initialized with the already computed results.

When using the GUI to continue a simulation, the front log file from a previous run must be loaded using the Log: Open button. The log file must be selected in the table and the “Continue from Log” button must be checked in the **Optimize** tab. When this is done, an optimization run can be performed that restarts from the selected log file or (in the Session scroll down menu) a run script can be created that specifies the appropriate parameter -frontlogfile.

- **-optimization\_parameter [begin\_val end\_val] integer**

An optional list of parameters from the process recipe file with values. Each parameter entry should begin with a hyphen (-).

All parameters with [Symbolic Names](#) must be listed. The names are case sensitive. Missing parameter specifications as well as parameter specifications that are not used in the recipe

are reported as an error during the initial stage of the optimization run. For each parameter, either one or three values must be specified.

If only *integer* is supplied, the (fixed) specified value is used in all simulations. Otherwise, two floating point values and one *integer* (*begin\_val*, *end\_val*, and the number of levels to search) must be specified, as they define the minimum and maximum bounds, and the number of steps for each parameter. These optimization parameters are varied within the specified bounds during the optimization process.

If the parameters force calculations out of range, the value 3.40282e+38 (FLT\_MAX -1) is returned in batch mode. When running in the GUI instead, the following message is generated:

```
ERROR: A try run failed calculate objective function.
```

## Description

The cmpoptimize command runs simulations and returns the RMS error of the simulated versus measured data. It also saves a log file and best recipe, along with any grids saved by the process recipe file, to a specified output directory.

If the process recipe file contains symbolic parameters (that is, parameters whose values are symbolic names or algebraic expressions that include symbolic names), then cmpoptimize also finds the best values for the symbolic parameters by simulating values in the indicated range against the measured data.

Running cmpoptimize without symbolic parameters is equivalent to running the [cmp](#) command, except that the RMS error is also returned.

## Examples

### Example 1: Compute RMS error for nominal settings

This example calculates the RMS Error using nominal model settings (no optimization parameters are present).

```
cmpoptimize -recipe myrecipe.in -mf mymeasured.dat \
-l layout.oas -outdir tmp -log out -recipeOut best \
-AAA 0.1 -ALPHA1 0.2
```

### Example 2: Model tuning

This example tunes the model parameters. It shows in bold the beginning and ending values and the step size for optimizing parameters AAA and ALPHA1.

```
cmpoptimize -recipe myrecipe.in -mf mymeasured.dat \
-l layout.oas -outdir tmp -log out -recipeOut best \
-AAA 0.1 1.1 5 -ALPHA1 0.2 0.8 4 \
-search front
```

### Example 3: Full factorial model evaluation

This example evaluates a model using the full factorial evaluation.

```
cmptimize -recipe myrecipe.in -mf mymeasured.dat \
-l layout.oas -outdir tmp -log out -recipeOut best \
-AAA 0.1 1.1 5 -ALPHA1 0.2 0.8 4
-search full
```

#### Example 4: Model tuning using multithreaded Newton

This example uses a multithreaded (-turbo) newton search with 10 exploration runs and an epsilon value of 0.005 to tune the model.

```
cmptimize -turbo -recipe myrecipe.in -mf mymeasured.dat \
-l layout.oas -outdir tmp -log out -recipeOut best \
-AAA 0.1 1.1 5 -ALPHA1 0.2 0.8 4 \
-search newton -explore 10 -epsilon 0.005
```

#### Example 5: Model tuning using distributed processing

This example tunes AAA and ALPHA1 using a partial factorial search (-explore value of 100 is less than the total number of parameter combinations that can be explored  $21 \times 16 = 336$ ). The model calibration is done in parallel on the local machine using 3 CPUs.

```
----- generate remote string
set mgc $env(MGC_HOME)
set remstr "REMOTE HOST localhost 3 MGC_HOME $mgc"
----- run optimization

cmptimize \
-recipe mysetup.in \
-mf mymeasured.dat \
-remotestring $remstr \
-outdir tmp \
-log out \
-search full -explore 100 \
-AAA 0.55 0.75 21
-ALPHA1 0.8 0.95 16
```

# Chapter 4

## CMP Modeling Reference

---

This chapter discusses the modeling commands for the process recipe file and the MD file.

The commands and the file format discussion use specific syntax described in “[Syntax Conventions](#)” on page 19.

<b>Symbolic Names .....</b>	<b>57</b>
<b>Numeric Expressions in Parameters .....</b>	<b>58</b>
<b>Measured Data File Format .....</b>	<b>59</b>
<b>Process Recipe File Format.....</b>	<b>65</b>
cmp .....	67
define_mask 1D .....	72
define_mask 2D .....	74
define_model .....	84
deposit .....	92
etch .....	102
initialize .....	110
save .....	114

## Symbolic Names

Symbolic names are parameter values represented as variables.

When used in commands, symbolic names serve two purposes:

- Distinguish constants from parameter values that can be optimized.
  - A numeric value indicates a constant that cannot be changed.
  - A symbolic name indicates a parameter value that is to be optimized to fit the data.
- Define dependencies between the various processes and steps. When you assign one variable to multiple parameters, the CMP model optimization process finds the optimal value that can be used in all locations.

Symbolic names are strings. To use a symbolic name as a parameter value, supply the string preceded by a dollar sign (\$). Only alphanumeric characters and underscores (\_) are allowed.

## Numeric Expressions in Parameters

Parameter values may contain expressions that use symbolic names, for example,  $(\$HEX+\$SF)^2$ , or references to the MD such as  $\$X$  or  $\$M\_THICK$  (measured thickness). In each simulation, the current value of the symbolic name is used to evaluate the expression.

Expressions can also use  $\$M\_PI$  to stand for  $\pi$ .

The following functions can be used:

**Table 4-1. Functions Supported for Expressions**

+ (addition)	- (subtraction)	* (multiplication)
/ (division)	log (natural logarithm)	sin (sine)
cos (cosine)	tan (tangent)	exp (e to the power of)
abs (absolute value)	asin (arcsine)	acos (arccosine)
sinh (hyperbolic sine)	cosh (hyperbolic cosine)	atan (arctangent)
tanh (hyperbolic tangent)	sqrt (square root)	log10 (base 10 logarithm)
asinh (hyperbolic arcsine)	acosh (hyperbolic arccosine)	atanh (hyperbolic arctangent)
pow (x to the power of y)	max (maximum)	min (minimum)
atan2 (arc tangent of two arguments)		( ) (grouping)

---

**Note**

 Spaces are not permitted.

---

# Measured Data File Format

Generated by: spreadsheet of measurements in .csv format, or **Measured > Testchip** dialog

Measured data (MD) files are spreadsheet-style ASCII files for storing measured and simulated data. They are required file for CMP model optimization. The file browser assumes a suffix of .dat, but this is not required.

One or more lines define a measured point. Each line defines up to five different model characteristics. When a measured point is represented by multiple lines, each line should have a different snapshot value.

The values in the MD file can be arbitrarily precise. However, most values are whole numbers because measurements are in angstroms by default. The CMP Model Builder window's **Measured** tab displays a maximum of 11 digits. Log files display up to six digits past the decimal point.

## Format

The values in the MD file are arranged as a table. The table can be preceded by an optional header. When written from the CMP Model Builder GUI, the header lists the Calibre version and column headings.

---

### Note

---

 Calibre CMP Model Builder can handle version 3 or version 4 MD files. Version 3 files do not include the comment column. Version 4 files, available from 2016.1, have the comment column as the first column.

---

The table has the following arrangement:

- Rows are the measured and simulated values for a particular point on a layout. There must be at least one non-comment row.
- Columns are separated by whitespace, and represent particular parameters such as coordinates or thickness at those coordinates. There must be between 14 and 34 columns.

An “empty” cell is given the value 0.

When an MD file is written by the CMP Model Builder GUI, non-table content, such as the header, must begin with a #. Lines beginning with # are ignored.

When an MD file is loaded to the **Measured** tab, the values are assigned to the parameters in the order listed following. To show hidden columns in the GUI, click the + button in the row of buttons above the table.

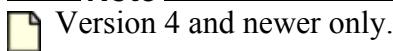
## Parameters

- Comment

---

**Note**

---



---

An optional value to use for naming measurement sites. The value must not contain whitespace or the characters # or \$.

The default value is //. This value is automatically supplied when loading a version 3 MD file.

- X

A required value giving the point's X coordinate. This is the horizontal offset in microns relative to the origin.

Each measurement should be at least one full grid point away from any other measurement, typically 20 microns. The grid size is defined by the initialize command.

- Y

A required value giving the point's Y coordinate. This is the vertical offset in microns relative to the origin.

- Snapshot

A required positive integer indicating when the manufacturing process data was collected (either measured or simulated). The snapshot reflects both a layer in the layer stack and the state that resulted from a physical action. For instance, the snapshot may represent a specific TaN layer after deposition.

The Snapshot number must match either a mask number for a metal layer or a Snapshot number defined within the corresponding commands in the Process Recipe File.

- To associate the data with a specific metal layer, the Snapshot number must match the layer number specified using the mask keyword in the corresponding etch command in the Process Recipe File.
- To associate the data with a particular process state, the Snapshot number must match the number specified using the Snapshot keyword in the corresponding deposit, etch, or cmp commands in the Process Recipe File.

- WE

A required number between 0 and 1 inclusive that specifies a weighting factor for erosion calculations. Low values imply low confidence in the measurement. Use 0 to remove the erosion value in this row from the model fitting.

- ES

An integer value that identifies another row that represents the field region within which the data was sampled. This parameter establishes a link from the current row in the Measured Data File to another row in the same file.

You typically collect profilometry measurements at multiple sampling sites within a single field region. The position (XY) indicated by the ES value defines the position in the non-trench region (a z1 value) that is taken as reference for the computation of the simulated erosion value.

- Erosion  
The measured value for erosion in angstroms at the X Y position.
- SErosion  
The simulated erosion value in angstroms. The user-supplied value is a placeholder that is replaced during simulation.
- W

**Note**



This is a version 3 column. Do not include it in MD files that have a Comment column.

---

A parameter that indicates which method of calculating trench width should be used. When a version 3 file is read by the current version of CMP Model Builder, this column's setting is ignored.

To retain this column when saving from the GUI with **Measured > Save**, set the File type field to “Old Format Measured (\*.odat).”

- WD  
A required number between 0 and 1 inclusive that specifies a weighting factor for dishing calculations. Low values imply low confidence in the measurement. Use 0 to remove the dishing value in this row from the model fitting.
- Dishing  
The measured value for dishing in angstroms at X Y.
- SDishing  
The simulated dishing value in angstroms. The user-supplied value is a placeholder that is replaced during simulation. It is computed as the difference of the vertical position over nontrench regions (Z1) and trench (Z2).
- WTT  
A required number between 0 and 1 inclusive that specifies a weighting factor for thickness calculations. Low values imply low confidence in the measurement. Use 0 to remove the thickness value in this row from the model fitting.
- ThickT  
The absolute thickness in angstroms of the layer at this location. Thickness is typically calculated based on Z1 and Z2 data. It represents the distance from the top of the previous layer, which is assumed to be completely flat, to the top of the Snapshot layer.  
Measurements are generally taken from ellipsometer data.

**Note**

 ThickT is the last column that requires an entry for ASCII MD files. SThickT and forward do not require a value when creating initial tables from spreadsheets.

---

- SThickT

The simulated thickness of the copper line in angstroms at the given location. The user-supplied value is a placeholder that is replaced during simulation.

- WTNT

A weighting factor that defines how much the thickness of the exposed material outside trench regions contributes to the objective function.

If a positive WTNT value is provided, the thickness of the exposed material is extracted from the simulation and considered in RMS calculations. If the WTNT value is not positive, the simulated thickness value (SThickNT) is extracted but not included in RMS calculations.

Do not specify a positive WTNT value for snapshots that represent the soft landing (platen 2) because there may or may not be thin residual films of copper or barrier layer in some frames and the exposed material may change from frame to frame.

- ThickNT

The thickness in angstroms of the exposed material outside of the trenches at the given location. This parameter can specify the absolute measured thickness of the exposed material. After electroplating and after platen 1, this is the absolute copper thickness, measured in field regions, between arrays of lines and spaces. In the final CMP snapshot (ILD overpolish) this is the (remaining) thickness of the topmost layer in the residual ILD stack.

Using WTNT and ThickNT eliminates the need for the initial position specification for the measured thickness values and simplifies your measured data files.

- SThickNT

The simulated thickness in angstroms of the exposed material outside of the trenches at the given location. It is computed as the difference ( $Z_1 - Z_{layer}$ ).  $Z_1$  is the height over nontrench regions.  $Z_{layer}$  is the topmost position of any layer less than  $Z_1$ .

Note that the exposed material may change from frame to frame. If there is some residual thin film of material left, its thickness is reported. This happens frequently during the soft landing, when some frames still may be covered by a thin film of barrier material while in other frames the barrier material has already been polished off. Therefore, it is not recommended to specify positive WTNT values for the snapshots that represent the soft landing CMP step. If no positive weight WTNT is specified, the simulated exposed material thickness value is still extracted from the simulation result, but it is not considered in RMS calculations.

- WZ1  
A number between 0 and 1 inclusive that specifies a weighting factor for the height of the nontrench regions in fitting the model. Low values imply low confidence in the measurement. Use 0 to remove the Z1 value in this row from the model fitting.
- Z1  
The measured position in angstroms of the surface of the nontrench regions at the given location. This is a local average reflecting the height in the test structure area, not the field area.  
Depending on the exposed material at the given location in the current layer stack, this can be the position of the copper, barrier layer, or oxide, though it is most commonly the oxide layer. The parameter Z1 can be used to specify absolute thickness measurements for copper layer after electroplating, platen 1, or platen 2 and absolute values for dielectric layer thickness at the end of the CMP step.
- SZ1  
The simulated position of the surface over nontrench regions in angstroms at the given location. The user-supplied value is a placeholder value that is replaced during simulation.
- WZ2  
A number between 0 and 1 inclusive that specifies a weighting factor for the height of the trench regions in fitting the model. Low values imply low confidence in the measurement. Use 0 to remove the Z2 value in this row from the model fitting.
- Z2  
The measured position in angstroms of the surface of the trench regions at the given location.
- SZ2  
The simulated position over trench regions in angstroms at the given location. The user-supplied value is a placeholder value that is replaced during simulation.
- TMP1 ... TMP10  
Optional columns for parameters to use in expressions. These columns are not included in MD files written by the GUI unless you enter a non-zero value in at least one of the cells.

## Examples

A minimum MD file contains only rows of 14 values. The following example shows the values for 6 sampling points associated with snapshot 21.

```
// 250 23800 21 1.0 -1 201.2 0.4 0.0 1.0 7.2 2100.0 1.0 0
// 1250 23800 21 1.0 -1 204.6 0.2 0.0 1.0 6.2 0.0 1.0 0
// 1950 23800 21 1.0 -1 206.2 0.1 0.0 1.0 5.1 0.0 1.0 0
// 250 22200 21 1.0 -1 2.8 3.3 0.0 1.0 71.4 2100.0 1.0 0
// 1250 22200 21 1.0 -1 0.6 1.7 0.0 1.0 71/4 0.0 1.0 0
// 1950 22200 21 1.0 -1 -0 0.4 0.0 1.0 69.4 0.0 1.0 0
```

## Related Topics

[Creating a Measured Data File](#)

[Optimizing Parameterized Expressions](#)

# Process Recipe File Format

Generated by: **Recipe** tab or ASCII text file

The process recipe file contains commands describing the process steps your silicon goes through in the manufacturing process.

## Format

- Process steps (deposition, etch, and CMP) must be presented in the same order as they are executed in manufacturing.
  - The first non-comment line must be the [initialize](#) command.
  - This must be followed by one or more process commands ([deposit](#), [etch](#), [cmp](#)) that mirror your manufacturing flow.
  - All model ([define\\_model](#)) and mask definition ([define\\_mask 2D](#)) commands must be supplied at the end of the file.
- Parameters are separated by whitespace. *Do not use a space between a value and its unit.*
- Commands that extend over several lines must use a backslash (\) as the final character at the end of a line to indicate the command continues on the next line.
- Comments are lines that begin with a pound sign (#).
- When representing physical quantities, you can provide the value and units of measure. If units are not specified, it is assumed to be in the default units for that parameter.

No spaces should separate the value and units.

Abbreviated units of measure for the process recipe file are shown in the following table.

**Table 4-2. Default Units of Measure**

Type of Data	Unit of Measure	As It Appears In the Process Recipe File
time	seconds	s
thicknesses/vertical positions	angstroms (Å)	A
lateral extensions	pixel size in microns (µm)	um
pressure and pad stiffness	kilo Pascals	kPa
rates and velocities	angstroms per second (Å/s)	A/s
reaction rates	Nanometers per second	nm/s
overPolish	percentage	%

- Any command may appear multiple times in the recipe, including initialize and save.

**Tip**

 To improve simulation time, after each simulation [save](#) the optimized process model and load it into the next simulation using the [initialize](#) command.

---

## Parameters

Command	Description
<a href="#">cmp</a>	Indicates a polishing step, typically for a copper or oxide layer. Each platen should have a cmp command.
<a href="#">define_mask 1D</a>	Defines a series of trench arrays that can be used as input instead of an existing layout. (The 2D form is preferred.)
<a href="#">define_mask 2D</a>	Defines a region of trench arrays that can be used as input instead of an existing layout.
<a href="#">define_model</a>	Defines models (sets of parameters) that are called by the cmp, deposit, and etch command.
<a href="#">deposit</a>	Adds a new layer of material to the simulated process.
<a href="#">etch</a>	Indicates a trench etching step.
<a href="#">initialize</a>	<b>Required.</b> Specifies the geometry at the start of the simulation. This command can also be used to load saved grids when modeling multiple layers.
<a href="#">save</a>	Saves the simulation information to a file.

# cmp

Defines a polishing step to be used within the recipe for a copper or oxide layer.

## Usage

```
cmp model=model_name \
[dt=time_step] [dtOver=value] [dz=value] [dzOver=value] \
[nsteps=integer] \
[OpenAreaFraction=ratio targetMaterial=material_name] \
[overPolish=floating_point_value] [overThickness=floating_point_value] \
[overTime=floating_point_value] \
[planarizationlength=length] [pressure=floating_point_value][PressureVersion=2] [restart] \
[Snapshot=snap_ID] [{SaveDt=time_step | SaveDz=height_step} File=file_name] \
[saveDishing] [saveErosion] [saveMaskData] [savePressure] [saveThicknessNT] \
[saveThicknessT] [saveZ1] [saveZ2] \
[targetMaterial=material_name] \
[targetThicknessNT=height [targetWeight=grid | targetX=coordinate targetY=coordinate]] \
[targetThicknessT=height [targetWeight=grid | targetX=coordinate targetY=coordinate]] \
[thickness=floating_point_value] [time=floating_point_value]
```

## Parameters

- **model=***model\_name*

A required keyword and argument supplying the name of the model to use when simulating the CMP process. The *model\_name* must be the name of a model defined using the [define\\_model](#) command.

You must specify either a model, or a thickness or a set of keywords that define the time, pressure and other factors that define the amount of material that is removed by this step.

- **dt=***time\_step*

An optional argument used to specify the time interval of the polished material, expressed in seconds. This parameter is used with the parameter nsteps.

- **dtOver=***value*

An optional parameter that defines the incremental time step in the overpolish stage.

- **dz=***height\_step*

An optional parameter that defines the thickness step per time step.

- **dzOver=***value*

An optional parameter that defines the incremental thickness steps in the overpolish stage.

- **File=***grid\_name*

An optional keyword and string supplying the base name to use for grid files generated for this process step. You must specify this keyword when using SaveDt or SaveDz.

- **nsteps=integer**  
An optional parameter defining the number of steps to polish. This parameter is used if dt is not specified.
- **OpenAreaFraction=ratio**  
An optional argument specifying the ratio of the “open” grid frames to total frames at which to stop polishing. “Open” grid frames are those in which the underlying material as specified by targetMaterial has been exposed. Total grid frames include only those with trenches that have targetMaterial underlying. When using OpenAreaFraction, you must also specify targetMaterial.  
Valid ratios are between 0 and 1 (0 - 100%). There is no default value.
- **overPolish=floating\_point\_value**  
An optional argument defining the fraction of the process time from start until the topmost layer is cleared in each frame. overPolish is expressed as a percentage.
- **overThickness=floating\_point\_value**  
An optional argument defining the thickness that must be removed during the over polish stage.
- **overTime=floating\_point\_value**  
An optional argument defining the length of time of over polishing.
- **planarizationlength=length**  
An optional argument specifying the length over which pattern density is averaged for the CMP simulation.
- **pressure=floating\_point\_value**  
An optional keyword and argument defining the pressure used in the CMP process, expressed in kilopascals.  
You must specify either a model, or a thickness or a set of keywords that define the time, pressure and other factors that define the amount of material that is removed by this step.
- **PressureVersion=2**  
An optional parameter that activates the Generation 2 pressure model. If creating the recipe in the GUI, use the “other” field. The Generation 2 pressure model is useful when simulating pressure over rough surfaces and requiring 200 iterations or more to converge.
- **restart**  
Before defining a CMP model definition for any platen, restart must be specified. This keyword indicates that the chip has been allowed to cool between previous CMP processes. Temperature effects from the previous simulations are no longer valid.  
This keyword is relevant only if the polishing rate is defined as time-dependent function. By default, the rate is computed by adding the time in the current simulation to the sum of the

times of all preceding cmp steps. Splitting the simulation of a cmp step into several cmp commands is used to report the intermediate state and to save files at intermediate times.

- Snapshot=*snap\_ID*

An optional keyword and positive integer used to assign a unique identifier to state after this polishing step is complete. The *snap\_ID* must match the value used in the Snapshot column in the measured data file to associate measured or simulated data with this state.

- SaveDt=*time\_step* / SaveDz=*height\_step*

An optional keyword and value used to instruct the tool to save simulation data periodically throughout the duration of the simulation. SaveDt saves every *time\_step* seconds. SaveDz saves every *height\_step* angstroms.

You must specify the File argument and one or more of the following keywords, which define the type of data to save for each grid:

saveDishing — saves the difference between the tops of trenches and non-trenches in angstroms.

saveErosion — saves the difference between the local maximum height and the z1 in angstroms.

saveMaskData — saves the simulated geometry.

savePressure — saves the calculated local pressure in kPA.

saveThicknessNT — saves the height of the local field region (non-trench) in angstroms.

saveThicknessT — saves the height of the local trench tops in angstroms.

saveZ1 — equivalent to saveThicknessNT.

saveZ2 — equivalent to saveThicknessT.

- targetMaterial=*material\_name*

An optional keyword specifying the layer to which OpenAreaFraction, targetThicknessT, or targetThicknessNT apply.

- targetThicknessNT=*height*

An optional parameter that specifies the non-trench (field) thickness at which to stop polishing. When this is set, the time setting is ignored.

The way the simulated thickness is calculated depends on the settings of targetX/targetY or targetWeight.

- If *neither* targetX/targetY nor targetWeight is set, the targetThicknessNT value is compared to the average of all NT grids. This is the default.
- If targetX and targetY are set, then the average thickness at the grid that includes that coordinate is used. If the tool detects that the coordinates are not in a field region, it issues an error.

- If targetWeight is set, then targetThicknessNT is compared to the weighted average of the thickness of all grids.

If *height* exceeds the current thickness an error is generated.

- **targetThicknessT=*height***

An optional parameter that specifies the thickness at which to stop polishing. The thickness is measured from the bottom of the trench to the top of the material within the trench. When targetThicknessT is set, the time setting is ignored.

The way the simulated thickness is calculated depends on the settings of targetX/targetY or targetWeight.

- If *neither* targetX/targetY nor targetWeight is set, the targetThicknessT value is compared to the average of all T grids. This is the default.
- If targetX and targetY are set, then the average thickness at the grid that includes that coordinate is used. If the tool detects that the coordinates are in a field region, it issues an error.
- If targetWeight is set, then targetThicknessT is compared to the weighted average of the thickness of all grids.

If *height* exceeds the current thickness an error is generated.

- **targetWeight=*grid***

An optional parameter naming a grid file with weights. The weights are applied to the simulated thickness for each pixel. The default weighting is 1 for all grids. Cannot be specified with targetX targetY.

- **targetX=*coordinate* targetY=*coordinate***

An optional pair of parameters that can be used with targetThicknessT or targetThicknessNT. The pair specify the coordinates at which to measure the simulated thickness. Both targetX and targetY must be specified when used. They cannot be used with targetWeight.

- **thickness=*floating\_point\_value***

An optional keyword and argument supplying the amount of material removed by this CMP process, expressed in angstroms.

You must specify a thickness or a set of keywords that define the time, pressure and other factors that define the amount of material that is removed by this step.

When you control cmp using the Thickness keyword, polishing is considered to be complete as soon as simulated thickness reaches the result specified here.

- **time=*floating\_point\_value***

An optional keyword and argument indicating the duration of this process in seconds. When specified, you must also specify the pressure.

You must specify either a model or a thickness or a set of keywords that define the time, pressure and other factors that define the amount of material that is removed by this step.

## Description

Defines a polishing step to be used within the recipe for a copper or oxide layer. Note that when multiple platens are required for a layer, each platen must use a separate cmp model command with a unique name.

The following parameters support [Symbolic Names](#):

OpenAreaFraction	overPolish	overThickness	overTime
pressure	rate	thickness	time

Symbolic names can be used in the command to indicate that the parameter is to be fitted to empirical data. Any parameter assigned a numeric value is untouched during model calibration. That is, this parameter is not fitted or optimized.

## Examples

### Example 1

This is the simplest form of the cmp command:

```
cmp model=platen1
```

When using only a model name, the model must be defined with [define\\_model](#) using the CMP model parameters.

### Example 2

A more typical cmp command defines several parameters:

```
cmp time=56s dz=0.1A model=platen2 pressure=2.8psi Snapshot=23 restart
```

The parameters here are for a second-stage polish that runs for 56 seconds at 2.8 PSI.

## Related Topics

[Measured Data File Format](#)

## define\_mask 1D

Defines a 1D mask in terms of a series of sections defined by their lengths, widths, and spaces.

### Usage

```
define_mask name=mask_name \
  {lengthn=len_n [widthn=width_n] [spacen=space_n] [density] \
  [dwidthn=d_width_n] [dspacen=d_space_n] [numbern=num_n]} ...
```

### Parameters

- **name=mask\_name**

A required keyword and string argument supplying a unique name for this mask. This is the name by which the mask is referenced in other commands.

- **lengthn=len\_n**

A required keyword and floating point value used to define the overall length of the test structure.

- **widthn=width\_n**

An optional keyword and floating point argument used to define the width of structure *n* in microns.

- **spacen=space\_n**

An optional keyword and floating point argument used to define the space from structure *n* to the next structure, in microns.

- **density**

The missing value of spaces or widths are computed from the value specified in density. If width and space are defined, density is computed. Each table entry must be cleared before calculations are carried out.

- **dwidthn=d\_width\_n**

An optional keyword and floating point argument used to define an increment to be used in calculating the width of the next feature created following this definition. The width of any feature in this structure is equal to (*widthn + dwidth*). dwidth must be expressed in microns. The default value is 0.

- **dspacen=d\_space\_n**

An optional keyword and floating point argument used to define an increment to be used to calculate the spacing in the next feature created following this definition. The space of any feature in this structure is equal to (*spacen + dspace*). dspace must be expressed in microns. The default value is 0.

- **numbern=num\_n**

An optional keyword and integer argument used to define the number of structures of this type to create.

## Description

Defines a 1D mask in terms of a series of sections defined by their lengths, widths, and spaces. By defining the mask directly in the process recipe file, you can run simulations without using a layout file. This is particularly useful during model calibration when simulating a few key features provided sufficient data. For each structure, you must specify at least one of the following values: length, width, or space.

Sections in the mask are indexed according to their position beginning with index=0. Each section definition must include one or more structure keywords, the name of which contains the index as a post-fix. For example, length0 is the length in the first section definition and length7 is the length in the eighth section definition.

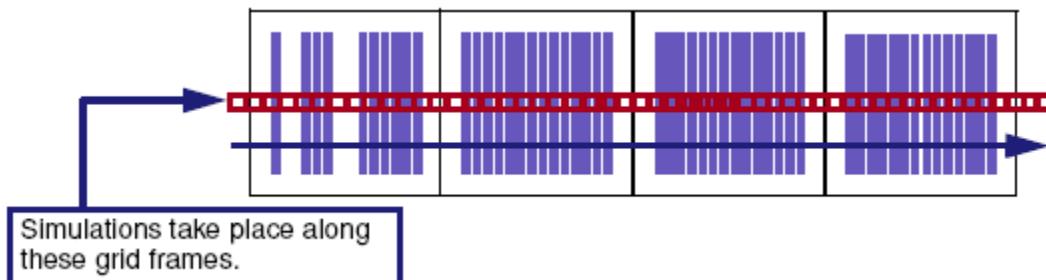
The first section in the mask must define a field area. It is defined as a large value for space0 and no width0 keyword. Subsequent sections are defined in terms of their width or space measurements. Including the keyword numbern creates a section comprised of a single repeated feature. Including numbern plus dwidtn or dspacecn creates a section comprised of a set of related features, where dwidtn or dspacecn supply values used to increment width and space. In this case widthn and spacen values define the first feature. The next feature is (widthn + dwidtn) and (spacen + dspacecn) and so on until the length of the test structures is satisfied.

For example, assume the following structure definitions are supplied as part of a define\_mask command:

```
define_mask name=mask_1D \
space0=20um \
length1=20um width1=0.2 space1=0.3 \
length2=20um width2=0.3 space2=0.2um \
length3=20um width3=0.4 space3=0.6 \
length4=20um width4=1 space4=1 \
length5=20um width5=2 space5=1
```

The mask command defines six sections of mask\_1D. Each structure is 20 um long. The first consists of only spaces (that is, a field region), followed by sections with different width and spaces. Mask\_1D can then be used in an etch step simulation. This is one of two mask schemes that characterize how simulations are carried out. A 1D mask is used for repeated arrays of lines and spaces defined along the X direction. There is no hard limit to how long these test structures can stretch.

**Figure 4-1. Mask1D**



## define\_mask 2D

Defines a 2D mask array of test structures defined by their lengths, widths, and spaces.

### Usage

```
define_mask name=mask_name LineUp={XY | YX} \
{Simple_Block | Isolated_Line | Asymmetrical_Block | Staggered_Block}...
```

### Parameters

- **name=mask\_name**

A required keyword and string argument supplying a unique name for this mask. This is the name by which the mask is referenced in other commands. The name must start with a letter.

- **LineUp={XY | YX}**

A required keyword that specifies whether the blocks are in row-column or column-row order. It does not affect the block identifier, which is always XY. It does affect how blocks are placed: in XY order, the column of blocks is drawn and then the next column starts at the maximum displacement to not overlap. In YX order, the row of blocks are drawn and the next start at the maximum height.

---

#### Note

 \_XY values in these parameters denote the column and row of the block in the overall mask. For example, ArrayWidth\_A1=400 specifies that the first test structure (row A, column 1) has a width of 400 um.

---

- **Simple\_Block**

Indicates a set of parameters that describe a block of trenches in an array. Simple\_Block sets can be created in the [Mask2D Dialog Box](#) without expanding the default fields. The syntax for a Simple\_Block is as follows:

```
{BlockWidth_XY=value BlockHeight_XY=value \
ArrayWidth_XY=value ArrayHeight_XY=value \
Width_XY=value Space_XY=value Orientation_XY={v | h}}
```

See “[BlockWidth\\_XY](#)” for more information.

- **Isolated\_line**

Indicates a set of parameters that describe a line that does not appear to be part of an array, as shown in [Figure 4-3](#) on page 77. The LineSpace field is hidden by default; to expose it, click the + in the Mask2D dialog box.

```
{BlockWidth_XY=value BlockHeight_XY=value \
ArrayWidth_XY=value ArrayHeight_XY=value \
[Width_XY=value] [Space_XY=value] [Orientation_XY={v | h}] \
LineSpace_XY=value}
```

See “[BlockWidth\\_XY](#)” for more information.

- **Asymmetrical\_Block**

Indicates a set of parameters that describe a block that can have different densities, spacings, and widths assigned to the left and right sides.

```
{BlockWidth_XY=value BlockHeight_XY=value \
ArrayWidth_XY=value ArrayHeight_XY=value \
[Width_XY=value] [Space_XY=value] [Orientation_XY={v | h}] \
[DWidth_XY=value] [DSpace_XY=value] \
[ArrayWidthLeft_XY=d_width_n] [SpaceLeft_XY=value]
[WidthLeft_XY=value] \
[ArrayWidthRight_XY=value] [SpaceRight_XY=value]
[WidthRight_XY=value] }
```

See “[BlockWidth\\_XY](#)” for more information.

- **Staggered\_Block**

Indicates a set of parameters that describe a block that may have rectangles placed in a staggered pattern within the array area. (If you use “Alignment=aligned”, polygons are placed beginning at the start of each row in the array and are not staggered.)

```
{BlockWidth_XY=value BlockHeight_XY=value \
ArrayWidth_XY=value ArrayHeight_XY=value \
Alignment={aligned | staggered} \
[Width_XY=value] [Space_XY=value] [Orientation_XY={v | h}] \
[DWidth_XY=value] [DSpace_XY=value] \
[Length_XY=value] [VSpace_XY=space_n] }
```

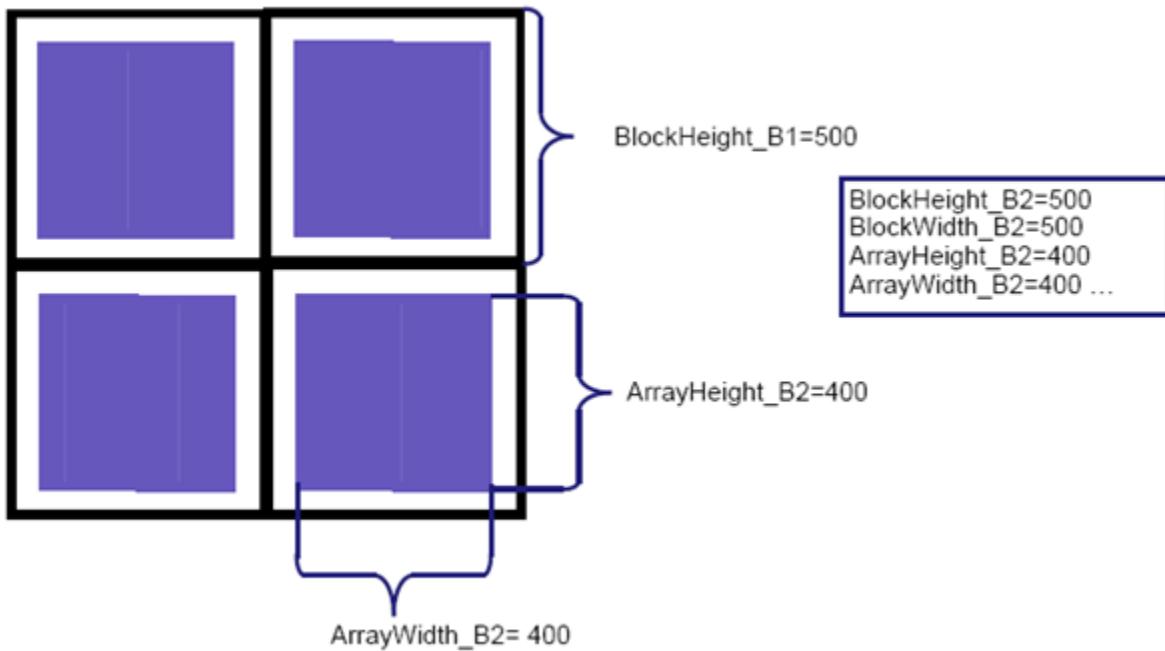
See “[BlockWidth\\_XY](#)” for more information.

- **BlockWidth\_XY=value BlockHeight\_XY=value**

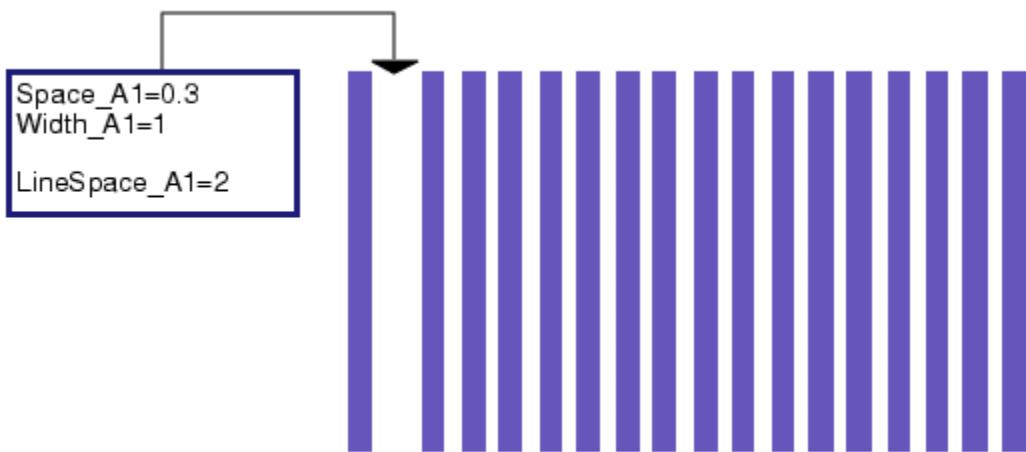
A required set of parameters specifying the width and height of the array block that contains the test structures. Differences between **BlockWidth**, **BlockHeight**, **ArrayWidth** and **ArrayHeight** can be seen in [Figure 4-2](#). It is desirable to leave the extra area around the array in order to have sufficient field region to properly calibrate the polishing models.

Prior to v2009.2, blocks were specified once for the entire mask using the parameter `blocksize`.

**Figure 4-2. Specifications for Block versus Array**



- **ArrayWidth\_XY=value ArrayHeight\_XY=value**  
A set of required keywords that specifies the width and height of the test structures.
- **Width\_XY=value**  
An optional keyword used to define the width of each line structure in the array block XY. *Value* is specified in microns.
- **Space\_XY=value**  
An optional keyword that specifies the separation between lines in the array block XY. *Value* is specified in microns.
- **Orientation\_XY={v | h}**  
An optional keyword that specifies the orientation of the trenches in the array block XY.
  - v — vertical, with widths and spaces alternating in the X direction. This is the default.
  - h — horizontal, with trenches lying along the X direction.
- **LineSpace\_XY=value**  
An optional keyword that defines an isolated line before the beginning of the array as in [Figure 4-3](#). *Value* is specified in microns.

