SIEMENS EDA

Calibre® WORKbench™: RET Flow Tool User's Manual

Software Version 2021.2

SIEMENS

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

Table of Contents

List of Figures

List of Tables

Chapter 1
Introduction to the Calibre RET Flow Tool

The Calibre® RET Flow Tool is a Calibre® WORKbench™ utility that is the central access point for all the major Calibre RET lithography tools.

The Calibre RET Flow Tool is a single launching point for frequently-used actions in the following tools:

- Calibre® WORKbench™ modeling
- Calibre® mbSRAF sub-resolution assist feature (SRAF) placement
- Calibre® nmBias™ table-driven OPC
- Calibre® nmMPC™ mask correction
- Calibre® MPCverify™ mask verification
- Calibre® nmOPC™ design correction for dense OPC
- Calibre® nmSRAF™ sub-resolution assist feature placement
- Calibre® OPCpro™ design correction for sparse OPC
- Calibre® OPCverify™ design verification
- Calibre® PRINTimage™ correction verification
- Calibre® pxOPC™ design correction
- Calibre® Directed Self-Assembly (DSA)

Calibre RET Flow Tool Fundamentals

There are a number of fundamental concepts that are used by the Calibre RET Flow Tool.

- **Flow** — A flow is a collection of tool sessions used to organize the collective runs for each tool for a specific end goal.

- **Litho Model** — A litho model is a directory of models (optical, resist, and so on) that Calibre tools can access through the *Lithomodel* file. It is a container for a set of process and mask conditions.
- **Process Window Condition** — The Calibre RET Flow Tool uses process window conditions (PWC) to represent image contours generated from polygons at different dose and focus conditions. Other variables, such as bias and shift, can be applied to the polygons to show the effects of different sizing and MEEF (Mask Error Enhancement Factor) on the design.
- **Session** — A session is the particular settings for an RET tool during a given run. These settings can be saved in a litho session file (*.lsf*). Changes to fields in the RET Flow Tool are not automatically saved. A session file differs from a setup file in that it cannot be used directly by batch mode commands.
- **Setup File** — A Calibre WORKbench configuration file. Calibre WORKbench uses these text files to set up important values and flags as input for multiple tools. Each tool may have its own version of a setup file. (*.setup* and *.in* extensions are typical.) It is important to note that each Calibre tool may have a setup file that is not cross-compatible with other Calibre tools.

Calibre RET Flow Tool Prerequisites

The Calibre RET Flow Tool is part of Calibre WORKbench and Calibre LITHOview. You must be able to run either Calibre WORKbench or Calibre LITHOview (referred to generically as “a viewer”). The RET Flow Tool does not require a separate license.

In addition, for easy use of the Calibre RET Flow Tool, the following are recommended:

- Calibrated optical and resist models for your process.
- A design file to run sessions on.
- Familiarity with Calibre viewers. This manual does not describe the viewer interface outside of the RET Flow Tool.
- For tuning tool flows and OPC corrections, familiarity with SVRF and the languages used by the particular tools.

You can use the RET Flow Tool without these items, but a mismatch between layouts, models, and setup files will produce poor results.

Related Topics

Calibre WORKbench Users and Reference Manual

Using the Calibre RET Flow Tool

The Calibre RET Flow Tool is organized along a flow of related tool sessions. A tool session is your particular settings for a Calibre lithography tool such as Calibre nmOPC or Calibre OPCverify.

Prerequisites

- A design layout
- A properly configured viewer

Procedure

1. Invoke Calibre WORKbench and load a layout.
2. In Calibre WORKbench, either click the **Litho** button or select **Litho > RET Flow Tool**. The Calibre RET Flow Tool appears with the **Flow** tab displayed.

Figure 1-1. Litho Button in Calibre WORKbench

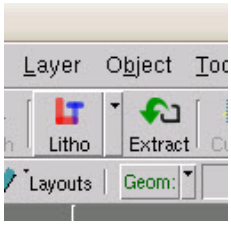
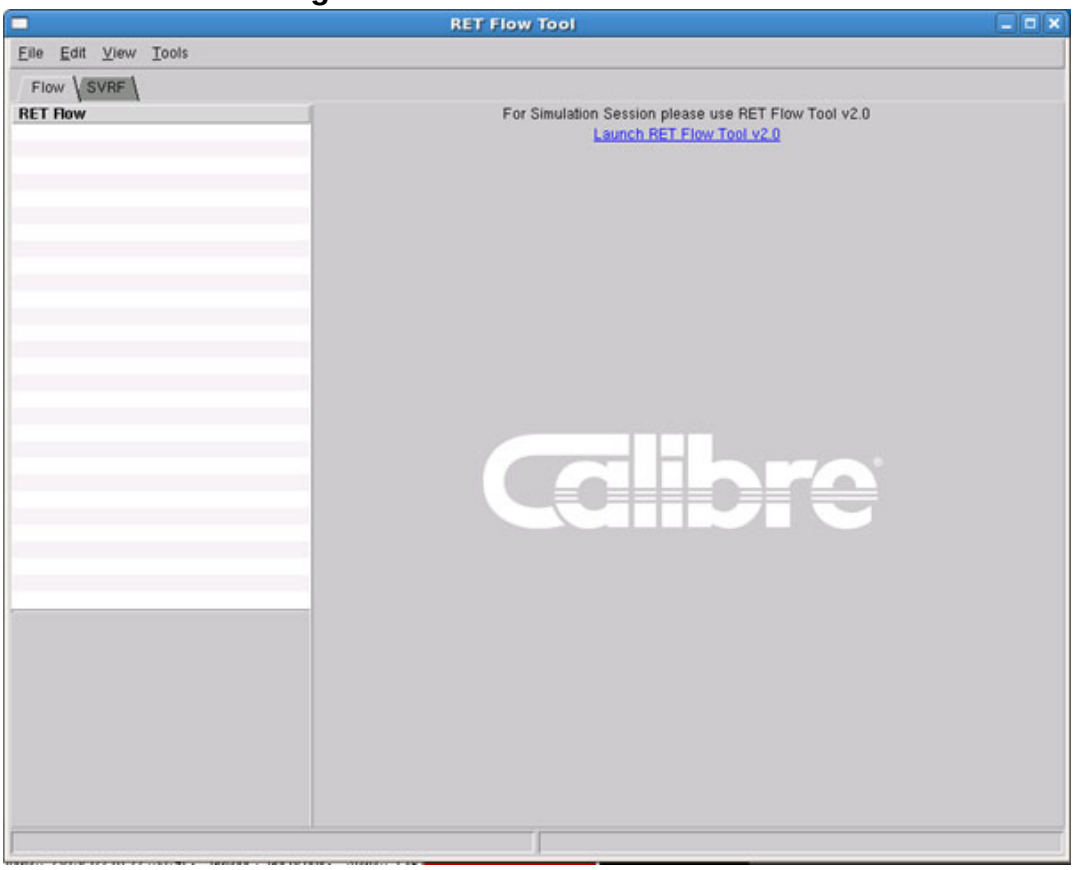


Figure 1-2. Calibre RET Flow Tool



Note

To keep the Calibre WORKbench window from closing in case of an execution error, select **Tools** > **Multi-Process**. Calibre WORKbench runs the RFT operation as a separate process, using an additional license.

3. Load or create a session flow. Sessions are the individual RET tools that you intend to use as well as their setup files. You can load or create sessions using one of the following methods:

   - Extract LITHO and RET command blocks from a pre-existing SVRF file or transcript. This reads file and populates the session with the RET tools that are required to run the command blocks. See “Extracting a Setup File from SVRF” on page 93 and “Extracting a Setup File from a Transcript” on page 92 for details.
   - Create sessions manually in the RET Flow Tool. See “Creating a Session” on page 38 for details.
   - Load pre-existing sessions and setup files for your RET Tools. See “Loading an Existing Setup File” on page 91 for details.

   After using any of these methods, a list of session icons appears in both the RET Flow Tool and the Calibre WORKbench window.

Figure 1-3. RET Tool Sessions in Calibre WORKbench

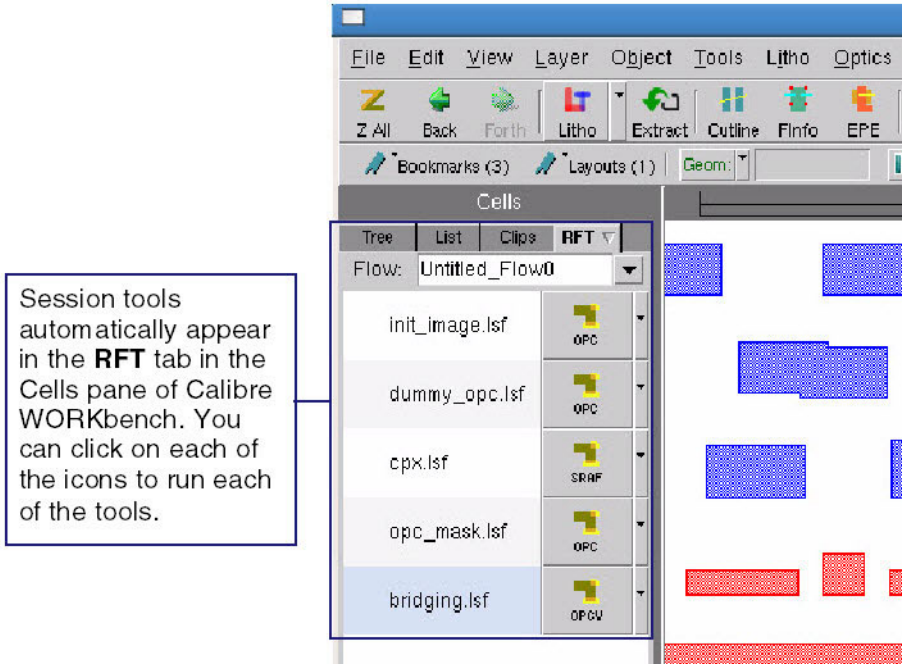
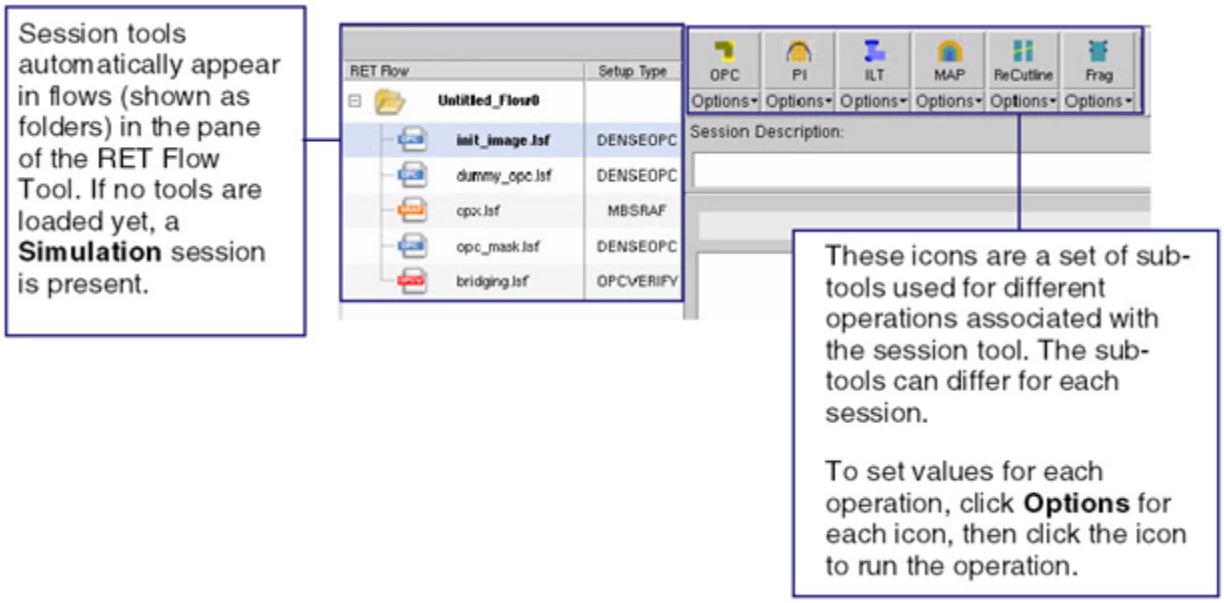


Figure 1-4. RET Tool Sessions in the RET Flow Tool



These icons represent a “flow” where you can run or re-run session files. This allows you to have multiple sequenced runs of RET tools, rather than having to manage individual tool sessions.

4. For several session tools, there are a set of sub-tools that you can use to produce output. You can change the options for these sub-tools, or directly edit the setup file.

5. Run your sessions to produce the desired RET output.

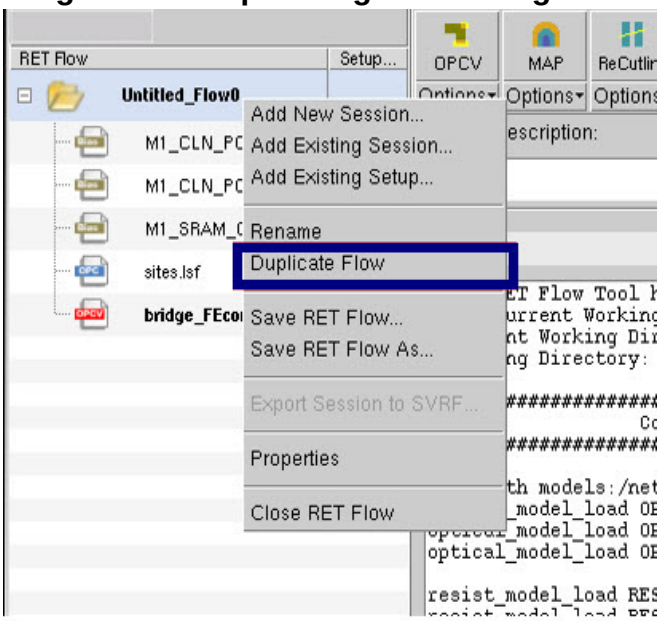
Tip

You can click **Clear Layers** to delete all layers generated by the RET Flow Tool.

6. Save the session flow (this includes all the settings for each of the session tools) for use at a later time by selecting **File > Save RET Flow** or **Save RET Flow As**.

   You can make a duplicate copy of a pre-existing session flow by right-clicking on an RET Flow folder and selecting **Duplicate** from the menu that appears.

Figure 1-5. Duplicating an Existing Flow



Results

When you save a session flow, the RET Flow Tool creates a directory by the name you specify. Inside the directory are directories for each flow. The session setups are saved in the flow directory with a *.lsf* extension.