**Figure 4-3. LineSpace Example**

- **DWidth\_XY=*value***

An optional parameter used to define a change to the width of a trench as successive rows are drawn within an array block XY. For example, a block with Width of 2.0 and DWidth of 0.1 results in trenches drawn with widths 2.0, 2.1, 2.2, etc., within the block. The default value is 0.0 microns, or no change.

- **DSpace\_XY=*value***

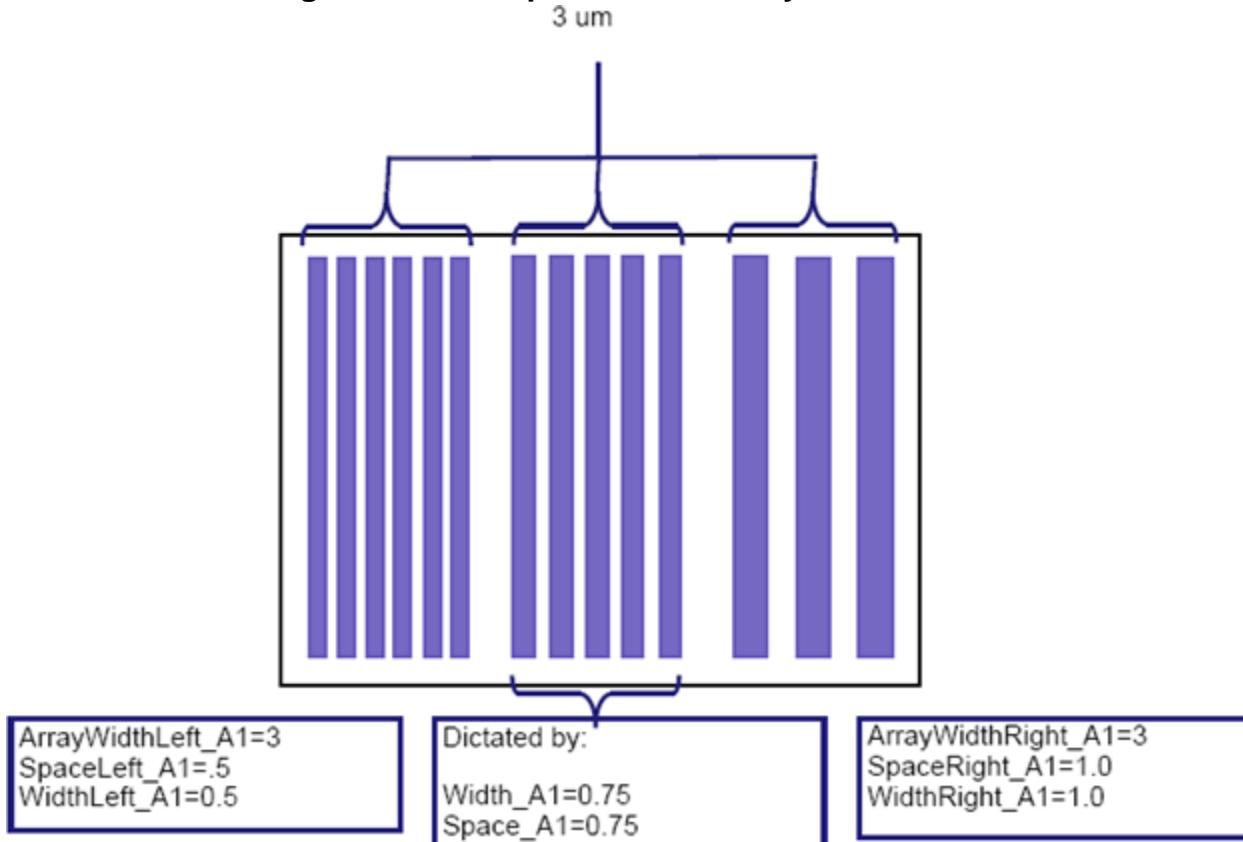
An optional parameter used to define a change to the spacing between trenches as successive rows are drawn within an array block XY. For example, a block with Space of 2.0 and DSpace of -0.01 results in trenches drawn with separations of 2.0, 1.99, 1.98, etc., within the block. The default value is 0.0 microns, or no change.

- **ArrayWidthLeft\_XY=*value* SpaceLeft\_XY=*value* WidthLeft\_XY=*value***

**ArrayWidthRight\_XY=*value* SpaceRight\_XY=*value* WidthRight\_XY=*value***

These keywords and value pairs define different degrees of separation and width sizes for three evenly divided blocks in the array, as shown in [Figure 4-4](#).

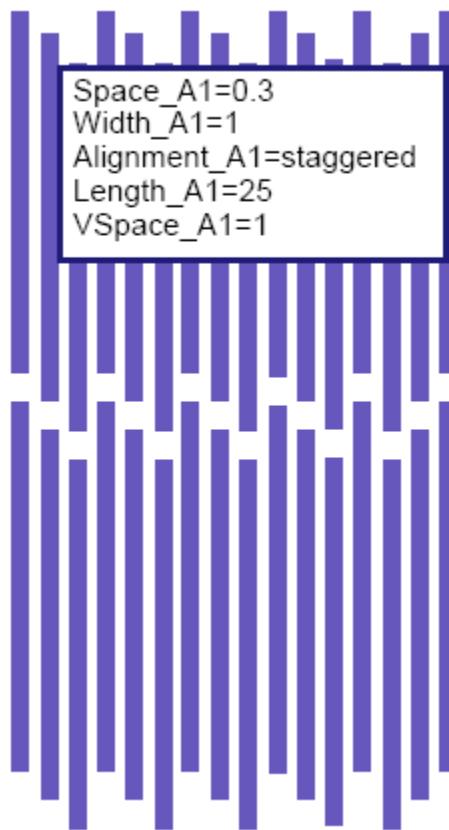
**Figure 4-4. Example of Block Array Distribution**



- Alignment\_XY={aligned | staggered}

An optional keyword indicating whether trenches are to be placed in block XY so that ends are at the same X or Y position, or they should be placed so that the spaces between them do not form a rectangular grid as in [Figure 4-5](#).

When you specify Alignment, you must also specify Length and VSpace.

**Figure 4-5. Example of Staggered Trench Distribution**

- **Length\_XY=*integer***  
An optional keyword and floating point argument used to define the overall length of the trenches in microns. The default length when Alignment is not specified is the length of the array.
- **VSpace\_XY=*integer***  
An optional keyword and floating point argument used to define the space between staggered trenches, in microns. There is no default value.

## Description

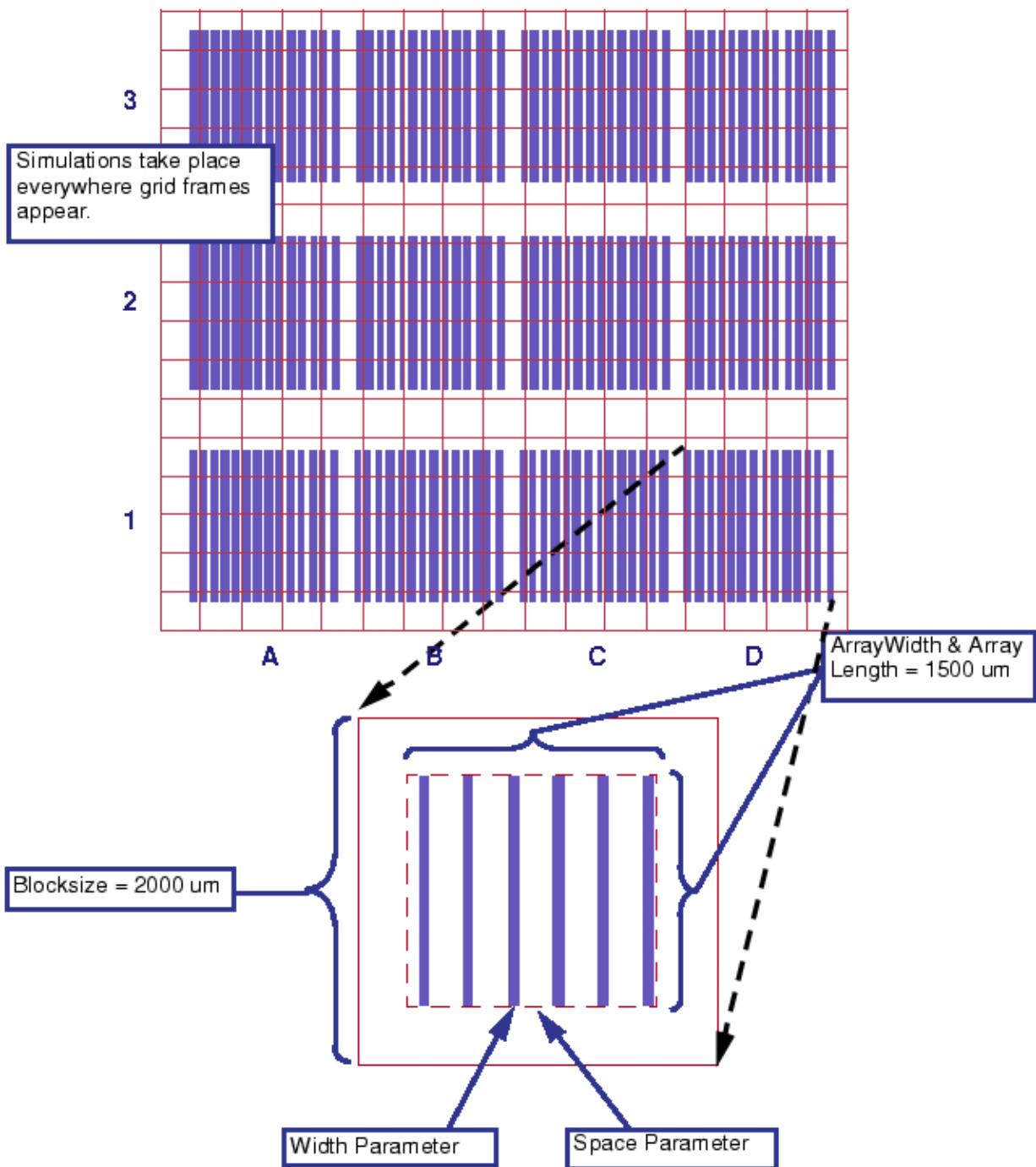
Defines a 2D mask array of test structures defined by their lengths, widths, and spaces. By defining the mask directly in the process recipe file, you can run simulations without using a layout file. This is useful during model calibration when simulating a few key features with only limited data.

For each structure, you must specify arguments for the following parameters: BlockWidth, BlockHeight, ArrayWidth, ArrayHeight, Space, and Width. The Array values should be less than the Block values to allow for field regions. Each argument identifies the feature it belongs to with a suffix \_XY, where X and Y are replaced by the row and column identifiers, for example, BlockWidth\_A1.

Sections in the mask are indexed according to their position beginning with index=0. Each section definition must include one or more structure keywords, the name of which contains the index as a post-fix.

Defining a mask pattern is performed with the define\_mask command in the process recipe file. A 2D mask extends in the X and Y directions. In CMP Model Builder, a mask with test structures can be defined in the process recipe file or a layout can be used for simulation.

Figure 4-6. Mask2D Example



## Examples

### Example 1

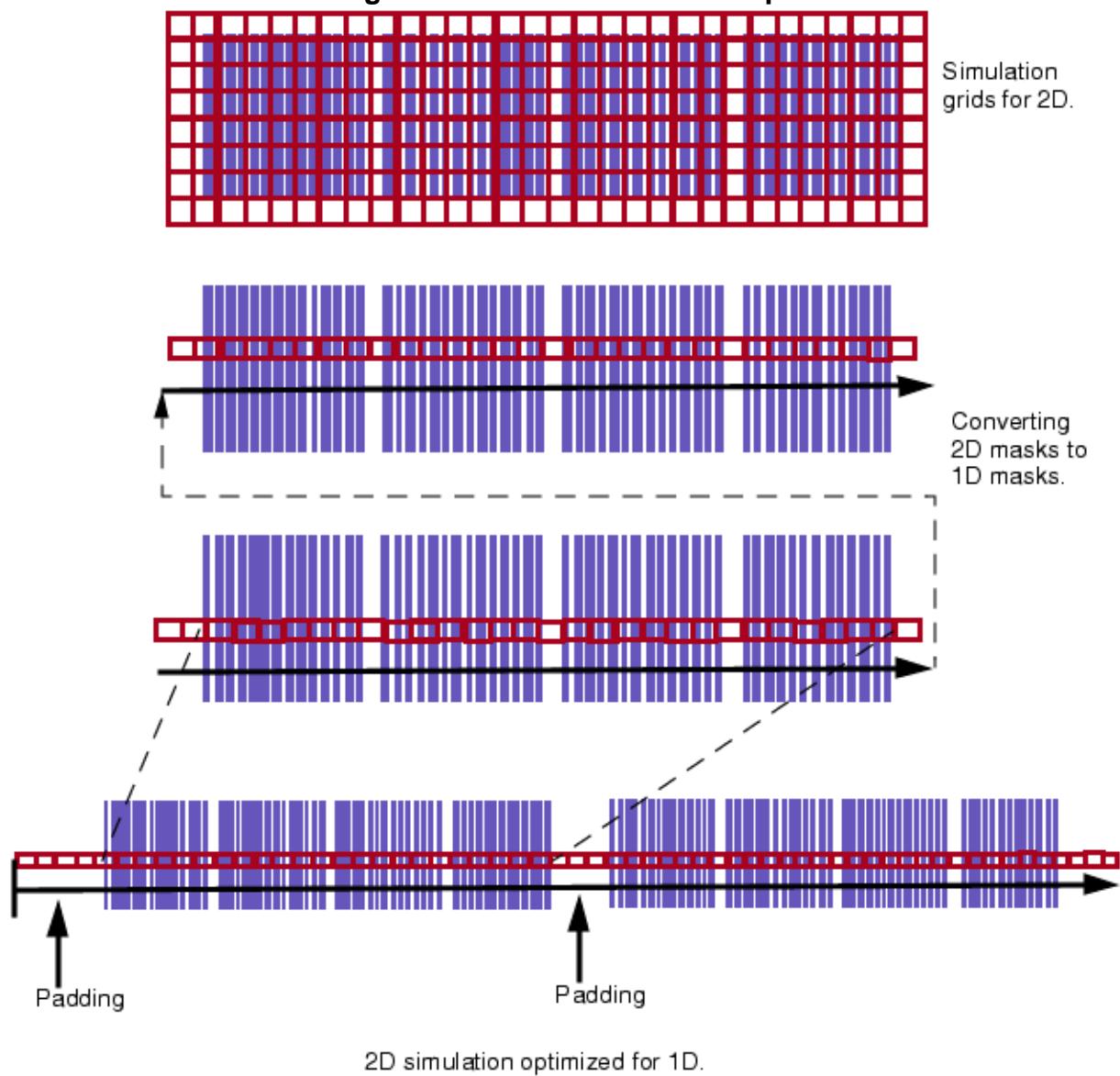
```
define_mask name=mask2D \
blocksize=500 \
ArrayWidth_A1=400 ArrayHeight_A1=400 Width_A1=0.5 Space_A1=0.5 \
ArrayWidth_B1=400 ArrayHeight_B1=400 Width_B1=0.6 Space_B1=1 \
ArrayWidth_C1=400 ArrayHeight_C1=400 Width_C1=0.7 Space_C1=2 \
ArrayWidth_A2=400 ArrayHeight_A2=400 Width_A2=0.8 Space_A2=0.5 \
ArrayWidth_B2=400 ArrayHeight_B2=400 Width_B2=0.9 Space_B2=0.5 \
ArrayWidth_C2=400 ArrayHeight_C2=400 Width_C2=1 Space_C2=0.5
```

The define\_mask 2D command creates a 3X2 array of test structures, each with different widths and spaces. Each array has a different width. The etch mask2D command can then be used in an etch step simulation.

### Example 2

For a mask that is 2000 um x 2000 um, with a default pixel size of 20 um, there are 10,000 grid frames used in simulations. For arrays of repeated structures like lines and spaces, using this many grid frames is redundant and unnecessary. Converting a 2D mask to a 1D mask accelerates runtime by only simulating along the center of the mask. In this example, a Mask2D requires 10,000 frames while a Mask1D only needs 100 frames.

Figure 4-7. ConvertTo1D Example



## define\_model

Creates a model of an ECD or CMP process.

### Usage

```
define_model name=model_name type={cmp | ecd | lookup} {param=value}...
```

### Parameters

- **name=***model\_name*

A required argument that assigns a name to the model. This is the name by which the model is referenced in other commands within the Process Recipe File.

- **type=**{cmp | ecd | lookup}

A required argument defining the type of model. Allowed values are lookup, cmp or ecd.

- **param=***value*

A keyword / value pair defining the parameters to be used for this model. You must supply at least one model parameter.

- Any parameter for which *value* is a numeric value is untouched during model calibration. That is, this parameter is not fitted or optimized.
- Any parameter for which *value* is a symbolic value (the name of the variable that stores the value) is fitted during model calibration. The tool attempts to identify the optimal value that fits the data.

---

#### Note



Assigning one symbolic value to multiple parameters, either within a single model command or within multiple commands, defines a dependency between these parameters. The tool optimizes for the best fit with regards to all occurrences.

---

The parameters you can specify vary according to the type of model being defined. For lookup tables, choose the parameters and values based on what argument is calling the table.

### ECD Model Parameters

Typically supplied parameters for an electroplating model are k1, k2, keq, and phistar.

Additional parameters supply values for the accelerator and suppressor concentrations, the acceleration factor and the model for the regular and superfill steps.

- **k1=dissolution\_rate**

A required argument defining a reaction accelerator constant. It represents the dissolution rate of the accelerator per second. Allowed initial values are between 0 and 1, inclusive. The default is 0.035. After optimization, allowed values may exceed 1.

- $k2=dissolution\_rate$

A required argument defining a reaction suppressor constant. It represents the dissolution rate of the suppressor per second. Allowed initial values are between 0 and 1, inclusive. The default is 0.005. After optimization, allowed values may exceed 1.

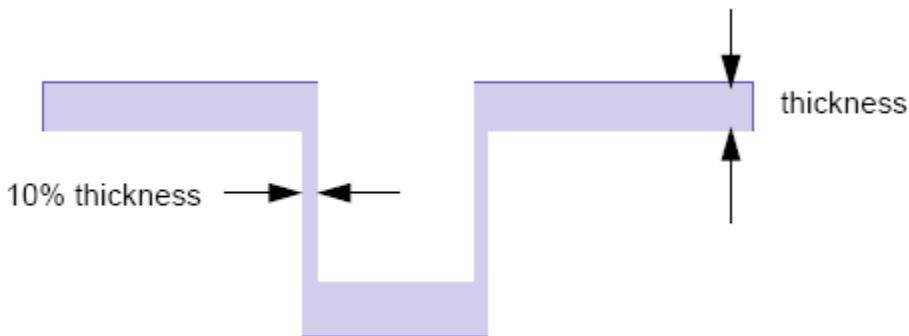
- $keq=equilibrium$

A required argument defining an equilibrium constant for suppressor capture and dissolution. Allowed values are between 0 and 20, inclusive. The default is 15.

- $phistar=equilibrium$

A required argument defining an equilibrium constant for the accelerator. Allowed values are between 0 and 1, inclusive. The default is 0.005.

- $anisotropyFactor=percentage$



An optional parameter representing the amount of sidewall accumulation of the deposited material. For example, specifying 0.1 means that ten percent of the final thickness in the field regions are deposited on the sidewalls. Specifying 10% or 0.1 for the anisotropyFactor carries out the same action.

When anisotropyFactor is set to 0, the model behaves like anisotropic deposition is applied and it deposits only horizontal surfaces. Material is not deposited on side walls in this case.

- $CuDepletionFactor=value$

An optional argument used to enable modeling of cupric ion depletion. By default, cupric ion depletion is not specifically modeled. This can result in greater uniformity in the field copper than really exists.

- $kacc=initial\_value$

An optional argument used to define the initial value for the acceleration factor  $k$ . The default value is 0.

- $k1s=value$

An optional argument used to define the reaction accelerator constant used during the superfill step. By default this is set to  $k1$ , which is used while the surface of the trench is lower than the surface of the non-trench.

- **k2s=value**  
An optional argument used to define the reaction suppressor constant used during the superfill step. By default this is set to k2, which is used while the surface of the trench is lower than the surface of the non-trench.
- **kexp=value**  
An optional argument that gives an exponential factor affecting the calculations for suppressor surface coverage and velocity calculation. The default value is 0.
- **LECD=length**  
An optional argument defining a diffusion length, expressed in um, to be used for Gaussian averaging when simulating cupric ion depletion. Allowed values are between 1000 and 4000, inclusive.
- **rmod=0**  
An optional argument used to indicate that this model represents a regular electroplating step. (Regular and superfill electroplating are modeled using different equations.)
- **sfactor=scale**  
An optional argument used to scale down the geometry effect during the superfill step. The default value is 1 (no scaling). Values can range from 0 (ignore geometry effect entirely) to 1. When set, the changing area of trench and sidewall changes the concentration of accelerator.
- **smod=0**  
An optional argument used to indicate that this model represents a superfill electroplating step.
- **wCrit=width**  
An optional argument indicating critical trench width for superfilling. This affects the k1s parameter. The critical width is the point at which the accelerator curve cuts off. For narrow trenches, the accelerator (k1s) depends linearly on width; for wide trenches, superfilling does not occur.

### CMP Model Parameters

When polishing occurs in stages, you must define a separate model for each stage or platen. Thus, a three stage polishing involving bulk copper removal, copper clear, and barrier polish requires a minimum of three cmp models.

Each cmp model must contain parameters defining the effects of polishing on each of the materials that are likely to be encountered.

- **material=material\_name**  
A required argument used to supply the name of the material to be polished. This keyword is followed by the model parameters for this material. This keyword must follow any parameters used to specify general characteristics of the model (not specific to any material) such as spad and pressure.

The CMP algorithm associates polishing effects with layers by comparing this material name to the material names specified in either the initialize command or one of the deposit commands.

You may specify as many materials as needed and there are no restrictions on material names. However, saved grid files may show incorrect thicknessNT values if the initialize command and the first deposit command use the same material name.

- **rate=height\_per\_second | ratefactor=floating\_point\_value**

Either **rate** or **ratefactor** must be specified once per material. At least the top material must use the **rate** parameter.

**rate** — defines the blanket copper removal rate for a platen, in angstroms per second.

The **rate** parameter establishes the overall polishing rate of the material (Cu, TiN, or TaN). This can be a user-specified value or an optimization variable.

**ratefactor** — The ratefactor parameter specifies a rate of polishing in relation to another material. That is, this expresses the removal rates of other materials as a percentage of the base material (for example, Cu or Tungsten) in different stages of polishing or overpolishing.

To display **ratefactor**, click the + in the upper right corner of the CmpModel dialog box.

- { **dmax=value dmin=value** } | **Hex=step\_height**

A required parameter set specifying the effect of trench depth on polishing rate. **Hex** is only used for blanket copper removal (single material polishing). The **dmax** and **dmin** parameters can be used for all materials, including copper.

---

**Tip**

For better results, set **Hex**, **dmax**, and **dmin** to a variable ([symbolic name](#)). The critical step height has a large effect on accuracy, so it should be among the first parameters optimized.

---

Each of the three parameters has three associated optional parameters for tuning the critical step height. Use alpha1, beta1, and s1 with **Hex**; alpha2, beta2, and s2 with **dmax**; and alpha3, beta3, and s3 with **dmin**.

**dmax=value**

A parameter specifying the critical step height at which the pressure at the bottom of a trench is zero during polishing. Below the critical step height, trench bottoms are polished at a slower rate than non-trench areas. Above it, trench bottoms are not polished.

The default value for the top material, assumed to be copper, is 0. For other materials, the default value is 300 angstroms.

**dmin=value**

A parameter similar to **dmax**, used for negative dishing.

The default value for the top material, assumed to be copper, is 0. For other materials, the default value is 50 angstroms.

**Hex=step\_height**

A parameter defining the critical step height for blanket copper removal (single material polishing), in angstroms. When the distance between top and bottom regions is greater than this, polishing does not touch the bottom regions.

The default value for the top material, assumed to be copper, is 10 angstroms. For other materials, **Hex** is not used; instead, use **dmax** and **dmin**.

- **alpha1=value**

An optional parameter defining the dependency between trench width and critical step height for **Hex**. Allowed values are between 0 and 1, inclusive.

The default value for the top material, assumed to be copper, is 0.15. For other materials, the default value is 0.

---

**Tip**

 For better results, set **alpha1** to a variable ([symbolic name](#)). The width/space dependency has a large effect on accuracy, so it should be one of the first ones optimized.

---

- **alpha2=value**

An optional parameter for tuning the dependency between trench width and **dmax**.

The default value for the top material, assumed to be copper, is 0. For other materials, the default value is 0.3. Values must be between 0 and 1, inclusive.

- **alpha3=value**

An optional parameter for tuning the dependency between trench width and **dmin**.

The default value for the top material, assumed to be copper, is 0. For other materials, the default value is 0.7. Values must be between 0 and 1, inclusive.

- **beta1=value**

An optional parameter defining the dependency between trench spacing and critical step height for **Hex**. Allowed values are between 0 and 1, inclusive.

The default value for the top material, assumed to be copper, is 0.1. For other materials, the default value is 0.

- **beta2=value**

An optional parameter for tuning the dependency between spacing and **dmax**.

The default value for the top material, assumed to be copper, is 0. For other materials, the default value is 0.29. Values must be between 0 and 1, inclusive.

- **beta3=value**

An optional parameter for tuning the dependency between spacing and **dmin**.

The default value for the top material, assumed to be copper, is 0. For other materials, the default value is 0.1. Values must be between 0 and 1, inclusive.

- **C=value**

An optional parameter for handling edge rounding in oxide removal. The default value is 0 (no special handling).

- **nD=value**

An optional argument that is used to model the non-linear pad bending due to pressure. The default is linear, equivalent to setting this to 1.

- **NonPrestonRateFactor=value**

An optional argument that is used by the non-Preston model. Values should be between -1 and 1 inclusive. When *value* is negative, it is interpreted as the pressure at which the removal rate falls to zero. When the value is positive or 0, it is interpreted as the removal rate when there is no pressure. The default behavior is the Preston model, equivalent to setting this to 0.

- **nP=value**

An optional argument that is used to model the removal rate's non-linear dependence on pressure. The default behavior is linear, equivalent to setting this to 1.

- **PowG2=value**

An optional argument defining an exponent to be used in calculating dishing.

- **pressure=force\_per\_area**

An optional argument used to specify the pressure for platen 1, specified in kPa. This argument must precede any instances of the material keyword.

- **rfb=value**

An optional argument specified together with the wb argument and used for the enhanced perimeter model (EPM). This argument is enabled when rfb > 0 and specifies a rate correction for the material due to transitions between trench and non-trench patterns. The value is a floating-point number that increases the polishing rate of the material (value > 1) or decreases the polishing rate of the material (value < 1). See “[Perimeter](#)” on page 211 for details on the perimeter model.

- **s1=value**

An optional parameter that can be that controls the simulation space used in fitting the critical step height portion of the polishing model for copper, when using **Hex**. The value is in um and must be  $\geq 0$ . The default value is 100 um.

- **s2=value**

An optional parameter that can be that controls the simulation space used in fitting the critical step height for a positive dishing (**dmax**) model. The value must expressed in um and must be  $\geq 0$ . The default value is 100 um.

- **s3=value**  
An optional parameter that controls the simulation space used in fitting the critical step height for a negative dishing (**dmin**) model. The value must be expressed in um and must be  $\geq 0$ . The default value is 100 um.
- **SC=value**  
An optional parameter for handling edge rounding in oxide materials that is dependent on line spacing. This parameter should be set to a symbolic name and matched to measured data.
- **singleRate=height\_per\_second**  
An optional argument that can only be specified for the first material, and that defines the polishing rate for that material while only it is being polished. When another material becomes exposed, the calculations use the corresponding Rate arguments for the first and new materials.
- **spad=stiffness**  
An optional argument used to specify the pad stiffness for platen 1, specified in kPa. This argument must precede any instances of the material keyword.
- **wb=value**  
An optional argument specified together with the rfb argument and used for the enhanced perimeter model (EPM). This argument is enabled when  $wb > 0$  and specifies a width bias value that defines the transition region between trench and non-trench patterns for the given material. It affects the removal rate of the material. The value is a floating-point number in microns. See “[Perimeter](#)” on page 211 for details on the perimeter model.

## Description

The duration of the CMP process is specified either as time or thickness, or a certain amount of over-polishing can be specified: a time as `overTime`, a thickness as `overThickness`, or a fraction of the process time from start until the topmost layer is cleared in each frame as `overPolish` in percent.

These models are called from the [deposit](#) or [cmp](#) commands. All keywords except name and type support [Symbolic Names](#). Symbolic names can be used in the command to indicate that the parameter is to be fitted to empirical data. Any parameter assigned a numeric value is untouched during model calibration. That is, this parameter is not fitted or optimized.

If a CMP model is specified, it must define model parameters for all materials that may be polished. If a material becomes exposed (locally), parameters for that material must be present. Otherwise the simulation stops with an error.

## Examples

### Example 1: A static ECD model (not impacted by calibration)

```
# electroplating model:  
define_model name=ecd1 k1=0.035 k2=0.05 Keq=10 phistar=0.05 \  
rmod=0 smod=0
```

### Example 2: An initial ECD model (fitted during calibration)

```
define_model name=ecd1 k1=$k1 k2=$k2 Keq=$KEQ phistar=$PHISTAR \  
rmod=0 smod=0
```

### Example 3: A static CMP model (not impacted by calibration)

```
define_model name=cmp1 spad=500kPa pressure=10psi \  
material=copper a1=200A/s a2=900A \  
Hex=3nm alpha1=0.15 beta1=0.1\  
s1=100um \  
material=TaN rate=20A/s \  
dmax=300A alpha2=0.3 beta2=0.28 s2=100um \  
dmin=40A alpha3=0.3 beta3=0.25 s3=100um \  
material=oxide rate=10A/s C=3 SC=22.5um \  
dmax=300A alpha2=0.3 beta2=0.28 s2=100um \  
dmin=40A alpha3=0.3 beta3=0.25 s3=100um
```

## deposit

Introduces a new layer at the top of the layer stack. If the material for this deposit command is the same as the material for the previous deposit command, the command only modifies the surface position and does not add a new layer.

### Usage

```
deposit type={isotropic | anisotropic | ecd | HDP_CVD | fill | SOD} \
[anisotropyFactor=value] [cornerBias=dist] [cornerBiasN=dist]\ \
[criticalSpace=width] [criticalSpaceN=width] \
[current=amperes] [currentDensity=amperes_per_cm2] [diameter=value] \
[dt=time_step] [Epsilon=fitting_factor] [fluxBalance] [HeightFactor=value] \
[mask_selective=layer_number] [maskFile_selective=filename] \
[material=material_name] [maxDishing=depth] [minW=min_width] \
[model=model_name] [nsteps=number_steps] \
[negativeSelective] \
[position=surface_position] [rate=rate] \
[saveDishing] [saveErosion] [saveThickness] [saveZ1] [saveZ2] [save2D] \
[sidewallAngle=degrees] \
[sidewallsOnly=thickness] \
[slopeAngle=degrees] [slopeAngleN=degrees] [Smax=s] [SmaxN=s] \
[snapshot=snap_ID] [spaceThreshold=distance] \
[targetThicknessNT=height [targetWeight=grid | targetX=coordinate targetY=coordinate]] \
[targetThicknessT=height [targetWeight=grid | targetX=coordinate targetY=coordinate]] \
[thickness=thickness] [time=time] \
[WidthBias=bias] [WidthFactor=value]
```

#### Typical syntax for each type:

```
deposit type={isotropic | anisotropic} [material=material_name] [thickness=thickness]
deposit type=ecd [model=model_name] [thickness=thickness] [time=time] [dt=time_step] \
[nsteps=number_steps]
deposit type=HDP_CVD [material=material_name] [cornerBias=distance] \
[slopeAngle=degrees] [targetThicknessT=height]
deposit type=fill [material=material_name] [position=surface_position]
deposit type=SOD [material=material_name] [thickness=thickness] [HeightFactor=value] \
[WidthFactor=value]
```

Each type has additional parameters, which may be useful for some situations but typically are not required for good results.

## Parameters

- **type={isotropic | anisotropic | ecd | HDP\_CVD | fill | SOD}**

A required parameter defining how deposition is applied. Allowed values are:

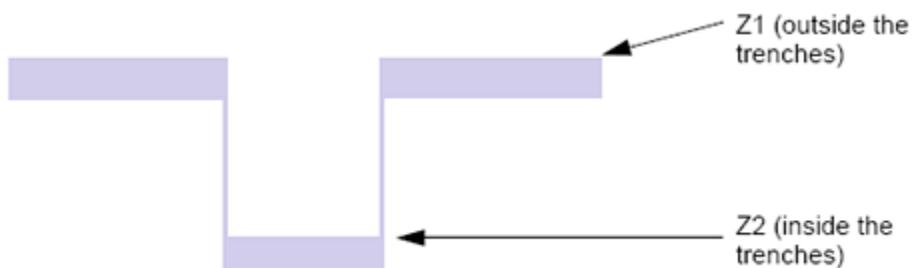
- **isotropic** — When specified, you must also specify the thickness and the material. Thickness can be supplied as a single value or by specifying the rate and time.

**Figure 4-8. Isotropic Deposition**



- **anisotropic** — When specified, you must also specify the thickness and the material. Thickness can be supplied as a single value or by specifying the rate and time.

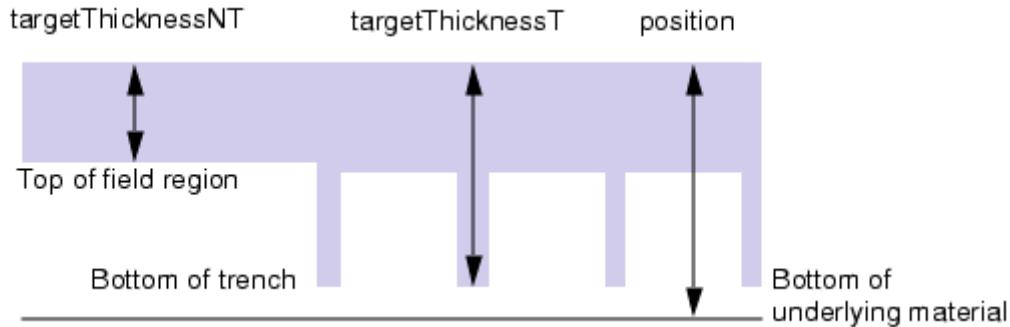
**Figure 4-9. Anisotropic Deposition**



- **ecd** — When specified, you must also specify the following:
  - The name of an electroplating model that must be specified elsewhere in the Process Recipe File using a `define_model` command.
  - Thickness information in the form of **thickness** and/or **time** and rate (**dt** and **nsteps**).
- **HDP\_CVD** (High Density Plasma Chemical Vapor Deposition) — When specified, you must also specify either **targetThicknessT** or **targetThicknessNT** and three of the following:
  - The **cornerBias** that defines how far from the edge of the sidewall the oxide overburden extends.

- The **criticalSpace** at which triangular pyramids start to become trapezoidal.
- Either dishing measurements in the MD file or **maxDishing**, which sets the depth of the difference between the highest and lowest point of the oxide surface over the trenched region.
- The **slopeAngle** of the pyramidal shape formed by the oxide overburden.
- **fill** — When specified, you must also specify the height. For single layers, use **position**, which defines the surface position after fill. For multilayer simulations, use either **targetThicknessT** or **targetThicknessNT**.

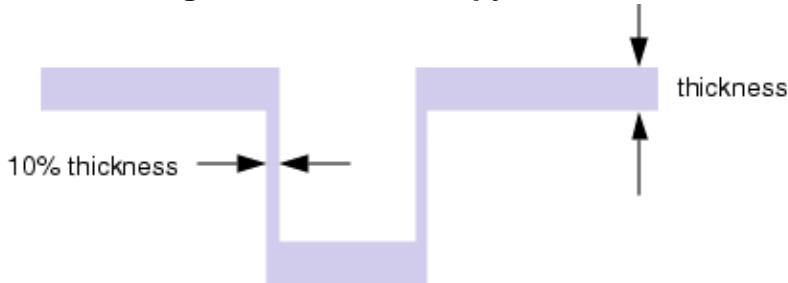
**Figure 4-10. Position Choices for Fill Deposition**



- **SOD (Spin-On Dielectric)** — When specified, you must also specify the following:
  - **Thickness**. Unlike the other types, you cannot use rate and time for thickness information. The thickness must be greater than the trench depth.
  - The **WidthFactor** and **HeightFactor** that calculate the dishing in the dielectric as the material settles into the trenches.
- **anisotropyFactor=*value***

A floating point value that represents the ratio of sidewall accumulation versus field thickness of the deposited material. For example, specifying 0.1 for this argument means that ten percent of the final thickness in the field regions are deposited on the sidewalls. Specifying 10% or 0.1 for the anisotropyFactor carries out the same action. See [Figure 4-11](#).

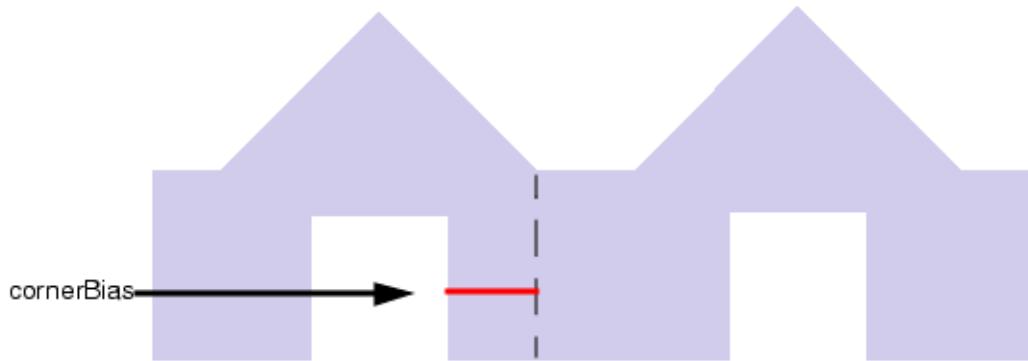
**Figure 4-11. Anisotropy Factor**



- `cornerBias=dist`

An optional argument used with HDP\_CVD deposition only. The value of *dist* indicates the distance between the edge of the trench and the start of the pyramidal build-up, and can be positive or negative. If *dist* is negative, its absolute value must be less than `criticalSpace/2`. It is measured in microns.

**Figure 4-12. Corner Bias**



- `cornerBiasN=dist`

An optional argument used with enhanced HDP\_CVD deposition only. When specified with `cornerBias` and `spaceThreshold`, `cornerBiasN` refers to narrow trench values and `cornerBias` to wide trench values.

- `criticalSpace=width`

An optional argument used with the HDP\_CVD deposition only. The critical space is the threshold width of active area at which the mound of deposited material starts to flatten at the top, changing shape from a triangle or pyramid to a trapezoid. The *width* value must be greater than 0.

- `criticalSpaceN=dist`

An optional argument used with enhanced HDP\_CVD deposition only. When specified with `criticalSpace` and `spaceThreshold`, `criticalSpaceN` refers to the critical space for narrow trenches and `criticalSpace` is restricted to wide trenches.

- `current=amperes`

An optional argument used with ecd deposition only. The current is the total current over the wafer. It is used to determine the nominal plating rate.

- `currentDensity=amperes_per_cm2`

An optional argument used with ecd deposition only. It is used to determine the nominal copper deposition rate.

- `diameter=value`

An optional argument used with ecd deposition only. The diameter refers to the diameter of the wafer.

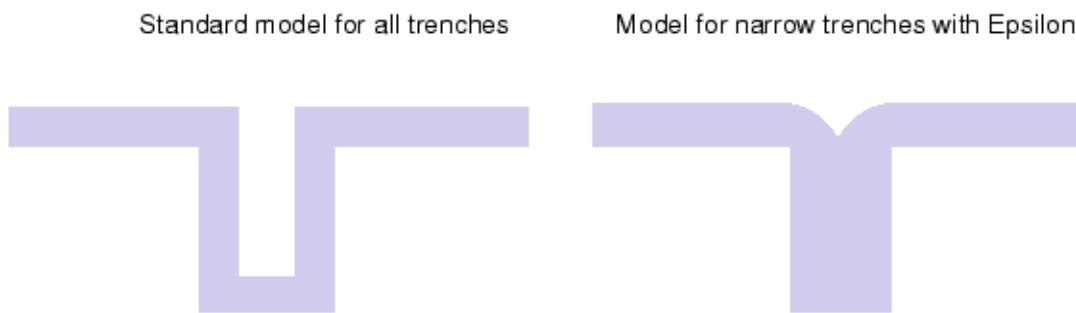
- `dt=time_step`

An optional argument used to specify the time interval of the deposited material, expressed in seconds. This argument is only used for electroplating. If `dt` is not specified, this value is derived from `time` and `nsteps`.

- `Epsilon=fitting_factor`

An optional parameter used to round the deposition of material into a trench. It may only be specified with isotropic depositions. It has no affect on wide trenches, but on narrow trenches it allows the trench bottom to be at a greater height. See [Figure 4-13](#).

**Figure 4-13. Deposition Rounding in Narrow Trenches**



When `Epsilon` is used, the `thickness` parameter can be specified as an expression.

- `fluxBalance`

An optional parameter that takes no value. When specified, includes dynamic modeling of flux when calculating isotropic deposition.

- `HeightFactor=value`

An optional argument used with SOD models primarily, where it is required. The value, which should be between 0 and 1, is used in conjunction with `WidthFactor` to model the overall depth of dishing. A value near 0 indicates very shallow dishing and near 1 very deep, nearly to the trench bottom on very wide trenches. In version 2010.1, this parameter was `anisotropyFactorZ`.

- `mask_selective=layer_number`

An optional argument that is only valid when `type=isotropic` or `type=fill`. It restricts the deposit to the areas indicated by the shapes on `layer_number`. If `negativeSelective` is specified, the areas outside the polygons are deposited rather than the areas inside the polygons.

- `maskFile_selective=filename`

An optional argument that is only valid when `type=isotropic` or `type=fill`. This argument contains the file name of the selective mask to read. When specified with `mask_selective`, the deposit is restricted to only pixels covered by the selective mask.

- `material=material_name`

An optional argument used to specify the name of the deposition material.

Material names allow you to define CMP models that behave differently depending on the type of material being polished. Refer to the [define\\_model](#) command for more information.

- `maxDishing=depth`

An optional argument used with HDP\_CVD deposition only. It specifies how far between targetThicknessT and the top of the deposition mound.

- `minW=min_width`

An optional parameter used to specify the minimum remaining width of a trench after isotropic deposition of a material into it. If the remaining width is smaller, the trench is considered filled and no electroplating is done for this trench.

- `model=model_name`

An optional parameter used to specify the electroplating model which must be specified elsewhere in the Process Recipe File using a [define\\_model](#) command. This parameter is required when type=ecd.

- `negativeSelective`

An optional keyword that is only used when `type=isotropic` or `type=fill`. It does not take a value.

The negativeSelective option indicates that the mask supplied by the `mask_selective` argument is a negative; that is, the shapes on the layer indicate the areas that are not deposited.

- `nsteps=number_steps`

An optional argument used to specify the number of steps of deposited material. This must be an integer value. This argument is only used for electroplating. The default value is 200. If dt and nsteps are not specified, nsteps is set to 200.

- `position=surface_position`

An optional parameter used to specify the position of the surface in vertical space, expressed in angstroms. When type=fill, the command must include one of position, targetThicknessT, or targetThicknessNT.

When anisotropyFactor is set to 0, the model behaves like anisotropic deposition is applied and it deposits only horizontal surfaces. Material is not deposited on side walls in this case.

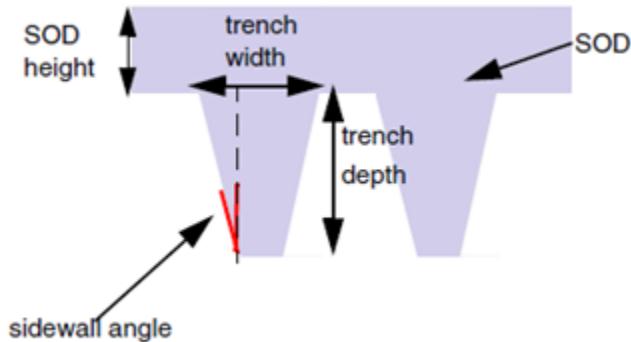
- `rate=rate`

An optional argument used to specify the deposition rate of the deposited material, expressed in angstroms per second. The HDP\_CVD model does not support this parameter.

- `sidewallAngle=degrees`

An optional argument used only with SOD deposition. It considers the effect of the angle of the trench sidewalls on the erosion of the deposited layer. The trench sidewalls are assumed to be symmetrical; `degrees` should be between 0 and 90. The default value is 0.

**Figure 4-14. Sidewall Angle**



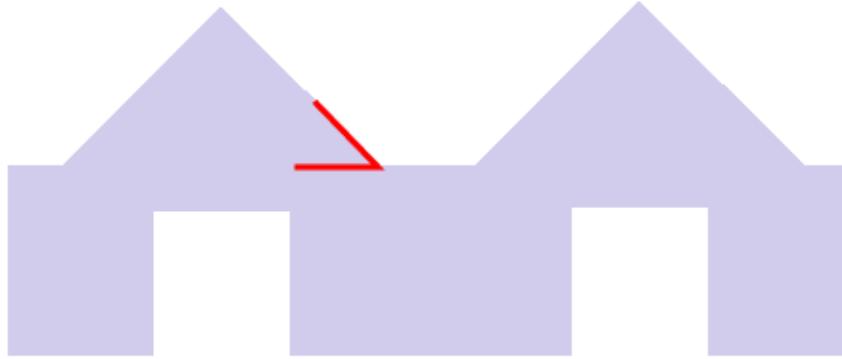
- `sidewallsOnly=thickness`

An optional argument used only with `type=anisotropic` to support material deposition only on the trench sidewalls. If enabled, the `thickness` value of the deposition in angstroms is used to update the geometry data. Thickness expressions are supported.

- `slopeAngle=degrees`

An optional argument used only with `HDP_CVD` deposition. It specifies the angle of the oxide sidewalls as shown in [Figure 4-15](#). The oxide overburden is assumed to be symmetrical; `degrees` should be between 0 and 90.

**Figure 4-15. Slope Angle**



- `slopeAngleN=degrees`

An optional argument used with enhanced `HDP_CVD` deposition only. When specified with `slopeAngle` and `spaceThreshold`, `slopeAngleN` specifies the angle for oxide associated with narrow trenches and `slopeAngle` is restricted to wide trenches.

- `Smax=s`

An optional argument that activates enhanced `HDP_CVD` deposition. It is equal to `cornerBias + (criticalSpace/2)`. For non-linear models, you only need either `Smax` or `cornerBias`.

- **SmaxN=*s***

An optional argument used only with enhanced HDP\_CVD deposition. When specified with Smax and spaceThreshold, SmaxN specifies the Smax value for narrow trenches and Smax itself is restricted to wide trenches.

- **Snapshot=*snapshot\_ID***

An optional keyword and positive integer used to assign a unique identifier to a state after this deposition step is complete. This identifier is used in the Snapshot column in the measured data file to associate measured or simulated data with this state.

- **spaceThreshold=*distance***

An optional argument used only with enhanced HDP\_CVD that indicates the threshold between wide and narrow trenches.

- **targetThicknessNT=*height***

An optional parameter that specifies the non-trench (field) thickness at which to stop the deposition. This argument can only be used when type is ecd, fill, or HDP\_CVD. When this is set, then the time setting is ignored.

The way the simulated thickness is calculated depends on the settings of targetX/targetY or targetWeight.

- If *neither* targetX/targetY nor targetWeight is set, the targetThicknessNT value is compared to the average of all NT grids. This is the default.
- If targetX and targetY are set, then the average thickness at the grid that includes that coordinate is used. If the coordinates are not in a field region an error is generated.
- If targetWeight is set, then targetThicknessNT is compared to the weighted average of the thickness of all grids.

When type=fill and targetThicknessNT=0, the deposit command keeps the current pattern density for all pixels.

- **targetThicknessT=*height***

An optional parameter that specifies the thickness at which to stop deposition. The thickness is measured from the bottom of the trench to the top of the deposited material within the trench. When targetThicknessT is set, the time setting is ignored.

When type=fill, the deposit command must include one of position, targetThicknessT, or targetThicknessNT. If targetThicknessT is used and it is less than the trench depth, the deposited material only partially fills the trenches. If targetThicknessT is greater than the trench depth, the surface position on the non-trench areas is equal to the difference between targetThicknessT and the average trench depth.

- **targetWeight=*grid\_file***

An optional parameter naming a grid file with weights. The weights are applied to the simulated thickness for each pixel. The default weighting is 1 for all grids. Cannot be specified with targetX targetY.

- **targetX=coordinate targetY=coordinate**

An optional pair of parameter that can be used with targetThicknessT or targetThicknessNT. The pair specify the coordinates at which to measure the simulated thickness. Both targetX and targetY must be specified when used. They cannot be used with targetWeight.

- **thickness=thickness**

An optional argument used to specify the thickness of the deposited material, expressed in angstroms.

When used with “type=isotropic” or “type=anisotropic”, *thickness* can be expressed as an expression that includes measured values (\$M\_width, \$M\_space, \$M\_area, and \$M\_perimeter) and calibration parameters. Expression-based thickness requires the process recipe includes at least one [etch](#) step preceding the deposit step; if the recipe would not otherwise include this etch step, set the etch thickness to 0. Expression-based thickness may be useful when modeling High-K Metal Gate (HKMG).

- **time=time**

An optional argument used to specify the length of time the material is being deposited, expressed in seconds. You must specify either the targetThickness or the rate and duration of the deposition (time with nsteps or dt alone) for all layers except those that are type=fill.

When the deposit command also includes targetThicknessT or targetThicknessNT, the time argument is ignored.

- **WidthBias=bias**

An optional parameter used primarily in SOD models for biasing geometry masks. The *bias* parameter represents an adjustment factor to help match the observed trench behavior. It can be negative (closes more trenches than otherwise predicted) or positive (reduces the number of trenches that otherwise close).

Use WidthBias when trenches are overetched, but you do not have an etch model in the recipe.

- **WidthFactor=value**

An optional parameter used only in SOD models. The *value* parameter is a number from 0 to 2000 that roughly correlates to the width of the dishing after the dielectric has been spun-on and settled into the trenches. The *value* is unitless. It is recommended to fit this to measured data.

This parameter is required when type=SOD. In the 2010.1 release, it was anisotropyFactor.

## Description

All keywords except material and snapshot support [Symbolic Names](#). Symbolic names can be used in the command to indicate that the parameter is to be fitted to empirical data. Any parameter assigned a numeric value is untouched during model calibration. That is, this parameter is not fitted or optimized.

## Related Topics

[Measured Data File Format](#)

## etch

Defines a trench etching step.

### Usage

```
etch type=etch_type
{thickness=angstroms | time=seconds rate=ang_per_sec | etchModel=calculation}
[anisotropyFactor=percent]
[extra=floating_point_value] [extractType= {0 | 1}]
[fluxBalance]
{[mask=layer_number [mask_selective=layer_number]] |
 [mask1D=filename] | [mask2D=filename] |
 [maskFile=filename [maskFile_selective=filename]]}
[material=material_name] [maxS=floating_point_value] [maxW=floating_point_value]
[minS=min_space] [minW=min_width]
[negative] [negativeSelective]
[overEtchThickness={floating_point_value / calculation}]
[overTime=floating_point_value]
[overThickness=floating_point_value]
[planarizationlength=microns]
[Snapshot=snap_ID]
[sidewallAngle=value]
[wCrit=critical_width]
```

### Parameters

- **type=etch\_type**

A required argument defining the type of etch process. Allowed values are:

**isotropic** — If isotropic etch is specified, width values are decreased because etching is performed in all directions.

**anisotropic** — If anisotropic etch is specified, width values remain unchanged. Space is reduced by the etched thickness. Area fraction and perimeter density are adjusted appropriately.

**all** — The etching removes a layer completely. When specified with material=material\_name, all layers above material\_name are also removed. When used with the arguments material=material\_name thickness=angstroms, the material\_name is etched to the specified thickness but not removed completely.

**empty\_trenches** — The etching process empties the trenches of material, as is typical in a Replacement Metal Gate (RMG) process. A CMP step is required before specifying empty\_trenches. When the argument thickness=angstroms is specified, the corresponding material thickness is reduced by the Thickness value for each window.

**etch\_back** — The etching process creates new trenches, simulating the reverse etch-back process. Typically etch-back occurs after depositing a thick conformal or isotropic oxide layer and before CMP. It creates new trenches over the deposited oxide profile.

- **etchModel=calculation**

A required keyword that can be used to define the thickness etched. The *calculation* can be either an algebraic expression or a call to a table with piece wise linear data. The table is specified in a “[define\\_model type=lookup](#)” command. Width and thickness pairs are used to interpolate the thickness for a given trench width. See “[Example 2: Using Lookup Tables or Algebraic Expressions](#)”.

In addition to the parameters defined elsewhere in the recipe and ones passed from the measured data file, algebraic expressions in the etch command can refer to layout data using the following reserved keywords: \$M\_width, \$M\_space, \$M\_area, and \$M\_perimeter. These keywords are treated as read-only variables and cannot be optimized.

- **thickness=angstroms**

A required keyword used to define the thickness to be etched. You must specify either thickness or time and rate for isotropic and anisotropic types.

- **time=seconds**

A required keyword and value used to define the length of time etching is performed. You must specify either thickness or time and rate.

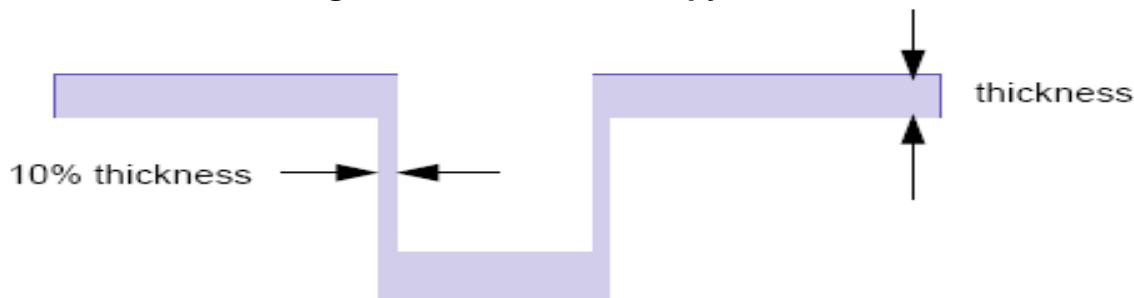
- **rate=ang\_per\_sec**

A required keyword used to define the length of time etch rate. You must specify either thickness or time and rate.

- **anisotropyFactor=percent**

An optional argument that takes a floating point value that represents the amount of sidewall etching of the deposited material. For example, specifying 0.1 for this argument means that ten percent of the final thickness in the field regions are left on the sidewalls. Specifying 10% or 0.1 for the anisotropyFactor carries out the same action.

**Figure 4-16. etch anisotropyFactor**



When anisotropyFactor is set to 0, the model behaves like anisotropic etching is applied and it etches only the horizontal surfaces. Material is not etched on side walls in this case.

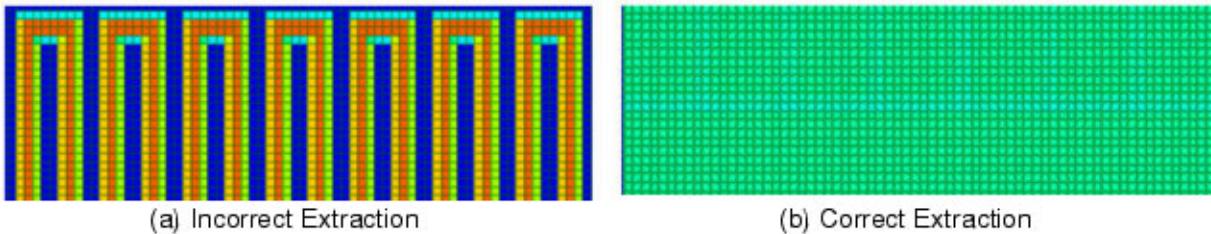
This argument is only valid with **type=isotropic**.

- `extra=floating_point_value`

An optional parameter that specifies a larger search range for grid frames that results in better extraction values. The default is 0 (zero).

If polygons fall within the specified search range, they are used to extract the width and lengths along with other mask data. If polygons are not found, the next nearest polygons are used for extractions. Incorrect extractions can be seen in (a) of [Figure 4-17](#). This shows the elevation mapping oscillations between widths and spaces in a test structure. Correct extraction values should look like (b). Generally, increasing the value of extra fixes the problem of oscillating conditions.

#### **Figure 4-17. Oscillatory Conditions From Incorrect Extractions**



Use this parameter to prevent the simulator from confusing test structures in the arrays for field regions when the width of your field regions are close to Space values.

- fluxBalance

An optional parameter that takes no value. When specified, includes dynamic modeling of flux when calculating trench etch.

- inversePattern

An optional parameter that takes no value. It causes the trench and non-trench regions to be swapped; that is, the z1 and z2 grids are interchanged. This parameter is only valid with **type=empty\_trenches**.

- mask=*layer\_number*

An argument used to specify the layer number in the layout to be used as the etch mask.

If the recipe contains more than one etch command, the mask parameter should only be used for the first instance. Calibre CMPAnalyzer allows only a single etch mask per model.

- mask selective=*layer\_number*

An optional argument that is only valid when `type=etch_back`. It restricts the etch-back etching to the areas indicated by the shapes on `layer_number`. If `negativeSelective` is specified, the areas outside the polygons are etched rather than the areas inside the polygons.

- `mask1D=mask_name | mask2D=mask_name`

A keyword used to instruct the tool to run the simulation based on user-supplied mask data rather than layout data.

- When a *mask\_name* is specified, it must be the name of a mask defined using the `define_mask` command in the Process Recipe File.
- When a *mask\_name* is not specified, CMP Model Builder displays the `define_mask` dialog box, prompting you to define the mask. The `define_mask` dialog box does not allow mask names that start with a number.

You must specify either a layer (using the `mask` keyword) or a mask file (using either the `mask2D` or the `maskFile` keywords).

- **maskFile=filename**

A keyword/value pair used to specify the base name for a set of grid files containing the 2D mask data.

- When specified without the `mask` keyword, `maskFile` instructs the tool to find mask data in the current working directory and use it in place of layout data. The tool expects to find a set of mask files having a base file name as specified.
- When both `mask` and `maskFile` are specified, the tool checks for the specified mask *filename* before extracting. If the mask files already exist, then the data in them is used instead of re-running extraction. If the files do not exist, CMP Model Builder uses the mask layer in the layout for simulation and saves mask data in the form of a set of grid files having a base file name as specified. These grid files are saved in the temporary directory specified in Calibre WORKbench. When used this way, the file extension for the base file name defines the grid file format:
  - *.txt* — order-independent ASCII. This is the default file format.
  - *.text* — order-dependent ASCII.
  - *.bin* — binary.

You must specify either a layer (using the `mask` keyword) or a mask file (using either the `mask1D/mask2D` or the `maskFile` keywords).

- **maskFile\_selective=filename**

An optional argument that is only valid when **type=etch\_back**. This argument behaves like `maskFile`, except that the grid files contains `mask_Selective` layer data.

- **material=material\_name**

An optional argument naming the material that is being etched. The default material supplied by the GUI is oxide. Material names allow you to define models that behave differently depending on the type of material being etched. Refer to the [define\\_model](#) command for more information.

Do not use this parameter when etching through multiple materials in one step.

- [maxS=*floating\_point\_value*] [maxW=*floating\_point\_value*]

Setting values for maxS and maxW along with the extra parameter helps the simulator set limits on the extraction window. The computation of the search ranges used for grid extractions are drawn from the values entered here.

Not setting maxS and maxW to the largest widths and spaces of your test structures can result in the simulator misreading spaces for widths and widths for spaces. The spatial relationship between structures is of paramount importance in identifying how the extraction is calculated.

Specifying these search limits ensures accurate readings in the extraction phases of model building. In test structures where widths and spaces vary by more than a ratio of 1:50, these parameters should be used to counteract any incorrect extractions. Values of maxS and maxW are in microns.

- minS=*min\_space*

An optional parameter indicating the technology-specific minimum value of spaces between trenches in microns. The default is 0.0. This can be used to cut off artifacts from the extraction, for example, a tiny piece of polygon clipped by a grid boundary.

- minW=*min\_width*

An optional parameter indicating the technology-specific minimum value of a trench width in microns. The default is 0.0.

- negative

An optional keyword that does not take a value. When specified, indicates that the mask is a negative; that is, that the shapes on the layer indicate the areas that are *not* etched. In a layout, generally polysilicon layers are negative and oxide and metal layers are not.

---

#### Note

 When looking at z1 and z2 grids for negative etch data, set the NegativeCutLine option under the **Grids** tab. (You may need to show more options by clicking the + button in the upper right corner.) When NegativeCutLine is selected, z1 is correctly assigned to the polygons and z2 to the space between.

The NegativeCutLine option does not have any assigned keyword and must be set using the CMP Model Builder window.

---

- negativeSelective

An optional keyword that is only used when **type=etch\_back**. It does not take a value.

The negativeSelective option indicates that the mask supplied by the mask\_selective argument is a negative; that is, the shapes on the layer indicate the areas that are not etched.

- overEtchThickness={*floating\_point\_value* | *calculation*}

An optional keyword that is only used when **type=anisotropic** or **type=empty\_trenches**. After normal etching, the value of z1 (the height of non-trench regions) is reduced by the overEtchThickness amount. The value is in angstroms.

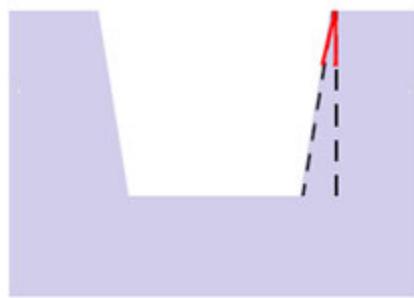
Use a lookup table or an algebraic expression to supply multiple values for overEtchThickness. The name specified by *table* is fully described in a [define\\_model](#) command. It should take the following form:

```
define_model type=lookup name=table width0=w0 value0=v0 \
width1=w1 value1=v1 ...
```

where width refers to the width of neighboring trench regions and value specifies the associated overEtchThickness value in angstroms.

- **overTime=floating\_point\_value**  
An optional keyword that specifies the over etch time in seconds (s).
- **overThickness=floating\_point\_value**  
An optional keyword that specifies the etch thickness beyond the regular **thickness** in angstroms ( $\text{\AA}$ ). When **type=isotropic**, the overThickness amount also affects etch biasing.
- **planarizationlength=length**  
An optional argument specifying the length over which pattern density is averaged for the etch simulation.
- **Snapshot=snap\_ID**  
An optional keyword and positive integer used to assign a unique identifier to a state after this deposition step is complete. This identifier is used in the Snapshot column in the measured data file to associate measured or simulated data with this state.
- **sidewallAngle=value**  
An optional keyword that considers the geometry effect of the angle of non-vertical trench sidewalls during material removal. The trench sidewalls are assumed to be symmetrical. The value can be either degrees between 0 and 90 (default is 0) or an algebraic expression as used in etchModel. See “[Example 2: Using Lookup Tables or Algebraic Expressions](#)” on page 109.

**Figure 4-18. etch sidewallAngle**



- **wCrit=critical\_width**  
Critical width before setting Z1=Z2 (used for oxide polishing where there are no larger field regions but perhaps huge trenches).

## Description

See “[Etch Dialog Box](#)” on page 147 for information on using the GUI to set up the etch command.

The following keywords support [Symbolic Names](#):

overEtchThickness	overThickness	overTime
rate	thickness	time

Symbolic names can be used in the command to indicate that the parameter is to be fitted to empirical data. Any parameter assigned a numeric value is untouched during model calibration. That is, this parameter is not fitted or optimized.

Etching may specify a mask in various ways:

- Using the number of the layer number: mask=33
- Using 2D mask, that is defined with a define\_mask command: mask2D=my2Dmask
- Using a grid file containing mask data: maskFile=my2Dmask.bin

If an etch step does not specify a mask, it changes only the vertical positions of the top layer.

See “[define\\_mask 1D](#)” on page 72 and “[define\\_mask 2D](#)” on page 74 to learn more about setting up the etch command with define\_mask.

## Examples

### Example 1: Multilayer Etch

The following recipe defines several base layers and then etches.

```
initialize x1=300 y1=300 x2=2200 y2=1700 material=nitride
deposit thickness=4100A material=oxide1 type=isotropic
deposit thickness=1000A material=oxide2 type=isotropic
deposit thickness=1000A material=TEOS type=isotropic
etch mask2D=mask thickness=3100A type=anisotropic
```

Notice that the etch command uses a single thickness (3100 A) rather than three separate etch calls each specifying a material. Starting from the top of the layer stack, the simulator etches down to the specified thickness. In this case, that means it removes all of TEOS and oxide2 and 1100 A of oxide1.

If you are using etch models or anisotropyFactor, it is better to have separate etch models. For example:

```
initialize x1=300 y1=300 x2=2200 y2=1700 material=nitride
deposit thickness=4100A material=oxide1 type=isotropic
deposit thickness=1000A material=oxide2 type=isotropic
deposit thickness=1000A material=TEOS type=isotropic
etch mask2D=mask material= TEOS thickness=1000A type=isotropic \
    anisotropyFactor=1.5
etch material=oxide2 thickness=1000A type=isotropic anisotropyFactor=1.2
etch material=oxide1 thickness=1100A type=isotropic anisotropyFactor=0.9
```

Notice that for the second and third etch models no mask is supplied; Calibre CMPAnalyzer does not allow stages of the same etch operation to use different masks.

### Example 2: Using Lookup Tables or Algebraic Expressions

Recipes can interpolate values based on tables. The table is given in a [define\\_model](#) command; the etch command calls the table with the etchModel parameter.

```
initialize x1=300 y1=300 x2=2200 y2=1700 material=nitride
deposit thickness=4100A material=oxide type=isotropic
etch mask2D=mask type=anisotropic etchModel=oxide_etch
...
define_model type=lookup name=oxide_etch \
    width0=1 value0=200 \
    width1=10 value1=500 \
    width2=50 value2=1000 \
    width3=100 value3=1500
...
```

Interpolation is piecewise linear. This means that if the mask contains a trench with a width of 8 um, the simulator returns 433.3 A, the interpolated value between value0 and value1. As the table does not specify units the default ones for width (lateral extension) and etch depth (vertical) are used as specified in [Table 4-2](#) in “[Process Recipe File Format](#)” on page 65.

The etch command could also be written as

```
etch mask2D=mask type=anisotropic \
etchmodel=(0.004*$M_width*$M_width*$M_width) - (0.67*$M_width*$M_width) +
(40.258*$M_width)+160.41
```

This yields a value of 480.2 for a trench width of 8. Note that the etchmodel expression must be entered all on the same line (the usual line end symbol is interpreted as division), but is forced here to wrap by the page width.

An expression can also be fitted to data using CMP Model Builder. See “[Optimizing Parameterized Expressions](#)” on page 193 for a procedure and example.

## initialize

Required. Defines the limits of the simulation domain with X1, Y1, X2, Y2. By default, initialize spans the extent of the loaded layout file or the extent of the define \_mask definition.

---

### Note

---

 This command must be the first command in the process recipe file.

---

## Usage

```
initialize{ [layout=file] | [grid=grid_file] } \  
  { [x1=x_coordinate] [y1=y_coordinate] [x2=x_coordinate] [y2=y_coordinate] } \  
  [pixel=grid_size] [pixelX=pixel_size pixelY=pixel_size] \  
  [position=v_position] [material=material_name] \  
  [compatibility=version] [layer_stack_topography=list] \  
  [materialNT(T)dfmdb]
```

## Parameters

- *layout*=*file*

An optional parameter used to define the name of the layout file containing mask geometries. A layout must be loaded in Calibre WORKbench and the layout currently displayed is the one used in simulations.

- *grid*=*grid\_file*

An optional parameter used to specify a grid file defining the surface position of the first layer. When specified in batch mode, a grid file must exist within the current working directory. When using the Calibre WORKbench interface, you can specify a directory to search for *grid\_file*. For a complete description of grid files, refer to “[Grid Files](#)” on page 202.

If a grid file is used, its size (number of cells in X and Y-directions) overrides the number of grid cells otherwise computed based on layout size and pixel size.

- *x1=x\_coordinate y1=y\_coordinate x2=x\_coordinate y2=y\_coordinate*

An optional parameter set identifying the portion of the layout to be used as the simulation area. When at least one coordinate is specified, the values of all non-specified coordinates default to *0*. When *y1* is 0, all grids have only one cell in the Y-direction.

If no coordinates are specified, the simulation area depends on other input parameters.

- If a **layout** is specified, the simulation area is the extent of the layout file.
- If **grid** is specified in either this command or an etch command, its size (number of cells in X- and Y-directions) overrides the number of grid cells otherwise computed from the input parameters.
- If **define\_mask** is specified, the extent of the data is used for simulation.

- `pixel=grid_size`

Optional parameter that defines the size of a frame for simulation. The default unit is in microns, but `grid_size` may specify a different unit. Pixel (grid) size is computed automatically by default. The default size is 20 um x 20 um.

- `PixelX=pixel_size PixelY=pixel_size`

Optional parameters used to define the pixel size to use for simulations. The value must include the units to be used. Pixel size is computed automatically by default. Users can prescribe a positive value if needed.

- `position=v_position`

An optional parameter indicating the bottom position of the first layer in the layer stack. The default is 0.

- `material=material_name`

An optional parameter used to define the material of the first layer in the layer stack. The default is oxide.

Material names allow you to define CMP models that behave differently depending on the type of material being polished. Refer to the [define\\_model](#) command for more information.

---

**Note**

 If the first [deposit](#) command after initialize uses the same material name, saved grid files could show incorrect thicknessNT (field thickness) values. Use a different material name for the first deposited layer and the initialize command.

---

- `compatibility=version`

An optional parameter that sets the version of Calibre to use for simulations. The behavior for some non-physical parameters differs in the various versions. Use this to revert back to an older version of CMP Model Builder. If creating the recipe using the GUI, the values 2007.4, 2008.1, 2009.2, and 2009.3 are already supplied in a list, but you can enter any version of the form NNNN.N such as 2011.2.

If the environment variable [CMPOPTIMIZE\\_COMPATIBILITY](#) is set in the shell at invocation, `cmptimize` ignores `version`.

- `layer_stack_topography=list`

An optional parameter that specifies the initial profile to use for simulating a stack of layers. The first layer is always assumed to be created on a flat surface, but subsequent layers reflect the topography of the layers underneath.

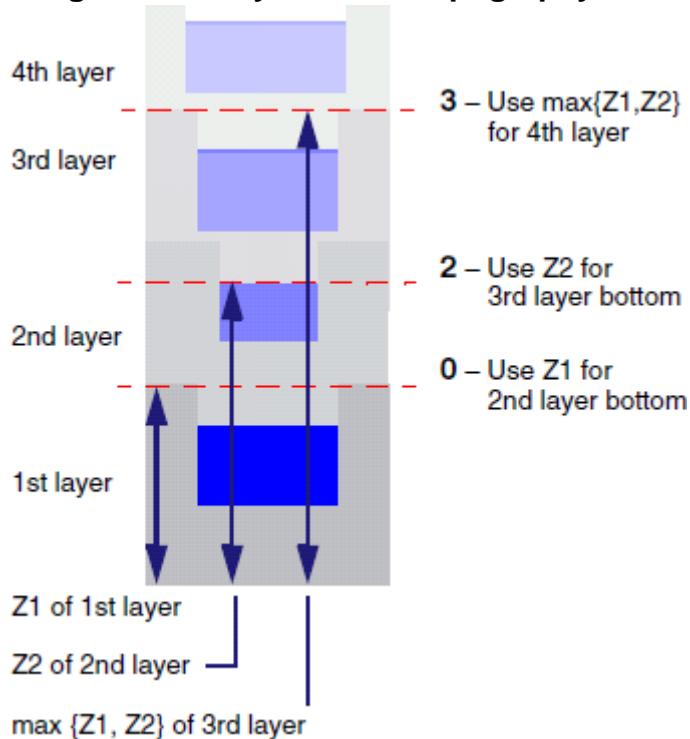
The `list` is a string composed of 0, 1, 2, 3, 4, or 5. Each digit corresponds to a layer in the stack, starting with the second layer. The leftmost value determines how the second layer's bottom (first layer top) is determined.

- 0 — Use Z1 (field, or nontrench height). This is the default.
- 1 — Use an average, Z1(1-density) + Z2(density), for each frame.

- 2 — Use Z2 (trench height).
- 3 — Use the greater of Z1 or Z2 for each frame.
- 4 — Use the smaller of Z1 or Z2 for each frame.
- 5 — Use a planar surface with the greatest Z1 or Z2 over all frames.

For example, a list for 4 layers could be entered as “layer\_stack\_topography 023”. As shown in [Figure 4-19](#) for a single grid frame, the bottom of the second layer would use the nontrench height at the top of the first layer; the bottom of the third layer would use the trench height of the second layer; and the bottom of the fourth layer would use the larger of the two heights at the top of the third layer.

**Figure 4-19. layer\_stack\_topography 023**



If there are more layers than digits in *list*, the last setting is used for the remaining layers. If there are fewer layers than digits, the values starting on the left of *list* are used.

- materialNT(T)dfmdb

An optional parameter that does not take a value. When specified, indicates that the MaterialNT (non-trench material) and MaterialT (trench material) properties are added to the DFM database (dfmdb). If this parameter is not specified, the default is to *not* add these properties to the dfmdb.

## Description

None of the parameters of the initialize command accept symbolic values.

See “[Initialize Dialog Box](#)” on page 144 for information on using the GUI to set up the initialize command.

## Save

Saves files containing data on grids for graphing in later stages.

### Usage

**save** {[state] | [dump] | [*property\_keywords*] } [file=*filename*]

### Parameters

- state

An optional argument instructing the tool to save the entire state of an optimized process model (simulation) in an ASCII format. Saving the state and re-loading with the [initialize](#) grid command is recommended after each simulation as it reduces optimization time for subsequent models.

- dump

An optional argument instructing the tool to save the entire state of an optimized process model in a binary format. Saving the state and re-loading with the [initialize](#) grid command is recommended after each simulation as it reduces optimization time for subsequent models.

- *property\_keywords*

An optional argument using one or more of the following optional keywords, specifying individual properties:

area — Average trench area, calculated as (trench area)/(total area).

dishing — The difference between the local z1 and the local z2 values.

erosion — Average erosion depth in microns in the frame.

length — The average trench length in the frame.

materialNT — The non-trench material index grid, where the material index is a number that defines the order of the deposited material in the layer stack.

materialT — The trench material index grid, where the material index is a number that defines the order of the deposited material in the layer stack.

---

### Note

 For a copper CMP process, the trench material index grid is the copper index. However, when TrenchLayerStack is enabled, the material index for the trench material can be different. See “[Multilayer Analysis Inside of Trenches](#)” on page 207 and “[Material Index Reporting in Grids](#)” on page 207.

---

perimeter — Average perimeter, calculated as (trench perimeter in frame)/(total area in frame).

pressure — The final pressure calculated for the layer.

preEtch — A binary mask generated by the etch type=all command.

space — The average trench space in the frame.

thicknessNT — The thickness over field regions.

thicknessT — The thickness over trenches, calculated as the difference between the top surfaces over trench regions and the trench bottoms.

topology — The thickness topology data related to the Thickness\_Topo property used for analysis in Calibre CMPAnalyzer.

width — The average trench width in the frame.

WL — The area per feature ratio.

z1 — The non-trench height values.

z2 — The trench height values.

z3 — The trench bottom values.

All values for thickness data grids Z1, Z2, ThicknessT, and ThicknessNT are reported in whole angstroms.

- `file=filename`

An optional argument that specifies the base name for the output files generated by this command. The file parameter defines only the base-name for the files. A prefix that identifies the particular field is prepended. For example, the following line saves two files, *z1\_cmp3.bin* and *erosion\_cmp3.bin*:

```
save z1 erosion file=cmp3.bin
```



# Chapter 5

## Environment Variable Reference

---

This chapter contains a dictionary of environment variables that can be used with CMP modeling. Environment variables are not saved as part of session files or recipe files. If you intend to use a particular setting across multiple sessions, set the environment variables in a shell script used to invoke Calibre WORKbench or run the cmptimize command.