Once the flow is saved, you can optionally invoke Calibre WORKbench with the RET Flow Tool loaded with the previously-saved flow using the following command:

```
$MGC_HOME/bin/calibrewb layout_file -rft flow_dir
```

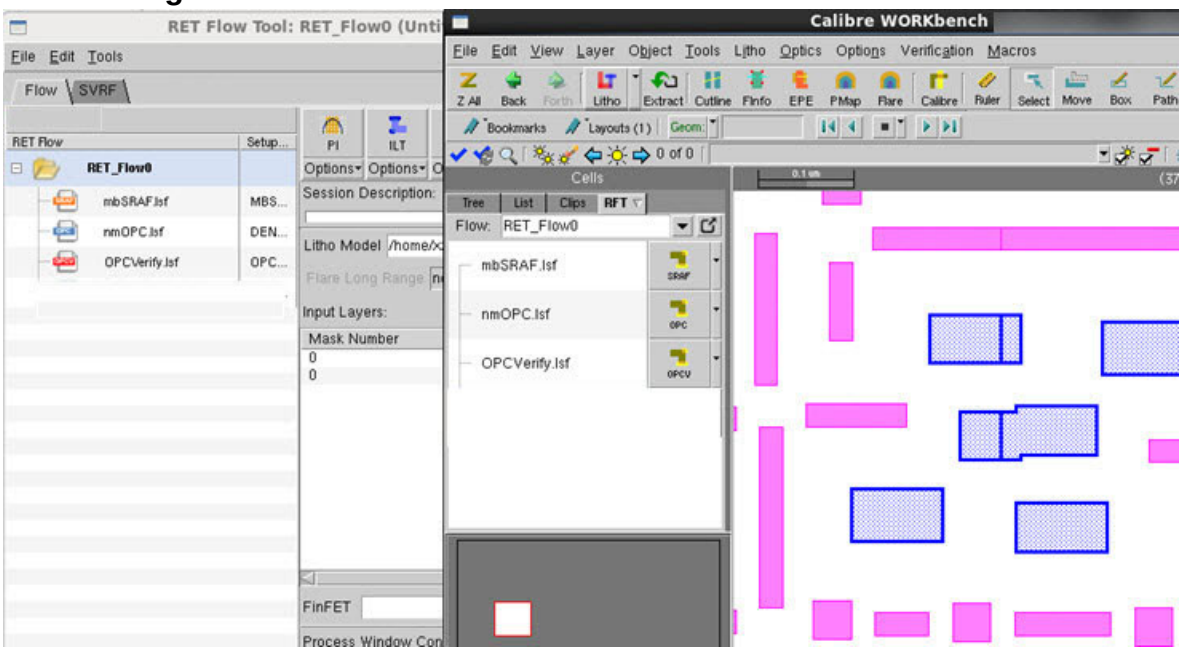
where *layout_file* is the layout file and *flow_dir* is the previously-saved RET Flow Tool flow directory.

For example:

```
$MGC_HOME/bin/calibrewb demo.oas -rft RET_Flow0
```

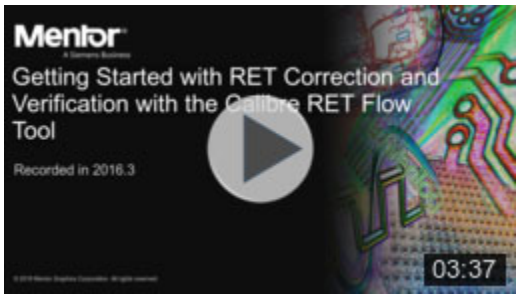
This command line invokes Calibre WORKbench and the RET Flow Tool as illustrated in Figure 1-6.

Figure 1-6. Calibre WORKbench with RET Flow Tool Loaded



Examples

You can watch this process of extracting setup files from an SVRF file and then configuring a run in the following video (extraction procedure begins at 00:56 in the video):



Related Topics

Calibre RET Flow Tool Fundamentals

Chapter 2
Calibre RET Flow Tool Workflows

The Calibre RET Flow Tool can be used in several key RET workflows.

Correcting Layouts and Verifying OPC

One of the common uses of the RET Flow Tool is to perform correction and verification on portions of a layout. This flow illustrates the use of the RET Flow Tool with the Calibre nmOPC and Calibre OPCverify products to perform correction and verification on an example layout.

The step-by-step example demonstrates the following basic correction and verification tasks in the RET Flow Tool:

- Importing setup files from an SVRF rule file.
- Modifying output options for a Calibre nmOPC session.
- Running Calibre OPCverify to check for bridging and pinching.
- Adding another setup file to an existing flow.
- Generating a print image.

Try It!	**Calibre Tools Quick Start and Example Kit**
	This Example Kit contains the design data used in this example flow, along with rules files and instructions for running Calibre. Go to this page on Support Center to download the eKit (Documentation tab, Document Types=Getting Started Guide). The link goes to the latest release.

Prerequisites

- An optical model and a resist model.
- A layout.

  The example in the procedure uses an OASIS®[1] layout named *fullchip_oas*. This design has a number of layers, but in this example only metal1 and poly layers are used.

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.

- An SVRF file.

  The example in the procedure uses *litho_demo.svrf*, included in full in the Examples section after the steps. The *litho_demo.svrf* file contains Calibre nmOPC and Calibre OPCverify setup files. This is primarily for the metal1 layer. Additionally, the example also uses the Calibre nmOPC setup file *poly.in* to correct the poly layer. The *poly.in* file is imported into the same flow.

  The example setup files used are intentionally rudimentary for demonstration purposes.

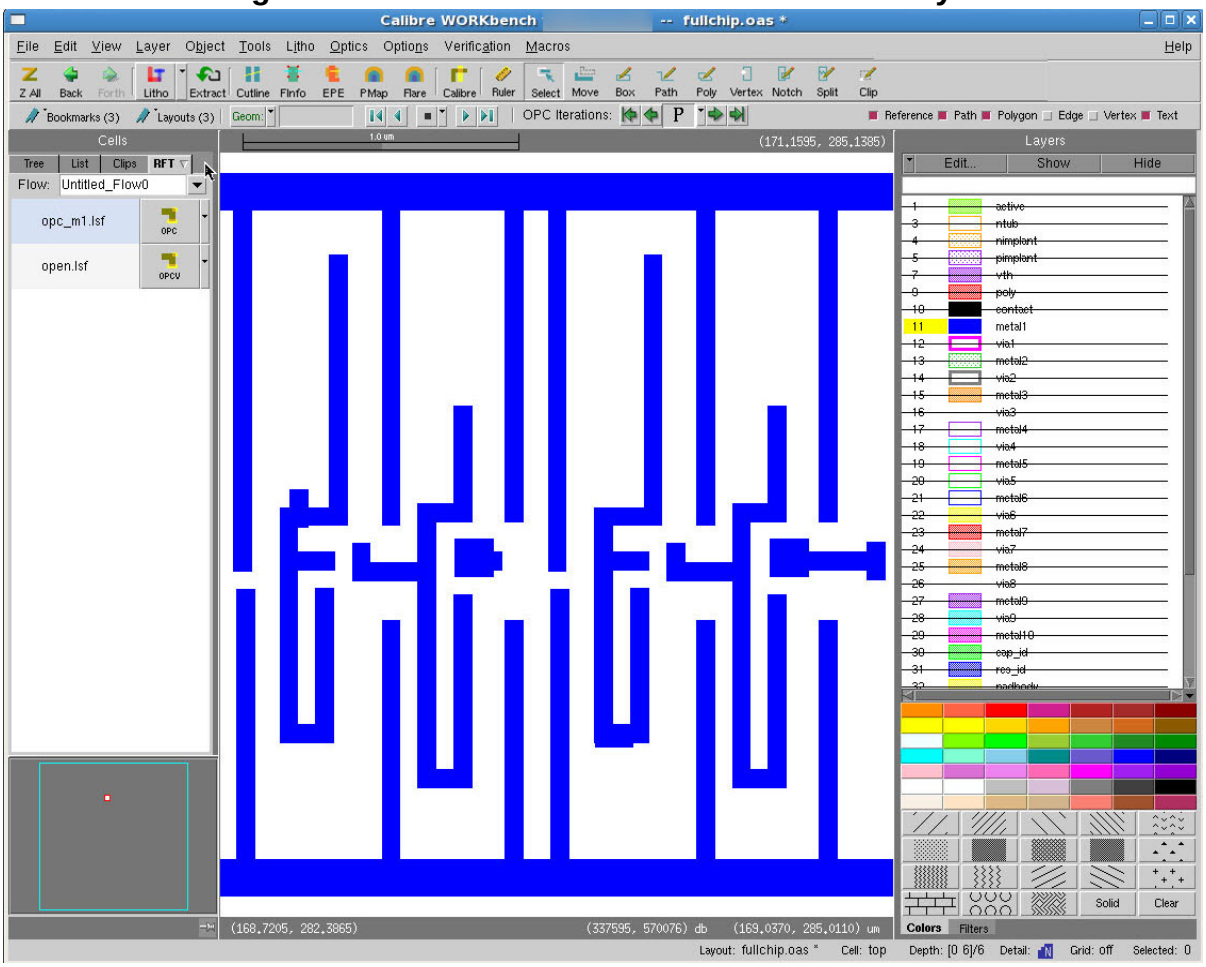
Procedure

1. From the command line, invoke Calibre WORKbench and load the *fullchip.oas* layout file.

   ```
   calibrewb fullchip.oas
   ```

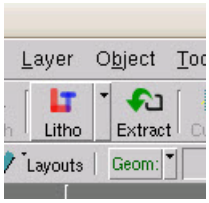
2. Zoom in on a portion of the metal1 layer for correction. Hide the other layers.

Figure 2-1. Calibre WORKbench with metal1 Layer



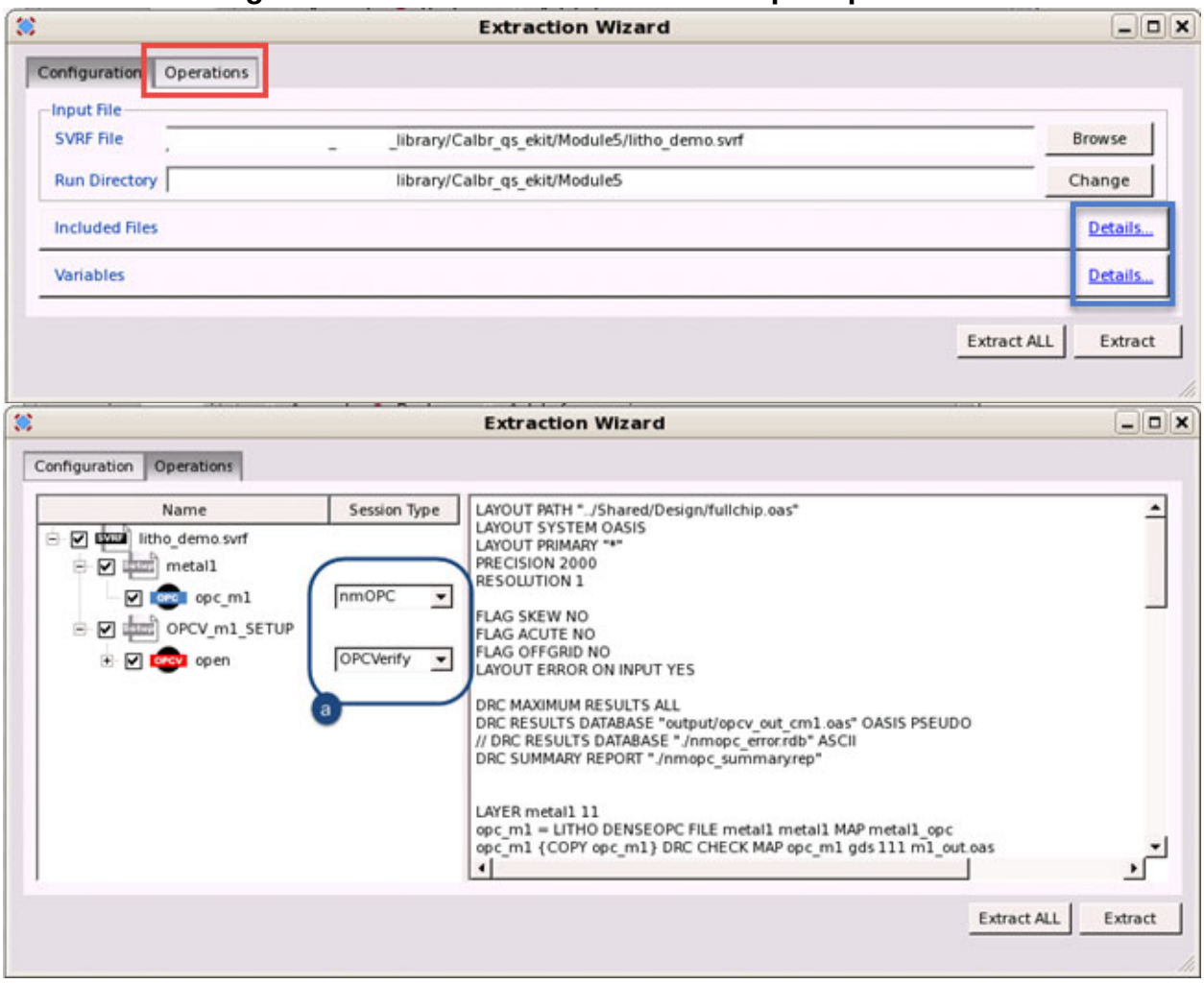
3. In Calibre WORKbench, click **Litho**. The RET Flow Tool appears.

Figure 2-2. Litho Button



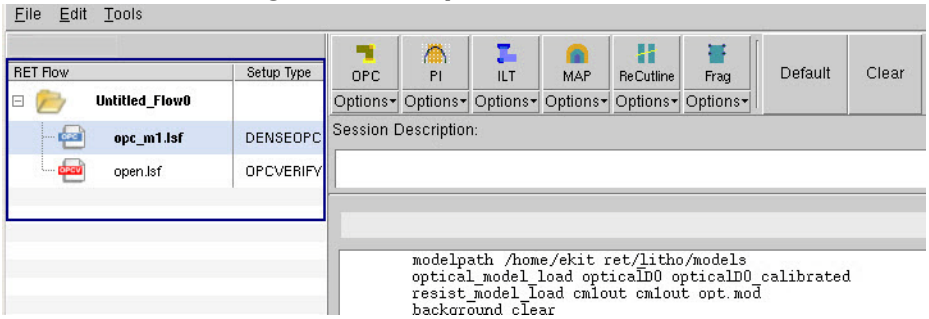
4. In the RET Flow Tool, select **File > Extract from SVRF**.
5. In the Extraction Wizard, click **Browse** and select *litho_demo.svrf*.
   - Select the **Operations** tab to see the different RET Flow Tool sessions (for example, Calibre nmOPC and Calibre OPCverify in (**a**) in the second half of Figure 2-3) that the RET Flow Tool can extract from the SVRF rule file.
   - (Optional) Expand the Included Files and Variables sections by clicking the associated Details link. Use these sections to verify your working environment is configured properly (all required files are loaded, and all variables are set correctly).

Figure 2-3. Extraction Wizard for Multiple Operations



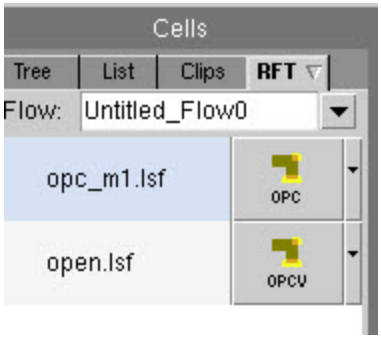
6. Click **Extract** or **Extract All** to extract sessions from the SVRF rule file. The RET Flow Tool changes to show each of the imported sessions displayed under a flow named “Untitled_Flow0”. The flow icon is a folder.

Figure 2-4. Imported Sessions



The sessions also appear as icons in the **RFT** tab of the Cells pane in Calibre WORKbench.

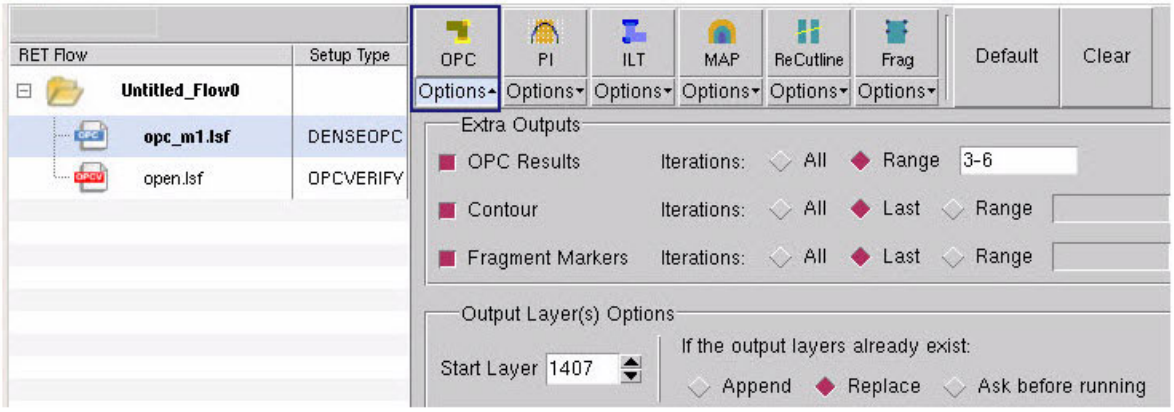
Figure 2-5. Cells Pane RFT Tab



7. In the RET Flow Tool, click the *opc_m1.lsf* entry to open the Calibre nmOPC session.

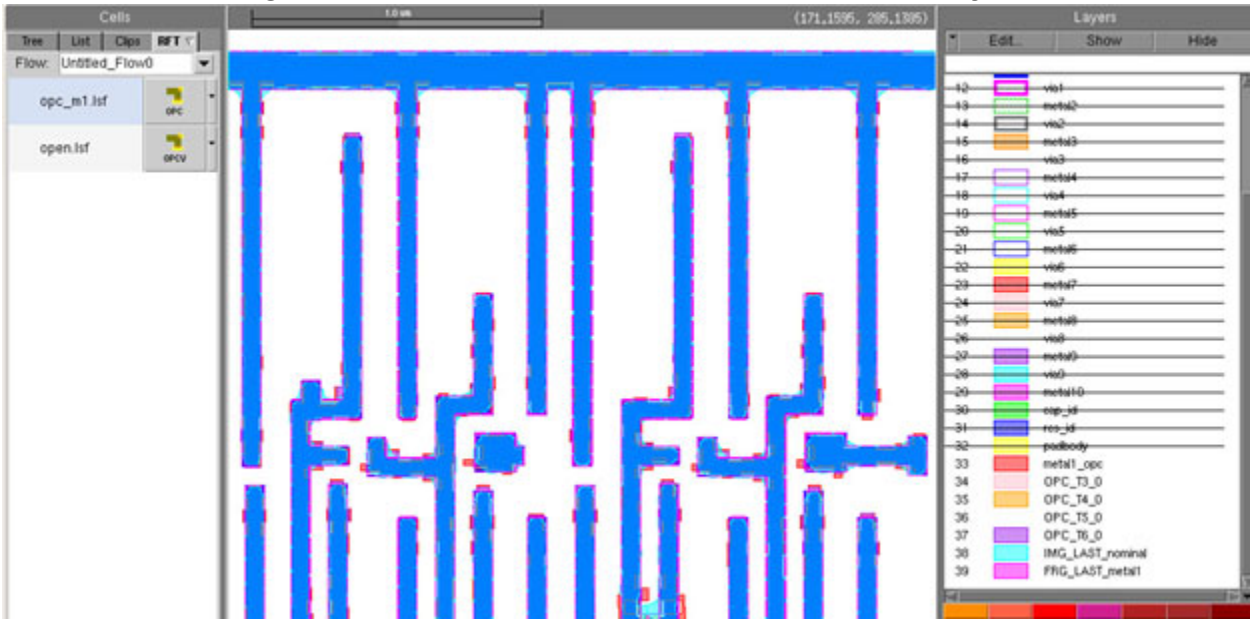
8. Under the **OPC** button, click **Options**, then set the options in the Extra Outputs section. In this example, layers are to be generated for OPC Results (iterations 3-6) as well as contour and fragment markers for the last OPC iteration.

Figure 2-6. OPC Outputs Example



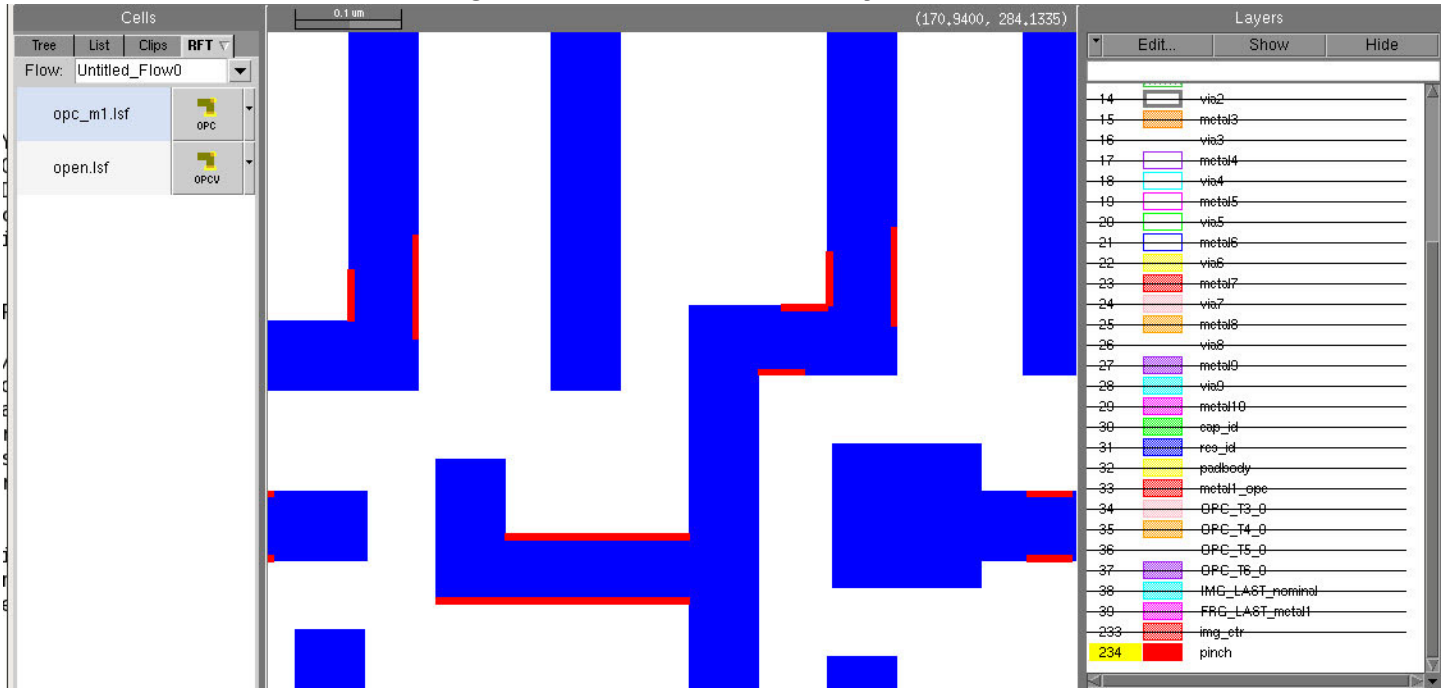
9. Click the **OPC** button. The metal1 layout area displayed in the Calibre WORKbench window is processed by Calibre nmOPC, producing the layers specified in the Extra Outputs section.

Figure 2-7. Calibre nmOPC Results for metal1 Layer



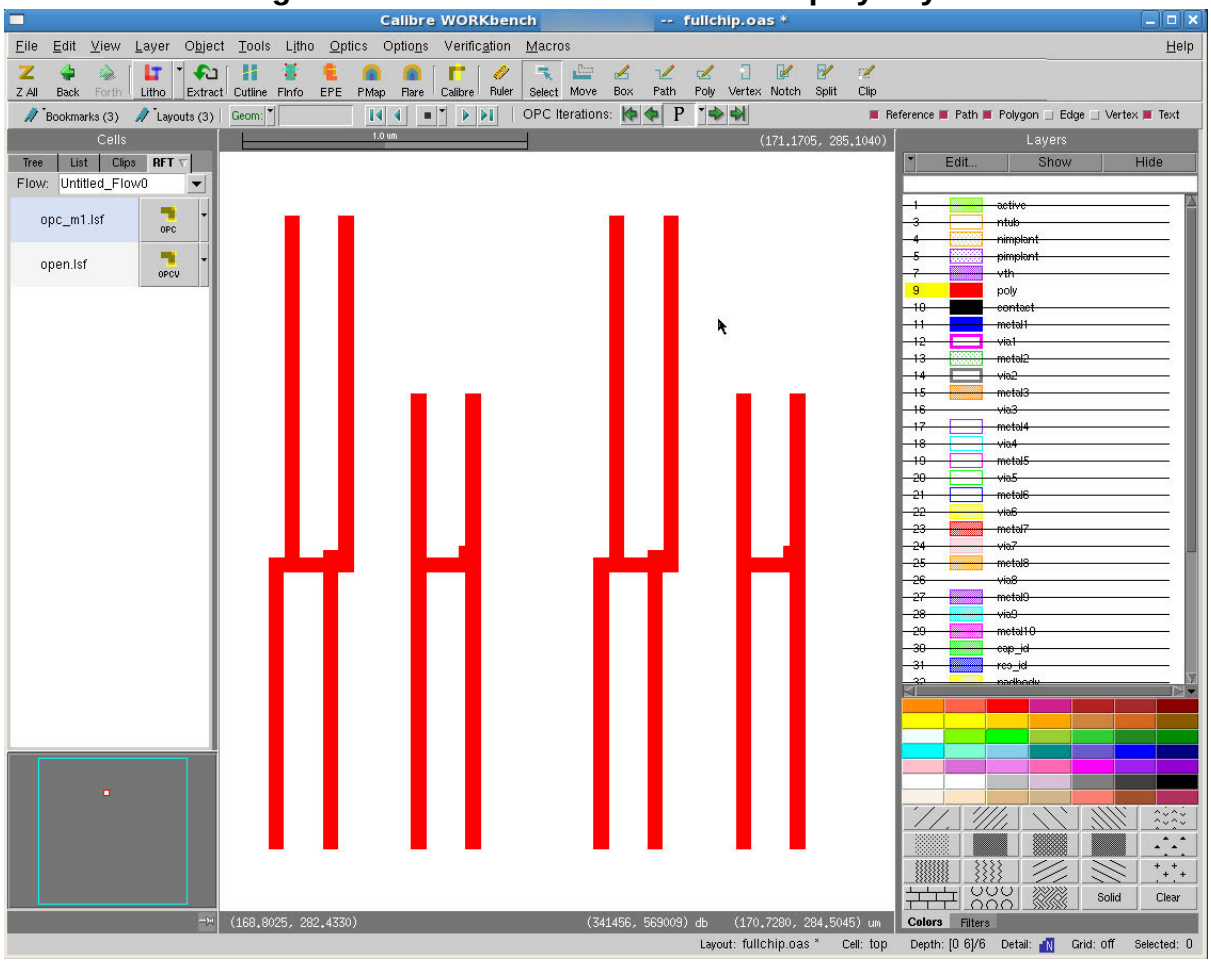
10. In the RET Flow Tool, click the *open.lsf* entry. The Calibre OPCverify setup file is displayed.
11. In the Setup Layers Mapping section, make sure that the layers are mapped.
12. Click the **OPCV** button. The same area processed by Calibre nmOPC is now verified by Calibre OPCverify. Two new layers are added for pinch and for bridging checks. In Figure 2-8, fragments that violated the bridging checks are highlighted in red.

Figure 2-8. Calibre OPCverify Results



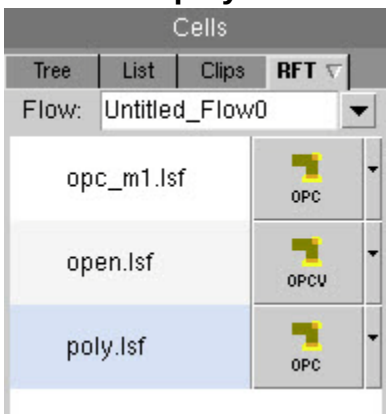
13. In the next part of this example, another Calibre nmOPC session is used to process the poly layer (layer 1). In Calibre WORKbench, display only the poly layer (layer 9 in Figure 2-9) and zoom into a section to correct.

Figure 2-9. Calibre WORKbench with poly Layer



14. In the RET Flow Tool, select **File > Add Existing Setup**. Select the file *poly.in* to load.
15. The *poly.in* file is added to the existing session, adding an extra Calibre nmOPC session (*poly.lsf*) icon to the Flow folder and to the **RFT** tab in Calibre WORKbench.