**Table 5-1. Environment Variables for CMP Modeling**

Variable	Description
CALIBRE_MGC_CMP_EXTRACTION_VERSION	Controls which form of extraction is used.
CMP_ENABLE_OAF_LOG_DT	Causes the OpenAreaFraction and polishing time to be reported.
CMP_EXTR_LARGE_METAL_REGION	Enables modified handling for metal shapes that are larger than the frame.
CMP_EXTRACT_POS_EXTRA_SPEEDUP	Turns on an extraction version optimized for many small shapes such as metal fill.
CMP_EXTRACTION_VERSION	Sets the extraction version.
CMP_OLD_EROSION	Uses the second-generation erosion definition.
CMP_PD_MIN_RANGE_FACTOR	Defines the range of minimal pattern density values for calculating erosion.
CMPOPTIMIZE_COMPATIBILITY	Sets the version of CMP Model Builder.
CMPOPTIMIZE_THICKNESSNT_MIN	Controls how SThickNT is calculated.
MGCMP_EROSION	Uses the first-generation erosion definition.

# **CALIBRE\_MGC\_CMP\_EXTRACTION\_VERSION**

Controls which form of extraction is used.

## **Usage**

```
setenv CALIBRE_MGC_CMP_EXTRACTION_VERSION 1
```

## **Arguments**

- **1**

The default value of the required argument. This setting runs the current version of the software.

## **Description**

This high-level variable controls which form of extraction is used. It was introduced in Calibre v2010.1.

---

### **Note**

 This variable has been replaced with [CMP\\_EXTRACTION\\_VERSION](#). Please update any scripts to use the new name.

---

## **CMP\_ENABLE\_OAF\_LOG\_DT**

Causes the OpenAreaFraction and polishing time to be reported.

### **Usage**

```
setenv CMP_ENABLE_OAF_LOG_DT interval
```

### **Arguments**

- *interval*

A required argument setting the number of seconds between reports of the OpenAreaFraction value.

### **Description**

Use the environment variable CMP\_ENABLE\_OAF\_LOG\_DT to report intermediate values of OpenAreaFraction and elapsed polishing time. Set CMP\_ENABLE\_OAF\_LOG\_DT to the interval in seconds at which to report.

This variable is ignored if OpenAreaFraction is not set in the cmp command.

### **Examples**

To report on the OpenAreaFraction every 2 minutes, use the following:

```
setenv CMP_ENABLE_OAF_LOG_DT 120
```

# **CMP\_EXTR\_LARGE\_METAL\_REGION**

Enables modified handling for metal shapes that are larger than the frame.

## Usage

```
setenv CMP_EXTR_LARGE_METAL_REGION any_value
```

## Arguments

- *any\_value*

Literally any value. The variable is checked to see whether it is set or not.

## Description

When this variable is set, CMP modeling identifies large metal areas and modifies parameters to handle the extraction more appropriately. “Large” metal areas are those filling at least 90% of a frame with a single polygon. The standard frame is 20 x 20 microns.

This variable was introduced in Calibre v2010.3 and is not set by default.

# **CMP\_EXTRACT\_POS\_EXTRA\_SPEEDUP**

Turns on an extraction version optimized for many small shapes such as metal fill.

## Usage

```
setenv CMP_EXTRACT_POS_EXTRA_SPEEDUP any_value
```

## Arguments

- *any\_value*

Literally, any value. This variable is checked to see whether it is set or not.

## Description

This variable activates a mode of extraction that runs faster on designs containing many small shapes such as dummy fill. It was introduced in Calibre v2010.3.

# **CMP\_EXTRACTION\_VERSION**

Sets the extraction version.

## Usage

```
setenv CMP_EXTRACTION_VERSION {1 | 2 | 3}
```

## Arguments

- **1**  
Specifies to run the original CMP extraction version.
- **2**  
Specifies to run a newer version for extraction. This version uses weighted averaging of intersecting geometries and sets extra to 0 or 100 for field or array regions, respectively.
- **3**  
Specifies to run the current extraction version. This version improves area density extraction for dummy fill shapes in negative extraction, and for cheating-like structures during positive extraction. No change to width and space data. It is the default.

## Description

This high-level variable controls which form of extraction is used. It was introduced in Calibre v2010.4. In Calibre v2013.2, the default extraction version changed from 2 to 3. Version 3, the newest, is recommended over version 2.

This variable was initially called [CALIBRE\\_MGC\\_CMP\\_EXTRACTION\\_VERSION](#).

# CMP\_OLD\_EROSION

Uses the second-generation erosion definition.

## Usage

`setenv CMP_OLD_EROSION any_value`

## Arguments

- *any\_value*

Literally, any value. The variable is checked to see whether it is set or not.

## Description

Set CMP\_OLD\_EROSION to use the erosion calculation of versions 2009.3 through 2010.2. This variable was introduced in Calibre v2010.3.

## Related Topics

[MGCMP\\_EROSION](#)

## **CMP\_PD\_MIN\_RANGE\_FACTOR**

Defines the range of minimal pattern density values for calculating erosion.

### **Usage**

**setenv CMP\_PD\_MIN\_RANGE\_FACTOR *range***

### **Arguments**

- ***range***

A required parameter specifying the spread of pattern densities that should be considered when calculating height. The default value is 0.1.

### **Description**

CMP\_PD\_MIN\_RANGE\_FACTOR defines the range of minimal pattern density values for calculating erosion. That is, it defines how dense a frame may be and still serve as a reference for erosion calculations. This variable allows you to control how similar the density of the frames considered for erosion should be.

The current erosion calculation includes a topography height factor. The average topography height is calculated from frames with a minimum density; the range in minimum density is  $(1 + \text{range}) \times (\text{minimum density})$ .

This variable was introduced in Calibre v2010.3.

# CMPOPTIMIZE\_COMPATIBILITY

Sets the version of CMP Model Builder.

## Usage

```
setenv CMPOPTIMIZE_COMPATIBILITY version
```

## Arguments

- *version*

A required version number of the form *NNNN.N*, such as 2015.2.

## Description

The CMPOPTIMIZE\_COMPATIBILITY environment variable instructs cmptimize to use the settings of the specified Calibre version for both simulation and extraction.

---

### Note

---

 You should only use this variable for validation purposes. To always use a particular version (not recommended), set it explicitly in the process recipe file using “initialize ... compatibility”. Environment variable settings are not saved as part of the session file.

---

This variable overrides any setting of the compatibility parameter in the [initialize](#) command.

If this variable is not set, the current version of cmptimize runs.

## Related Topics

[CMP\\_EXTRACTION\\_VERSION](#)  
[CMPOPTIMIZE\\_THICKNESSNT\\_MIN](#)

## CMPOPTIMIZE\_THICKNESSNT\_MIN

Controls how SThickNT is calculated.

### Usage

```
setenv CMPOPTIMIZE_THICKNESSNT_MIN any_value
```

### Arguments

- *any\_value*

Literally, any value. The variable is checked to see whether it is set or unset.

### Description

This variable controls how simulated thicknessNT (SThickNT in the MD) is calculated. To calculate the thickness of the exposed material in the field region requires a stored reference grid.

Without CMPOPTIMIZE\_THICKNESSNT\_MIN, CMP Model Builder finds the thickness of the topmost layer strictly below the local z1 value. This is reported as SThickNT, and is the default behavior.

When CMPOPTIMIZE\_THICKNESSNT\_MIN is set, CMP Model Builder locates the global z1 minimum. It then uses the topmost layer below the z1 value and uses it as a reference for SThickNT.

This variable was introduced in Calibre v2009.2.

# MGCMP\_EROSION

Uses the first-generation erosion definition.

## Usage

`setenv MGCMP_EROSION any_value`

## Arguments

- *any\_value*

Literally, any value. The variable is checked to see whether it is set or not.

## Description

Set this variable to use the original definition of erosion, which was changed in v2009.2. This variable was introduced in Calibre v2009.2.

## Related Topics

[CMP\\_OLD\\_EROSION](#)



# Chapter 6

## CMP Model Builder GUI Reference

---

This chapter describes the function of the major menus, tabs, and secondary dialog boxes. Common dialog boxes such as the file browser are not listed.

The images of the graphical user interface (GUI) used in this chapter are taken from Calibre WORKbench and are for example purposes only.

<b>CMP Model Builder Main Window</b> .....	<b>130</b>
<b>Measured Tab</b> .....	<b>134</b>
Sort Dialog Box .....	136
LineScan Dialog Box .....	138
<b>Recipe Tab</b> .....	<b>142</b>
Initialize Dialog Box .....	144
Etch Dialog Box .....	147
Deposit Dialog Box .....	153
Cmp Dialog Box .....	158
Save Dialog Box .....	162
EcdModel Dialog Box .....	164
CmpModel Dialog Box .....	167
LookUpModel Dialog Box .....	172
Mask1D Dialog Box .....	173
Mask2D Dialog Box .....	175
Expression Dialog Box .....	178
<b>Optimize Tab</b> .....	<b>179</b>
<b>Remote Tab</b> .....	<b>182</b>
<b>Grids Tab</b> .....	<b>183</b>

# CMP Model Builder Main Window

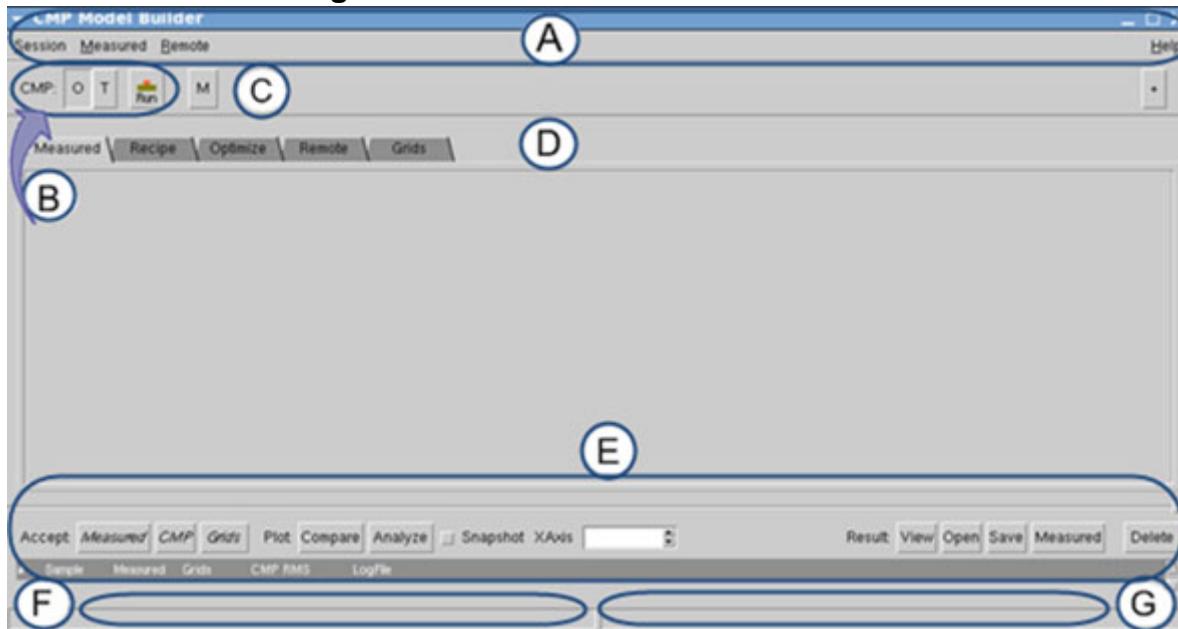
To access: From Calibre WORKbench, **Tools > CMP Model Builder**

Use this tool window to set up CMP recipes, weight data, and analyze simulation results.

## Description

The CMP Model Builder Main Window is an elaborate interface that contains multiple tabs and spawns multiple dialog boxes. Following is the initial view of the CMP Model Builder tool. There are three global menus that work in conjunction with the main tabbed work areas.

**Figure 6-1. CMP Model Builder Window**



### Session Menu

Save and restore the state of all tabs in the CMP Model Builder window. See following field descriptions in the “Session Menu” table.

### Measured Menu

Create measurement data and manipulate the mask or layout upon which measurements are taken. See following field descriptions in the “Measured Menu” table.

### Remote Menu

Load remote configuration files for distributed runs. See following field descriptions in the “Remote Menu” table.

## Objects

**Table 6-1. CMP Model Builder Global Contents**

Field	Description
A	Menus. <b>Session</b> — Save and restore the state of all tabs in the CMP Model Builder window. <b>Measured</b> — Create measurement data and manipulate the mask or layout upon which measurements are taken. <b>Remote</b> — Load remote configuration files for distributed runs. <b>Help</b> — Opens this manual. Many commands supply additional information in area F when you over the cursor over the command for a few seconds.
B	Simulation controls. <b>O</b> and <b>T</b> — Toggle the run between Optimization (automatically varying parameterized arguments to find the best match to measured data) and Testing (using the “best” values, or minimum values if no best has been set). <b>Run</b> — Starts the simulation in the currently selected mode, optimization or test. Runs may take a long time. Area G shows messages indicating progress.
C	<b>Measurement Sites button</b> — Click this to start placing measurement sites in the layout. Measurement sites should be placed before running simulations.
D	<b>Tabs</b> — The primary work areas for CMP Model Builder. See the specific tab pages: “ <a href="#">Measured Tab</a> ” on page 134 “ <a href="#">Recipe Tab</a> ” on page 142 “ <a href="#">Optimize Tab</a> ” on page 179 “ <a href="#">Remote Tab</a> ” on page 182 “ <a href="#">Grids Tab</a> ” on page 183
E	<b>Analysis controls</b> — Various controls for comparing, accepting, and importing simulation results. The table below the buttons lists log files.
F	Menu help tips.
G	Status messages. Check this area for further helpful information when the system is not responding as you expected.

**Table 6-2. Session Menu**

Menu Item	Description
Open	Loads a previously saved session file. This restores the data from all tabs. It does not load a layout into Calibre WORKbench.
View	Shows what would be saved to a file if you were to save a session.
New	Erases the current session information, including the recipe, measured data, and loaded grids. If the information was previously saved in a file, it can be restored.
Save	Saves a session file so you can restore the tab content later, after having made changes.
Export Run Script	Creates two files: a Tcl script and measured data file, both prefixed as you specify. The files can be run from the command line. The Tcl script either optimizes the recipe or tests it, depending on whether O or T is enabled in the CMP Model Builder window.
Close Window	Closes the CMP Model Builder window. The data is restored when the window is opened next.

**Table 6-3. Measured Menu**

Menu Item	Description
Open	Opens a file browser. Once you have selected a file, the CMP window attempts to load the file and switches to the <b>Measured</b> tab. If the file is not in <a href="#">Measured Data File Format</a> , the Calibre WORKbench transcript prints an error, “cmpconvert:cannot read measured data file.”
Append	Opens the measured data file in a new tab in the <b>Measured</b> tab. The tabs for the different measured data files are at the bottom of the table display area.
Save	Saves the currently displayed data file to a name you specify.
New	Switches to the <b>Measured</b> tab and creates a measured data tab with only one row and minimum columns.
Interactive	Adds rows to the currently shown measured data file. The XY coordinates are set by clicking the layout in the main Calibre WORKbench window.
Delete	Deletes the currently displayed measured data tab. Does not delete files.
Export Markers	Places markers in a new layer, “measured”, in the design displayed in the Calibre WORKbench main window. The markers indicate the XY coordinates of the currently displayed measured data table.
Clip Layout	Generates a new OASIS file from the displayed layout. The new file is clipped at the coordinates you specify. It includes any CMP Modeling-generated data that was displayed before the clipping.
Testchip	Opens the Test Chip dialog. Use this dialog box to automatically generate a design suitable for making measurements to calibrate your recipe.

**Table 6-3. Measured Menu (cont.)**

Menu Item	Description
Image	Opens the Image tool for making measurements from AFM or SEM scans.

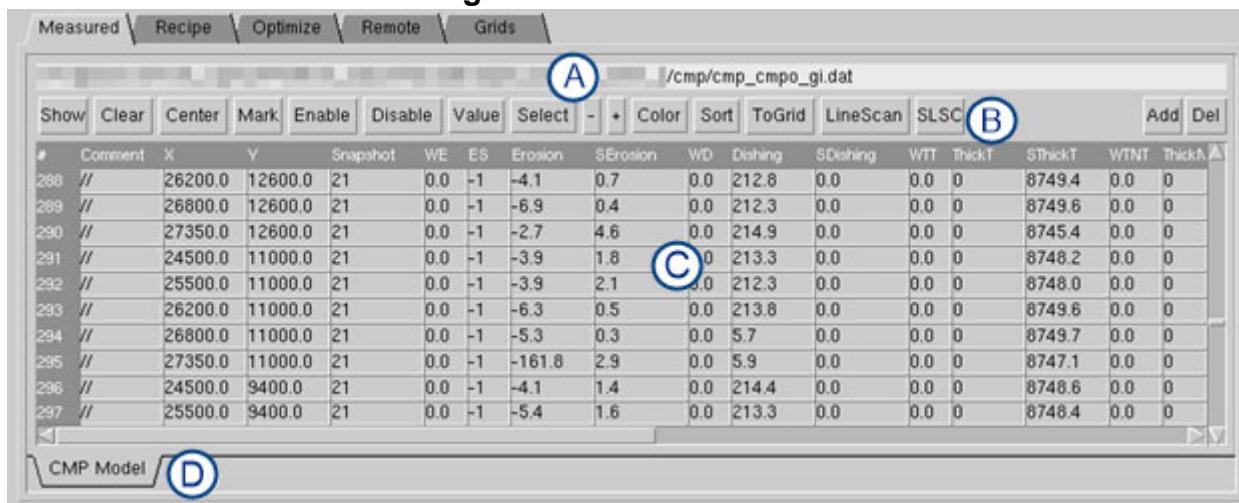
**Table 6-4. Remote Menu**

Menu Item	Description
Open	Opens a file browser. Once you have selected a file, the CMP window loads it and switches to the <b>Remote</b> tab. It does not check file contents.
Save	Saves the contents of the <b>Remote</b> tab to a file. If you do not specify a file suffix, “.rcf” is automatically appended. There is no validation of syntax or file type; whatever is in the tab’s display is saved under the specified filename.
Delete	Clears the contents of the <b>Remote</b> tab. Does not delete saved configuration files from the file system.

## Measured Tab

To access: From Calibre WORKbench, **Tools > CMP Model Builder**, click the **Measured** tab.  
Use this tab to input SEM or AFM measurement data (the measured values in the MD file) and apply weighting to individual measurements.

**Figure 6-2. Measured Tab**



## Objects

**Table 6-5. Measured Tab Contents**

Field	Description
A	File name of the MD file. This is not editable. Use <b>Measured &gt; Open</b> and <b>Measured &gt; Save</b> .

**Table 6-5. Measured Tab Contents (cont.)**

<b>Field</b>	<b>Description</b>
B	<p>Buttons to interact with data.</p> <p><b>Show, Clear, Center, Mark</b> — Manipulate the view in WORKbench.</p> <p><b>Enable, Disable</b> — Change the weighting (WE) of selected rows.</p> <p><b>Value</b> — Menu of functions that can be used to create expressions.</p> <p><b>Select</b> — Highlights other rows that have a matching value in the selected column type. For example, you can highlight all cells with the same Y.</p> <p>–, + — Show and hide columns. The default is to show fewer columns.</p> <p><b>Color</b> — Set a background color to the indicated cells.</p> <p><b>Sort</b> — Sorts the table based on the currently selected cells or columns. See “<a href="#">Sort Dialog Box</a>” on page 136.</p> <p><b>ToGrid</b> — Converts selected rows to grids.</p> <p><b>LineScan</b> — Opens the Line Scan tool, which lets you import dishing and erosion data from graphical measurements. See “<a href="#">LineScan Dialog Box</a>” on page 138.</p> <p><b>SLSC</b> — Simulates a line scan based on selected rows.</p> <p><b>Add, Del</b> — Adds and deletes rows to the table.</p>
C	The measurement data. Click in dark gray areas to select a row or a column. Click in cells to edit data. Empty cells are not allowed; if you delete a value, it is replaced with 0.
D	Tabs for navigating between files.

## Usage Notes

For a description of the columns and the format for the MD file, see “[Measured Data File Format](#)” on page 59.

## Related Topics

[Creating a Measured Data File](#)

[Measured Data File Format](#)

## Sort Dialog Box

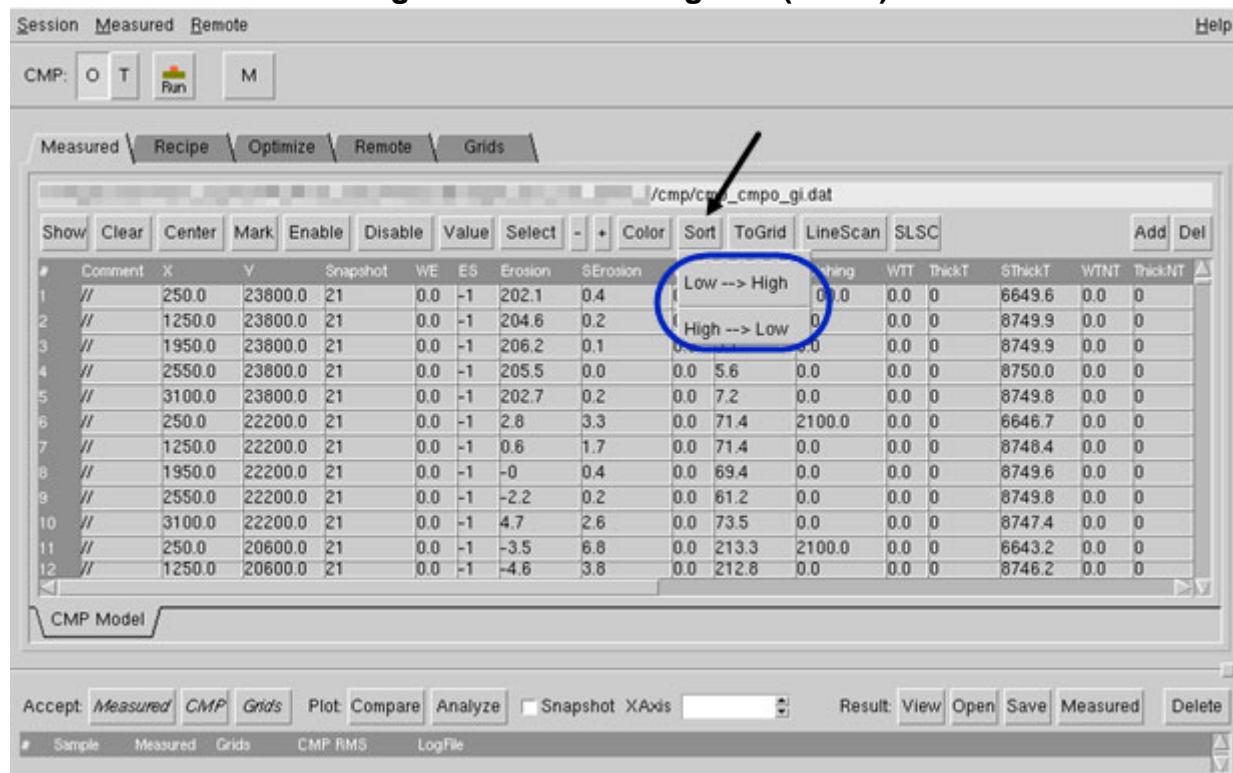
To access: Select data cells or column(s) in the measured data table and click **Sort**.

Use this dialog box to sort and order measured data when reviewing your CMP measured data table.

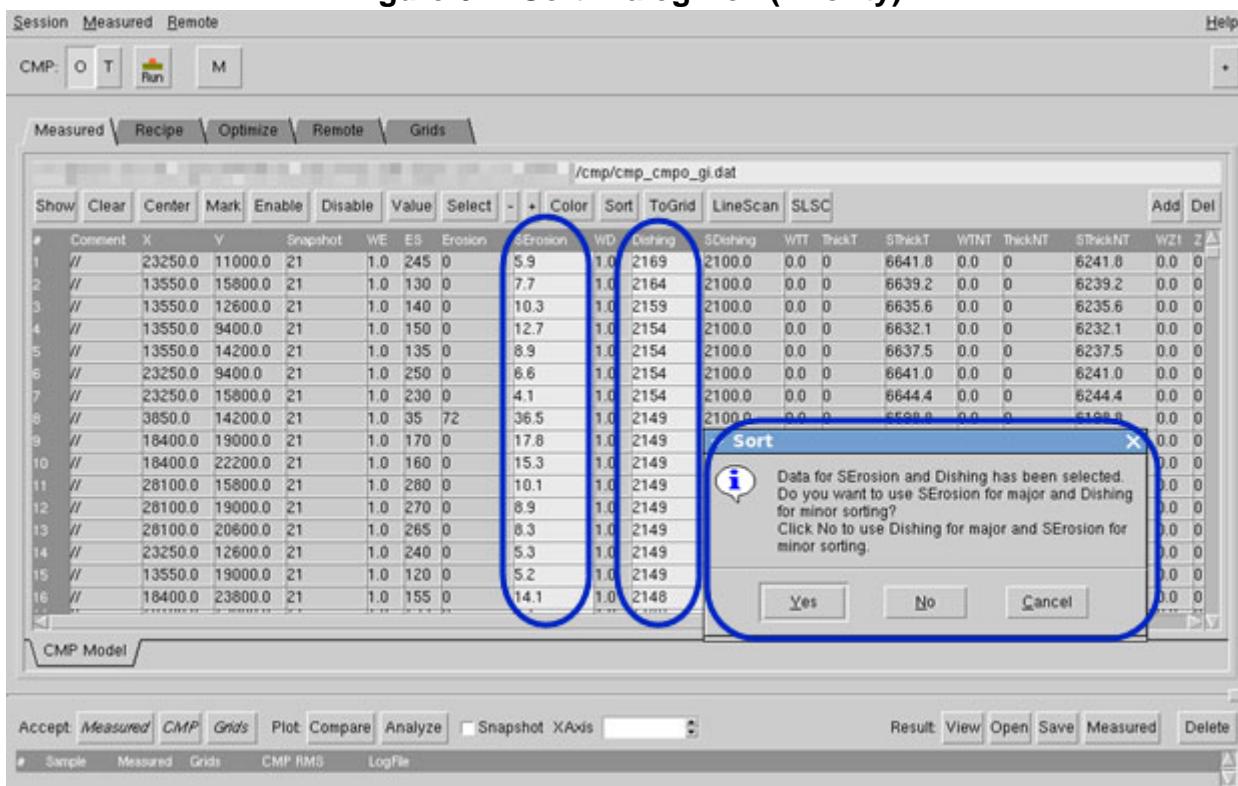
### Description

The contents of the Sort Dialog Box can be used to control the sorting order and priority of the displayed data in the measured data table.

**Figure 6-3. Sort Dialog Box (Order)**



**Figure 6-4. Sort Dialog Box (Priority)**



## Objects

**Table 6-6. Sort Dialog Box Contents**

Field	Description
Low --> High	Select Low --> High to sort data by increasing order.
High --> Low	Select High --> Low to sort data by decreasing order.
Sort major or minor query popup window	Allows you to choose the sorting priority (major or minor) between two selected columns of data. Review the query and choose either Yes, No, or Cancel.

## Usage Notes

The **Sort** function in the measured data table has the following constraints:

- Sorting the measured data requires you to select at least two cells in one column.
- Sorting data for more than two columns is not supported.

## LineScan Dialog Box

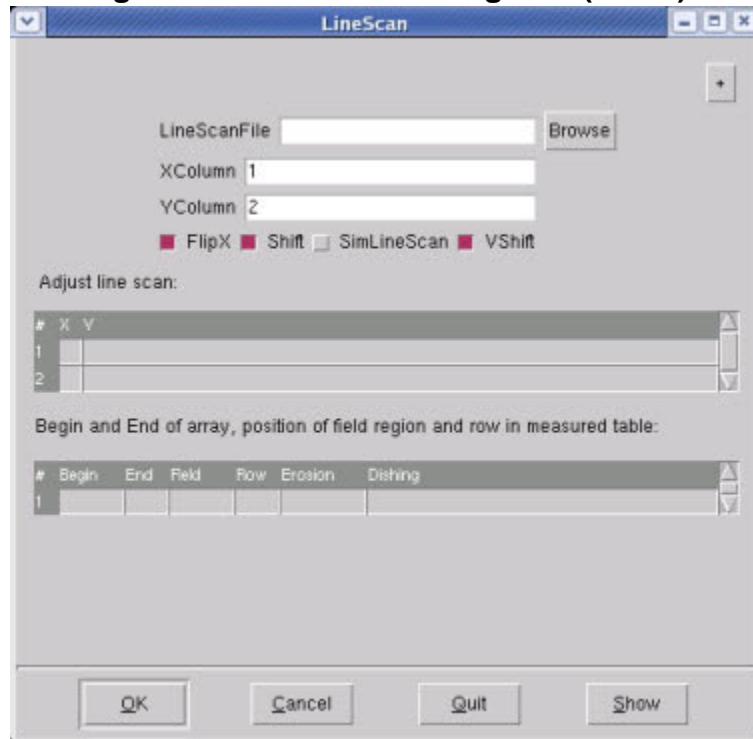
To access: Either draw a cutline on the layout or in the **Measured** tab select rows in the measured data table and click **LineScan**.

Use this dialog box to create plots that can be used when analyzing your CMP models.

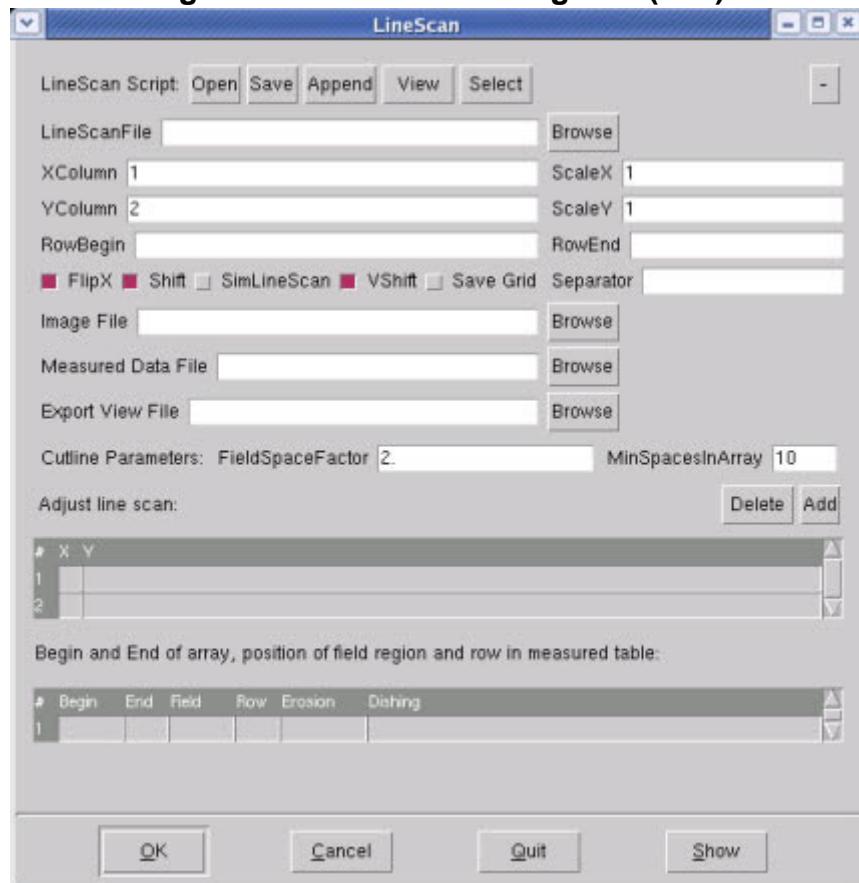
### Description

The LineScan dialog box is available in basic mode and full mode. See field descriptions in the tables following the figures.

**Figure 6-5. LineScan Dialog Box (Basic)**



**Figure 6-6. LineScan Dialog Box (Full)**



## Objects

**Table 6-7. LineScan Dialog Box (Basic) Contents**

Field	Description
Select	Only visible when a session file that contains linescan data has been loaded. The Select button displays a table with available data that may be used in the current session.
+	Reveals additional fields shown in <a href="#">Figure 6-6</a> .
LineScan File	Specify a grid file or a spreadsheet.
XColumn YColumn	For data with multiple rows, indicate which column (line) is to be used for erosion and dishing measurements. The first column is referenced as "1" or "A". If your file begins with coordinates, use "1" and "2".
FlipX	Select the FlipX option if the measurements are reversed on the X-axis.
Shift	Select the Shift option if the LineScan dialog box was launched from a cutline. Shift moves the beginning of the line scan to 0.
SimLineScan	Select the SimLineScan option to display measured and simulated data together when you plot the data using <b>Show</b> .

**Table 6-7. LineScan Dialog Box (Basic) Contents (cont.)**

Field	Description
VShift	Select the VShift option to shift simulated data vertically in the plot so that the first data points are at the same height.
Adjust line scan: table	Apply offset values to rotate skewed X and Y data to lie along the same orthogonal axes as the MD file. Refer to “ <a href="#">Importing Values to the MD File</a> ” on page 38.
Begin and End columns	Coordinates of the beginning and ending of the array, in microns, for adjusting the boundaries of line scan data to match with that of the MD file.
Field column	Size of the field region surrounding the array but within the block, in microns.
Erosion and Dishing columns	Reports line scan data for erosion and dishing, if present.

**Table 6-8. LineScan Dialog Box (Full) Contents**

Field	Description
ScaleX	Use if the line scan data is on a different resolution than the MD file. The line scan X or Y data is multiplied by the value in the ScaleX or ScaleY field.
ScaleY	Use if the line scan data is on a different resolution than the MD file. The line scan X or Y data is multiplied by the value in the ScaleX or ScaleY field.
RowBegin	Restricts the consideration of the line scan data to just the rows between RowBegin and RowEnd.
RowEnd	Restricts the consideration of the line scan data to just the rows between RowBegin and RowEnd.
SaveGrid	Select to save the line scan data as a grid file. It appears in the <b>Grids</b> tab.
Separator	Use to change the characters separating columns from the default spaces, tabs, and new lines (“whitespace”).
Image File	Specify an image of SEM or AFM data to import.
Measured Data File	Specifies a file to receive the erosion and dishing measurements. By default, this is the file currently displayed in the <b>Measured</b> tab.
Export View File	Export the line scan data as a graph.
FieldSpaceFactor	A multiplier indicating how much larger than the trench-to-trench spacing must a non-trenched area be to be considered “field”.

**Note**

 This table does not include fields that also appear in the basic version. Those fields are covered in the previous table.

## Usage Notes

For best simulation results, cutlines should meet the following requirements:

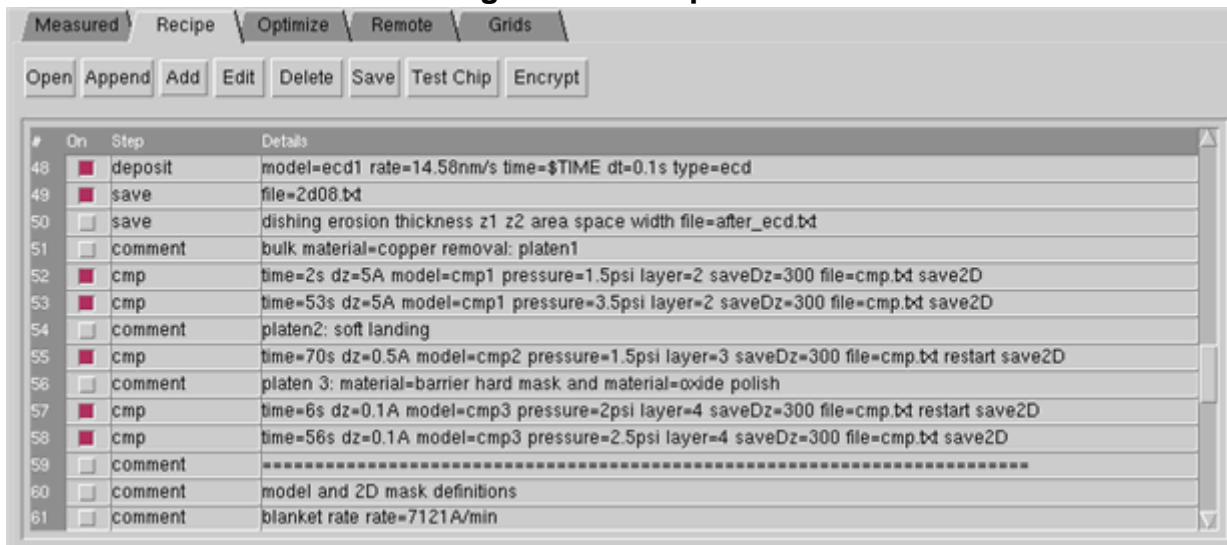
- The line should be perpendicular to the array, not at an angle to the X or Y axis.
- The drawn line should cross the field regions on either side of the arrays. The field (“not trench”) regions are used as reference values for erosion and dishing.

## Recipe Tab

To access: From Calibre WORKbench, **Tools > CMP Model Builder**, click the **Recipe** tab.

Use this tab to create a process recipe file.

**Figure 6-7. Recipe Tab**



## Objects

**Table 6-9. Recipe Tab Buttons**

Button	Description
Open	Opens an existing recipe file.
Append	Adds a recipe to another recipe. This may be useful if you are refining the model for a multi-stage polishing step, or want more detailed modeling of higher layers.
Add	Adds a line to the recipe. This button is a menu, with an item for each CMP command and also a way to enter hotspot and expression information. Selecting an item from this menu opens a dialog box for entering command parameters.
Edit	Opens a dialog box to edit the parameters of the currently selected command.
Delete	Deletes the selected lines.
Save	Saves the recipe file.
Test Chip	Creates a test chip based on any define_mask commands in the recipe. If there are none, prompts you to define one.
Encrypt	Encrypts the recipe on disk. Encrypted recipes can be used in CMPAnalyzer but are not human readable.

## Usage Notes

Unselecting the **On** button causes the line to be commented out in the file when it is saved.

## Related Topics

[Creating a Process Recipe File](#)

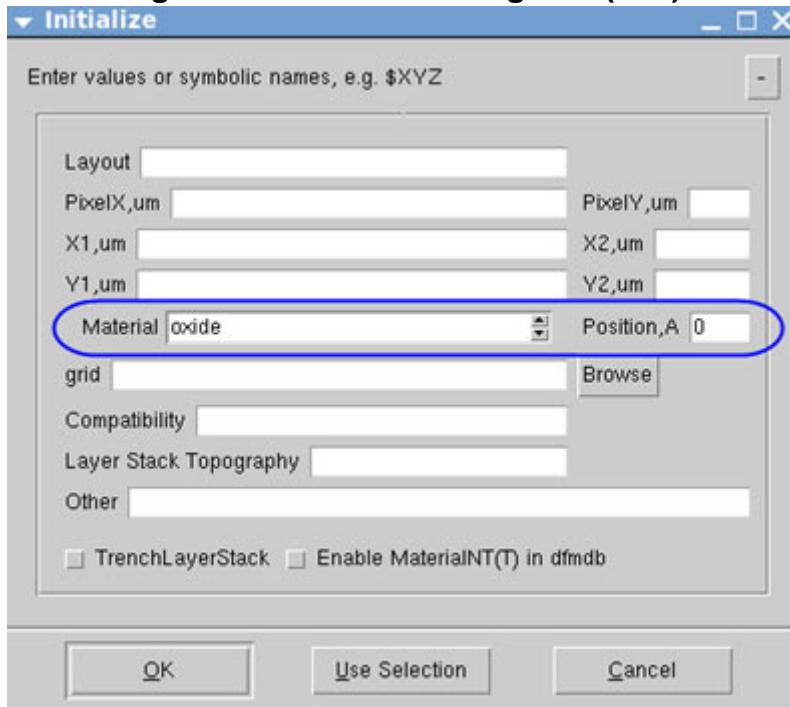
[Finishing the CMP Model](#)

## Initialize Dialog Box

To access: In the **Recipe** tab, select **Add > initialize**.

Use this dialog box to set up or edit the initialize command in a process recipe.

**Figure 6-8. Initialize Dialog Box (Full)**



## Objects

**Table 6-10. Initialize Dialog Box Contents**

Field	Description
+	Toggles the dialog box to show or hide additional fields.
-	
Material	(Required) Names the material of the first layer in the layer stack. You can enter your own name, or use the standard material names accessible through the arrow keys. The material name is also used in defining etch and cmp models.  <b>Note:</b> Do not use the same material name in the initialize command and the first deposited layer.
Position,A	(Required) The bottom position of the first layer in the stack in angstroms.
Layout	Identifies a layout file (GDS or OASIS) containing mask geometries. Note that if a different layout file is loaded in Calibre WORKbench, the layout being displayed is used for simulations, not the one specified here.

**Table 6-10. Initialize Dialog Box Contents (cont.)**

<b>Field</b>	<b>Description</b>
PixelX,um PixelY,um	Sets the size of a frame for simulation. The default size is 20 um x 20 um.
X1,um Y1,um X2,um Y2,um	Restricts the simulation area to the portion of the layout contained within the specified coordinates. The default is to use the entire layout currently displayed in Calibre WORKbench.  Note that the coordinates are in microns and not DBUs.
grid	Creates an uneven initial surface layer based on the named grid file. The file must be in the working directory when a run is started.  The grid file is also used as the grid for simulation and extraction, overriding the number of grid cells otherwise computed based on layout size and pixel settings.
Compatibility	Causes CMP Model Builder to use older versions for extraction and simulation. Provided for backwards compatibility, but not recommended for new recipes.
Layer Stack Topography	Defines the topography to use for the bottom of a layer when multiple metal layers are included in a single recipe. Accepts a space-separated list of integers. The first value is assigned to the second layer. <ul style="list-style-type: none"> <li>• 0 — Use Z1 (field, or nontrench height). This is the default.</li> <li>• 1 — Use an average, Z1(1-density) + Z2(density), for each frame.</li> <li>• 2 — Use Z2 (trench height).</li> <li>• 3 — Use the greater of Z1 or Z2 for each frame.</li> <li>• 4 — Use the smaller of Z1 or Z2 for each frame.</li> <li>• 5 — Use a planar surface with the greatest Z1 or Z2 over all frames.</li> </ul> For example, a list for 4 layers could be entered as “0 2 3”. The lowest layer is always assumed to be on a flat surface. The second layer would use the nontrench height of the first layer as its starting surface; the third layer would use the trench height; and the fourth layer would use the larger of the two.
Other	Use for entering additional command parameters such as pixel manually.
TrenchLayerStack	Enables multilayer stack support inside of the trench regions.
Enable MaterialNT(T) in dfmdb	Enables the reporting and saving of MaterialNT (non-trench) and MaterialT (trench) properties to a DFM database for use in Calibre CMP Analyzer results analysis for colormaps, histograms, and reported properties.  If enabled, this option updates the “initialize” statement in the process recipe file and is not backwards compatible with older versions of Calibre CMP Model Builder (before 2018.1).

**Table 6-10. Initialize Dialog Box Contents (cont.)**

Field	Description
Use Selection	Populates the X1, Y1, X2, and Y2 fields with the coordinates of the currently displayed area in Calibre WORKbench.

## Usage Notes

See “[initialize](#)” on page 110 for more information on how the parameters that define extent interact. Generally you should define only one of Layout or grid.

## Related Topics

[Grid Files](#)

[Grid Frames \(Pixels\)](#)

## Etch Dialog Box

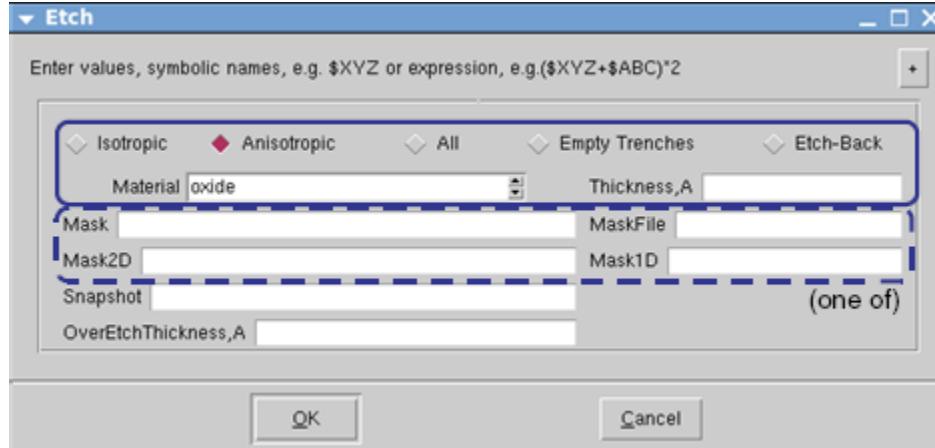
To access: In the **Recipe** tab, select **Add > etch**.

Use this dialog box to set up or edit an etch command in a process recipe.

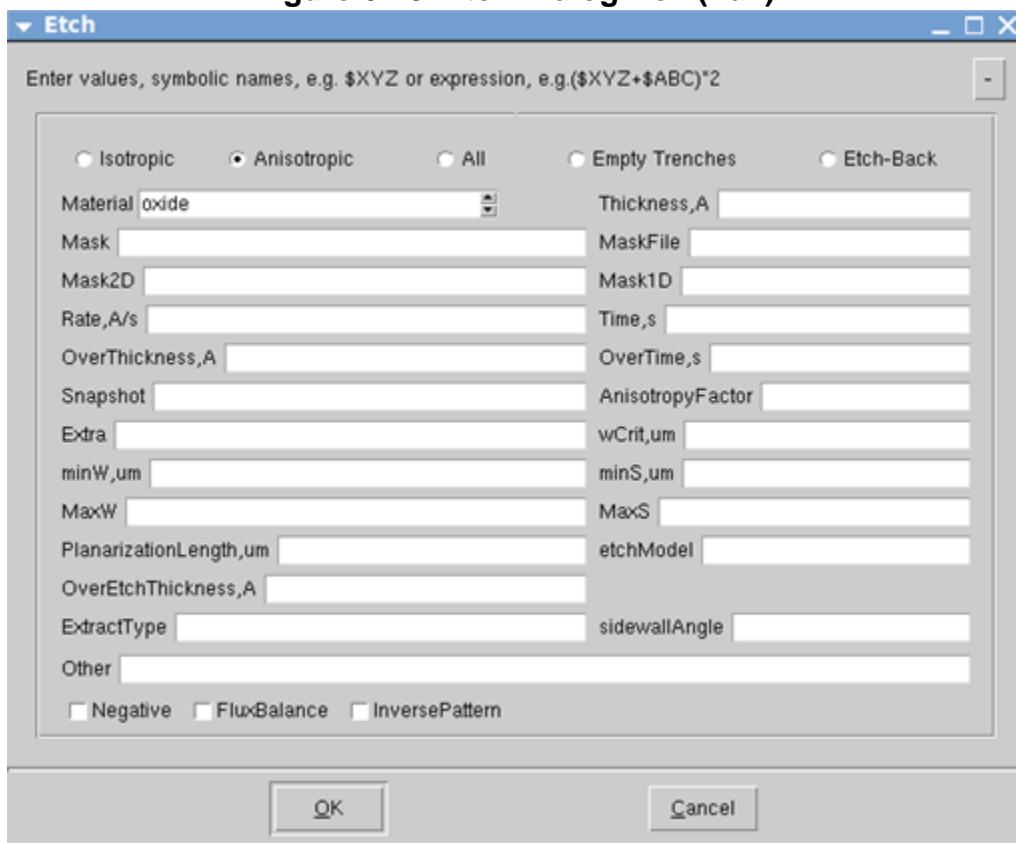
### Description

The Etch dialog box is available in basic mode and full mode. The available fields change based on the type (isotropic, anisotropic, and so on). See field descriptions in the tables following the figures.

Figure 6-9. Etch Dialog Box (Basic)



**Figure 6-10. Etch Dialog Box (Full)**



## Objects

**Table 6-11. Etch Dialog Box Contents (Basic)**

Field	Description
+	Toggles the dialog box to show additional fields.
Isotropic	(Required) Indicates the type of etch to simulate.
Anisotropic	<ul style="list-style-type: none"> <li>• Isotropic etches all surfaces equally.</li> </ul>
All	<ul style="list-style-type: none"> <li>• Anisotropic etches walls less rapidly than the material top.</li> </ul>
Empty Trenches	<ul style="list-style-type: none"> <li>• All indicates all material is to be etched away.</li> </ul>
Etch-Back	<ul style="list-style-type: none"> <li>• Empty Trenches removes material from trenches, as for an RMG process.</li> <li>• Etch-Back simulates the reverse etch-back process.</li> </ul>
Material	<p>(Required except for Empty Trenches and Etch-Back.) Names the material on top when etching begins. You can enter your own name, or use the standard material names accessible through the arrow keys.</p> <p>Note that if etching is defined as being deeper than the material thickness, etching continues to the underlying layers using the etch parameters defined for those materials.</p>

**Table 6-11. Etch Dialog Box Contents (Basic) (cont.)**

Field	Description
Thickness, A	( <i>Required</i> if not using Time and Rate.) Available in Empty Trenches. The depth in angstroms at which the simulation should halt etching. To base depth on other conditions, enter a variable name preceded by a dollar sign (\$); for example, \$thick.  To use Time and Rate instead, open the full dialog box by clicking +.
Mask	Uses a layer in the layout as the mask for simulating etching. The value must be a number, not a layer name.
MaskFile	Uses the grid data contained in the named file, which must be present in the current working directory, as the etch mask. The named file should be a base name without property prefixes.
Mask2D Mask1D	(Both unavailable in All and Etch-Back. Mask1D unavailable in Empty Trenches. Specify only one.) Uses a mask defined in the process recipe itself as the etch mask. The masks are defined using <a href="#">Mask1D Dialog Box</a> or <a href="#">Mask2D Dialog Box</a> . The value must not start with a number. Note that Mask2D is preferred over Mask1D; Mask1D is retained for compatibility with older process recipes.
MaskSelective	(Available in Etch-Back only.) Use with Mask when restricting reverse etch-back to specific areas of the layout. The selective mask indicates the areas to be etched. If used without Mask, the reverse etch-back is performed on the entire selected area.
MaskFileSelective	(Available in Etch-Back only.) Use with MaskFile when restricting reverse etch-back to specific areas of the layout. The mask in the file indicates the areas to be etched. If used without MaskFile, the mask in the file indicates that the reverse etch-back is performed on the entire selected area of the layout.
Snapshot	(Unavailable in All, Empty Trenches, and Etch-Back.) Uses the supplied number to tag the results of the etch step in the <a href="#">Measured Tab</a> .
OverEtchThickness,A	(Visible with Anisotropic and Empty Trenches.) Use to further reduce the height of field regions. You can supply either a single value, an algebraic expression, or a lookup table. Lookup tables are set up with the <a href="#">LookUpModel Dialog Box</a> .

**Table 6-12. Etch Dialog Box Contents (Full)**

Field	Description
-	Hides the additional fields and returns the dialog box to Basic view.
Rate,A/s	(Isotropic and Anisotropic only.) Use with Time to specify when etching should stop. Cannot be used with Thickness. The units are angstroms per second. To calculate the rate based on other conditions, enter a variable name preceded by a dollar sign (\$); for example, \$RATE.

**Table 6-12. Etch Dialog Box Contents (Full) (cont.)**

Field	Description
Time,s	(Isotropic and Anisotropic only.) Use with Rate to specify when etching should stop. Cannot be used with Thickness. The default units are seconds.  To calculate time based on other conditions, enter a variable name preceded by a dollar sign (\$); for example, \$TIME.
OverThickness,A	(Isotropic and Anisotropic only.) Adds additional overetching depth to the regular etching. In Isotropic mode, it also affects etch bias. To base overetch on other conditions, enter a variable name preceded by a dollar sign (\$); for example, \$OThick.
OverTime,s	(Isotropic and Anisotropic only.) Sets the simulation to overetch for the indicated number of seconds. To base overetch on other conditions, enter a variable name preceded by a dollar sign (\$); for example, \$OTime.
AnisotropyFactor	(Isotropic and Anisotropic only.) Sets sidewall thickness to be a fraction of the overall final thickness. Values can be entered as either percent (for example, 10%) or a decimal (for example, 0.10).
Extra	(Unavailable for All.) Sets a distance in microns for a halo to be used if a grid is apparently empty. If a grid contains polygons, the extra area is not checked.
wCrit,um	(Unavailable for All and Etch-Back.) Sets how large a shape can be before it is treated as a field region. This is typically used on oxide layers, where huge trenches may exist.
minW,um	(Unavailable for All.) The minimum trench width in microns possible for the manufacturing process. “Trenches” smaller than this are assumed to be incomplete.
minS,um	(Unavailable for All.) The minimum space between trenches for the manufacturing process. “Spaces” smaller than this are assumed to be artifacts.
MaxW	(Unavailable for All.) Set to the size in microns of the largest trench widths in your test structure if the simulator has been misidentifying spaces and widths. Use with MaxS and Extra.
MaxS	(Unavailable for All.) Set to the size in microns of the largest spaces in your test structure if the simulator has been misidentifying spaces and widths. Use with MaxW and Extra.
PlanarizationLength, um	(Unavailable for All and Etch-Back.) Specifies the length in microns to average pattern density.
etchModel	(Unavailable for All and Etch-Back.) Use to specify a lookup table that uses width and thickness pairs to interpolate etch. If you provide a name that is not already defined, the GUI opens the <a href="#">LookUpModel Dialog Box</a> for you to enter the data.

**Table 6-12. Etch Dialog Box Contents (Full) (cont.)**

Field	Description
ExtractType	(Unavailable for All and Etch-Back.) Set to 1 to clip trenches and spaces at the grid boundary and adhere strictly to the defined mask. The default behavior (set to 0 or leave blank) is to use a halo when computing the width, length, and perimeter of shapes within the grid.
sidewallAngle	(Always available.) Set this option to consider the geometry effect of the angle of non-vertical trench sidewalls during material removal. The trench sidewalls are assumed to be symmetrical; enter either degree values between 0 and 90 (default is 0) or an algebraic expression.
Other	(Unavailable for Etch-Back.) Use for entering additional command parameters manually.
Negative	(Always available.) Set this option to indicate that the shapes on the mask are areas that are not etched. In layouts, generally polysilicon layers are negative and oxide and metal layers are not.
NegativeSelective	(Etch-Back only.) Set this option to indicate that the shapes on the selective mask are the regions that are not etched.
FluxBalance	(Isotropic and Anisotropic only.) Set this option to include dynamic modeling of flux when calculating trench etch.
InversePattern	(Empty Trenches only.) Sets trench areas to not-trench, and not-trench areas to trench. For use with the Empty Trenches option only.

**Note**

 This table does not include fields that also appear in Basic mode. Those are described in the previous table.

**Usage Notes**

See “[etch](#)” on page 102 for more details on the parameters to which the fields correspond.

If no mask is specified (that is, all of Mask, MaskFile, Mask1D, and Mask2D are empty), etching is applied equally. This does nothing but change the vertical position of the top layer.

When you specify both a MaskFile and one of Mask, Mask1D, or Mask2D, that mask is used for etching and the grid results are saved to MaskFile.

Use type All to define a pre-etch mask. A pre-etch mask is a binary mask used inside the simulator. The grid is included with “save state” and “save dump” output. For “etch type=all”, if material is not defined then the top material is etched. If material is defined, all materials above it are etched. The specified material is etched to the specified thickness, or completely etched if thickness is not defined.

## Related Topics

- [Symbolic Names](#)
- [Numeric Expressions in Parameters](#)
- [Mask 1D and 2D](#)
- [Grid Frames \(Pixels\)](#)

## Deposit Dialog Box

To access: In the **Recipe** tab, select **Add > deposit**.

Use this dialog box to set up or edit deposit commands.

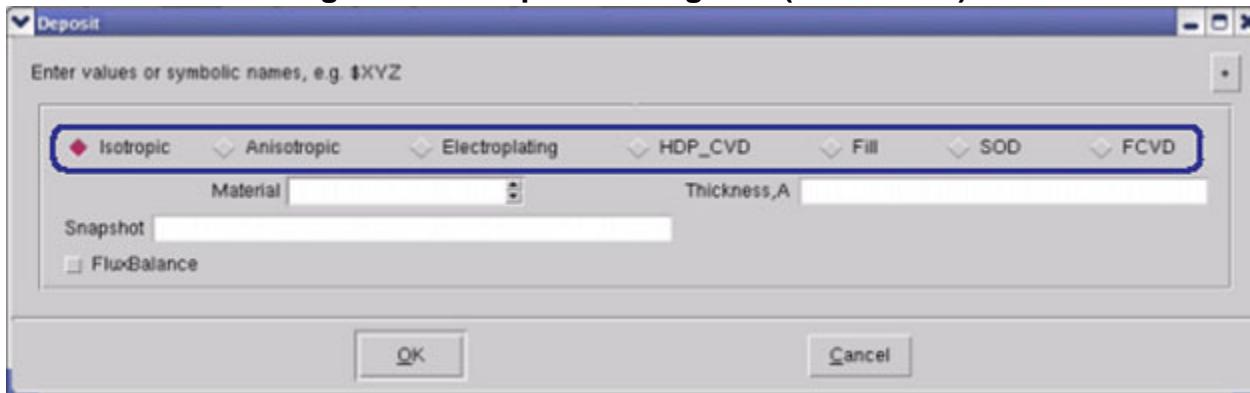
### Description

The Deposit dialog box by default shows only the required fields for the type of deposit chosen. This varies significantly between types.

#### Note

Because each deposit type changes the available fields significantly, the Deposit Dialog Box Contents table lists all fields in alphabetical order instead of dialog box order.

Figure 6-11. Deposit Dialog Box (Initial View)



### Objects

Table 6-13. Deposit Dialog Box Contents

Field	Description
+	Toggles the dialog box to show or hide additional fields. Unlike other dialog boxes, + only shows fields applicable to the deposit type.
-	

**Table 6-13. Deposit Dialog Box Contents (cont.)**

Field	Description
Isotropic Anisotropic Electroplating HDP_CVD Fill SOD	Indicates the type of deposit, and also controls which fields are displayed. <ul style="list-style-type: none"> <li>• Isotropic deposits material evenly over field, trench walls, and trench bottom.</li> <li>• Anisotropic deposits material more thinly along trench walls.</li> <li>• Electroplating dynamically models deposition as an interplay of reaction accelerators and suppressants.</li> <li>• HDP_CVD models high density plasma chemical vapor deposition.</li> <li>• Fill should be used for simple modeling. You define only the position of the top surface after fill; it is assumed to be flat.</li> <li>• SOD deposits material as a spin-on dielectric. The material settles into the trenches from an initially even deposition.</li> </ul>
anisotropyFactor	(Isotropic+, Electroplating+) The ratio of sidewall accumulation versus field thickness for deposited material. For example, a value of 0.5 indicates the sidewall coating is half as thick as the top surface coating.
Corner Bias, um	(HDP) A positive or negative distance between the edge of the trench and the start of build-up. See <a href="#">Figure 4-12</a> on page 95 of the deposit command.
Corner Bias N, um	(HDP) As with Corner Bias, but restricted to narrow trenches.
Critical Space, um	(HDP) The width of the active area at which deposited material starts to flatten at the top of the mound.
Critical Space N, um	(HDP) As with Critical Space, but restricted to narrow trenches.
Current,Amp	(Electroplating+) Total current over the wafer in amperes during electrochemical deposition (ECD). It affects the nominal plating rate.
CurrentDensity,Amp /cm <sup>2</sup>	(Electroplating+) Local current in amperes per square centimeter. It affects the nominal copper deposition rate.
Diameter,mm	(Electroplating+) The diameter of the wafer.
Dishing	(Electroplating+) When SaveDt and File are also set, includes dishing data in the intermediate grid files.
dt,s	(Electroplating+) The interval in seconds to use when simulating ECD.
Epsilon	(Isotropic+) An amount by which to round the corners of deposited edges. For narrow trenches, this raises the trench bottom more rapidly.
Erosion	(Electroplating+) When SaveDt and File are also set, includes erosion data in the intermediate grid files.
File	(Electroplating+) Provides a base filename for saving information at intermediate steps of electroplating.

**Table 6-13. Deposit Dialog Box Contents (cont.)**

<b>Field</b>	<b>Description</b>
FluxBalance	(Isotropic) Set this option to include dynamic modeling of flux when calculating isotropic deposition.
HeightFactor	(SOD) Required for SOD models. Indicates the relative depth of dishing. Set to a value between 0 (very shallow) and 1 (very deep).
Mask	(Electroplating+) When SaveDt and File are also set, includes mask data in the intermediate grid files.
MaskFileSelective	(Isotropic+, Anisotropic+, Fill+) Contains the file name of the selective mask to read. Use with MaskSelective when restricting deposit to only pixels covered by the selective mask. The mask in the file indicates the areas to be deposited.
MaskSelective	(Isotropic+, Anisotropic+, Fill+) Contains the layer number used for the selective mask. It restricts the deposit to the areas indicated by the shapes on the layer number. If NegativeSelective is specified, the areas outside the polygons are deposited rather than the areas inside the polygons.
Material	(All) The name of the material being deposited.
Maximum Dishing, A	(HDP) The difference in angstroms between the highest and lowest point of the deposited surface, measured over the trenched region.
minW	(Isotropic+, Electroplating+) The minimum width of a trench before it is considered filled.
Model	(Electroplating) Required for Electroplating. Specify the name of an ECD model. If the model does not already exist, the GUI prompts you with the <a href="#">EcdModel Dialog Box</a> after you click <b>OK</b> .
NegativeSelective	(Isotropic+, Anisotropic+, Fill+) Use this option to indicate that the mask supplied by MaskSelective is a negative; that is, the shapes on the layer indicate the areas that are not deposited.
NSteps	(Electroplating+) Sets how many time steps the electroplating calculation should use. Total time of electroplating is NSteps x dt. The default is 200.
Other	(All+) Use for entering additional command parameters manually.
Position	(Fill) Required for Fill. Sets the top of the surface to the specified height in angstroms.
Rate,A/s	(Electroplating) The deposition rate of the material in angstroms per second. If you specify this, you must also specify deposit duration, either with time or nsteps and dt.
SaveDt,s	(Electroplating+) The interval at which to save intermediate states during electroplating. Set to a multiple of dt.

**Table 6-13. Deposit Dialog Box Contents (cont.)**

Field	Description
Sidewall Angle, degree	(SOD) Use this option only with SOD deposition. It considers the effect of the angle of the trench sidewalls on the erosion of the deposited layer. The trench sidewalls are assumed to be symmetrical; enter degree values between 0 and 90. The default is 0.
SidewallsOnly, A	(Anisotropic) Use this option to support material deposition only on the trench sidewalls. If enabled, the thickness value of the deposition in angstroms (Thickness, A) is used to update the geometry data.
Slope Angle, degree	(HDP) The angle of the deposited material, in degrees. See <a href="#">Figure 4-15</a> on page 98 of the deposit command.
Slope Angle N, degree	(HDP) The angle of the deposited material for narrow trenches. Also requires Slope Angle and Space Threshold.
Smax, um	(HDP) Enables enhanced HDP_CVD deposition. Set to a value roughly equal to Corner Bias + 0.5(Critical Space). (Corner Bias and Critical Space do not need to be set.)
Smax N, um	(HDP) Smax for narrow trenches.
Snapshot	(All) Uses the supplied number to tag the results of the etch step in the <a href="#">Measured Tab</a> .
Space Threshold, um	(HDP) Ignored unless Smax is also set. Indicates the threshold between wide and narrow trenches.
TargetThicknessNT, A	(Electroplating+, HDP, Fill) The thickness in angstroms for field regions at which deposition should halt. Causes Time to be ignored.
TargetThicknessT,A	(Electroplating+, HDP, Fill) The trench thickness at which deposition should halt. Thickness is measured from the bottom of the trench to the top of the deposited material and is in angstroms. Causes Time to be ignored.
TargetWeight	(Electroplating+) Specify a grid file that contains a weighting factor for each frame. The default weighting is 1 (no derating).
TargetX,um TargetY,um	(Electroplating+) Coordinates in microns for a specific point at which to check thickness, for use with TargetThicknessNT or TargetThicknessT. Must both be specified or left blank.
Thickness,A	(Isotropic, Anisotropic, Electroplating+, SOD) The intended thickness of the deposition in angstroms.
ThicknessT	(Electroplating+) When SaveDt and File are also set, includes trench thickness in the intermediate grid files.
ThicknessNT	(Electroplating+) When SaveDt and File are also set, includes field thickness in the intermediate grid files.

**Table 6-13. Deposit Dialog Box Contents (cont.)**

Field	Description
Time,s	(Electroplating) The length of time in seconds for which material is being deposited.
WidthBias, um	(SOD) An empirical bias factor, useful when trenches are overetched but you do not have an etch model in the process recipe.
WidthFactor	(SOD) Required for SOD models. Indicates the relative width of dishing. Recommended to fit this to measured data (use a symbolic name such as \$WIDTH), but you may set to a value between 0 (narrow) and 2000 (wide).
Z1	(Electroplating+) When SaveDt and File are also set, includes field height in the intermediate grid files.
Z2	(Electroplating+) When SaveDt and File are also set, includes trench height in the intermediate grid files.

## Usage Notes

All fields except Material and Snapshot accept variables (strings beginning with \$ such as \$THICK) as well as set values.

See “[deposit](#)” on page 92 for more details on the parameters to which the fields correspond.

## Related Topics

[Save Dialog Box](#)

## Cmp Dialog Box

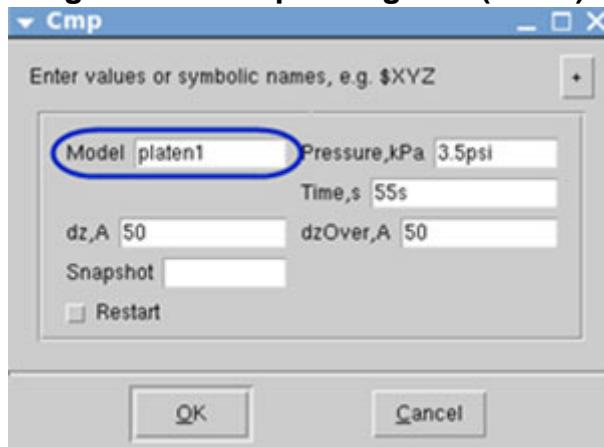
To access: In the **Recipe** tab, select **Add > cmp**.

Use this dialog box to set up or edit cmp commands.

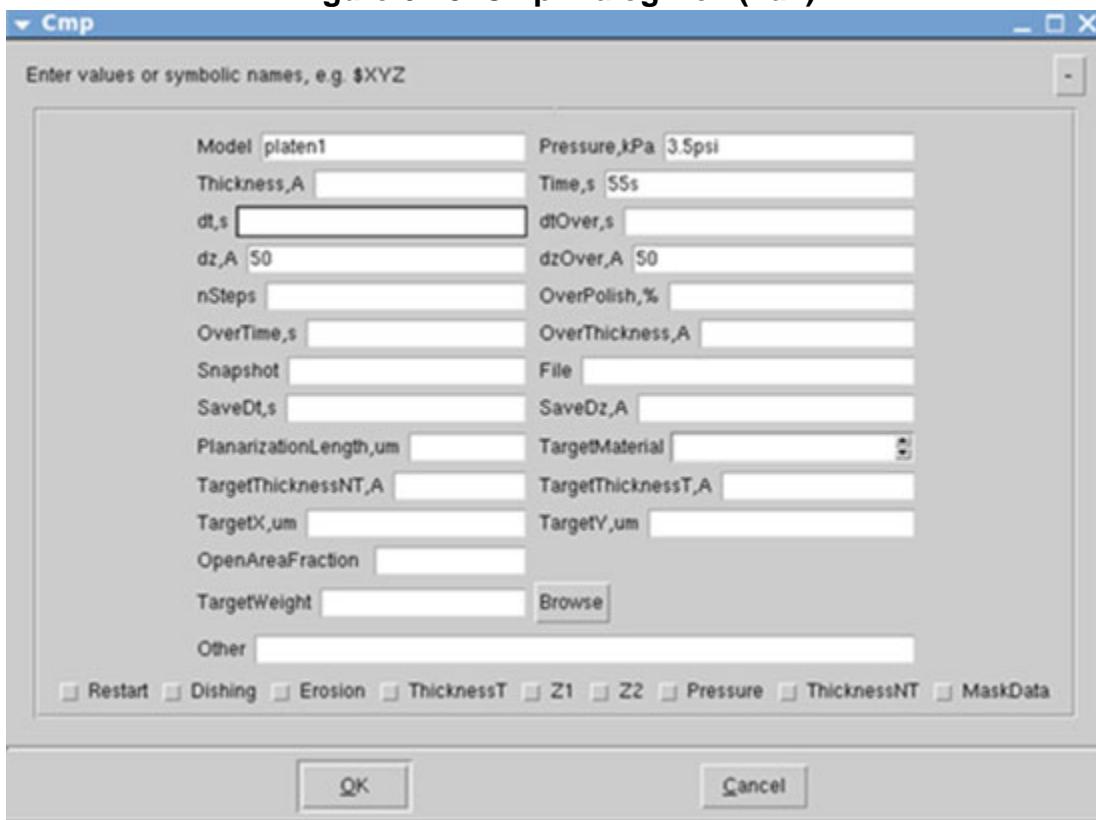
### Description

The Cmp dialog box is available in basic mode and full mode (Figure 6-13). See the field descriptions in the tables following the figures.

**Figure 6-12. Cmp Dialog Box (Basic)**



**Figure 6-13. Cmp Dialog Box (Full)**



## Objects

**Table 6-14. Cmp Dialog Box Contents (Basic)**

Field	Description
+	Toggles the dialog box to show additional fields.
Model	(Required) Specify the name of a CMP model. If the model does not already exist, the GUI prompts you with the <a href="#">CmpModel Dialog Box</a> dialog box after you click OK.
Pressure, kPa	The pressure used in polishing, in kilopascals. You can use other units by including them after the value; for example, 3.5psi. To calculate the pressure that best matches measurements, enter a variable name preceded by a dollar sign (\$); for example, \$PRESSURE.
Time,s	How many seconds polishing is applied. To calculate the duration that best matches measurements, enter a variable name preceded by a dollar sign (\$); for example, \$TIME.
dz,A	Thickness increment to use per iteration.
dzOver,A	Thickness increment to use per iteration during overpolish.
Snapshot	Uses the supplied number to tag the results of the etch step in the <a href="#">Measured Tab</a> .

**Table 6-14. Cmp Dialog Box Contents (Basic) (cont.)**

Field	Description
Restart	Indicates the start of a platen model. Many recipes split the CMP step into several cmp commands. Setting Restart starts the time at 0 instead of resuming from the last cmp command.

**Table 6-15. Cmp Dialog Box Contents (Full)**

Field	Description
Thickness,A	The number of angstroms removed by polishing. To calculate the thickness that best matches measurements, enter a variable name preceded by a dollar sign (\$); for example, \$THICK.
dt,s	An interval of time in seconds to use as a time-step. Combined with nSteps, can be used instead of Time.
dtOver,s	An interval of time in seconds to use during the overpolish stage. Overpolish refers to polishing after underlying material has been exposed.
nSteps	How many increments (either time or thickness) to use. Can be used in place of Time.
OverPolish,%	The point at which overpolish begins. That is, the percentage of overall CMP time spent from start until the topmost layer is cleared in all frames. To calculate the amount of overpolish that best matches measurements, enter a variable name preceded by a dollar sign (\$); for example, \$OP.
OverTime,s	An absolute time measurement in seconds indicating how long overpolish continues.  To calculate the duration of overpolishing that best matches measurements, enter a variable name preceded by a dollar sign (\$); for example, \$OT.
OverThickness,A	How much material is removed during overpolish, in angstroms. Note that underlying materials almost always have a different removal rate than the original top material.  To calculate the thickness that best matches measurements, enter a variable name preceded by a dollar sign (\$); for example, \$OT.
File	Required with SaveDt or SaveDz. Specifies the base name to use for generated grid files.
SaveDt,s	Generates grid files saved at the specified time intervals during CMP. The name of the grid files is determined by the File setting.
SaveDz,A	Generates grid files saved at the specified material removal intervals during CMP. The name of the grid files is determined by the File setting.
Planarization Length,um	The length over which pattern density is averaged.
TargetMaterial	If using OpenAreaFraction, targetThicknessT, or targetThicknessNT, specify the material to which those apply. Ignored if none of those are set.

**Table 6-15. Cmp Dialog Box Contents (Full) (cont.)**

<b>Field</b>	<b>Description</b>
TargetThicknessNT,A	Halts polishing when the field region equals the specified thickness. Overrides any time settings.
TargetThicknessT,A	Halts polishing when the trench region equals the specified thickness. Overrides any time settings.
TargetX,um TargetY,um	Coordinates in microns for a specific point at which to check thickness, for use with TargetThicknessNT or TargetThicknessT. Must both be specified or left blank.
OpenAreaFraction	Halts polishing when the percentage of frames with TargetMaterial exposed reaches the specified value. (Frames that do not have any underlying TargetMaterial are not included as part of the total.) Can be specified as a number between 0 and 1 (0.1) or as a percentage (10%).
TargetWeight	Specify a grid file that contains a weighting factor for each frame. The default weighting is 1 (no derating).
Other	Use for entering additional command parameters manually.
Dishing Erosion ThicknessT Z1 Z2 Pressure ThicknessNT MaskData	Various properties that can be included in the grid files saved when File and either SaveDt or SaveDz are set. At least one property must be selected.

**Note**

 This table does not include fields that also appear in Basic mode. Those are described in the previous table.

**Usage Notes**

See “[cmp](#)” on page 67 for more details on the parameters to which the fields correspond.

There is some overlap between Cmp and CmpModel dialog box parameters. Generally, it is best to define all material characteristics in the CmpModel parameters, and then call the model from Cmp commands that identify target thicknesses and materials.

**Related Topics**

[CMP Model](#)

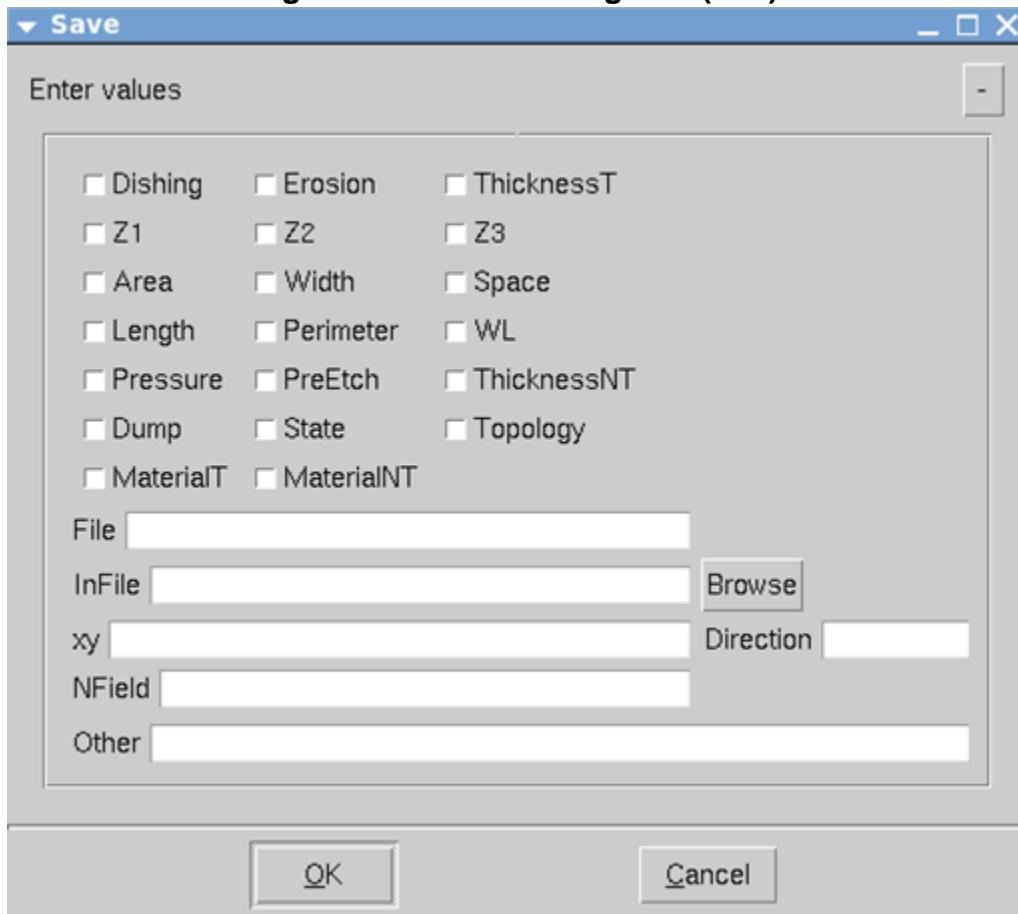
[CmpModel Dialog Box](#)

## Save Dialog Box

To access: In the **Recipe** tab, select **Add > save**.