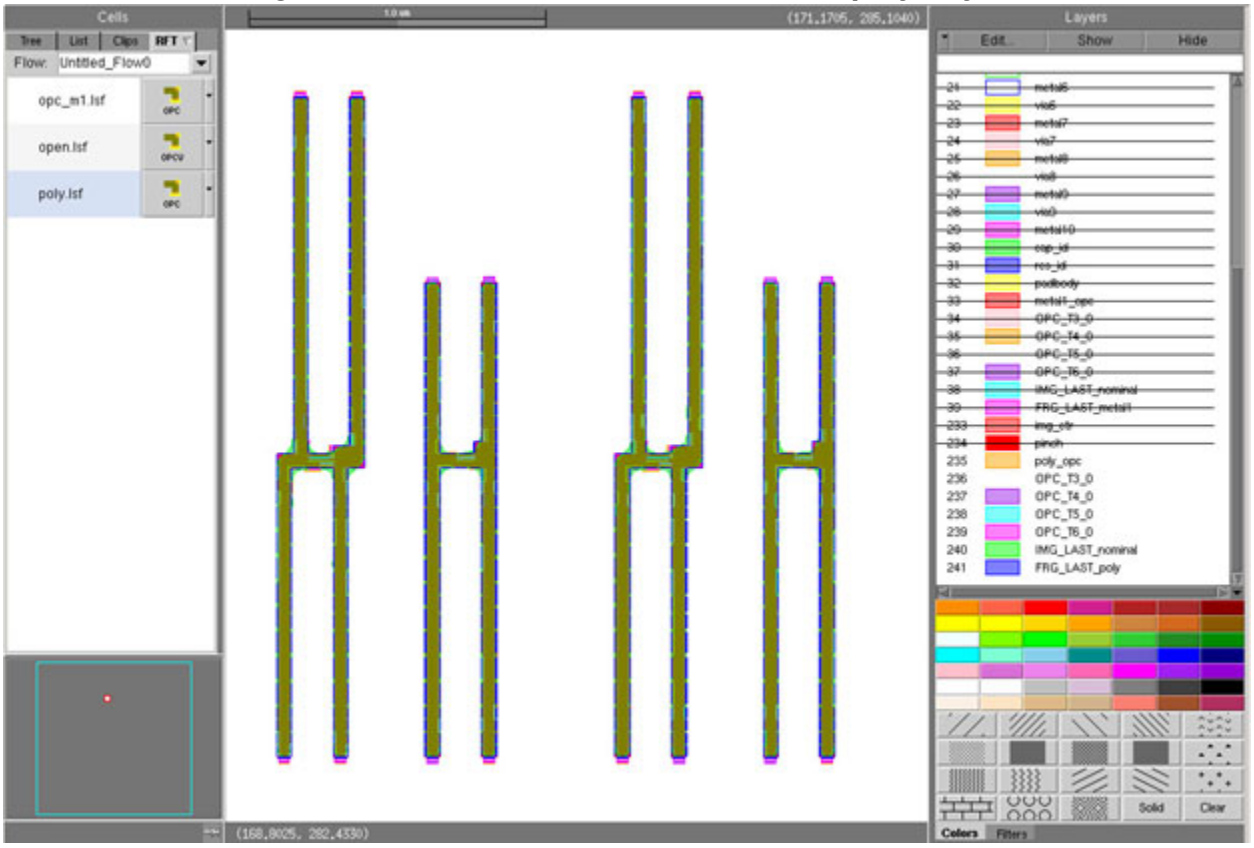
Figure 2-10. Additional poly.lsf Added to RFT Tab



16. Set the Calibre nmOPC run options as they were in Figure 2-6. However, in the Setup Layers Mapping section, the poly layer must be set to layer 9.

17. Click **OPC** to run Calibre nmOPC on the selected area of the poly layer. Output layers are generated for OPC results, fragmentation, and image.

Figure 2-11. Calibre nmOPC Results for poly Layer



18. Generate a print image by clicking the **PI** button in the RET Flow Tool. A nominal print image layer is generated in Calibre WORKbench. You can also switch the **OPC** icon to **PI** in the **RFT** tab in Calibre WORKbench using the pulldown menu next to the icon.

Figure 2-12. Changing to the PI Icon

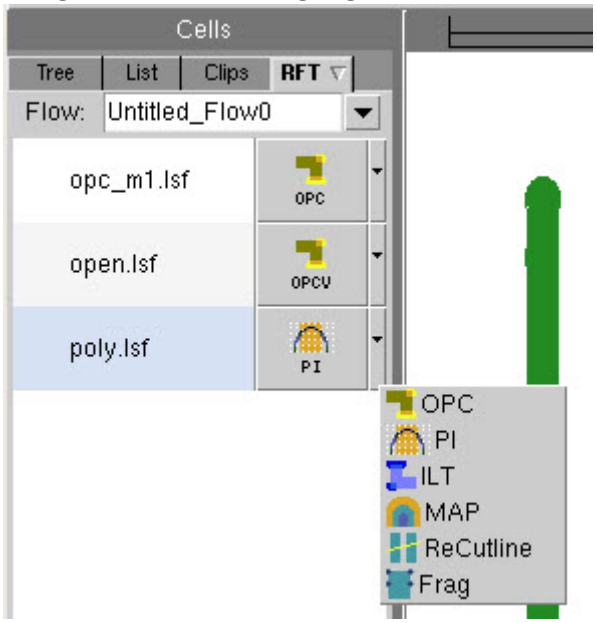
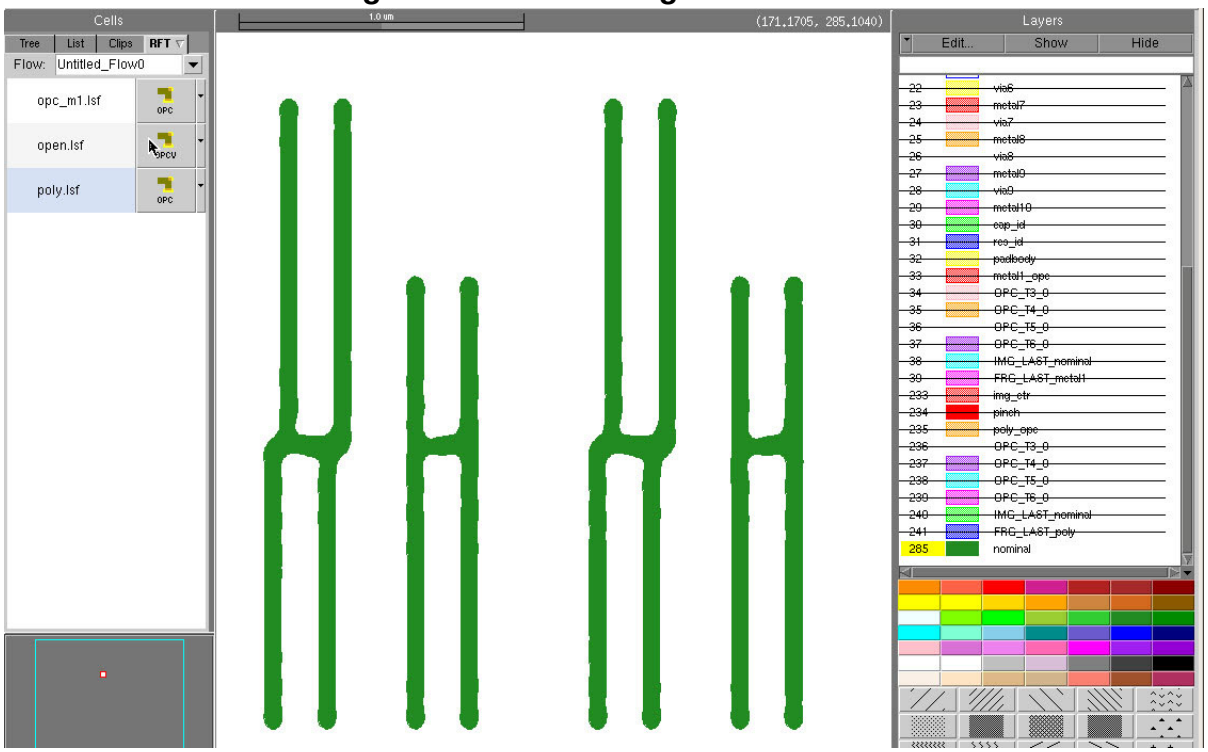


Figure 2-13 shows the results of the PI operation.

Figure 2-13. Print Image Generation



19. Save the entire flow by selecting **File > Save RET Flow**. All the current settings in the RET Flow tool are saved in a directory. Each of the sessions are saved as *.lsf* files (Litho Session Files).

Examples

This is the text of the SVRF file used in the procedure:

```
LAYOUT PATH "../Shared/Design/fullchip.oas"
LAYOUT SYSTEM OASIS
LAYOUT PRIMARY "*"
PRECISION 2000
RESOLUTION 1

FLAG SKEW NO
FLAG ACUTE NO
FLAG OFFGRID NO
LAYOUT ERROR ON INPUT YES

DRC MAXIMUM RESULTS ALL
DRC RESULTS DATABASE "output/opcv_out_cm1.oas" OASIS PSEUDO
DRC SUMMARY REPORT "./nmopc_summary.rep"


LAYER metal1 11
opc_m1 = LITHO DENSEOPC FILE metal1 metal1 MAP metal1_opc
opc_m1 {COPY opc_m1} DRC CHECK MAP opc_m1 gds 111 m1_out.oas

// The Calibre nmOPC setup file.

LITHO FILE metal1 [
   modelpath <modelpath>/models
   optical_model_load opticalD0 opticalD0_calibrated
   resist_model_load cm1out cm1out_opt.mod
   background clear

   layer metal1 hidden atten 0.06

   denseopc_options dopc {
      version 1
      background clear
      layer metal1 opc atten 0.06
      image optical opticalD0 resist cm1out
   }

   setlayer metal1_opc = denseopc metal1 MAP metal1 OPTIONS dopc
]

// The Calibre OPCverify section.

img_ctr= LITHO OPCVERIFY metal1 FILE OPCV_m1_SETUP MAP   img_ctr
img_ctr {COPY img_ctr} DRC CHECK MAP img_ctr 21

short= LITHO OPCVERIFY metal1 FILE OPCV_m1_SETUP MAP   short
open = LITHO OPCVERIFY metal1 FILE OPCV_m1_SETUP MAP   open
short{COPY short} DRC CHECK MAP short 25
open {COPY open} DRC CHECK MAP open 26

short_rdb { DFM RDB short "./ascii/poly_out.ascii" \
   MAXIMUM ALL ALL CELLS NOEMPTY CHECKNAME short  }
open_rdb {DFM RDB open "./ascii/poly_out.ascii" \
```

```
   MAXIMUM ALL ALL CELLS NOEMPTY CHECKNAME open}

pinch = LITHO OPCVERIFY metal1 FILE OPCV_m1_SETUP MAP pinch
bridge = LITHO OPCVERIFY metal1 FILE OPCV_m1_SETUP MAP bridge
pinch {COPY pinch} DRC CHECK MAP pinch 51
bridge {COPY bridge} DRC CHECK MAP bridge 52
pinch_rdb {DFM RDB pinch "./ascii/poly_out.ascii" \
   MAXIMUM ALL ALL CELLS NOEMPTY CHECKNAME pinch}
bridge_rdb {DFM RDB bridge "./ascii/poly_out.ascii" \
   MAXIMUM ALL ALL CELLS NOEMPTY CHECKNAME bridge}

LITHO FILE OPCV_m1_SETUP [/*
   modelpath <modelpath>/models
   optical_model_load opticalD0 opticalD0sim_modified
   resist_model_load cm1out cm1out1.mod

   background clear
   layer metal1 visible dark

   setlayer img_ctr  = image optical opticalD0 dose 1.00 \
   resist_model cm1out

setlayer short = bridge metal1 img_ctr <= 0 \
   separation 0.2 output_expand 0.005 property { min }
setlayer open = pinch  metal1 img_ctr <= 0 \
   separation 0.2 output_expand 0.005 property { min }
setlayer bridge = bridge metal1 img_ctr < 0.060 \
   separation 0.2 max_edge 0.050 output_expand 0.005 property {min}
setlayer pinch = pinch  metal1 img_ctr < 0.050 \
   separation 0.2 max_edge 0.050 output_expand 0.005 property {min}
*/]
```

The following example shows a separate file, *poly.in*, for running dense OPC (nmOPC) on the poly layer.

```
modelpath <modelpath>/models
optical_model_load opticalD0 opticalD0_calibrated
resist_model_load cm1out cm1out_opt.mod
background clear

layer poly hidden atten 0.04
denseopc_options dopc {
   version 1
   background clear
   layer poly opc atten 0.04
   image optical opticalD0 resist cm1out
}
setlayer poly_opc = denseopc poly MAP poly OPTIONS dopc
```

Related Topics

Calibre nmOPC in the RET Flow Tool (nmOPC Session)

Calibre OPCverify in the RET Flow Tool (OPCverify Session)

Using Sub Resolution Assist Features

The RET Flow Tool performs SRAF placement development within a portion of a layout.

This flow illustrates RET Flow Tool usage with Calibre nmSRAF.

Prerequisites

- An optical model and a resist model.
- A layout.

  The example layout used in this procedure is a double-patterning design. The only layer used is metal.
- An LSF file.

  The example LSF file, *dp_mbsraf.lsf*, used in the procedure is shown in the Examples section after the steps. This file contains the Calibre nmSRAF setup. This is for the two metal1 masks of the double-patterning layout.

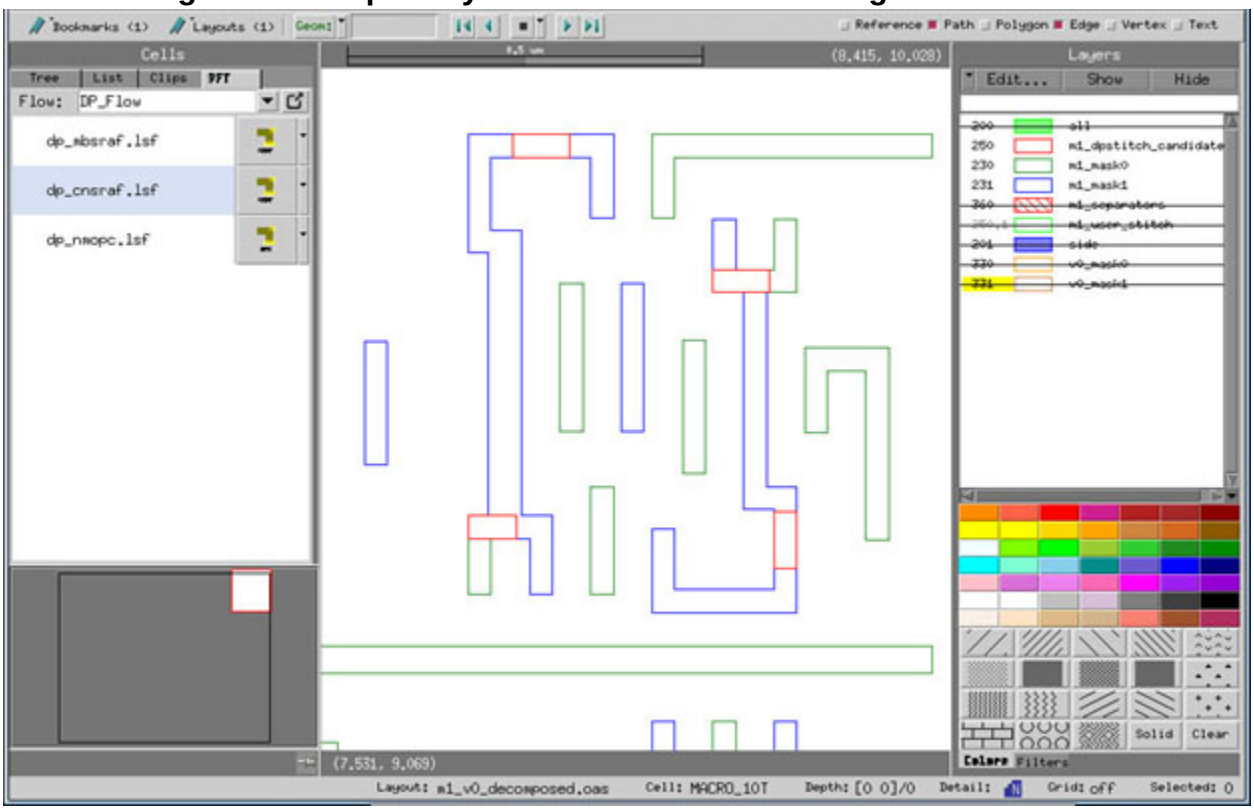
Procedure

1. Invoke Calibre WORKbench and load the layout file.

   ```
   calibrewb dp_layout.oas
   ```
2. Zoom in on a portion of the metal1 layer for SRAF. In this example, two masks representing the entirety of the metal layer have already been decomposed.
3. Display only the metal1 layer.

   Figure 2-14 on page 29 shows an example of a zoomed-in metal1 layer.

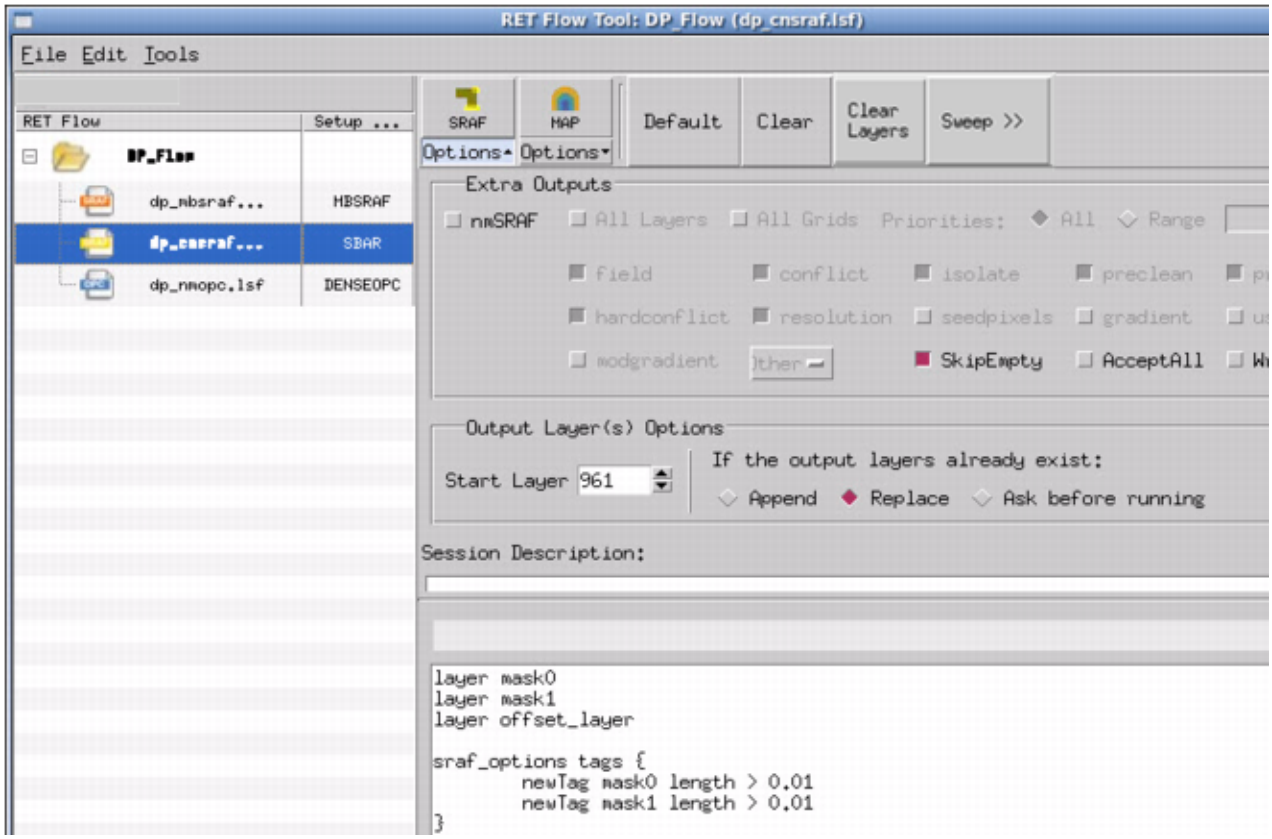
Figure 2-14. Input Layout with Double-Patterning metal1 Masks



4. In Calibre WORKbench, click the **Litho** button. The RET Flow Tool appears.
5. In the RET Flow Tool, select **File > Open RET Flow**.
6. Select the *dp_mbsraf.lsf* file and click **OK**.

   The flow opens. Figure 2-15 shows three sessions in the flow: MBSRAF, SBAR, and NMOPC. The MBSRAF session is selected.

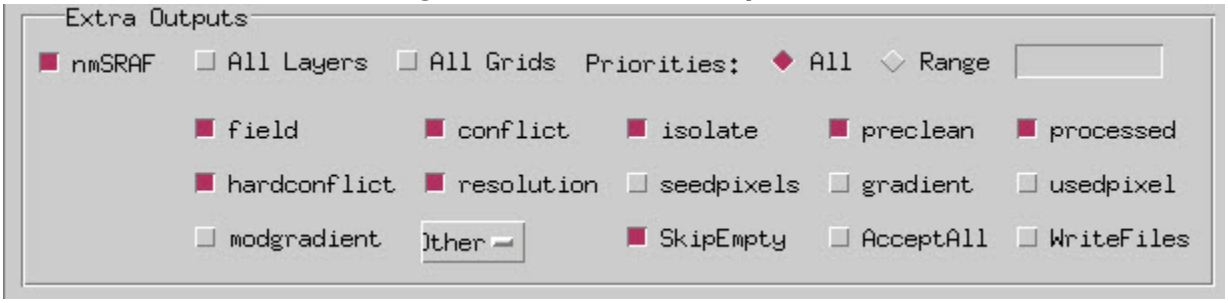
Figure 2-15. Flow Session Opened



7. Click the nmSRAF checkbox in the Extra Options section of the SBAR session.

   Various options are selected by default.

Figure 2-16. nmSRAF Options

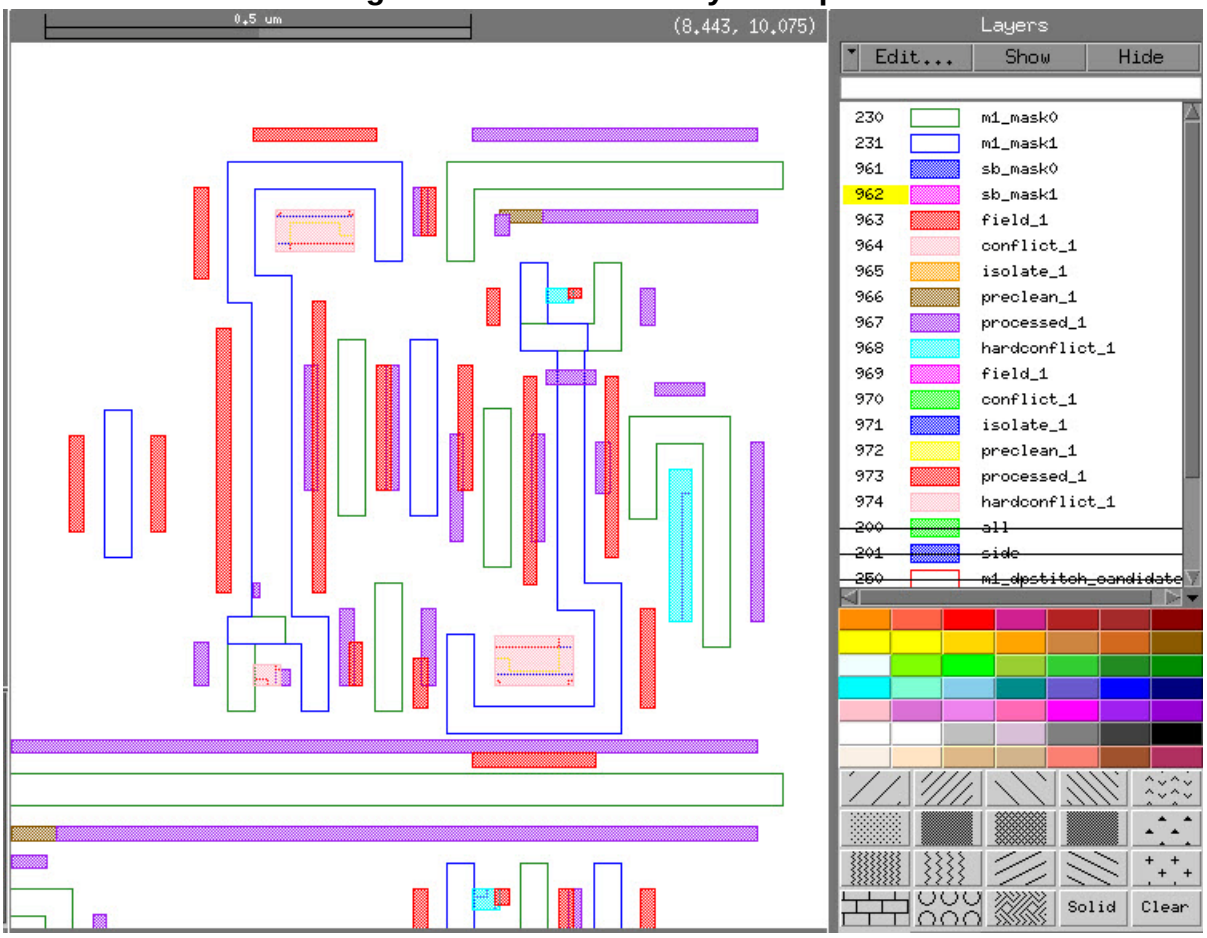


   The Extra Outputs section on the nmSRAF session provides debugging layers and, for mbsraf only, various pixel-based maps.

8. In the nmSRAF session, click the **SRAF** button.

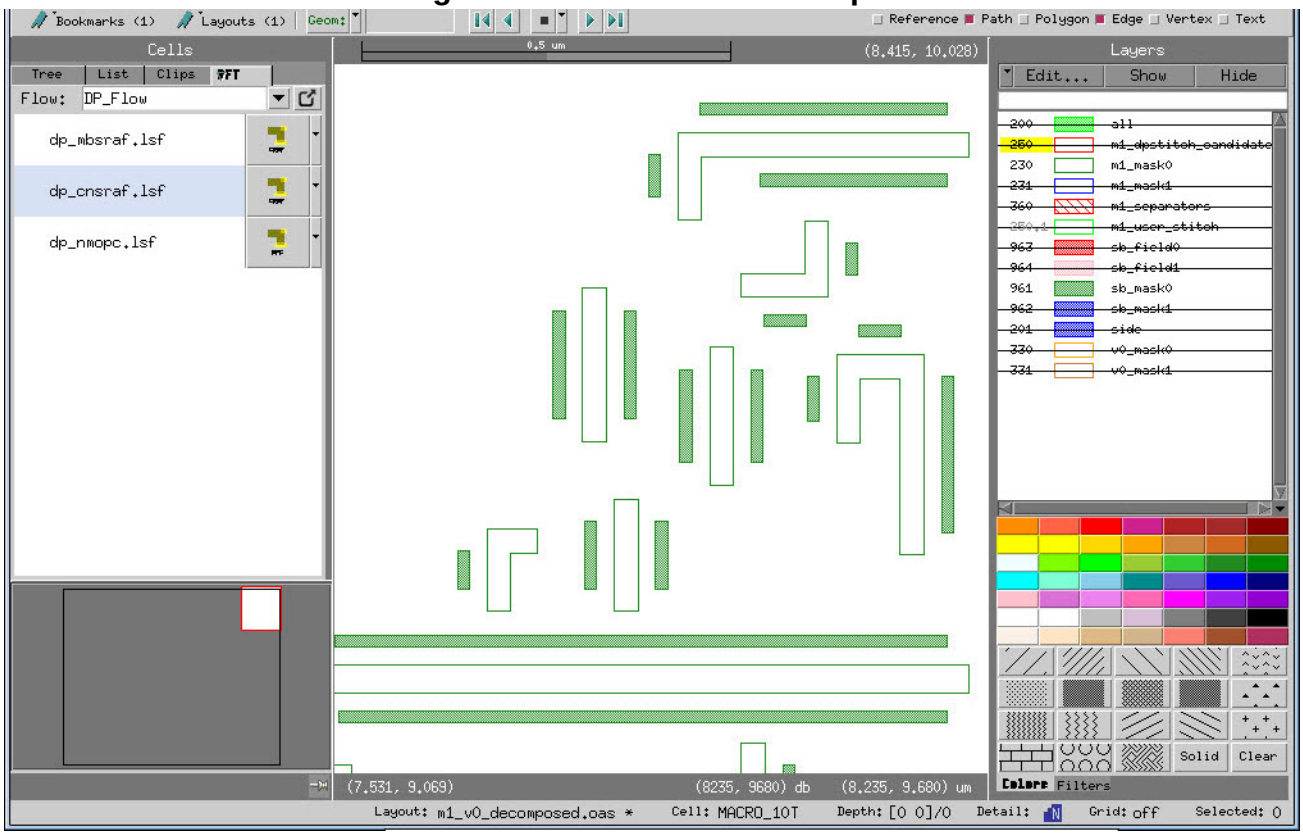
   Calibre nmSRAF runs on the layout. The resulting SRAFs and debug layers are placed into the design for both double-patterning metal masks as depicted in Figure 2-17.