Use this dialog box to set up or edit a save command in a process recipe file. This dialog box is not used for saving sessions or process recipes.

**Figure 6-14. Save Dialog Box (Full)**



## Objects

**Table 6-16. Save Dialog Box Contents**

Field	Description
+	Toggles the dialog box to show or hide additional fields.
-	

**Table 6-16. Save Dialog Box Contents (cont.)**

Field	Description
Property options	Various properties that can also be chosen to appear. The full list is documented under “ <a href="#">save</a> ” on page 114. <ul style="list-style-type: none"><li>• Dump saves the entire state in a binary format.</li><li>• State saves the entire state in an ASCII format.</li><li>• WL is the frame area divided by number of polygons (area per feature).</li><li>• Z1 is the non-trench, or field, height.</li><li>• Z2 is the trench height.</li><li>• Z3 is the height of the trench bottom.</li></ul>
File	A base name for the saved grid files. You can control the grid file type by including the <i>.txt</i> , <i>.text</i> , or <i>.bin</i> suffix as part of the base filename. If you are saving “dump” or “state”, the filename automatically gets a <i>.dump</i> or <i>.sta</i> file extension.
InFile	A grid file that indicates the frames to save by weighting them with a value of 1. CMP Model Builder looks for <i>&lt;file&gt;.txt</i> , <i>&lt;file&gt;</i> , and <i>cmp/&lt;file&gt;.txt</i> in that order.
Other	Use for entering additional command parameters manually.

## Usage Notes

See “[save](#)” on page 114 for more details on the property definitions.

For simulations involving multiple models, it is recommended to save the state after each model’s simulation and re-load it with [initialize](#). This reduces optimization time.

## Related Topics

[Grid Files](#)

## EcdModel Dialog Box

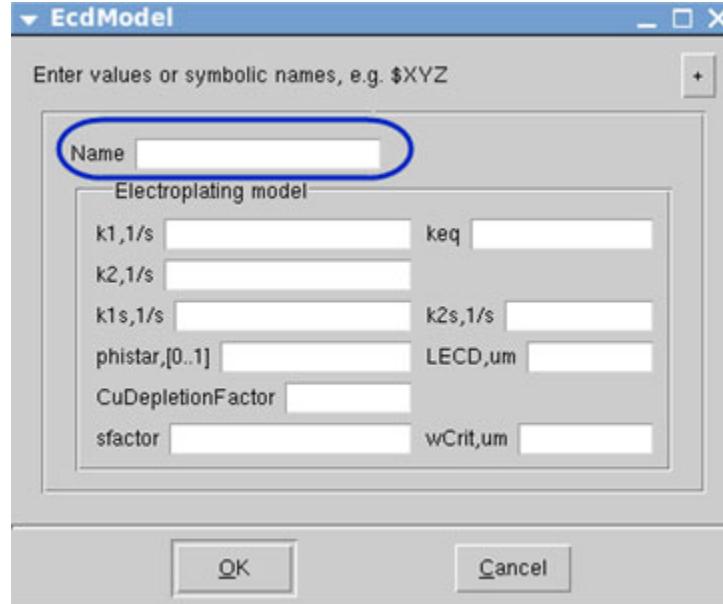
To access: In the **Recipe** tab, select **Add > ecdModel**. It also appears when specifying an undefined model name in an Electroplating [Deposit Dialog Box](#).

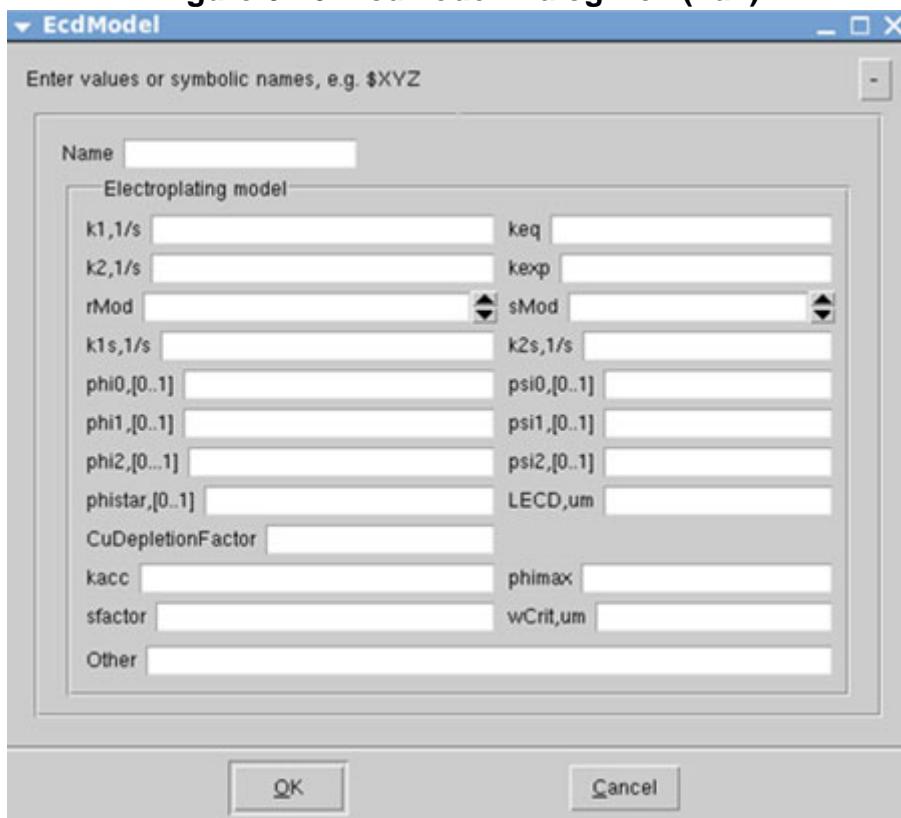
Use this dialog box to set up or edit the parameters for an ECD define\_model command.

### Description

The EcdModel dialog box is available in basic mode and full mode. See field descriptions in the tables following the figures.

**Figure 6-15. EcdModel Dialog Box (Basic)**



**Figure 6-16. EcdModel Dialog Box (Full)**

## Objects

All fields except Name accept variables (strings beginning with \$ such as \$K1) as well as numeric values.

**Table 6-17. EcdModel Dialog Box Contents (Basic)**

Field	Description
+	Toggles the dialog box to show additional fields.
Name	(Required) The name of the ECD model. This should be identical including capitalization to the value in the Model field in the <a href="#">Deposit Dialog Box</a> .
k1,1/s	The dissolution rate per second of the accelerator. The default is 0.035. Values must be between 0 and 1, inclusive.
keq	The equilibrium constant for the suppressor. The default is 15. Values must be between 0 and 20, inclusive.
k2,1/s	The dissolution rate per second of the suppressor. The default is 0.005. Values must be between 0 and 1, inclusive.
k1s,1/s	The dissolution rate per second of the accelerator during superfilling. Values must be between 0 and 1, inclusive.
k2s,1/s	The dissolution rate per second of the suppressor during superfilling. Values must be between 0 and 1, inclusive.

**Table 6-17. EcdModel Dialog Box Contents (Basic) (cont.)**

Field	Description
phistar,[0..1]	The equilibrium constant for the accelerator. The default is 0.005. Values must be between 0 and 1, inclusive.
LECD,um	Diffusion length for simulating cupric ion depletion. Values must be between 1000 and 4000 microns, inclusive.
CuDepletionFactor	Enables modeling of cupric ion depletion. (Off by default.)
sfactor	A scaling factor for modifying geometry during superplating. The default is 1.
wCrit,um	Trench width in microns beyond which trenches are too wide to superfill.

**Note**

 The “Full” table does not include fields that also appear in Basic mode.

**Table 6-18. EcdModel Dialog Box Contents (Full)**

Field	Description
-	Toggles the dialog box to hide the fields listed in this table.
kexp	An exponential factor affecting suppressor surface coverage and velocity. The default is 0, or non-exponential behavior.
rmod	Trench filling models. The default is 0.
smod	Superplating models. The default is 0.
phi0,[0..1]	Used when rmod or smod are set to something other than 0.
psi0,[0..1]	Values must be between 0 and 1, inclusive.
phi1,[0..1]	
psi1,[0..1]	
phi2,[0..1]	
psi2,[0..1]	
kacc	An initial value for the acceleration factor. The default is 0.
Other	Use for entering additional command parameters manually.

## Usage Notes

If you are simulating cupric ion depletion, it is best to optimize CuDepletionFactor, k1, and k2 together, and after fitting those optimize other parameters.

## CmpModel Dialog Box

To access: In the **Recipe** tab, select **Add > cmpModel**. It also appears when specifying an undefined model name in a [Cmp Dialog Box](#).

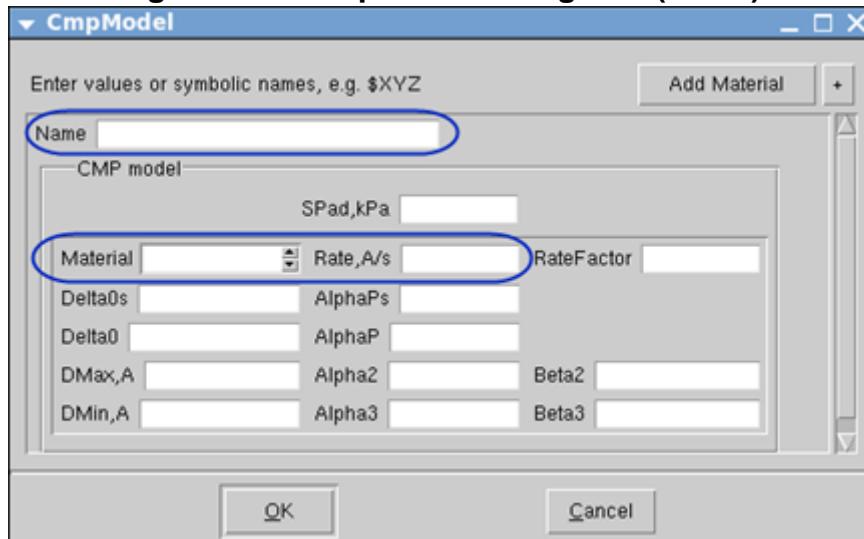
Use this dialog box to set up or edit the parameters for a CMP define\_model command.

### Description

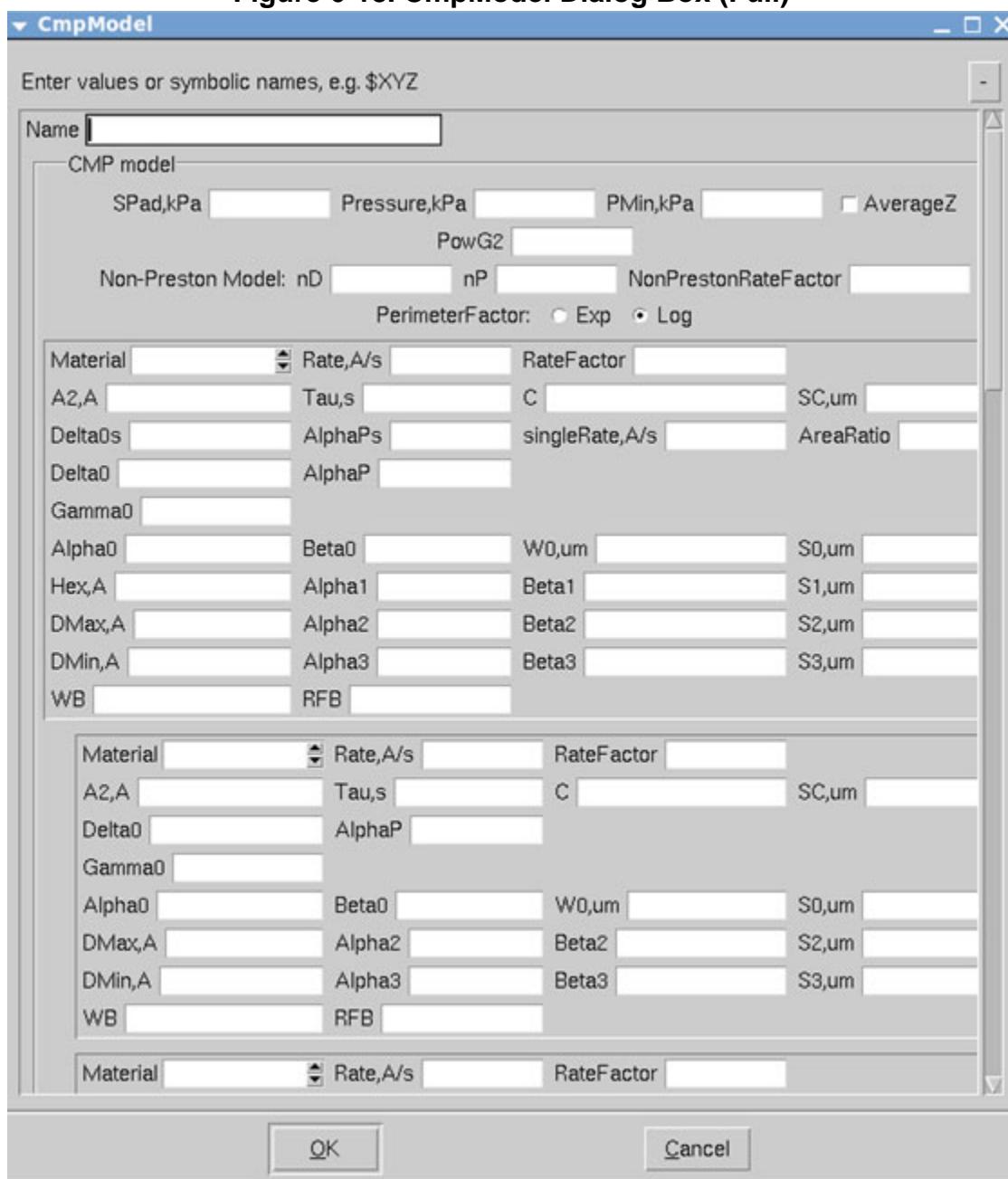
The CmpModel dialog box is available in basic mode and full mode. See the field descriptions in the tables following the figures.

All fields except Name and Material accept variables (strings beginning with \$ such as \$K1) as well as numeric values.

**Figure 6-17. CmpModel Dialog Box (Basic)**



**Figure 6-18. CmpModel Dialog Box (Full)**



## Objects

**Table 6-19. CmpModel Dialog Box Contents (Basic)**

Field	Description
Add Material	A button for adding a material and its parameter settings (initially, only the settings for one material are displayed).
+	Toggles the dialog box to show additional fields.

**Table 6-19. CmpModel Dialog Box Contents (Basic) (cont.)**

<b>Field</b>	<b>Description</b>
Name	(Required) The name of the CMP model. This should be identical including capitalization to the value in the Model field in the <a href="#">Cmp Dialog Box</a> .
SPad,kPa	Pad stiffness in kilopascals.
Material	(Required) The name of a previously deposited material.
Rate,A/s	(Required) The amount of height per second removed by polishing.
RateFactor	Sets the polishing rate of underlying materials as a ratio of the topmost material's rate. For example, 0.85 indicates the material is removed at 85% of the speed of the topmost material.
Delta0s	The delta ( $\delta$ ) value to be used for the topmost material. When the second material is exposed, the model uses Delta0 instead. Value calculations are exponential (Exp: $0 \leq \delta \leq 1$ ) or default logarithmic (Log: $0 \leq \delta \leq 5$ ) as indicated by PerimeterFactor. See “ <a href="#">Perimeter</a> ” on page 211 for model details.
AlphaPs	The alpha ( $\alpha$ ) value to be used for the topmost material. When the second material is exposed, the model uses AlphaP instead. Values must be $0 \leq \alpha \leq 5$ . See “ <a href="#">Perimeter</a> ” on page 211 for model details.
Delta0	The delta value to be used for the exposed material. If Delta0s is not specified, Delta0 is also used for the topmost material. Values calculations are exponential (Exp: $0 \leq \delta \leq 1$ ) or default logarithmic (Log: $0 \leq \delta \leq 5$ ) as indicated by PerimeterFactor. See “ <a href="#">Perimeter</a> ” on page 211 for model details.
AlphaP	The alpha value to be used for exposed material. Values must be $0 \leq \alpha \leq 5$ . See “ <a href="#">Perimeter</a> ” on page 211 for model details.
DMax,A	Characteristic height for maximum positive dishing, in angstroms. Can be used in conjunction with DMin in place of Hex.
Alpha2	A parameter for tuning the dependency between trench width and DMax. Values must be between 0 and 1, inclusive.
Beta2	A parameter for tuning the dependency between spacing and DMax. Values must be between 0 and 1, inclusive.
DMin,A	Characteristic height for maximum negative dishing, in angstroms. Can be used in conjunction with DMax in place of Hex.
Alpha3	A parameter for tuning the dependency between trench width and DMin. Values must be between 0 and 1, inclusive.
Beta3	A parameter for tuning the dependency between spacing and DMin. Values must be between 0 and 1, inclusive.

**Table 6-20. CmpModel Dialog Box Contents (Full)**

<b>Field</b>	<b>Description</b>
–	Toggles the dialog box to hide the fields listed in this table.

**Table 6-20. CmpModel Dialog Box Contents (Full) (cont.)**

Field	Description
PMin,kPa	Specifies the pressure value below which the polishing rate drops to 0.
AverageZ	Use the average height of material when optimizing pressure parameters.
PowG2	Sets an exponent to be used in the nonlinear dishing model.
Non-Preston Model parameters	Fields particular to the non-Preston pressure model, which polishes two exposed materials. It is also used when there is a difference in pressure because of a rough surface. See “ <a href="#">define_model</a> ” for details on nD, nP, NonPrestonRateFactor, and PerimeterFactor.
PerimeterFactor	Sets the type of calculation to be used. Choose either exponential or logarithmic (default). See “ <a href="#">Perimeter</a> ” on page 211 for details on the perimeter model.
Material	Groups properties by material type. Supply materials in the same order they are exposed by polishing. The topmost material group is the same one shown in Basic mode, and includes some fields not available to underlying materials.
Hex,A	For blanket copper removal, the step height at which the polishing pad does not reach the bottom. The default value for copper is 10 angstroms.
Alpha1	A parameter tuning the dependency between trench width and Hex. Values must be between 0 and 1, inclusive.
Beta1	A parameter tuning the dependency between spacing and Hex. Values must be between 0 and 1, inclusive.
S1,um	A lower bound for the simulation space used in fitting Hex. The default value is 100 microns.
S2,um	A lower bound for the simulation space used in fitting DMax. The default value is 100 microns.
S3,um	A lower bound for the simulation space used in fitting DMin. The default value is 100 microns.
WB	Width bias (wb) parameter value for the enhanced perimeter model (EPM). This parameter is enabled when wb > 0. Specifies the transition region between trench and non-trench patterns for a given material. It affects the removal rate of the material and must be used together with RFB. See “ <a href="#">Perimeter</a> ” on page 211 for details on the perimeter model.
RFB	Rate-factor bias (rbf) parameter value for the enhanced perimeter model (EPM). This parameter is enabled when rfb > 0. Specifies the rate correction for the material due to transitions between trench and non-trench patterns. It affects the removal rate of the material and must be used together with WB. See “ <a href="#">Perimeter</a> ” on page 211 for details on the perimeter model. <ul style="list-style-type: none"> <li>• rfb &gt; 1 increases the polishing rate by rfb.</li> <li>• rfb &lt; 1 reduces the polishing rate by rfb.</li> </ul>

**Table 6-20. CmpModel Dialog Box Contents (Full) (cont.)**

<b>Field</b>	<b>Description</b>
A2,A Tau,s	Use only when modeling a copper removal rate that depends on time.
C SC,um	Changes removal rate to depend on line spacing. This has been observed to cause edge rounding in oxide removal. These should be set to a variable to be optimized by matching measured data.
singleRate, A/s	Overrides Rate when only the topmost material is on the surface, if different from the overall polishing rate.

**Note**

 This table does not include fields that also appear in Basic mode. Those are described in the previous table.

**Usage Notes**

Although you can define properties for only a single material, if the polishing might expose underlying materials you should expand the dialog box and define properties for them as well. Model parameters are organized into functional groupings:

- General parameters that apply to all materials, such as pad pressure (pressure) and pad stiffness (spad).
- Parameters describing polishing effects for copper.
- Parameters describing polishing effects for oxide.

When polishing occurs in stages, you must define a separate model for each stage or platen. Thus, a three stage polishing involving bulk copper removal, copper clear, and barrier polish requires a minimum of three cmp models.

For more accurate results, calibrate DMax, DMin, Alpha, and Beta instead of accepting default values; these parameters are very dependent on the manufacturing process, and also affect the optimization of other parameters.

**Related Topics**

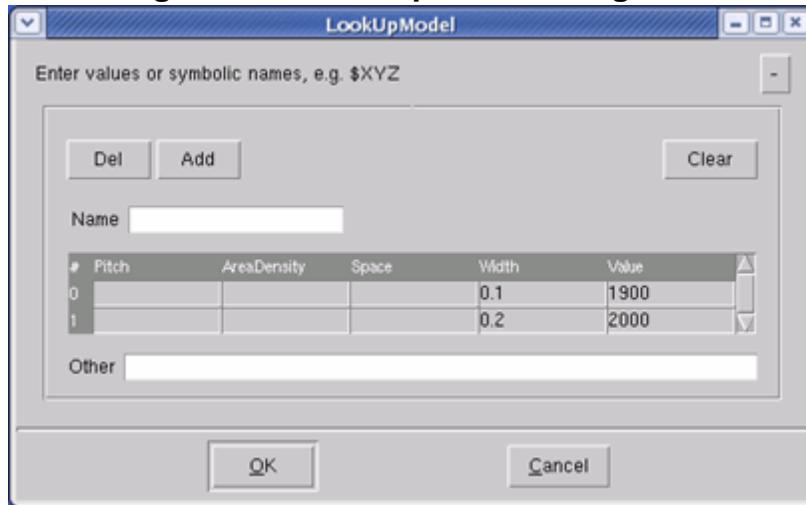
[CMP Model](#)

## LookUpModel Dialog Box

To access: In the **Recipe** tab, select **Add > lookupModel**. Also appears when you provide a lookup model name in a field that accepts them.

Use this dialog box to set up or edit a lookup table to use for non-constant parameter values.

**Figure 6-19. LookUpModel Dialog Box**



## Objects

**Table 6-21. LookUpModel Dialog Box Contents**

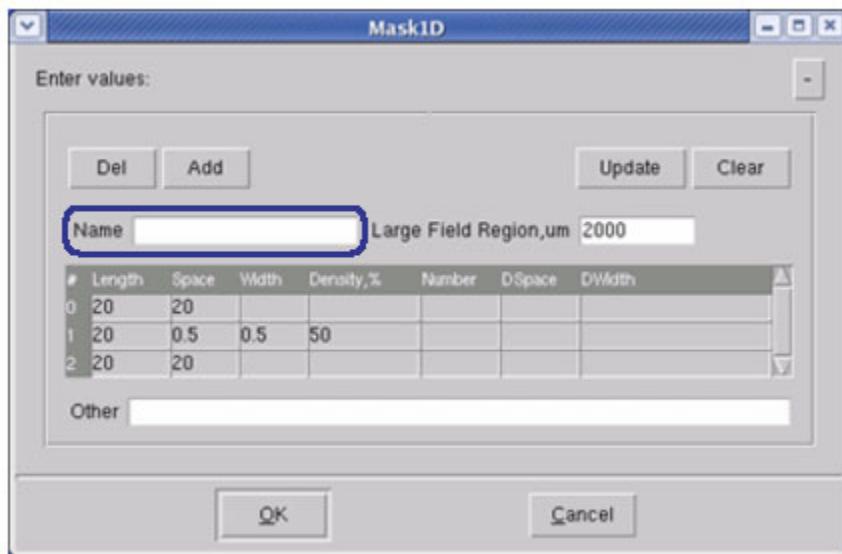
Object	Description
+	Toggles the dialog box to show or hide additional fields.
-	
Del	Buttons that act on rows that are selected in the table. Del removes the row entirely. Clear removes the values in the row.
Add	
Clear	
Name	The name must be identical, including capitalization, to the spelling in the fields that call it. Names must begin with a letter but can include numbers and underscores.
Table data	The properties and resulting values. The Value column can contain calculations or references to symbolic names used in other commands. Values are not interpolated but used stepwise.
Other	Use for entering additional command parameters manually.

## Mask1D Dialog Box

To access: In the **Recipe** tab, select **Add > define\_mask 1D**.

Use this dialog box to set up a 1-dimensional (linear) mask.

**Figure 6-20. Mask1D Dialog Box**



### Objects

**Table 6-22. Mask1D Dialog Box Contents**

Field	Description
+	Toggles the dialog box to show or hide additional fields.
-	
Del	Buttons that act on rows selected in the table.
Add	<ul style="list-style-type: none"> <li>• Del removes the row entirely.</li> </ul>
Update	<ul style="list-style-type: none"> <li>• Clear removes the data in the rows.</li> </ul>
Clear	<ul style="list-style-type: none"> <li>• Update updates density based on length, space, and width.</li> </ul>
Name	(Required) The name to use in other command's Mask field. Valid names may include letters, numbers, and underscores.
Table data	The values to use in creating a mask. See “ <a href="#">define_mask 1D</a> ” on page 72 for a detailed description.
Other	Use for entering additional command parameters manually.

### Usage Notes

No fields in the mask dialog boxes may take symbolic names or mathematical expressions.

This command dialog box is preserved for backwards compatibility. Generally two-dimensional masks are required to fit the large number of test structures on a chip.

## Related Topics

[Mask2D Dialog Box](#)

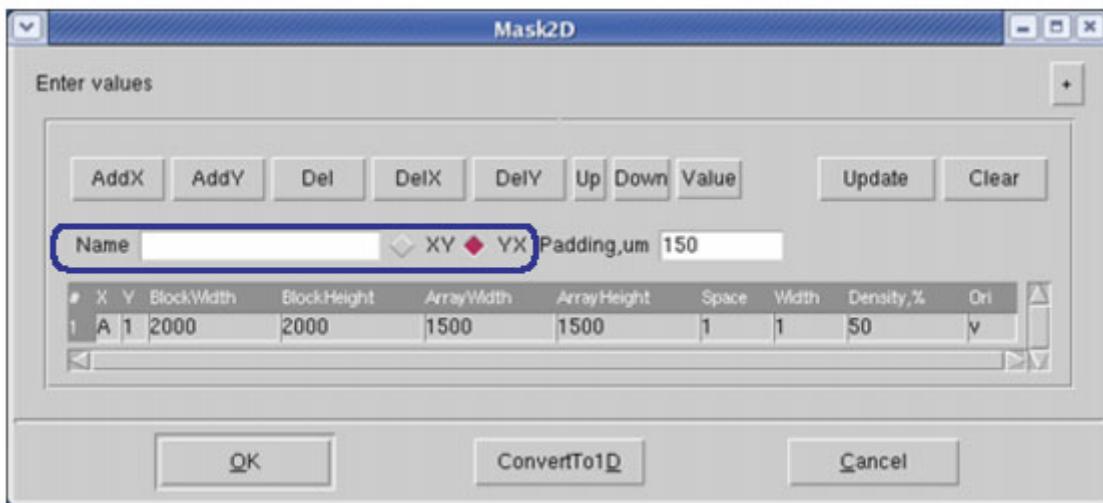
[Mask 1D and 2D](#)

## Mask2D Dialog Box

To access: In the **Recipe** tab, select **Add > define\_mask 2D**.

Use this dialog box to set up a two-dimensional mask; that is, a mask in which the test structures form a grid. For Mask2D, expanding the dialog box reveals additional columns in the table data but does not expose more fields or controls.

**Figure 6-21. Mask2D Dialog Box (Basic)**



## Objects

**Table 6-23. Mask2D Dialog Box Contents**

Field	Description
+	Toggles the dialog box to show or hide additional columns in the table. The expanded data table requires horizontal scrolling.
-	
AddX	Adds an entire row or column to the grid of test structures. The new lines have appropriate X and Y values, but all other parts of the table are empty.
AddY	
Del	Del removes the selected lines of the table. Clear removes the values of the selected lines of the table.
Clear	
DelX	Removes all lines matching the X (or Y) value of the selected lines in the table.
DelY	Allows you to remove entire rows or columns from the grid.
Up	Move the selected lines up or down in the table. Note that the X and Y values do NOT move with the other data.
Down	
Value	A dropdown menu allowing you to construct algebraic expressions for use in the table.
Update	Updates any dependent cells in the table such as density based on changes.

**Table 6-23. Mask2D Dialog Box Contents (cont.)**

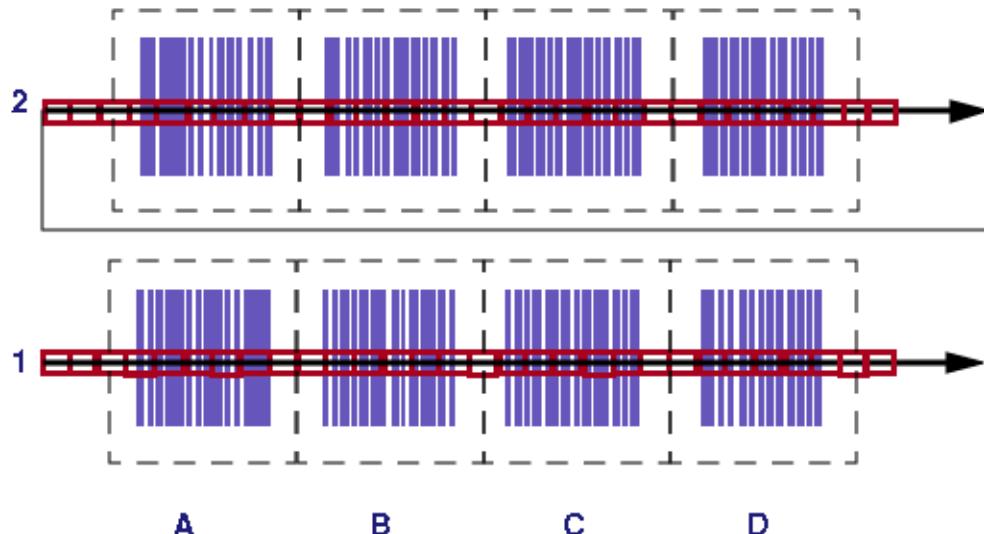
Field	Description
Name	(Required) The name to use in the Mask field of other commands. The name cannot start with a number.
XY YX	Specifies whether the blocks are in row-column or column-row order. Does not affect the block identifier, which is always XY.
Padding,um	How much space to leave between the array edge and the block boundary.
Table data	The values to use in creating a mask. See “ <a href="#">define_mask 2D</a> ” on page 74 for a detailed description. The minimum parameters you must set are BlockHeight, BlockWidth, ArrayHeight, ArrayWidth, Space, and Width.
ConvertTo1D	Converts the two-dimensional grid to an array of blocks according to the XY or YX setting. See <a href="#">Figure 6-22</a> .

### Usage Notes

When setting up your test grid, be sure to leave sufficient space around the arrays to serve as a field region. Field regions are necessary for calibrating model parameters.

See “[define\\_mask 2D](#)” on page 74 for extensive visual explanations of what the parameters in the table mean.

**Figure 6-22. Details of Mask 2D to 1D Conversion**



Converting 2D masks to 1D masks with the XY option (first along the X direction then along the next row).

## Related Topics

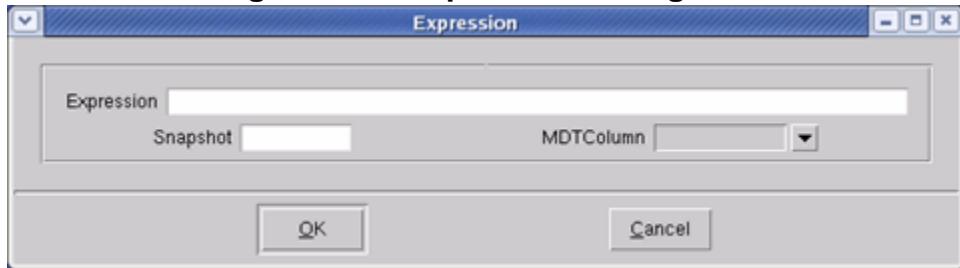
[Mask1D Dialog Box](#)

## Expression Dialog Box

To access: In the **Recipe** tab, select **Add > Expression**.

Use this dialog box to add expressions that populate the measured data table with calculated values. The expression can be used as an optimization parameter. The value computed by the expression is considered to be a simulated value.

**Figure 6-23. Expression Dialog Box**



### Objects

**Table 6-24. Expression Dialog Box Contents**

Field	Description
Expression	Enter an algebraic expression to assign to a parameter. You can use the functions mentioned in “ <a href="#">Numeric Expressions in Parameters</a> ” on page 58 as well as parentheses for grouping.
Snapshot	Restricts the expression to using only the values with a matching snapshot. This is a required value.
MDTColumn	A list of available measured data values to which the final value of the expression should be compared. Only one column can be used per expression.

### Usage Notes

X and Y are independent variables in the expression, not the X and Y columns of the measured data file.

The value of the expression appears in the MDT in the simulated column for the MDTColumn property. For example, if MDTColumn is set to Erosion, then the expression value appears in SErosion.

When the expression command is used to optimize parameters for use in a function, it must be the only step in the recipe.

### Related Topics

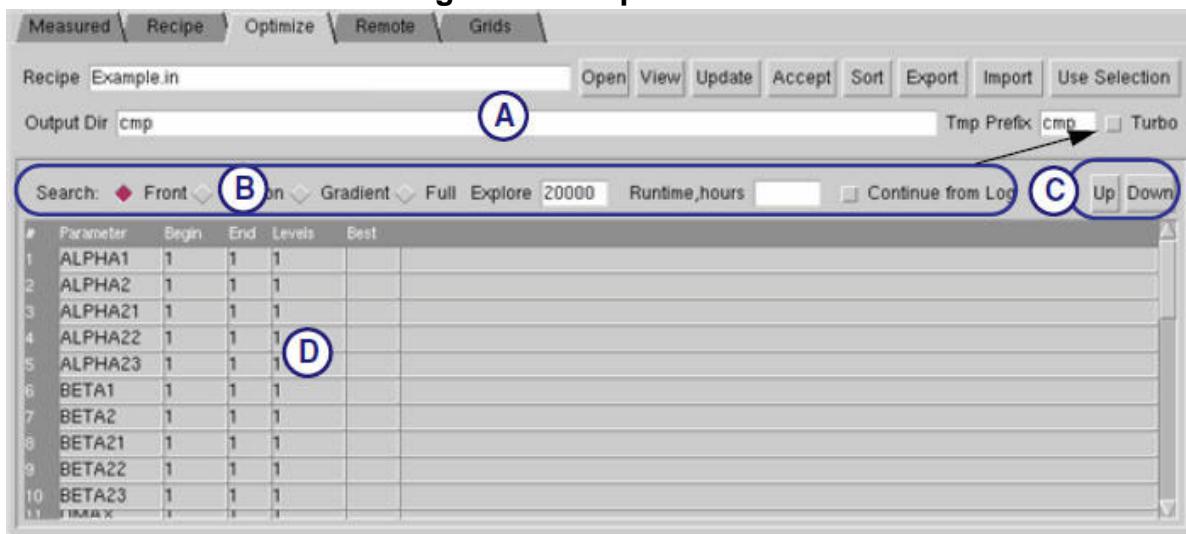
[Optimizing Parameterized Expressions](#)

# Optimize Tab

To access: From Calibre WORKbench, **Tools > CMP Model Builder**, click the **Optimize** tab.

Use this interface to adjust optimization runs and view the results. If you click the + button in the upper right corner of the window, additional controls are displayed.

**Figure 6-24. Optimize Tab**



## Objects

**Table 6-25. Optimize Tab Contents**

Field	Description
A	<p>Higher level controls for data manipulation.</p> <p><b>Recipe</b> — The filename of the recipe from which the parameters are drawn. You can enter a new filename in the field, but it is not automatically loaded.</p> <p><b>Open</b> — Loads a new recipe file and updates the parameter table (D). This action also changes the recipe displayed in the Recipe tab.</p> <p><b>View</b> — Opens the file shown in the Recipe field in a text editor.</p> <p><b>Update</b> — Updates the parameter table (D) with the parameters from the file shown in the Recipe field. Use this after editing a recipe to pick up changed parameters.</p> <p><b>Accept</b> — Changes the current recipe to use the best value for the selected parameters. The original process recipe file is not changed until you save the recipe.</p> <p><b>Sort</b> — Arranges the parameters in alphabetical order. This is not recommended.</p> <p><b>Export</b> — Saves the parameter table in a text file.</p> <p><b>Import</b> — Loads a previously saved parameter table.</p>

**Table 6-25. Optimize Tab Contents (cont.)**

Field	Description
A (cont.)	<p><b>Use Selection</b> — Draw the area in the WORKbench window to use for simulation. The default is to use all portions of the layout currently showing in WORKbench.</p> <p><b>Output Dir</b> — The location for files created during optimization.</p> <p><b>Tmp Prefix</b> — The prefix to identify temporary files.</p>
B	<p>Optimization controls, only visible when the run is set to O (Optimization).</p> <p><b>Search Front/Newton/Gradient/Full</b> — Algorithm to use for optimization. The default is <u>Front</u>. See “Usage Notes” for more description.</p> <p><b>Explore</b> — Maximum number of runs allowed in the exploration stage. The default is <u>20,000</u>.</p> <p><b>Runtime, hours</b> — Maximum number of hours to run. The default is unlimited.</p> <p><b>Continue from Log</b> — If you have logs in the analysis area (E in “<a href="#">CMP Model Builder Main Window</a>” on page 130), you can use this to continue an optimization run based on previous work.</p> <p><b>Turbo</b> — Enables MT processing for optimizations. Must be selected before clicking Run.</p>
C	<p><b>Parameter ordering</b> — Use the <b>Up</b> and <b>Down</b> buttons to move selected parameters in the table.</p>
D	<p>Parameters to be optimized.</p> <p><b>Parameter</b> — The parameter name as it appears in the recipe, stripped of the \$ character.</p> <p><b>Begin, End</b> — The limiting range of values to try while optimizing.</p> <p><b>Level</b> — The number of values to try when optimizing. A value of 1 (the default) results in only the parameter’s Begin value being tried.</p> <p><b>Best</b> — The recommended setting post-optimization.</p> <p>The parameters listed first are optimized first and given more weight for the final solution.</p>

**Table 6-26. Additional Optimize Tab Contents**

Field	Description
Epsilon	Ignored except when using <b>Newton</b> search. Specifies how tightly optimization converges.
ExploreFullFront	Ignored except when using <b>Front</b> search. Check this button to simulate all possible combinations in the Front search experiment space. The number of experiments may slightly exceed Explore value. If this button is not checked, the Front search exploration stops as soon as the number of experiments equals the Explore value.

## Usage Notes

### Best Practices

To ensure reasonable time to completion, limit the number of parameters per optimization run to 10 or fewer. If you have more than 10 parameters you want to optimize together, you can optimize parameters with the largest effect first, then load their log files and optimize secondary parameters.

### Optimization Methods

The optimization algorithms are the same as those used for Calibre WORKbench.

- Front optimization is the default. It uses a global search optimization method that is less dependent on a starting point than the Gradient and Newton options, which is why it is the recommended method. Front optimization performs search iterations based on the explore or runtime options, and uses up to 16 CPUs in -turbo mode.
- Newton optimization uses a quasi-Newton optimization method, in which the second derivatives are evaluated in terms of the sequence of gradients from successive iterations using an update formula for the Hessian matrix.
- Gradient optimization uses a simple conjugate gradient descent method. It is faster than the other methods but often gives different results.
- Full optimization causes a full factorial search over all combinations of variables. It is the slowest of the optimization methods.

## Remote Tab

To access: From Calibre WORKbench, **Tools > CMP Model Builder**, click the **Recipe** tab.

Use this tab to view the contents of remote configuration files and set the run to be distributed (MTflex or MTflex in hyperscaling mode).

**Figure 6-25. Remote Tab**



## Objects

**Table 6-27. Remote Tab Contents**

Area	Description
A	File name of the remote configuration file. This is not editable. Use <b>Remote &gt; Open</b> and <b>Remote &gt; Save</b> .
B	Select the Run Distributed check box for faster runs. This causes optimization to use multiple CPUs.
C	File viewing area. The area can be used for creating a file, but it only offers cut, copy, and paste. Remote configuration files are text files that can be created in any text editor.

## Usage Notes

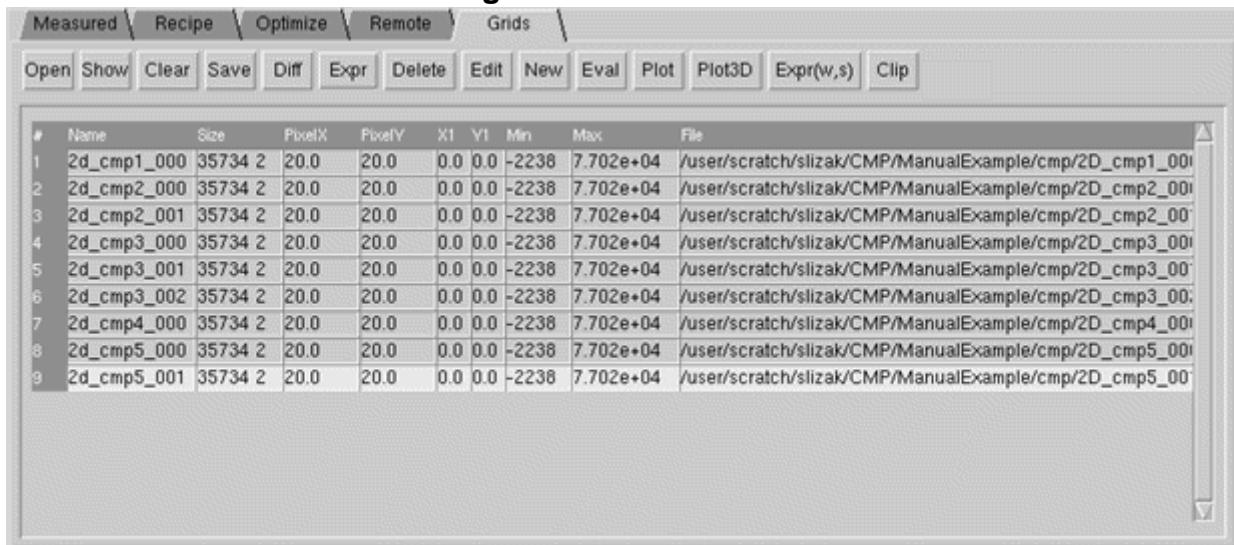
The *Calibre Administrator's Guide* describes MTflex and hyperscaling. It also has a complete description of all possible controls for a remote configuration file.

## Grids Tab

To access: From Calibre WORKbench, **Tools > CMP Model Builder**, click the **Recipe** tab.

Use this tab to view and modify saved data from a simulation. If you click the + button in the upper right of the window, additional controls are displayed.

**Figure 6-26. Grids Tab**



## Objects

**Table 6-28. Grids Tab Contents**

Buttons	Description
Open	Opens a file browser for selecting grid files to list in the table. Another way to list grids in the table is to open a log file using the analysis controls (E in “ <a href="#">CMP Model Builder Main Window</a> ” on page 130) and then accept the grids.
Show	Displays the selected grid or grids in the Calibre WORKbench layout as a colormap.
Clear	Removes the colormap from the layout.
Save	Saves the selected grids to new files. The filename you supply is used as a prefix. For example, if the selected grid is “z1Andz2” and the file name is “run1”, the saved file is “z1Andz2_run1.txt”.
Diff	Creates a new grid based on at least one other grid. If you have only one grid selected and press Diff, you are prompted for a value which is then subtracted from the grid in a uniform manner. If you have two grids selected, the one listed later is always subtracted from the one listed first in the table. To get the effect of subtracting in the other order, use <b>Expr &gt; Invert</b> on the result to change the order.

**Table 6-28. Grids Tab Contents (cont.)**

<b>Buttons</b>	<b>Description</b>
Expr	Opens a dropdown list of operations you can apply to the selected grid. This creates a new grid in the table.
Delete	Removes the selected grids from the table.
Edit	Opens a dialog box displaying the numerical data in the grid and controls for changing the values. This changes the data in memory, but not in the grid file.
New	Creates a new grid with a single data value.
Eval	Evaluates the selected grids at the coordinates selected in the MD table in the <b>Measured</b> tab, and adds them to the MD table as a new column. If the <b>Measured</b> tab is empty, Eval returns an error.
Plot	Opens a new dialog box showing an XY plot of the selected grids. The plots use standard Calibre WORKbench zoom and pan mouse strokes.
Plot3D	Opens a new dialog box showing a topological plot of the selected grids.
Expr(w,s)	Opens a dialog box for creating a new grid based on a detailed calculation that may use width, space, perimeter, area, and other parameters.
Clip	Creates a new grid (or grids, if multiple grids are selected) that includes only the data pertaining to sites currently displayed in the WORKbench layout.

**Table 6-29. Additional Grids Tab Contents**

<b>Field</b>	<b>Description</b>
Toggle	Toggles the display of grid data in the Calibre WORKbench main window.
min	Minimum grid value to show in the Calibre WORKbench main window. Values less than the min value are shown as the min value.
max	Maximum grid value to show in the Calibre WORKbench main window. Values greater than the max value are shown as the max value.
log	Graphs grid data using a natural log scale.
tick	Defines the width of the frame border when grid data is shown in the WORKbench main window.
X1 X2 Y1 Y2	Can be used instead of the layout extent shown in the Calibre WORKbench main window in conjunction with the <b>Clip</b> button.
Negative Cutline	Use to obtain a correct LineScan when drawing a cutline over negative mask data in the Calibre WORKbench main window.
Keep Cut	Keeps grids generated by cutlines.
Configure Plot	Displays a dialog box for configuring the graph when you click Plot. Most significantly the Configure Plot dialog box lets you control whether grids are plotted on a single graph or individual graphs.

## Usage Notes

The easiest way to access grids is by loading a log file in the analysis controls table (area E in the [Figure 6-1](#) on page 130).

You can display grid data either in the graph window or as a color gradient map in the Calibre WORKbench main window.

When you open matrix grid files, you must select entire sets of three matched files ( $<\text{filename}>.\text{matX}$ ,  $<\text{filename}>.\text{matY}$ , and  $<\text{filename}>.\text{matZ}$ ). If you also select non-matrix files (.txt, for example) or omit one from a matrix set, the action results in a missing files error.

## Related Topics

[Viewing ThickNT, Erosion, and Dishing Plots](#)



# Appendix A

## Process Recipe File Examples

---

The file examples in this appendix demonstrate preferred style. As every semiconductor manufacturing process is proprietary, all values used are fictitious. The files can serve as templates for your calibration but do not attempt to preserve the values.

<b>File Examples .....</b>	<b>187</b>
<b>Optimizing Parameterized Expressions.....</b>	<b>193</b>
<b>Creating a Surface Plot .....</b>	<b>195</b>

## File Examples

The four file examples are provided for illustrative purposes only. They do not represent a real foundry's process.

### Deposition Model for Spin-On Dielectric

This model layers silicon, polysilicon, and silicon nitride (SiN). The layers are then etched, receive another coating of SiN and silicon oxynitride before a final layer of spin-on dielectric. The recipe saves its state before and after the last deposition.

The format of this recipe is typical of those created entirely with the CMP Model Builder GUI.

#### Example A-1. Spin-On Dielectric (SOD) Deposit Model

```
initialize material=silicon position=0
deposit thickness=2120 type=isotropic material=polysilicon
deposit thickness=540 type=isotropic material=SiN
etch mask=35 maskFile=mask thickness=540 material=SiN \
    type=anisotropic extra=150
etch thickness=2120 material=polysilicon type=anisotropic
deposit thickness=35 type=isotropic anisotropyFactor=$afDep material=SiN
deposit thickness=35 type=isotropic anisotropyFactor=$afDep material=SiON
save z1 z2 area space width length file=postSiON
deposit thickness=5000 type=SOD WidthFactor=$WF material=SOD \
    Snapshot=5 HeightFactor=$HF
save z1 z2 z3 area space width length file=PostSOD
```

### Electroplating Model

The electroplating model following makes good use of comments. Comments and whitespace are recommended for ease of maintenance.

The model starts with a base of oxide, which is immediately etched. Tantalum nitride (TaN) is added, followed by copper. The last step is an ECD deposit using symbolic names. It does not specify the material.

### **Example A-2. 2D Electroplating Model**

```
initialize pixel=20um material=oxide position=0
# Typical trench depths M1 3000 M2 3000 M3 3500
etch oxide mask2D=etchmask2 thickness=3000A type=anisotropic

# deposit barrier layer
deposit type=isotropic material=TaN thickness=10nm

# deposit seed layer
deposit type=isotropic material=copper thickness=15nm

# ecd step description
# typical deposited metal thickness M1 8500 M2 8500 M3 9500
deposit type=ecd model=ecd1 thickness=8500A time=100s dt=0.01s

=====
# model and 2D mask definitions
define_model name=ecd1 k2=$k2 Keq=$KEQ phistar=$PHISTAR \
rmod=0 smod=0
```

### **Recipe for a CMP Flow**

The following recipe includes both deposition and polishing. The bulk copper removal is done in stages, saving between each stage. All the polishing steps (cmp keyword) use the same CMP model, cmp1, but pass different parameters. The model definitions are at the end of the file.

### **Example A-3. Example with ECD and CMP Model Definitions**

```
initialize pixel=20um material=oxide position=0 y2=340

# Typical trench depths M1 3000 M2 3000 M3 3500
etch oxide mask2D=etchmask2 thickness=3000A type=anisotropic

# Using test_chip.oas
# etch oxide mask=0 thickness=3000A type=anisotropic
# deposit barrier layer
deposit type=isotropic material=TaN thickness=10nm

# deposit seed layer
deposit type=isotropic material=copper thickness=15nm

# ecd step description
# typical deposited metal thickness M1 8500 M2 8500 M3 9500
deposit type=ecd model=ecd1 thickness=8500A time=100s dt=0.01s
save file=after_ecd dishing erosion z1 z2
```

```
# bulk copper removal
cmp thickness=2000A dt=0.01s model=cmp1
save file=cmp1.bin dishing erosion z1 z2

cmp thickness=2000A dt=0.01s model=cmp1
save file=cmp2.bin dishing erosion z1 z2

cmp thickness=2000A dt=0.01s model=cmp1
save file=cmp3.bin dishing erosion z1 z2

cmp thickness=2000A dt=0.01s model=cmp1
save file=cmp4.bin dishing erosion z1 z2

cmp thickness=1000A dt=0.01s model=cmp1
save file=cmp5.bin dishing erosion z1 z2

# copper clear
cmp thickness=1000A dt=0.01s model=cmp1
save file=cmp6.bin dishing erosion z1 z2

# overpolish
cmp thickness=100A dt=0.01s model=cmp1
save file=cmp7.bin dishing erosion z1 z2

# overpolish
cmp thickness=100A dt=0.01s model=cmp1
save file=cmp8.bin dishing erosion z1 z2

# =====
# model and 2D mask definitions
# spad=-500A critical step height
# spad=5000kPa pad stiffness
define_model name=cmp1 spad=5000kPa pressure=10psi \
    material=copper a1=200A/s a2=900A/s \
    Hex=3nm alpha1=0.15 beta1=0.1 \
    s1=100um \
    material=TaN rate=20A/s \
    dmax=300A alpha2=0.3 beta2=0.28 s2=100um \
    dmin=40A alpha3=0.3 beta3=0.25 s3=100um \
    material=oxide rate=10A/s C=3 SC=22.5um \
    dmax=300A alpha2=0.3 beta2=0.28 s2=100um \
    dmin=40A alpha3=0.3 beta3=0.25 s3=100um

define_model name=ecd1 k1=0.035 k2=0.05 Keq=10 phistar=0.05 \
rmod=0 smod=0
```

## Three-Platen Example

The process recipe describes all three platens of a typical production CMP model.

### Example A-4. Full Example with All Three Platens

```
initialize material=nitride pixel=20um
# real initial flat layer stack
# hard mask 350A (assume nitride)
# oxide 2 1000A
# oxide 1 3200A (or 3100A)
# Cu1 (full sheet)
# barrier
# oxide

deposit thickness=4100A material=oxide type=isotropic
save file=2d00.txt

deposit thickness=350A material=nitride type=isotropic
save file=2d01.txt # etch hard mask
etch mask2D=etchmask2 thickness=350A material=nitride type=anisotropic

save file=2d02.txt

# etch oxide underneath hard mask
etch thickness=2710A material=oxide type=anisotropic

# save dishing erosion thickness z1 z2 area space width
save file=2d03.txt

# deposit barrier layer
deposit isotropic material=TaN thickness=100A

save file=2d04.txt

# deposit seed layer
deposit isotropic material=copper thickness=150A

save file=2d05.txt

# deposit type=ecd model=ecd1 thickness=7900A time=100s dt=0.1s
# in real process
# 6.1s at 4.5A
# 30s at 6.75A
# 32s at 33.75A
# assume 300mm wafer --> 706.8cm2 area
# 6.1s at 1.943nm/s veq
# 30s at 2.914nm/s veq
# 32s at 14.58nm/s veq
# --> total of 5659A + 150A seed ==> 5809A nominal
# --> written nominal is 7000A + 600A
# --> assume 150A seed plus 7450 ecd stretching the time of the last step
# --> 7600A - 150A - 992.7A = 6457.277A / 14.58nm/s --> 44.28s
#
deposit model=ecd1 rate=1.943nm/s time=6.1s dt=0.1s type=ecd
save file=2d06.txt

deposit model=ecd1 rate=2.19nm/s time=30s dt=0.1s type=ecd
save file=2d07.txt
```

```

deposit model=ecd1 rate=14.58nm/s time=$TIME dt=0.1s type=ecd
save file=2d08.txt

# save dishing erosion thickness z1 z2 area space width file=after_ecd.txt
# bulk copper removal: platen1

cmp time=2s dz=5A model=cmp1 pressure=1.5psi layer=2 saveDz=300 \
    file=cmp.txt save2D

cmp time=53s dz=5A model=cmp1 pressure=3.5psi layer=2 saveDz=300 \
    file=cmp.txt save2D

# platen2: soft landing
cmp time=70s dz=0.5A model=cmp2 pressure=1.5psi layer=3 saveDz=300 \
    file=cmp.txt restart save2D

# platen 3: barrier hard mask and oxide polish
cmp time=6s dz=0.1A model=cmp3 pressure=2psi layer=4 saveDz=300 \
    file=cmp.txt restart save2D

cmp time=56s dz=0.1A model=cmp3 pressure=2.5psi layer=4 saveDz=300 \
    file=cmp.txt save2D

# =====
# model and 2D mask definitions
# blanket rate rate=7121A/min

define_model name=cmp1 type=cmp spad=$SPAD material=copper rate=$RATE \
    Hex=$HEX1 alphal=$ALPHA1 betal=$BETA1 s1=100um material=nitride \
    rate=0.001 material=TaN rate=0.001 material=oxide rate=0.001

#
define_model name=ecd1 type=ecd k1=$K1 k2=$K2 k1s=$K1S k2s=$K2S \
    phistar=$PHISTAR keq=$KEQ rmod=0 smod=0 LECD=$LECD

#
# turn off acc/supp: isotropic deposition
# all trenches either filled isotropically or closed and flat
# define_model name=ecd1 k1=0 k2=0 k1s=0 k2s=0 Keq=0 phistar=0 \
#    rmod=0 smod=0 type=ecd
#
# several mask sections with different width and space
# in-between two field regions for reference
# the line/space regions 1...30 are 1.5 mm each
# in-between there are 500um field regions
# the regions separating the groups are made 3000um wide to separate
# the line/space/field groups

```

## Process Recipe File Examples

### File Examples

---

```
define_mask name=etchmask2 L0=2000um L1=1500um s1=4 w1=4 \
L2=500 s2=500 L3=1500um s3=0.5um w3=0.5 L4=500 s4=500 L5=1500um \
s5=20 w5=20 L6=2000 s6=2000 L7=1500um s7=1 w7=4 L8=500 s8=500 \
L9=1500um s9=9.3 w9=4 L10=500 s10=500um L11=1500um w11=4 s11=2.7 \
L12=2000 s12=2000 L13=1500um w13=8 s13=5.3um L14=500 s14=500 \
L15=1500um w15=2 s15=2 L16=500 s16=500 L17=1500um w17=12 s17=3 \
L18=2000 s18=2000 L19=1500um w19=2 s19=0.5 L20=500 s20=500um \
L21=1500um w21=4 s21=22.7 L22=500 s22=500 L23=1500um w23=20 \
s23=8.6um L24=2000 s24=2000 L25=1500um w25=2 s25=1.3 L26=500 \
s26=500 L27=1500um w27=0.5 s27=1.2 L28=500 s28=500 L29=1500um \
w29=4 s29=0.5 L30=2000 s30=2000um L31=1500um w31=8 s31=18.7 \
L32=500 s32=500 L33=1500um w33=8 s33=2um L34=500 s34=500 L35=1500um \
w35=8 s35=0.9 L36=2000 s36=2000 L37=1500um w37=8 s37=8 L38=500 \
s38=500 L39=1500um w39=2 s39=11.3 L40=500 s40=500um L41=1500um \
w41=20 s41=2.2 L42=2000 s42=2000 L43=1500um w43=12 s43=8um L44=500 \
s44=500 L45=1500um w45=12 s45=28 L46=500 s46=500 L47=1500um \
w47=12 s47=1.3 L48=2000 s48=2000 L49=1500um w49=12 s49=68 L50=500 \
s50=500um L51=1500um w51=2 s51=4.7 L52=500 s52=500um L53=1500um \
w53=40 s53=17.1 L54=2000 s54=2000um L55=1500um w55=12 s55=12 \
L56=500 s56=500um L57=1500um w57=0.5 s57=2.8 L58=500 s58=500um \
L59=1500um w59=40 s59=4.4 L60=2000 s60=2000um

# platen 2
# blanket rate Cu rate=2718A/min
# blanket rate TaN rate=31A/min
define_model name=cmp2 type=cmp spad=$SPAD2 material=copper rate=$RATE2 \
material=TaN rate=$RATE2TAN dmax=$DMAX \
alpha2=$ALPHA2 beta2=$BETA2 s2=100um dmin=$DMAX \
alpha3=$ALPHA2 beta3=$BETA2 s3=100um
material=nitride rate=1 dmax=$DMAX \
alpha2=$ALPHA2 beta2=$BETA2 s2=100 dmin=$DMAX \
alpha3=$ALPHA2 beta3=$BETA2 s3=100 \
material=oxide rate=1 dmax=$DMAX \
alpha2=$ALPHA2 beta2=$BETA2 s2=100 dmin=$DMAX \
alpha3=$ALPHA2 beta3=$BETA2 s3=100

# platen 3
# blanket rate Cu rate=1155A/min
# blanket rate TaN rate=663A/min
# blanket rate Oxide rate=664A/min
# blanket rate Nitride rate=1896A/min
define_model name=cmp3 type=cmp spad=$SPAD3 material=copper rate=$RATE3 \
material=TaN rate=$RATE3TAN dmax=$DMAX31 \
alpha2=$ALPHA21 beta2=$BETA21 s2=100um dmin=$DMAX31 \
alpha3=$ALPHA21 beta3=$BETA21 s3=100um \
material=oxide rate=$RATE3OX dmax=$DMAX32 \
alpha2=$ALPHA22 beta2=$BETA22 s2=100um dmin=$DMAX32 \
alpha3=$ALPHA22 beta3=$BETA22 s3=100um \
material=nitride rate=$RATE3NI dmax=$DMAX33 \
alpha2=$ALPHA23 beta2=$BETA23 s2=100um dmin=$DMAX33 \
alpha3=$ALPHA23 beta3=$BETA23 s3=100um
```

# Optimizing Parameterized Expressions

Expression-based optimizations work differently from typical multi-parameter optimizations. A major difference is that the recipe must contain only the expression step.

The following example walks through fitting a function for trench depth and dishing, with a dependency on trench width. What makes this situation fit well to an expression recipe is that a complex function has a dependency on a measured property.

## Prerequisites

- Basic Measured Data (MD) file.

## Procedure

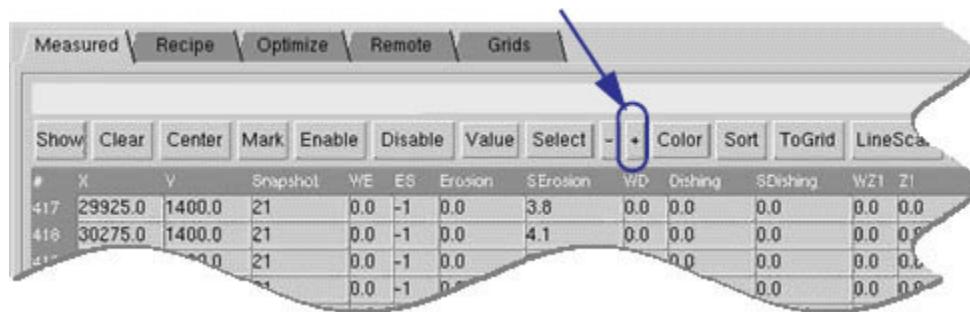
1. Fill in the MD file with the following values:

- The data you want to fit. In general, this would be Erosion, ThickT (thickness in the trench region), ThickNT (thickness in the non-trench region), Z1, or Z2.
- A positive weight in the associated weighting column. For Erosion, the associated weighting column is WE; for ThickT, it is WTT; and so on.
- The initial values for the parameters you intend to use. These should go in the TMP1, TMP2, TMP3, ... TMP10 columns.

For this example, because the goal is to fit trench depth and dishing, the data to fit goes in the Dishing column.

Because the data is in the Dishing column, the WD column values are set to 1. In the CMP Model Builder window, you can set all values with two clicks in the **Measured** tab: select the column, and then click the **Enable** button.

Because the results depend on trench width, the trench width values go in TMP1. If you are using the GUI, you may need to click + to display the TMP columns.



#	X	Y	Snapshot	WE	ES	Erosion	SErosion	WD	Dishing	SDishing	W21	Z1
417	29925.0	1400.0	21	0.0	-1	0.0	3.8	0.0	0.0	0.0	0.0	0.0
418	30275.0	1400.0	21	0.0	-1	0.0	4.1	0.0	0.0	0.0	0.0	0.0
419	30275.0	1400.0	21	0.0	-1	0.0		0.0	0.0	0.0	0.0	0.0
420	30275.0	1400.0	21	0.0	-1	0.0		0.0	0.0	0.0	0.0	0.0

If X and Y coordinates are not used in the expression and they are not already in the initial MD, their values can be the same for all rows as they will be ignored.

2. Create a process recipe whose only step is expression. This can be done by editing a text file or using the GUI.

When expression is used, it should be the only command in the recipe.

For this example, the expression to optimize is

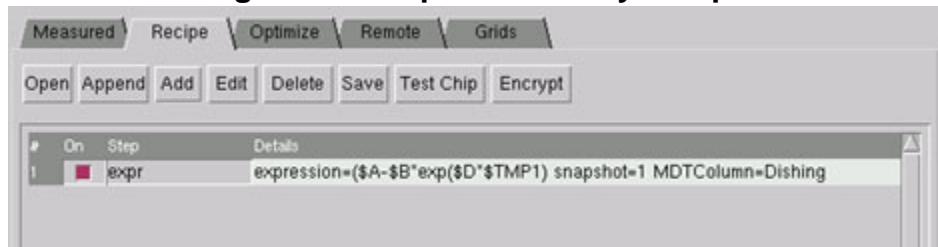
`(\$A+\$B*exp (\$D*\$TMP1)`

Parameters A, B, and D are values that will be fitted during optimization. \$TMP1 refers to the data in the MDT in the TMP1 column. In Step 1, this was filled with measured trench width values. Any standard Tcl math operators can be used, such as those listed in “[Numeric Expressions in Parameters](#)” on page 58.

The snapshot in this example is set to 1. You should set yours to a value used by data in the MDT. If the snapshot values do not match, optimization does not run.

The MDTColumn is set to Dishing, which means simulated values will be compared to measured Dishing data, and the expression’s values will appear in SDishing in the table.

**Figure A-1. Expression-Only Recipe**



The expression can reference data from more than one column of the MDT using the form “`$colname`”.

3. Set the optimization ranges for the parameters. This can be done at the command line (see the [cmpoptimize](#) shell command, especially the model tuning examples), but the GUI is recommended.
  - a. In the CMP Model Builder window, load the MD file in the **Measured** tab and the recipe, if you have not already.
  - b. Click the **Optimize** tab. The expression parameters are listed in the table.
  - c. Adjust the parameter order of optimization with the Up/Down buttons and specify each parameter’s range (Begin, End, and Level columns).

In the expression shown in [Figure A-1](#), the parameters that need to be set are A, B, and D. The data referred to by \$TMP1 will not be optimized.

4. Run the optimization.

## Results

The parameter values that produce the lowest error, measured as RMS between the MDTColumn values and the overall value of the expression, appear in the “Best” column in the **Optimize** tab.

After the run completes, the MD Table shows the simulated values in the indicated column. In this example, the values appear in SDishing because MDTColumn was set to Dishing.

To test the fit of the values, accept the Best values. Replace the fitted parameters (A, B, and D) with the Best values and then use the expression in a model in a typical recipe.

## Related Topics

[Expression Dialog Box](#)

# Creating a Surface Plot

Although Calibre WORKbench plots can show a surface plot, many engineers are more comfortable with other graphing software. The CMP Model Builder GUI can save standard surface grids in a tabular form that is easily viewed in MathWorks® MATLAB™, gnuplot, Octave, or other plotting tools.

## Prerequisites

- Grid data from a previous run.
- Preferred graphing software.

## Procedure

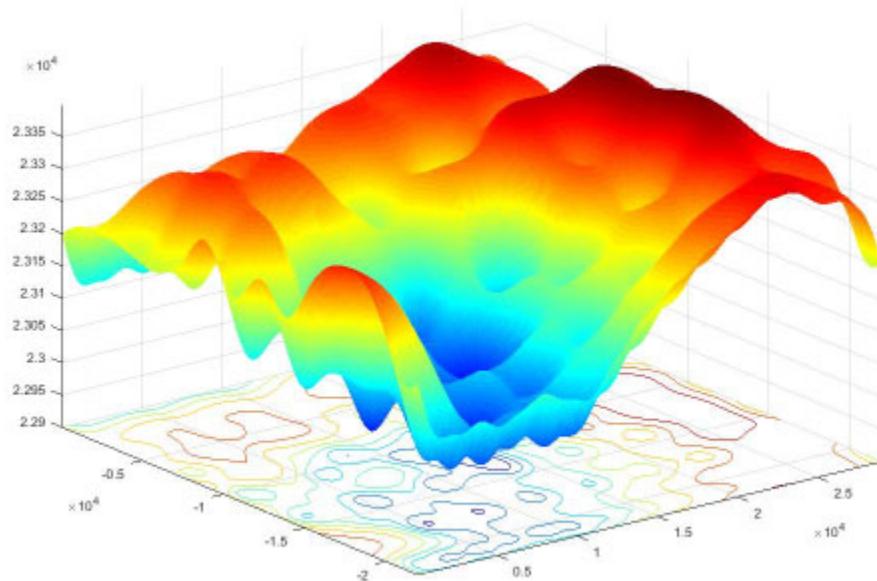
1. In the **Grids** tab of the CMP Model Builder window, click **Open** to load at least one grid file.
  2. Click the row of the grid file to work with.
  3. Export the grid in matrix format.
    - a. Click **Save**.
    - b. In the Save grid dialog box, change the File type to “Export matrix (\*.matZ, \*.matX, \*.matY).”
    - c. Supply a file name and click **Save**.
- The grid data is saved to three files, *<filename>.matX*, *<filename>.matY*, and *<filename>.matZ*.
4. Import the matrix files into your preferred software. In MATLAB, run the following script after substituting your filename for z1.

For example, in MathWorks MATLAB, you might run a script like the following, which uses z1.

```
Z=dlmread('z1.matZ');  
x=dlmread('z1.matX');  
y=dlmread('z1.matY');  
[X Y]=meshgrid(x,y);  
surf(X,Y,Z)  
%pcolor(X,Y,Z)  
%mesh(X,Y,Z)  
%surf(X,Y,Z)  
shading flat  
colormap jet
```

## Results

The script produces a surface plot like the one following:



# Appendix B

## CMP Model Builder Components

---

There are many components of a CMP model and it is imperative to know how they fit together. Some files are required inputs for an optimization run.

Other files are used primarily for storing temporary information for optimizations. The following components are the major components of CMP modeling.

<b>CMP Models</b> . . . . .	<b>197</b>
<b>Layouts</b> . . . . .	<b>198</b>
<b>Measured Data Files (MD Files)</b> . . . . .	<b>199</b>
<b>Mask 1D and 2D</b> . . . . .	<b>200</b>
<b>Snapshots</b> . . . . .	<b>201</b>
<b>Grid Files</b> . . . . .	<b>202</b>
Grid File Types . . . . .	203
Extracting Grids for Large Test Chips . . . . .	203
<b>Grid Frames (Pixels)</b> . . . . .	<b>205</b>
<b>Optimized CMP Model File</b> . . . . .	<b>209</b>

## CMP Models

CMP models are required for running Calibre CMPAnalyzer.

CMP models take the form of a process recipe file and define the process steps used to fabricate a chip. These models provide simulation capabilities for the following types of process steps:

- isotropic (conformal) deposition (for example, barrier layer, copper seed layer and dielectric oxide layers)
- anisotropic etching (for example, the oxide layers, using mask information)
- electrochemical deposition (ECD) of copper, including the nearly isotropic (conformal) deposition for wide trenches and the super-filling and bulge formation for narrow trench/narrow space regions.
- CMP of copper, barrier layer, and oxide, including the following stages:
  - a. bulk copper removal
  - b. copper clear (soft landing)
  - c. barrier removal (overpolish)

CMP Model Builder uses parameterized equations to represent electrochemical deposition and chemical mechanical polishing. Developing an accurate CMP model involves calibrating the models for both steps to measured line scans, copper thicknesses and oxides for wafers with exposed test structures.

## Related Topics

[Process Recipe File Format](#)

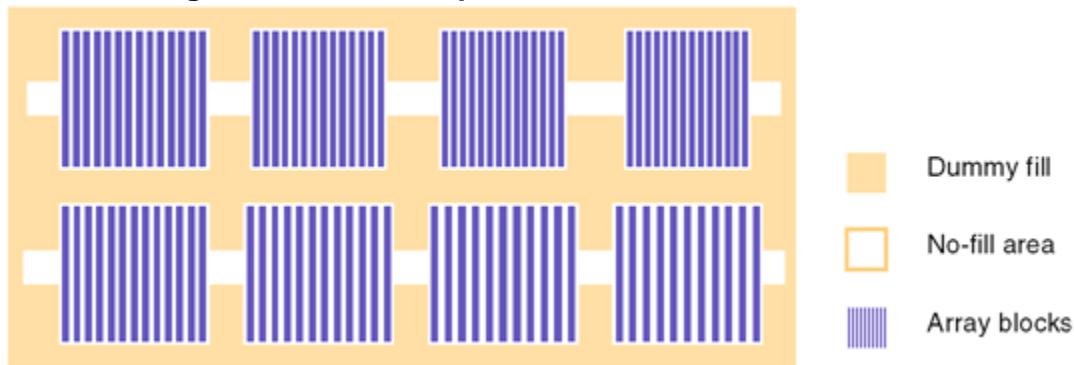
# Layouts

Layouts used in CMP Model Builder should be flat (non-hierarchical) and in GDS or OASIS format.

## Test Structures

Test structures should be a grid of arrays of long trenches, with each array using a different spacing or width value. Separate the arrays with an empty area (the field) that is large enough to minimize CMP long-range effects between the array blocks. The array blocks may be surrounded by fill, but there should be an area that excludes fill between the array blocks. Take linescan measurements along the no-fill band.

**Figure B-1. Test Chip General Structure**



Restrictive design rules introduced at the 20 nm technology node improve layout uniformity but prohibit long parallel trenches in array blocks for FEOL CMP. Instead of trenches you can use a regular pattern of similarly oriented rectangles separated by a variety of spacing values.

The array blocks should cover the full range of allowable density, spacing, and width values. Within each array block, hold the values constant. Typically, there are 50 or more array blocks on a test chip.

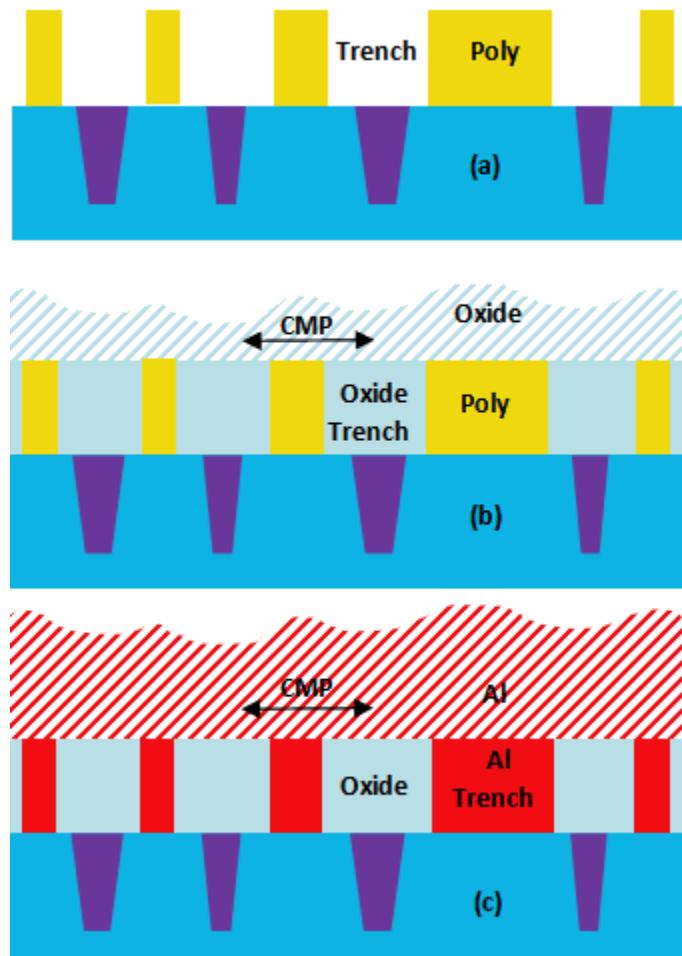
## Physical Layer Reuse

Some processes (for example, HKMG and RMG) include “sacrificial layers”; that is, an etched layer that is filled and then removed, with the resulting trenches filled by another material. In

these situations, the same layout layer may be used for two different material layers, with one specifying to use the negative of the layout layer.

For example, test chip patterns of poly-open planarization (POP) with aluminum RMG are correlated. (See [Figure B-2](#).) In aluminum RMG technology, the sacrificial polysilicon (poly) layer is etched (a). The trenches are filled with oxide (b) and then the wafer is polished. The poly layer is then replaced with aluminum (c) and polished. For the CMP model, the oxide deposition and polishing commands an inverse or negative of the poly layer. The aluminum deposition uses the unmodified poly layer from the layout.

**Figure B-2. Cross Section View of Aluminum RMG Technology**



## Measured Data Files (MD Files)

MD files contain thickness information about your test structures and patterns after they have gone through the manufacturing process. The MD file has up to 34 columns that contain simulation and measurement values.

For initial runs, only seven columns are required. Simulation and optimization runs create data for columns that begin with “S” (that is, SErosion, SDishing, SThickT, and SThickNT). Relevant columns for typical simulations are displayed. Other columns that hold optimized values are initially hidden.