Figure 2-17. nmSRAF Layer Output



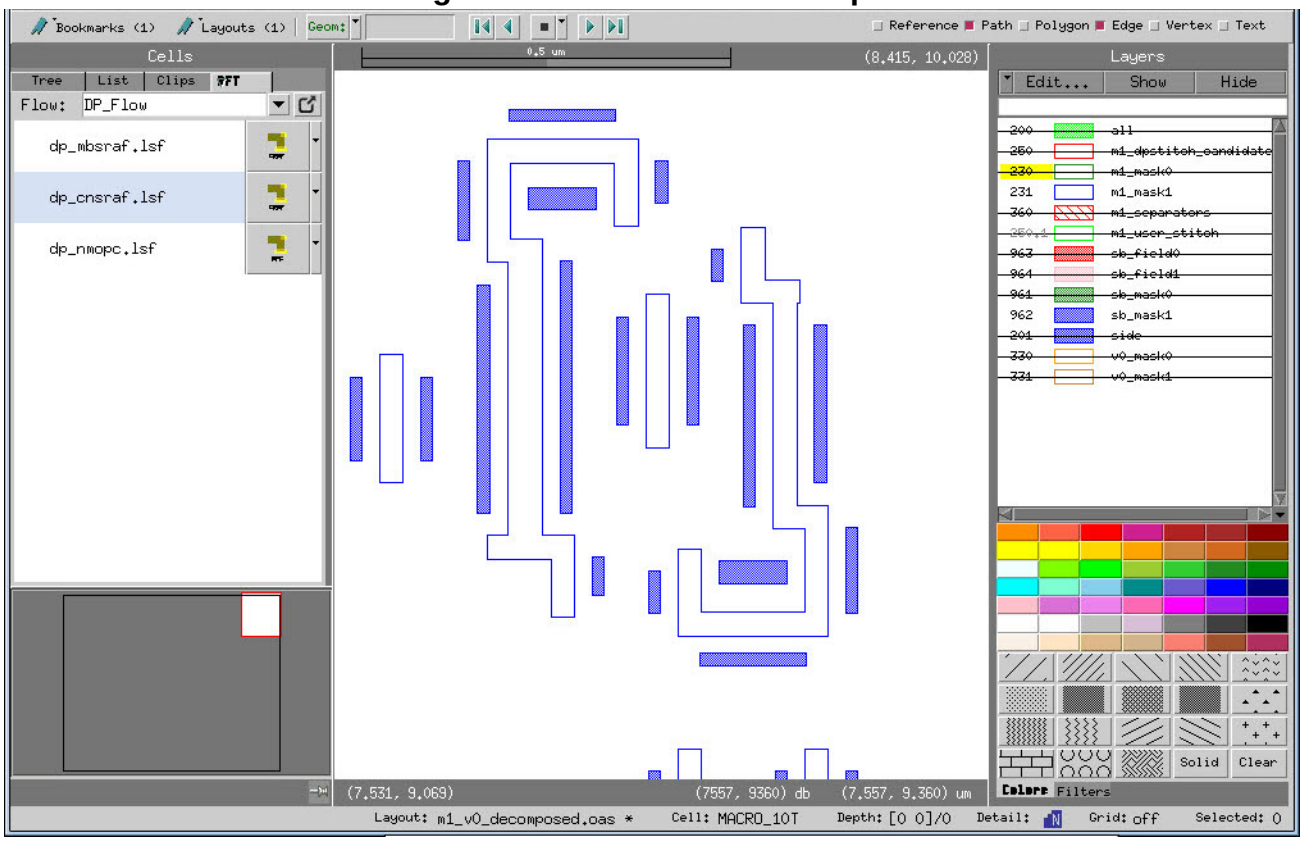
Viewing the SRAF for masks independently provides greater clarity. The SRAF for Mask0 only is shown in Figure 2-18.

Figure 2-18. Mask0 SRAF Output



The SRAF for mask1 only is shown in Figure 2-19.

Figure 2-19. Mask1 SRAF Output



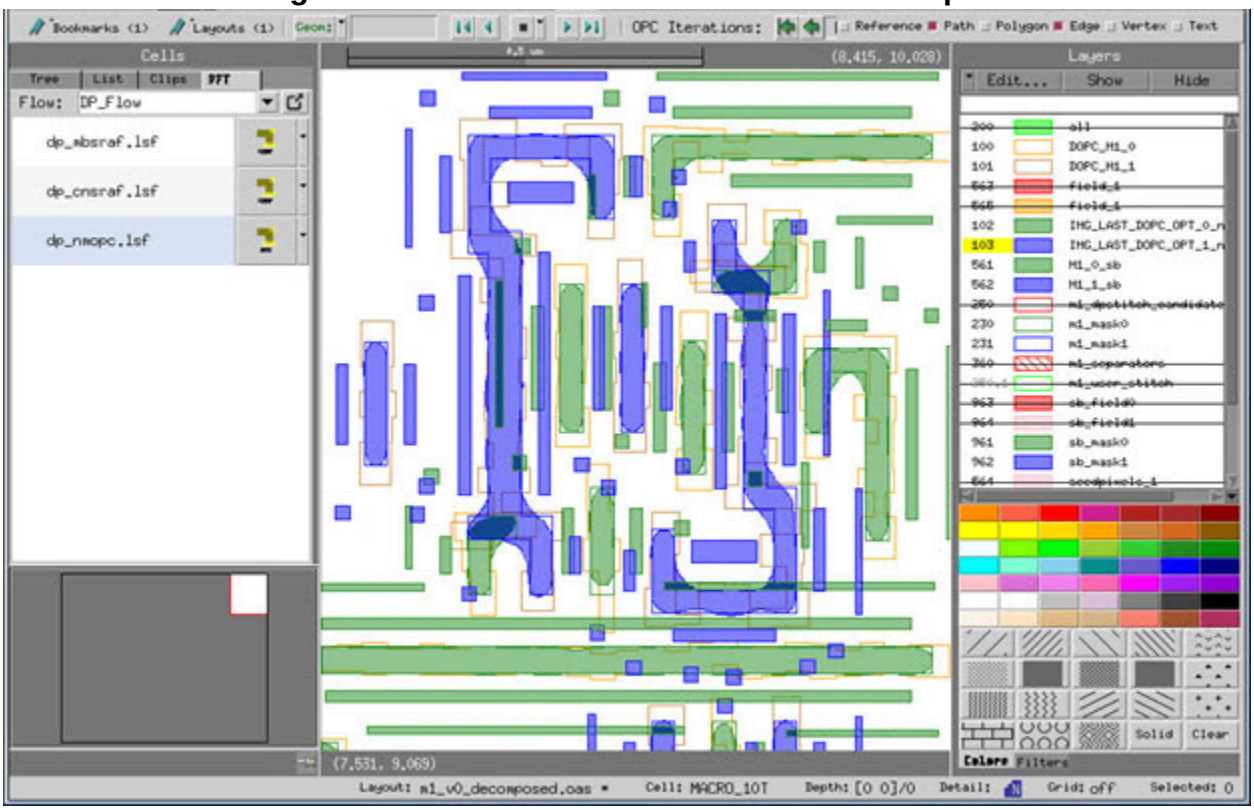
9. In the same layout window, select the **MBSRAF** session and click the **SRAF** button.

   The model-based SRAF and debug layers appear. All SRAF are now complete.

10. Select the NMOPC session and click the **OPC** button.

    The OPC treatment and resulting contour layers appear as shown in Figure 2-20.

Figure 2-20. OPC and Contour Simulated Output



Examples

The following file, *dp_mbsraf.lsf*, is used for the Calibre nmSRAF procedure for a double-patterning design:

```
############## SESSION ##############
LITHO_SESSION MBSRAF_TEXT_SESSION

############## SETUP TEXT ##############
LITHO_FILE {
# The RET Flow Tool has modified the "modelpath" command.
# The Current Working Directory is different from the Running Directory.
# Current Working Directory: "/user/work/testcases/RFT/litho"
# Running Directory: "/user/work/testcases/RFT/litho/DP_Flow".

# ---------------- Simulation models ---------------
   modelpath ./models:/user/work/.calibrewb_workspace/models

#---Layer info ------
   background clear
   layer target_0 visible dark 0 0.7
   layer target_1 visible dark 0 0.7
   layer opc_layer_0 visible dark 0 0.7
   layer opc_layer_1 visible dark 0 0.7
   layer sb_0 visible dark 0 0.7
   layer sb_1 visible dark 0 0.7

   tilemicrons 30

   image_options M0 {
      litho_model 32nm_C1
      layer target_0 visible mask_layer 0
      layer sb_0 visible mask_layer 0
   }
   image_options M1 {
      litho_model 32nm_C1
      layer target_1 visible mask_layer 0
      layer sb_1 visible mask_layer 0
   }

   sraf_options M1_0 {
      sblayer sb_0
      correctionLayer opc_layer_0
      image_options M0
      cornerPref $env(CP)
      sortSeed gradient
      minSquareLength 0.020 minSquareArea 0.0004
      minOffset $env(MO)
      peaktype $env(P)
      minfeaturespace $env(MF)
      localMaxGradient width 0.010 squareWidth 0.020 prior 1 mode $env(M)
   }

   sraf_options M1_1 {
      sblayer sb_1
      correctionLayer opc_layer_1
      image_options M1
```

```
      cornerPref $env(CP)
      sortSeed gradient
      minSquareLength 0.020 minSquareArea 0.0004
      minOffset $env(MO)
      peaktype $env(P)
      minfeaturespace $env(MF)
      localMaxGradient width 0.010 squareWidth 0.020 prior 1 mode $env(M)
   }

   setlayer M1_0_sb = mbsraf target_0 OPTIONS M1_0 \
      maxOffset 0.090 minWidth 0.016 minLength 0.060 minArea 0.000625 \
      dynamicSizing 0.05

   setlayer M1_1_sb = mbsraf target_1 OPTIONS M1_1 \
      maxOffset 0.090 minWidth 0.015 minLength 0.060 minArea 0.000625 \
      dynamicSizing 0.05

   setlayer seedpix_0 = mbsraf debug M1_0_sb seedpixels 1
   setlayer seedpix_1 = mbsraf debug M1_1_sb seedpixels 1
   setlayer tiles = tilegen 0.01
}

############### MAPPING ###############
INPUTLAYERS_MAPPING {target_0 230 target_1 231 \
   opc_layer_0 8192 opc_layer_1 8193 sb_0 961 sb_1 962}
VARIABLES_MAPPING {{CP env square} {MO env 0.020} {P env 3dir} \
   {MF env 0.024} {M env nearestno45}}
```

Related Topics

Calibre nmOPC in the RET Flow Tool (nmOPC Session)

Calibre nmSRAF in the RET Flow Tool (cnsraf and mbsraf Sessions)

Chapter 3
Using Sessions

Each supported RET product has its own session type in the RET Flow Tool. Each session in a flow represents a particular configuration of a major RET tool for a particular run.

Creating a Session

Sessions correspond to running one of the Calibre RET tools. A flow consists of a set of sessions. Although it is more typical to extract sessions from an SVRF rule file, you can also create sessions with no previous setup file.

Restrictions and Limitations

Some sessions may not currently support all functionality of the related tool in this release.

Prerequisites

Some tools may require you to have an existing command file, setup file, or SVRF file to properly configure the session.

Procedure

1. Invoke Calibre WORKbench or Calibre LITHOview and open the RET Flow Tool (**Litho > RET Flow Tool**).
2. In the RET Flow Tool, create a flow by selecting one of the following options from the **File** menu. Depending on your choice, you may need to navigate to a file location.

Table 3-1. RET Flow Tool Flow Creation Options (File Menu)

Option	Description
New RET Flow	Creates a new Untitled_Flow folder in the RET Flow list.
Open RET Flow	Opens a previously-created RET Flow via a file navigator.
Extract from SVRF	Opens a specialized navigator that lists layer output rules in a SVRF rule file. You select the rules to import. See “Extracting a Setup File from SVRF” on page 93 for more information.
Extract all from SVRF	The RET Flow Tool imports all the rules it recognizes without prompting the user. See “Extracting a Setup File from SVRF” on page 93 for more information.

Table 3-1. RET Flow Tool Flow Creation Options (File Menu) (cont.)

Option	Description
Extract from Transcript	Opens a specialized navigator that reads the rules from a transcript generated by a previous SVRF run. See "Extracting a Setup File from a Transcript" on page 92 for more information.
Extract all from Transcript	The RET Flow Tool reads the transcript and imports all the rules it recognizes without prompting the user. See "Extracting a Setup File from a Transcript" on page 92 for more information.

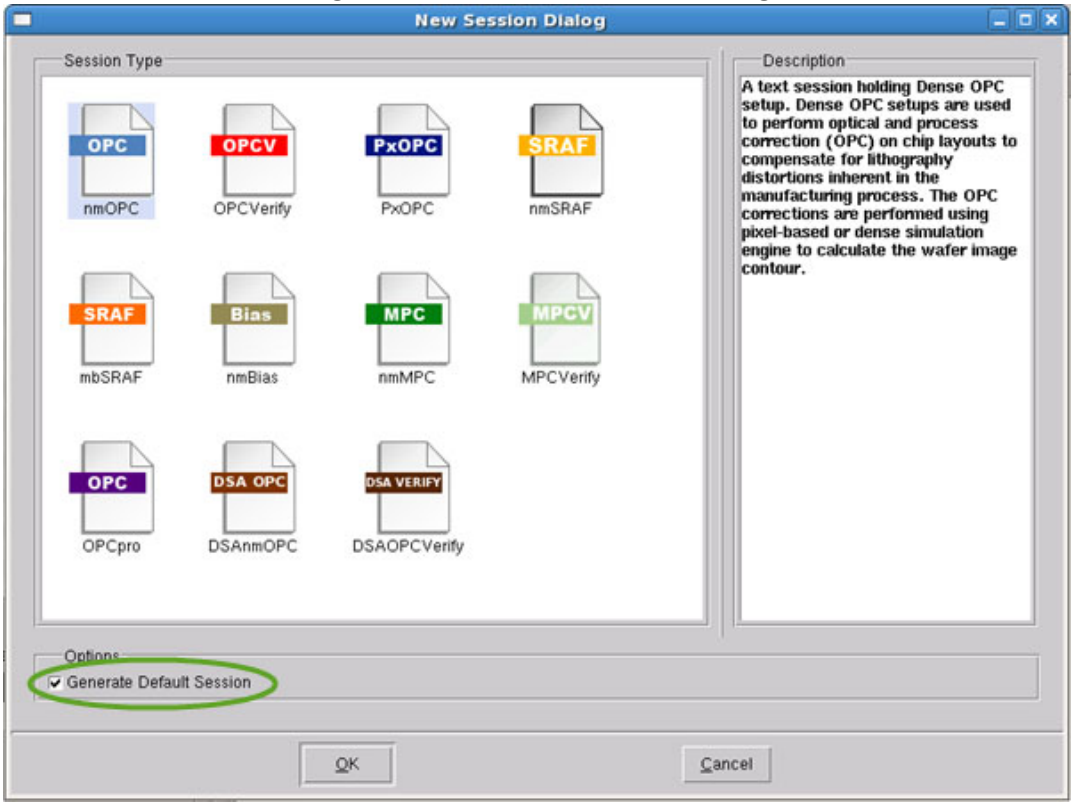
3. Rename the new flow by right-clicking its name in the list and selecting **Rename**.
4. The **Extract** menu options automatically create sessions from the source file. However, if you used the **New RET Flow** option, you must create a session.