**Figure B-3. Measured Data File in CMP Model Builder**

The screenshot shows the CMP Model Builder interface with the 'Measured' tab selected. The main window displays a table of data with the following columns: #, Comment, X, Y, Snapshot, WE, ES, Erosion, SErosion, WD, Dishing, SDishing, WTT, ThickT, SThickT, WTNT, ThickN. The data consists of 12 rows, each starting with a double slash (//). The data is as follows:

#	Comment	X	Y	Snapshot	WE	ES	Erosion	SErosion	WD	Dishing	SDishing	WTT	ThickT	SThickT	WTNT	ThickN
268	//	26200.0	12600.0	21	0.0	-1	-4.1	0.7	0.0	212.8	0.0	0.0	0	8749.4	0.0	0
269	//	26800.0	12600.0	21	0.0	-1	-6.9	0.4	0.0	212.3	0.0	0.0	0	8749.6	0.0	0
290	//	27350.0	12600.0	21	0.0	-1	-2.7	4.6	0.0	214.9	0.0	0.0	0	8745.4	0.0	0
291	//	24500.0	11000.0	21	0.0	-1	-3.9	1.8	0.0	213.3	0.0	0.0	0	8748.2	0.0	0
292	//	25500.0	11000.0	21	0.0	-1	-3.9	2.1	0.0	212.3	0.0	0.0	0	8748.0	0.0	0
293	//	26200.0	11000.0	21	0.0	-1	-6.3	0.5	0.0	213.8	0.0	0.0	0	8749.6	0.0	0
294	//	26800.0	11000.0	21	0.0	-1	-5.3	0.3	0.0	5.7	0.0	0.0	0	8749.7	0.0	0
295	//	27350.0	11000.0	21	0.0	-1	-161.8	2.9	0.0	5.9	0.0	0.0	0	8747.1	0.0	0
296	//	24500.0	9400.0	21	0.0	-1	-4.1	1.4	0.0	214.4	0.0	0.0	0	8748.6	0.0	0
297	//	25500.0	9400.0	21	0.0	-1	-5.4	1.6	0.0	213.3	0.0	0.0	0	8748.4	0.0	0

## Related Topics

[Measured Data File Format](#)

## Mask 1D and 2D

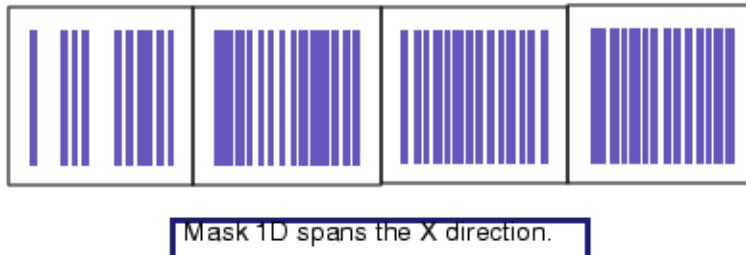
A mask is an automatically generated pattern for a test chip. Based on parameters you provide, it generates sets of trenches (arrays) laid out in a one-dimensional (1D) or two-dimensional (2D) pattern.

The test structures should always include at least three spacing variations for each trench width, and at least three width variations for each trench-to-trench spacing. For best data, use smaller steps in increasing the minimum values than you would for the larger values.

Defining a mask pattern can be done with the define\_mask command in the process recipe file. This command allows you to define a 1D mask or a 2D mask. These two mask schemes characterize how simulations are carried out.

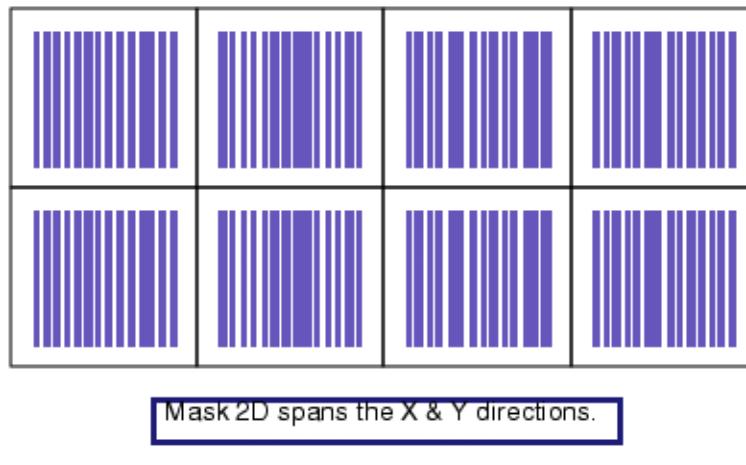
A 1D mask is used for repeated arrays of lines and spaces defined along the X direction. There is no limit to how long these test structures can stretch.

**Figure B-4. Mask1D**



A 2D mask extends in the X and Y directions. Grid frames are calculated from the `ArrayWidth` and `ArrayHeight` values. They are divided by the pixel size defined in the initialize command to define how many grid frames are used for simulation. In CMP Model Builder, a mask with test structures can be defined in the process recipe file or a layout can be used for simulation.

**Figure B-5. Mask2D Example**



## S<sub>n</sub>apshots

A snapshot is a set of grids that represents the current state of the simulation. Snapshots track the surface position and pressure of a CMP simulation at specified times or after a process step. These grids can be displayed in CMP Model Builder.

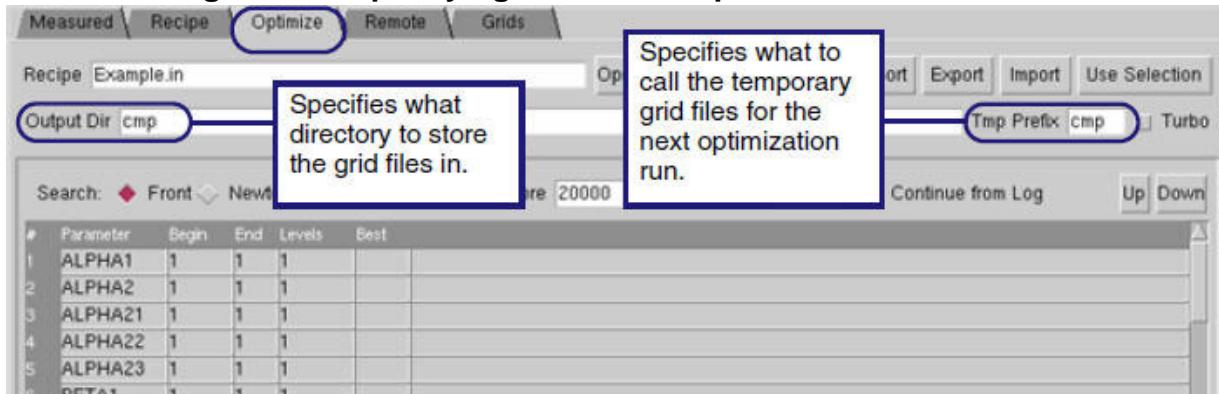
The tool compares snapshots from the MDT with created grids of the same snapshot number. Created grids that do not have a corresponding snapshot number in the MDT are ignored for calculating the simulation error. If no snapshots are specified for an optimization run, the program halts with the error, “No snapshot(s) specified in recipe. Cannot calculate RMS. Please correct recipe.” To fix the error, add snapshot arguments to your process recipe that match snapshot values in the MDT at appropriate steps for your process.

## Grid Files

Once a simulation (or optimization) run is started, layout data is read into CMP Model Builder and mask data is extracted and stored into grids.

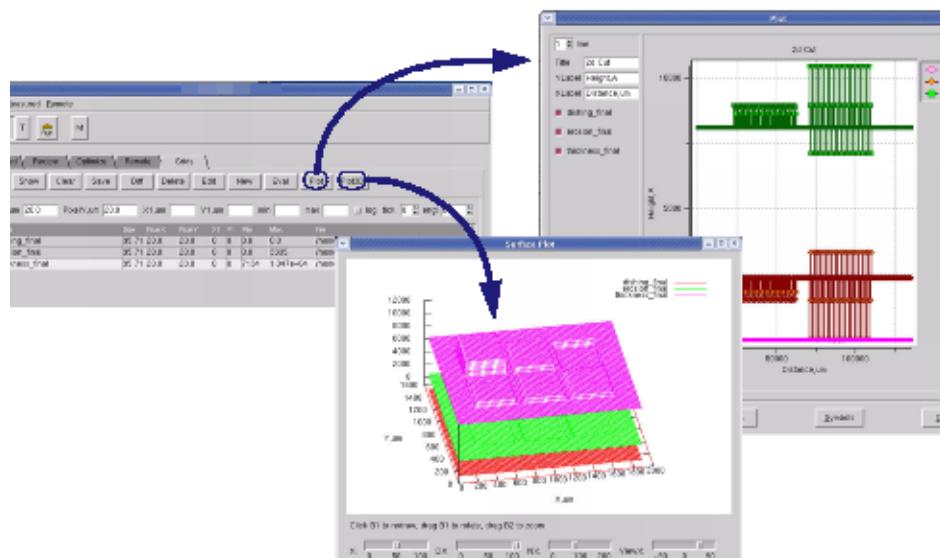
Grid files are stored in a temporary directory specified in **Outdir** under the **Optimize** tab. These files are given a prefix specified in Tmp Prefix field. If an optimization or simulation run was run from a previous session, CMP Model Builder reads the extracted mask data from grid files as input instead of the layout, saving you time in future simulations.

**Figure B-6. Specifying Grid File Outputs From Simulations**



Grid files are viewed as plots from the **Grids** tab. Vertical profiles of your grid simulations are generated with the Plot button. 3D surface plots are available from the Plot3D button.

**Figure B-7. Viewing Vertical Profiles and 3D Profiles**



<b>Grid File Types.....</b>	<b>203</b>
<b>Extracting Grids for Large Test Chips .....</b>	<b>203</b>

## Grid File Types

Grid files are generated from simulations in CMP Model Builder.

The file extension supplied as part of the file name defines the format of the file created. The following list shows the file types that CMP Model Builder can create.

- **.txt** — default ASCII format
- **.bin** — binary format
- **.text** — alternative ASCII file format
- **.rdb** — results database format is an ASCII file that can be viewed using Calibre RVE
- **.dump** — binary format containing information on the entire simulation
- **.sta** — ASCII format containing information on the entire simulation

---

### Note

 Only use **.dump** and **.sta** when saving the entire state using “save state” or “save dump” in the recipe.

---

- **.matX .matY .matZ** — “matrix” grid files in a tabular format that can be imported into graphing tools

---

### Note

 Matrix files cannot be created with the save command; they can only be exported from another grid using the **Grids** tab **Save** button.

---

All three matrix files must have the same base filename to be recognized as a set, and must be opened or plotted as a set.

---

---

### 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 Grids tab. The selection list of supported formats is also shown: GP (\*.bin, \*.txt, \*.rdb, \*.sta, \*.dump, \*.grid).

---

The file type extension defines the reader that is used to open the file. If a file cannot be read, all available readers are tried.

## Extracting Grids for Large Test Chips

When the test chip is large (either wider than a centimeter on one side, or when there are billions of shapes), it is faster to use Calibre CMPAnalyzer to extract the grid file and then load it into

CMP Model Builder than it is to use CMP Model Builder directly because Calibre CMPAnalyzer supports multithreading.

## Prerequisites

- Test chip
- Output recipe with the following conditions:
  - The etch command uses “mask=*layer\_number*” rather than mask2D.
  - The recipe includes a **save** command that saves at least the properties area, space, width, length, perimeter, and WL.

## Procedure

1. In Calibre WORKbench, invoke Calibre CMPAnalyzer:
  - a. Select **Verification > Run DFM** from the WORKbench menu.
  - b. In the Calibre® Interactive™ window, provide the inputs shown in [Table B-1](#):

**Table B-1. Calibre CMPAnalyzer Settings**

Pane	Field: Setting
Inputs	Run: CMP Analyzer CMP tab > Layer Number: Layout’s equivalent layer CMP tab > Recipe File: Output recipe file
Rules	DFM Run Directory: An empty directory
Outputs	Results Format: DFM Database Directory: new directory to store DFM results

- c. Click **Run CMP**.

The grid files are saved. The output names depend on the recipe you provided. For example, if the save command is as follows:

```
save area space width length perimeter WL file=postEtch
```

Then the grid files are named as follows:

<i>area_density_postEtch.txt</i>	<i>width_postEtch.txt</i>
<i>space_postEtch.txt</i>	<i>length_postEtch.txt</i>
<i>perimeter_density_postEtch.txt</i>	<i>WL_postEtch.txt</i>

To build a model, the CMP Model Builder engine needs at least these six grids.

2. Copy the grid files to the CMP modeling work area, renaming them to use CMPO prefixes as shown in [Table B-2](#):

**Table B-2. CMPAnalyzer to CMP Model Builder Conversion**

CMPAnalyzer Prefix	CMP Model Builder Prefix
area_density_*	a_*
width_*	w_*
space_*	s_*
length_*	l_*
perimeter_density_*	p_*
WL_*	WL_*

If the `etch maskFile` value is different from the `save` file value, you also need to change the \* portion of the file name.

## Related Topics

[Declaring Optimization Parameters for Calibration Runs \(Optimize Tab\)](#)

# Grid Frames (Pixels)

The simulation grids are regular grids of a user-specified size (typically 20 um x 20 um) that divides the simulation area into uniform regions.

## Grid Frame Evaluation

When simulating the surface evolution for the chip, CMP Model Builder evaluates one grid frame (the square area between grid lines as shown in [Figure B-8](#)) at a time. (Grid frames are also referred to as pixels.) The size is set in the initialize command using the PixelX and PixelY parameters. The default pixel setting is 20 um x 20 um. Smaller sizes require increased processing times. The pixel size should be large enough that a single frame contains a number of shapes.

**Figure B-8. Grid Overlaid On Design**



The simulator calculates the following values for each grid frame at every stage in the process recipe file:

- Vertical position of the exposed material in the non-trench region (Z1). The exposed material changes during the process: it can be copper, barrier material (TaN), or oxide.
- Vertical position of copper in the trench region (Z2).

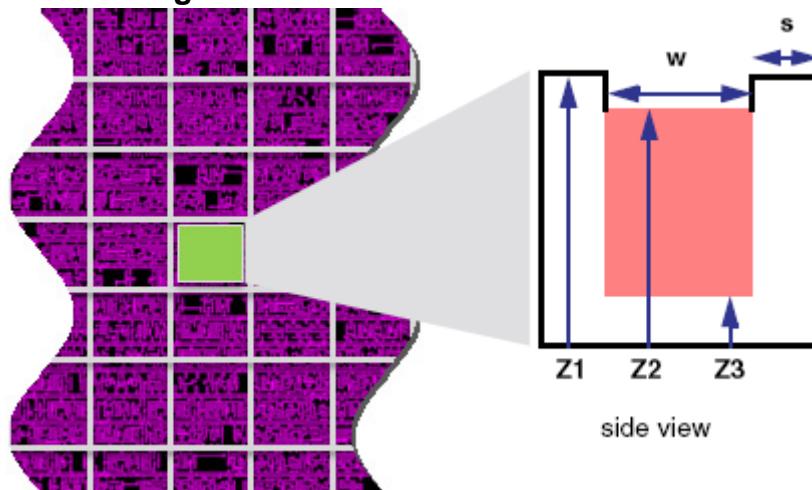
All thickness data grids for Z1, Z2, ThicknessT, and ThicknessNT are reported in whole angstroms.

The simulator calculates the following values for each grid frame for etch stages:

- Pattern density as the fraction of the area of the frame covered by trenches (a).
- Width of the trenches (w).
- Spacing between trenches (s).
- The length of the trenches (l).
- The perimeter density of the trenches (p).
- The average area per feature (WL).

This mask information on the grid is used for the simulation instead of the individual polygons of the trenches. This approximation is referred to as the effective trench. The Z1, Z2, and other values are shown in plots of the grid as though the surface across the grid frame were uniform.

**Figure B-9. Effective Trench Values**



## Multilayer Analysis Inside of Trenches

For multilayer stack analysis inside of trenches, you can select the **TrenchLayerStack** button in the **Recipe** tab [Initialize Dialog Box](#). When this mode is enabled, CMP Model Builder considers multiple layers of different materials inside of trench regions and uses different removal rates for material inside of trenches. This mode is important for the polishing of wide and narrow trenches, where the layer stack information may be different, and the polishing dynamics depend on the material inside of the trench. In the resulting analysis, each grid frame (pixel) has its own layer stack data for inside of trench regions.

## Material Index Reporting in Grids

The material index is a number that defines the order of the deposited material in the layer stack. Reporting the material index in grid data provides information for understanding the CMP Model Builder simulation flow. The material index has the following conventions:

- Material index starts from 0 for a material defined in the initialize command.
- Same materials get the same material index within a layer (M1, M2, ...).
- Material index of the first deposited material after CMP is 0. This means that for each layer, the material index of the first deposited material starts from 0.

The **MaterialNT** and **MaterialT** property parameters of the [save](#) command (**materialNT** and **materialT**) are used to detect the non-trench and trench material indexes across the simulated area. You can setup these property parameters for a process recipe file in the **Recipe** tab by selecting **Add > save**, and then choosing either **MaterialNT** or **MaterialT** or both. This saves the indexes of the non-trench and trench materials to the grid data. See “[Save Dialog Box \(Full\)](#)” on page 162. Colormaps for these property parameters can be displayed in a Calibre WORKbench layout to indicate the exposed materials across the die.

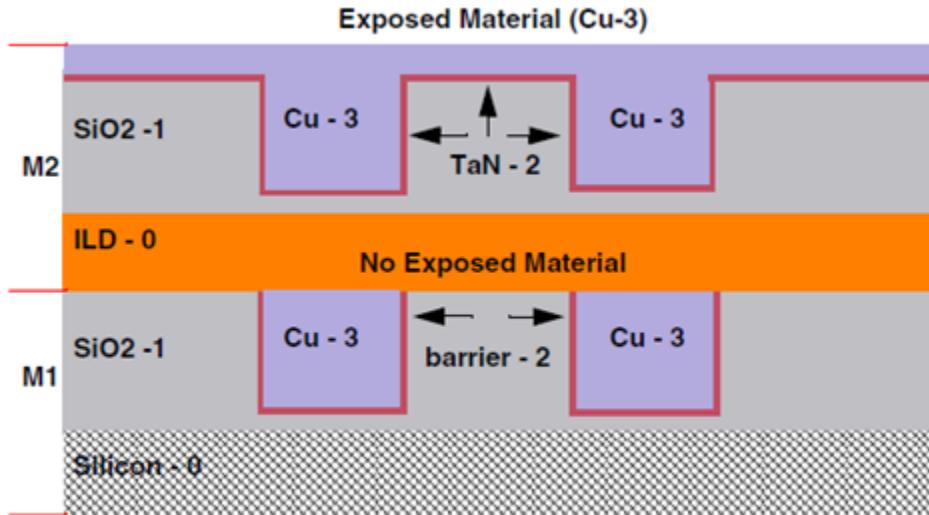
To use MaterialNT and MaterialT properties for analysis in Calibre CMPAnalyzer, you can select the **Enable MaterialNT(T) in dfmdb** button in the CMP Model Builder Initialize dialog box. This updates the process recipe file [initialize](#) command to enable the reporting and saving of these property parameters to a DFM database. The DFM database can then be used in Calibre CMPAnalyzer results analysis in Calibre RVE for DFM for colormaps, histograms, and reported properties to identify regions of exposed non-trench and trench materials and potential hotspots in a design. See “[Initialize Dialog Box](#)” on page 144.

The material index in the MaterialNT and MaterialT grids provides useful information for model building by reporting if certain materials are exposed during a process step. For example, the MaterialNT and MaterialT grids can be used for building a model for the platen 2 step to check if the barrier layer is exposed or if other materials are exposed. Similarly, the MaterialNT and MaterialT grids can be used for building a model for the platen 3 step.

In [Figure B-10](#) on page 208, the deposited material in the layer stack and corresponding material index can be used to report exposed material. The exposed material in the non-trench region can be copper, barrier material (TaN), or oxide. The material index is the number shown to the right of the material name.

- If a material with index 2 or 3 is exposed, then barrier/copper is exposed.
- If a material with index 1 is exposed, then only oxide is exposed.

**Figure B-10. Exposed Material in Multilayer Stack**



The recipe file statements with material indexes are shown in the following table.

**Table B-3. Recipe File Statements and Material Indexes.**

Recipe File	Material Name	Material Index
initialize material=silicon position=0	silicon	0

**Table B-3. Recipe File Statements and Material Indexes. (cont.)**

<b>Recipe File</b>	<b>Material Name</b>	<b>Material Index</b>
deposit material=SiO2 type=isotropic ...	SiO2	1
etch thickness=100 ...	...	...
deposit material=barrier type=isotropic ...	barrier	2
deposit material=copper type=isotropic ...	copper	3
deposit material=copper type=ecd ...	copper	3
cmp ...	...	...
deposit material=ILD type=isotropic ...	ILD	0
deposit material=SiO2 type=isotropic ...	SiO2	1
etch thickness=100 ...	...	...
deposit material=TaN type=isotropic ...	TaN	2
deposit material=copper type=isotropic ...	copper	3
deposit material=copper type=ecd ...	copper	3

### Grid File Data Save

To inspect data frame-by-frame use the [save](#) command. It saves simulated data in the form of grid files. You load these grid files in CMP Model Builder then click **Show** to display the grid over the layers. You can also create a 3D surface plot using **Plot3D**.

## Optimized CMP Model File

After calibration runs have been completed and meet your specifications, the optimized model along with the best values for your parameters are located in the directory specified in Outdir. The optimized model has the *.in* file extension.

For example, if the Outdir and Tmp Prefix fields have “cmp” and “final” respectively, the optimized model is the file name *cmp\_finalo\_so.in*. Look for this file in the directory specified in Outdir along with the grid files created from simulations. This is the model file passed into CMPAnalyzer.

### Related Topics

[Optimize Tab](#)

[cmpoptimize](#)



# Appendix C

## Details on Theoretical Models

---

A CMP Model is composed of various theoretical models that are used by the CMP Optimization tool.

The appendix is organized by the type of physical model each theoretical model is associated with.

<b>CMP Model .....</b>	<b>211</b>
<b>Deposit Model.....</b>	<b>215</b>

## CMP Model

The CMP model is an overall model used for optimizing parameters related to the CMP step. It has several interacting submodels.

The CMP model is run after deposition. Its initial input is a single-material surface of varying height and the geometry of the trench region. The geometry may include trench width, trench spacing, length, area, and density depending on which values are provided.

The CMP simulation takes into account long-range effects such as pad deformation and short-range effects such as varying depths of trenches. It computes both erosion and dishing variations of the surface based on the local pressure.

## Perimeter

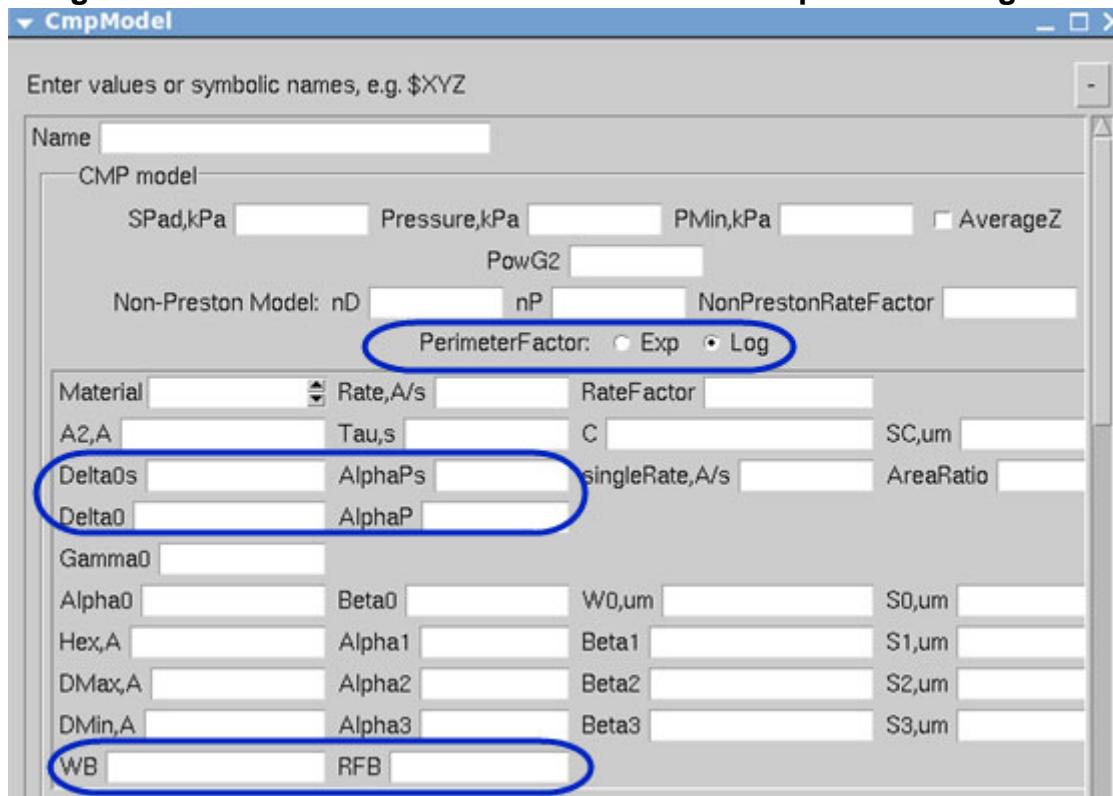
The perimeter model is useful when your measured data suggests that areas with many narrow trenches are more polished than initial simulations predicted. This most commonly occurs when designs have large variations in trench width while maintaining a consistent density.

The perimeter model is part of the overall model for material removal for CMP of platen. The perimeter model has two forms, exponential and logarithmic. Generally the logarithmic form has a smoother response.

The enhanced perimeter model (EPM) is used to modify the removal rate of the material depending on the contact area of the different materials along the pattern perimeter. It is a generalization of the removal rate correction by the perimeter model.

## Parameters

**Figure C-1. Parameters for Perimeter Model in CmpModel Dialog Box**



More information on the effect of the fields is provided in the following table.

**Table C-1. Perimeter Model Fields**

Field	Description	Values
PerimeterFactor	Indicates whether to use the exponential or logarithmic calculation.	<ul style="list-style-type: none"> <li>• Exp</li> <li>• Log (default)</li> </ul>
Delta0s	The delta ( $\delta$ ) value to be used for topmost material. When the second material is exposed, the model uses Delta0 instead.	<ul style="list-style-type: none"> <li>• Symbolic name (value to be optimized)</li> <li>• Exp: <math>0 \leq \delta \leq 1</math></li> <li>• Log: <math>0 \leq \delta \leq 5</math></li> </ul>
Delta0	The delta value to be used for exposed material. If Delta0s is not specified, Delta0 is also used for the topmost material.	<ul style="list-style-type: none"> <li>• Symbolic name (value to be optimized)</li> <li>• Exp: <math>0 \leq \delta \leq 1</math></li> <li>• Log: <math>0 \leq \delta \leq 5</math></li> </ul>
AlphaPs	The alpha ( $\alpha$ ) value to be used for topmost material. When the second material is exposed, the model uses AlphaP instead.	<ul style="list-style-type: none"> <li>• Symbolic name (value to be optimized)</li> <li>• <math>0 \leq \alpha \leq 5</math></li> </ul>

**Table C-1. Perimeter Model Fields (cont.)**

<b>Field</b>	<b>Description</b>	<b>Values</b>
AlphaP	The alpha value to be used for exposed material.	<ul style="list-style-type: none"> <li>Symbolic name (value to be optimized)</li> <li><math>0 \leq \alpha \leq 5</math></li> </ul>
WB	An optional width bias (wb) parameter value in microns used for the enhanced perimeter model (EPM). It affects the removal rate of the material. Specify together with RFB.	<ul style="list-style-type: none"> <li>Argument name (and numeric value to be applied)</li> <li><math>wb &gt; 0</math> enables parameter</li> </ul>
RFB	An optional rate-factor bias (rbf) parameter value used for the enhanced perimeter model (EPM). Specifies a rate correction for the material due to transitions between trench and non-trench patterns. It affects the removal rate of the material. Specify together with WB.	<ul style="list-style-type: none"> <li>Argument name (and numeric value to be applied)</li> <li><math>rbf &gt; 1</math> means to increase the polishing rate</li> <li><math>rbf &lt; 1</math> means to decrease the polishing rate</li> <li><math>rbf &gt; 0</math> enables parameter</li> </ul>

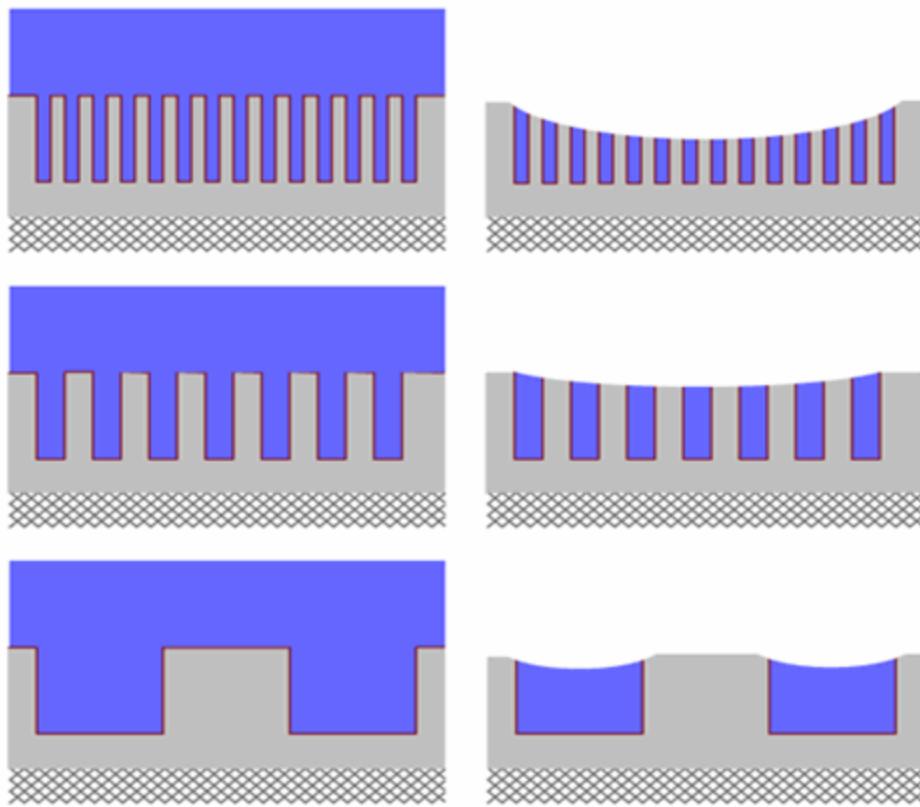
**Usage Notes**

The perimeter model increases the removal rate in areas with narrow trenches.

Those same sites are also affected by the Chekina theoretical model which calculates global pressure. The Chekina model is trying to reduce pressure there. The interaction between these models can significantly increase the pad stiffness parameter, SPAD.

The perimeter model reflects how chip topography affects dishing and erosion. The three cross-sections shown in [Figure C-2](#) all have 50% density. The top layout has many shallow trenches; the middle layout has several wide trenches; and the bottom one is a single wide metal area. The perimeter is calculated as the sum of the perimeters of the metal areas in the frame being modeled. (Note that this is not the same as the perimeter of the full shape.) This means that the top layout with many trenches has a larger perimeter than the fewer wide trenches, which is wider still than the single metal area. Another term for this value is “surface roughness.”

**Figure C-2. Same Density, Different Perimeters**



## Preston

The Preston model is a theoretical model that is used by default for simulating bulk copper polishing. It reduces initial height variations to achieve relative planarity between trench and non-trench regions. There are no controls to enable it, as it is always on.

The CMP process initially deals with a structure that is completely covered by copper. During polishing, the original height variations decrease towards zero.

## Equations

The Preston equation specifies the relation between removal rate and the pad's pressure and motion.

$$R = k_{cu} PV$$

where

- $R$  is the removal rate
- $k_{cu}$  is a constant
- $P$  is the pressure, and varies by height
- $V$  is the relative velocity between pad and wafer

### GUI Fields

In the CmpModel dialog box, the only field that affects the Preston model is Hex. The Hex field indicates the variation in height ( $|z_1 - z_2|$ ) at which the pad no longer applies direct pressure to the lowest copper areas. This field should usually be set to a symbolic name so that its value will be optimized.

## Deposit Model

The deposit models consist of several different ways of describing how fast material appears on the surface and how it fills in the trenches.

The basic models are isotropic and anisotropic. These models assume a traditional form of chemical vapor deposition. The primary difference between isotropic and anisotropic is whether material accumulates on the side walls as quickly as it does the field region. The transition area at trench walls is sharp with square corners, unless you trigger the “thick deposit” model by setting epsilon. The thick deposit model rounds the corners.

The ecd model handles electroplating. Unlike the others, the details of the model are defined not in a deposit command but in a define\_model.

HDP\_CVD is the high-density plasma model. It forms mounds (pyramids in our models) over the high points between trenches.

The SOD model handles spin-on dielectric or spin-on glass. The material is deposited all at once; rate and dispersion are not modeled. The material is initially at a certain thickness, but then fills in the trenches. Over trench regions, the thickness decreases but must still exceed trench height.



# Glossary

---

## **dishing**

A value representing the local difference between the tops of trenches and non-trenches (field region).

## **effective trench**

A simplified representation of the layout used by the simulator so that each grid frame has a single Z1, Z2, and Z3 value. The effective trench is calibrated by comparing with physical measurements.

## **erosion**

A value representing the local field region's z1 minus the maximum height inside the grid cell, regardless of the exposed material in the grid cells.

## **pixel**

An area of simulation. Also called "frame" or "grid frame". See "[Grid Frames \(Pixels\)](#)" on page 205.

## **planarization**

A manufacturing process that smooths the wafer surface. The goal is to achieve as flat a surface as possible.

## **superfill**

The process that occurs during material deposition after trenches have filled. This can occur either when the trench bottom has filled until equal with the non-trench area, or when the width of a trench approaches zero.



# Index

---

## — Symbols —

[]<sup>19</sup>  
{}<sup>19</sup>  
|<sup>19</sup>

## — Numerics —

3D plots, [195](#)

## — A —

Absolute thickness measurements, [36](#)  
Analyzing results, [44](#)  
Anisotropic deposition, [215](#)  
Anisotropic etch, [102](#)  
Anisotropic etching, [103](#)  
Array blocks, [198](#)

## — B —

Backwards compatibility, [122](#), [123](#), [125](#), [127](#)  
Batch command, [52](#)  
BEOL, [13](#)  
Best practices, [141](#)  
Bold words, [19](#)

## — C —

CALIBRE\_HOME environment variable, [18](#)  
cmp command, [67](#)  
CMP model  
    Calibre usage, [15](#), [197](#)  
    components, [197](#)  
    creating, [15](#)  
    definition, [13](#)  
    requirements, [17](#)  
CMP Model Builder  
    batch command, [52](#)  
    invocation, [18](#)  
CMP Optimize  
    invocation, [18](#)  
Command syntax, [19](#), [52](#)  
Copper layer, [67](#)  
Courier font, [19](#)

Creating a model, [84](#)

Creating a new layer on top of stack, [92](#)

Cutlines, [141](#)

CVD model, [215](#)

## — D —

dat files, [59](#)  
Data collection, [15](#)  
Defaults, [65](#)  
Define simulation limits, [110](#)  
define\_mask 1D command, [72](#)  
define\_mask 2D command, [74](#)  
define\_model command, [84](#)  
Defining a 1D mask, [72](#)  
Defining a 2D mask array, [74](#)  
deposit command, [92](#)  
Deposit models, [215](#)  
Dishing, [37](#)  
DOF, [14](#)  
Double pipes, [19](#)  
dump files, [203](#)

## — E —

ECD model, [215](#)  
    parameters, [84](#)  
EDP, [15](#)  
Electroplating, [215](#)  
Environment variables  
    CALIBRE\_HOME, [18](#)  
    CMP\_ENABLE\_OAF\_LOG\_DT, [119](#)  
    CMP\_EXTR\_LARGE\_METAL\_REGION\_N, [120](#)  
    CMP\_EXTRACT\_POS\_EXTRA\_SPEED\_UP, [121](#)  
    CMP\_EXTRACTION\_VERSION, [122](#)  
    CMP\_OLD\_EROSION, [123](#)  
    CMP\_PD\_MIN\_RANGE\_FACTOR, [124](#)  
    CMPOPTIMIZE\_COMPATIBILITY, [125](#)  
    CMPOPTIMIZE\_THICKNESSNT\_MIN, [126](#)

---

MGCMP\_EROSION, 127  
Erosion, 37, 124, 127  
Erosion site, 37  
etch command, 102  
Etch examples, 108  
Etch models, 103  
Etch-back, 102  
Expressions, 58  
    optimizing, 193  
Extraction oscillating, 104  
Extraction version, 122, 125

## — F —

FEOL, 14  
Fill shapes, 121  
Font conventions, 19  
Functions  
    fitting values, 193  
Functions in parameters, 58

## — G —

Grid files  
    etch, 105  
    formats, 202

## — H —

Heavy font, 19  
HKMG, 14

## — I —

Initialize command, 110  
Interpolation, 109  
Isotropic deposition, 215  
Isotropic etch, 102  
Italic font, 19

## — L —

Licensing, 17  
Linescan delineation, 141

## — M —

Mask, 72  
Mask array, 74  
Mathematical functions, 58  
Matrix files, 195  
matX files, 203  
Measured data files

definition, 34, 199  
format, 59  
preparing, 35  
MGC\_HOME, *see* CALIBRE\_HOME

## — O —

od at files, 59  
Open area fraction log, 119  
Optimizing calibration runs, 42  
Oxide layer, 67

## — P —

Pads, 120  
Parentheses, 19  
Pattern density, 124  
Pipes, 19  
Planarization, 13  
Plot surface, 195  
Polishing, 67  
POP, 14  
Process Recipe File (PRF)  
    preparing, 22  
Process types, 102

## — Q —

Quotation marks, 19

## — R —

rdb files, 203  
Report time, 119  
Reverse etchback  
    *see* Etch-back  
RMG layers, 198

## — S —

save command, 114  
Simulation grid, 205  
Slanted words, 19  
Snapshots, 60, 201  
SOD model, 215  
Space definition, 106  
Square parentheses, 19  
sta files, 203  
STI, 13

## — T —

Test structures, 198

---

Trench etching, [102](#)

— **U** —

Underlined words, [19](#)

Units, [65](#)

Usage syntax, [19](#)

— **W** —

Width definition, [106](#)



## **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.