   Add a session to the flow by selecting one of the following options from the **File** menu or the popup menu that appears when you right-click a flow.

Table 3-2. RET Flow Tool Session Creation Options

Option	Description
Add New Session	Brings up the New Session Dialog (Figure 3-1). Select one of the session types from the list. If you select the Generate Default Session check box, the session is pre-populated with a standard template. Otherwise, the new session is added with its data fields empty.
Add Existing Session	Loads a previously-saved *.lsf* file to re-create the session.
Add Existing Setup	Loads a previously-created setup (*.in*) file. The RET Flow Tool creates a session that most closely resembles the contents of the setup file.

Figure 3-1. New Session Dialog



5. Configure the new session using one of the session tasks in the Related Topics.

Related Topics

Basic Simulation in the RET Flow Tool (Simulation Session)

Calibre nmOPC in the RET Flow Tool (nmOPC Session)

Calibre OPCverify in the RET Flow Tool (OPCverify Session)

Calibre OPCpro in the RET Flow Tool (OPCpro Session)

Calibre nmSRAF in the RET Flow Tool (cnsraf and mbsraf Sessions)

Calibre nmBias in the RET Flow Tool (nmBIAS Session)

Calibre nmMPC in the RET Flow Tool (nmMPC Session)

Calibre MPCverify in the RET Flow Tool (MPCverify Session)

Calibre DSA in the RET Flow Tool

Basic Simulation in the RET Flow Tool (Simulation Session)

The Basic Simulation tool for Calibre WORKbench functionality has been converted to the new RET Flow Tool v2.0.

Note

For more information on the simulation tool for Calibre WORKbench, see the *Calibre WORKbench: RET Flow Tool (RFT) v2.0 User's Manual*.

Calibre nmOPC in the RET Flow Tool (nmOPC Session)

When you select nmOPC as one of your sessions, you access the Calibre nmOPC tool. This session uses a Calibre nmOPC-based setup file and contains settings for simulation tools.

Configuring a Calibre nmOPC Run

You must properly set up a Calibre nmOPC run in order to produce corrected results for your layout.

Prerequisites

- A valid litho model file or the following:
  - An optical model.

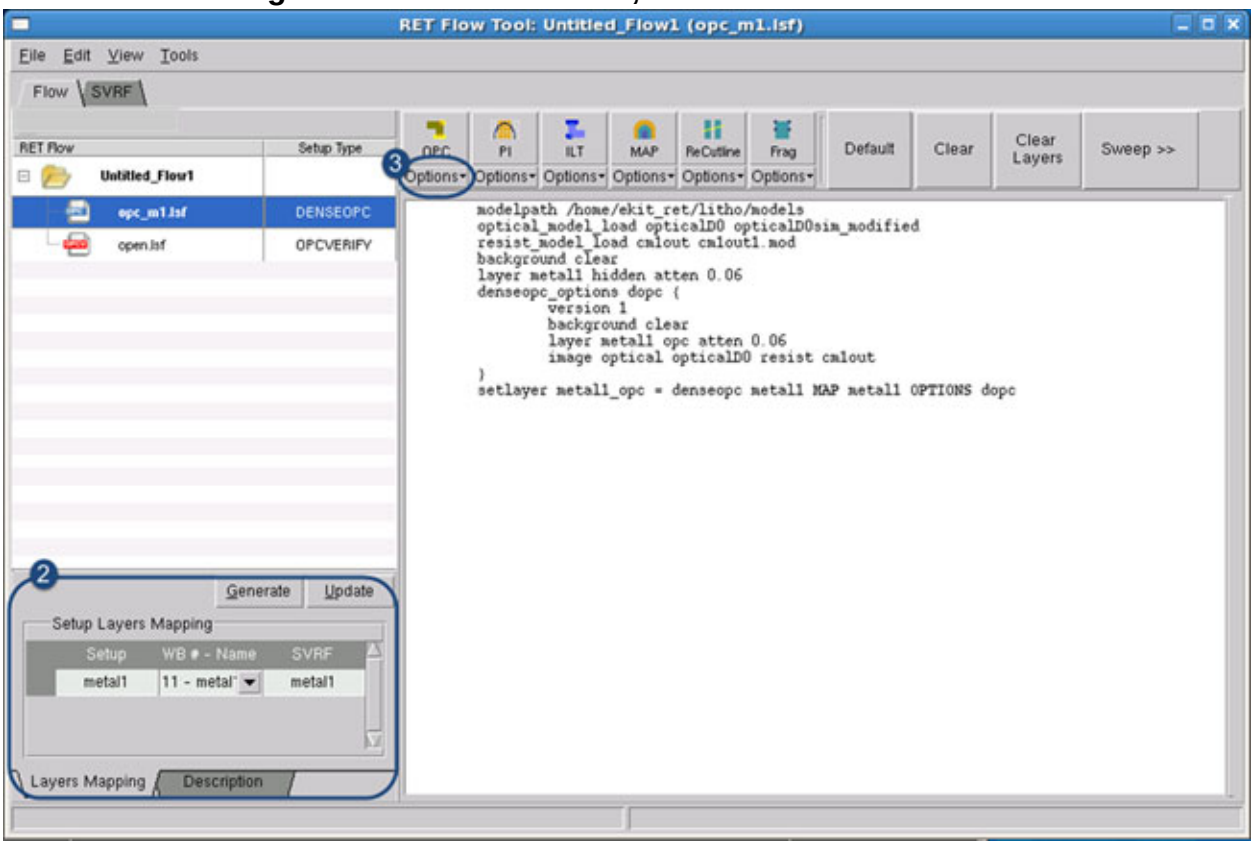
    If this is an EUV optical model, you will also need a flare model file.
  - A resist model if you are not using aerial image mode.

  Refer to “Specifying Models” on page 99 for information.
- A layout loaded in the viewer main window.

Procedure

1. In the RET Flow Tool, select a session of type “DENSEOPC” or add a Calibre nmOPC session.
2. In the Setup Layers Mapping section, set the design layer numbers to the corresponding Calibre nmOPC derived layer name.

Figure 3-2. RET Flow Tool, Calibre nmOPC Session



3. Click **Options** under the **OPC** button to expose the Extra Outputs pane (see Figure 3-3 for an example).
4. In the Extra Outputs pane, set output options such as OPC iterations, contours, edge movement, and fragment information for the OPC run.
    - **OPC Results** — Outputs the resulting OPC edge movements for the specified iterations. This is useful to track how edges move during OPC. It generates one consecutive output layer per iteration.
    - **Contour** — Outputs a contour based on the EPE measurements for the specified iterations. It generates one consecutive output layer per iteration. Selecting Pre-OPC includes pre-OPC contour information in the output.
    - **Fragment Markers** — Outputs fragmentation information such as final EPE, position, length, type, feedback, displacement, target (goal) position, and the metric used to measure the EPE. It generates one consecutive output layer per iteration. This is required if you want to view fragmentation information in the post-OPC analysis phase. Selecting Pre-OPC includes pre-OPC fragment information in the output.

Note

Annotated tags, as defined by the -annotated or -aout command options, can also be displayed in the Calibre nmOPC Fragment Information window.

- **Color** — Changes the color of the output type.

5. Click the **OPC** button to run OPC.

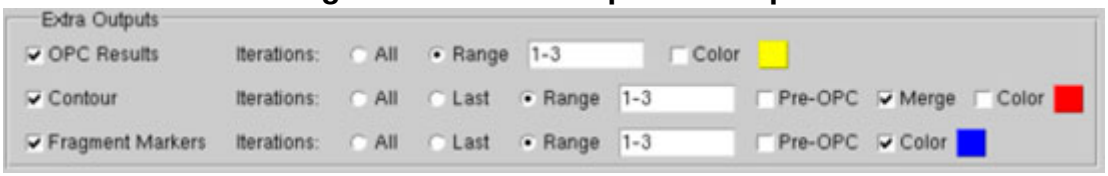
Results

Output layers are generated depending on the settings in the Extra Outputs pane prior to the run (see Figure 3-4 for an example). The output can be analyzed using a number of different tools and operations. See "Calibre nmOPC Operations" on page 45 for a list of operations that can be performed.

Examples

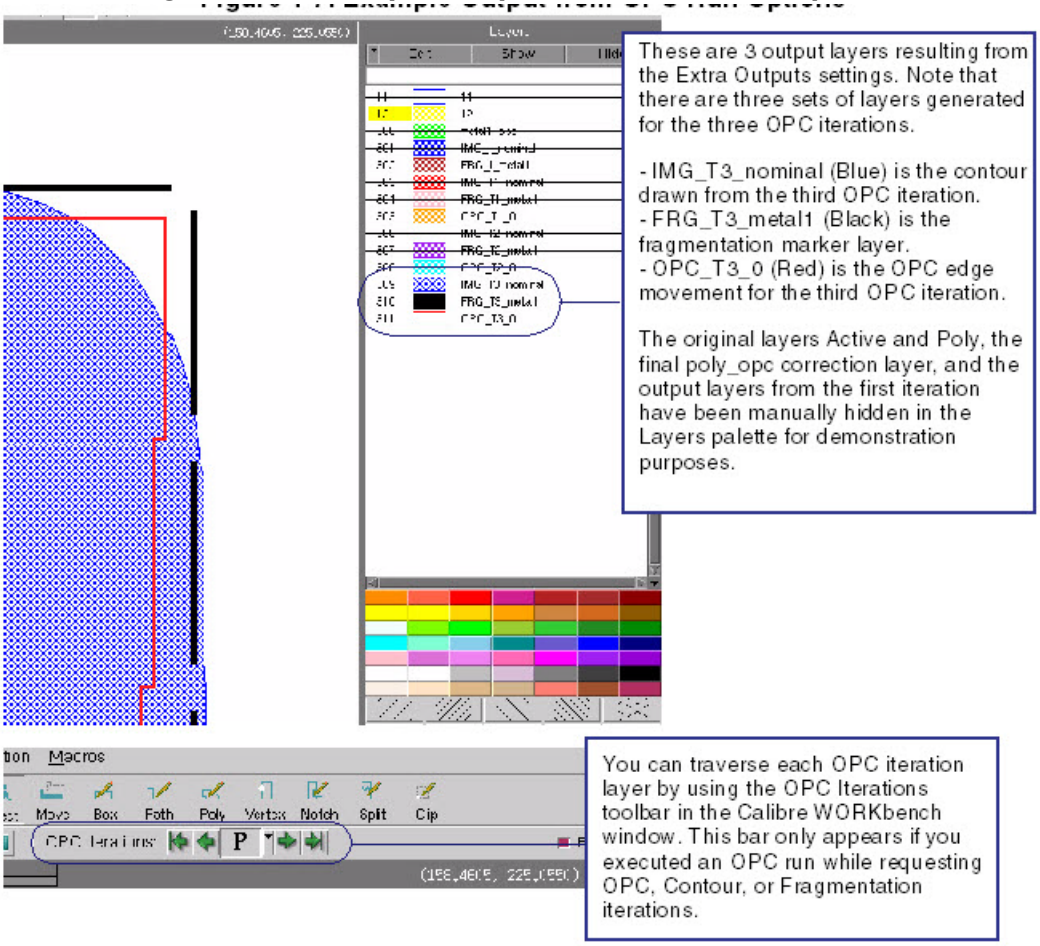
In Figure 3-3, the OPC Results, Contour, and Fragment Markers have been enabled for the first three iterations of the OPC run.

Figure 3-3. Extra Outputs Example



The output looks similar to Figure 3-4.

Figure 3-4. Example Output from OPC Run Options



Calibre nmOPC Operations

Calibre WORKbench and Calibre LITHOview support a number of core Calibre nmOPC operations. For instance, there are different types of analysis that can be performed on the results of an OPC run, as well as options to adjust how data is displayed.

The operations and where their controls are located are summarized in the following table.

Table 3-3. Calibre nmOPC Operations

Operation	Control	Location	Description
“Generating a Print Image” on page 105	**PI**	RET Flow Tool toolbar	Simulate OPC effects on geometries on opc layers that lie partially or fully within a selected area.
“Configuring an ILT Run” on page 100	**ILT**	RET Flow Tool toolbar	Use inverse lithographic transforms (ILT) to perform OPC.

Table 3-3. Calibre nmOPC Operations (cont.)

Operation	Control	Location	Description
"Fragment Information" on page 120	**FInfo**	Calibre WORKbench toolbar	Output marker layers and inspect data for individual fragments, including EPE, length, position, biasing movement, and so on.
"Mapping EPE from Correction Layers" on page 46	**OPC Mapping**	Fragment Information dialog box, (after using **FInfo**)	(Used only for multi-mask exposures) Display how EPEs are re-mapped from the correction layer to fragments on the opc layer.
"Fragment Generation" on page 116	**Frag**	RET Flow Tool toolbar	Calculate and show how edges are fragmented for OPC on the layout. Adjust the display properties of fragmentation points, mapping and measurement locations, and sites in a user-selected area.
"Cutline Operations" on page 107	**Cutline** on the Calibre WORKbench icon bar	Calibre WORKbench toolbar	Compare one or more simulations along cutlines.
"Setting Cutline Properties" on page 108	**Re-Cutline**	RET Flow Tool toolbar	Change the display properties of cutlines.
"Creating an Aerial Image Contour Map" on page 101	**MAP**	RET Flow Tool toolbar	Create a contour map of the light intensity at the surface of the resist.
"Running Tests on Multiple Values With the Sweeping Tool" on page 131	**Sweep**	RET Flow Tool toolbar	Examine multiple values for specified command parameters.

Related Topics

Using the Calibre RET Flow Tool

Mapping EPE from Correction Layers

For processes that require multiple masks to create a desired image (as in phase-shift masks or double-exposure masks), use correction layers.

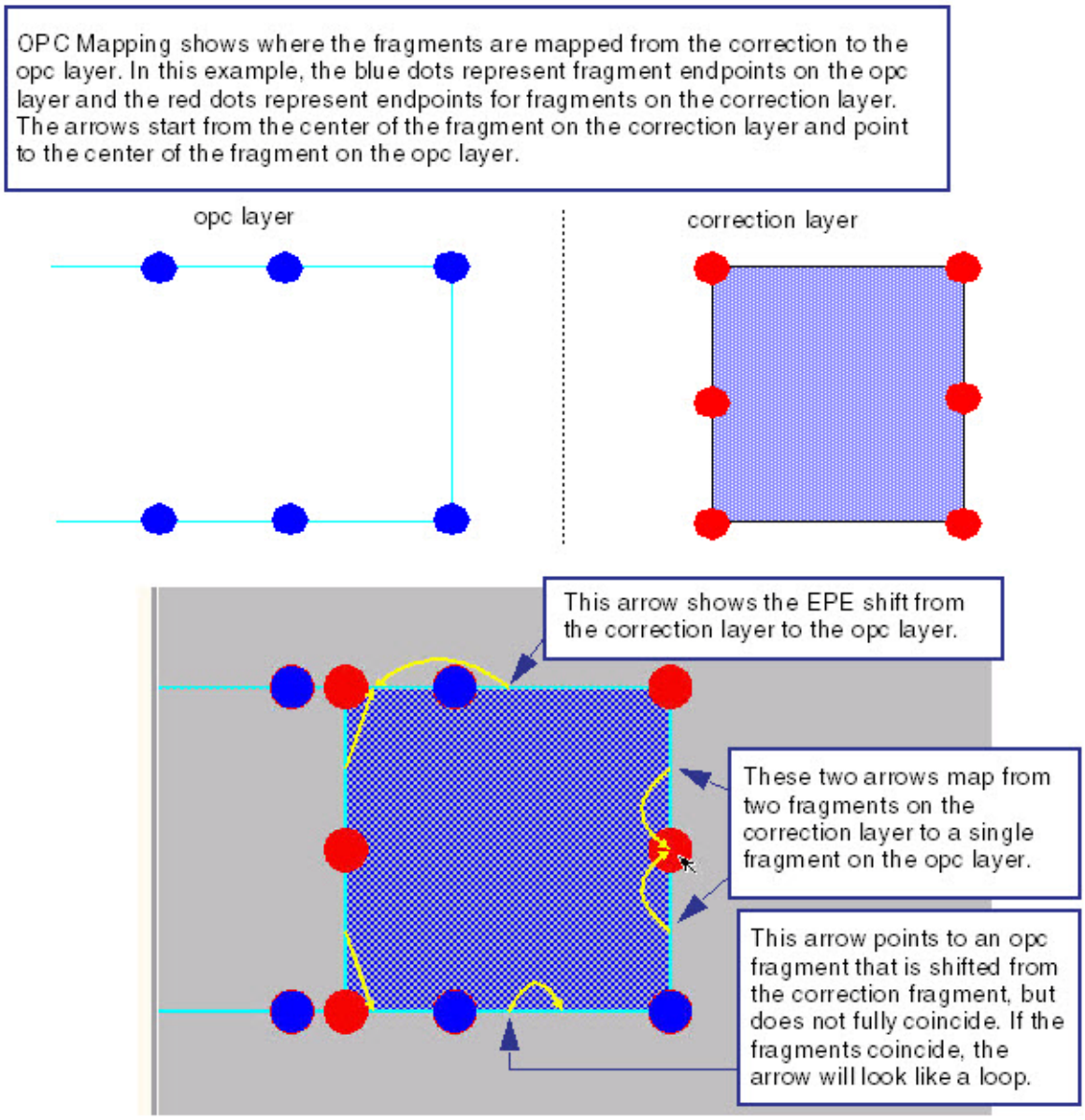
Correction layers are special layers containing edges that are moved during OPC. During the OPC process with a correction layer, fragments on a correction layer are mapped to fragments

on the opc layer. This means that EPEs are effectively measured from one layer and are applied to adjust fragment movements on the other layer.

Procedure

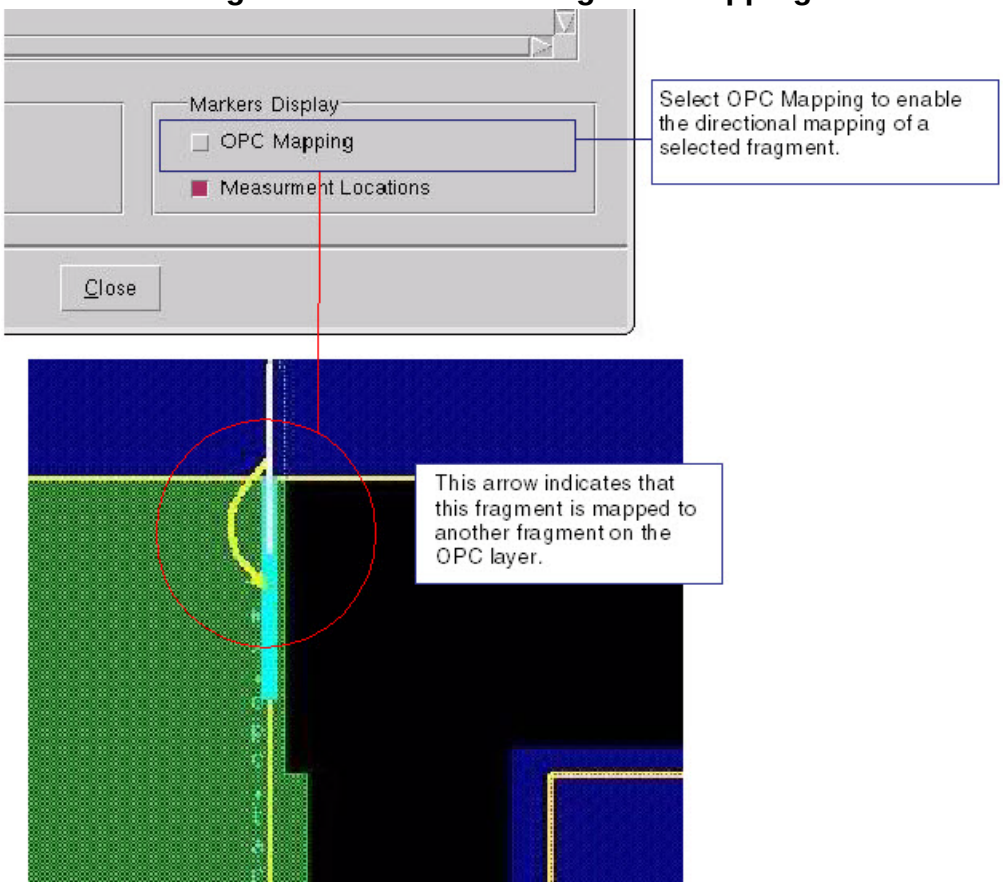
1. Run OPC from the RET Flow Tool with the **Fragment Markers** option enabled.
2. You can view OPC mapping using one of the following methods:
   - In the RET Flow Tool, click the **Frag** button following the OPC run to display the fragments. If there is a correction layer present, Calibre WORKbench automatically displays the OPC mapping as directional arrows (see Figure 3-5).

Figure 3-5. OPC Mapping Example



- You can also view the mapping of individual fragments by clicking the **OPC Mapping** option (see Figure 3-6) in the Fragment Information dialog box (see "Viewing Fragment Information" on page 120 for information).

Figure 3-6. Individual Fragment Mapping



Related Topics

Fragment Information

Calibre OPCverify in the RET Flow Tool (OPCverify Session)

When you select OPCverify as one of your sessions, you access the Calibre OPCverify tool. Sessions of type OPCVERIFY use a Calibre OPCverify-based setup file and contain settings for simulation tools.

Configuring a Calibre OPCverify Session

A Calibre OPCverify session must be correctly loaded and configured before you can use it to perform verification tasks.

Restrictions and Limitations

- image_sets are not recognized by the Calibre RET Flow Tool.

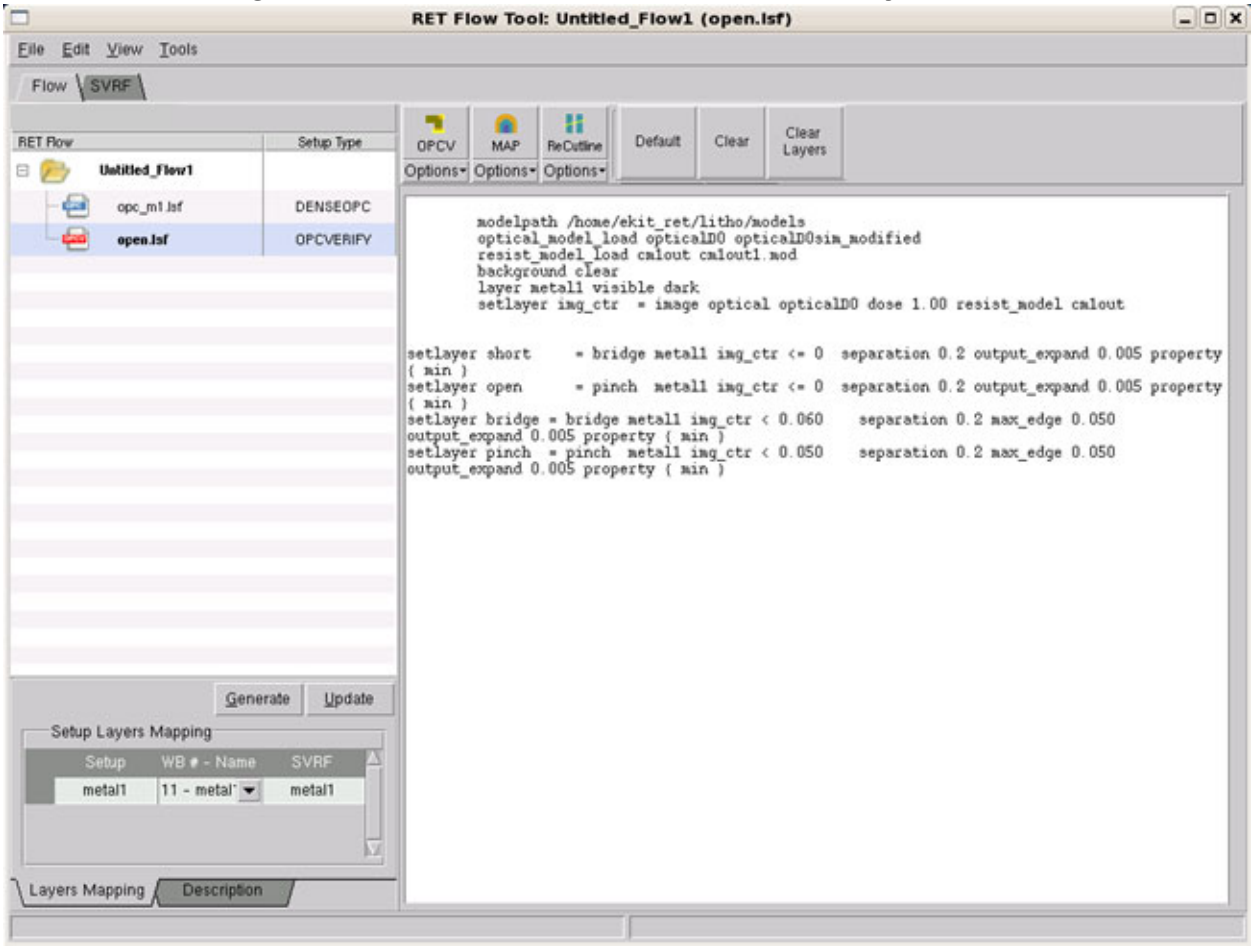
Prerequisites

- A known working Calibre OPCverify rule file in one of the following formats:
  - An SVRF file containing the Calibre OPCverify command file inline
  - A transcript from a Calibre OPCverify run
- The relevant design layout loaded into Calibre WORKbench

Procedure

1. Follow the steps in "Creating a Session" to create a Calibre OPCverify session.
   - If you used one of the methods that loaded an existing file, your RET Flow Tool should look similar to Figure 3-7.
   - If the session text entry pane is empty, you can use the **Copy** and **Paste** options from the **Edit** menu or right-mouse context menu inside the text pane to copy directly from a Calibre OPCverify rule file.

Figure 3-7. RET Flow Tool, Calibre OPCverify Session



2. Assign layers in the Setup Layers Mapping list from the active layout list to the listed layer outputs from the Calibre OPCverify file.
   - If you leave a layer set as Dummy Layer, it is considered to be hidden. Only layers assigned from the list are considered visible.
   - If layers are incorrectly mapped, running Calibre OPCverify may give unexpected results.
3. Test Calibre OPCverify by clicking the **OPCv** button.

Results

Calibre OPCverify runs the script, and any layers that created result markers or contours are added to the active layout in the viewer main window.

Calibre OPCverify Session Operations

A Calibre OPCverify session contains a number of GUI controls that run tools using the conditions in defined in the session.

The operations and where their controls are located are summarized in Table 3-4.

Table 3-4. Calibre OPCverify Session Operations

Operation	Control	Location	Description
See the *Calibre OPCverify User's and Reference Manual*	**OPCv**	RET Flow Tool toolbar	Run Calibre OPCverify.
"Creating an Aerial Image Contour Map" on page 101	**MAP**	RET Flow Tool toolbar	Create a contour map of the light intensity at the surface of the resist.
"Cutline Operations" on page 107	**Cutline** on the Calibre WORKbench toolbar	Calibre WORKbench toolbar	Compare one or more simulations along cutlines.
"Setting Cutline Properties" on page 108	**Re-Cutline**	RET Flow Tool toolbar	Change the display properties of cutlines.

Calibre pxOPC in the RET Flow Tool (PxOPC Session)

When you select PxOPC as one of your sessions, you access the Calibre pxOPC inverse lithography tool. This session uses a Calibre pxOPC-based setup file to perform simulation.

Calibre pxOPC produces its own SRAFs. The overall flow that includes a PxOPC session should not include an SRAF-generating session, though you may want to compare output between an nmOPC-SRAF flow and Calibre pxOPC.

Configuring a Calibre pxOPC Session

You must set up a session in order to run Calibre pxOPC on the layout.

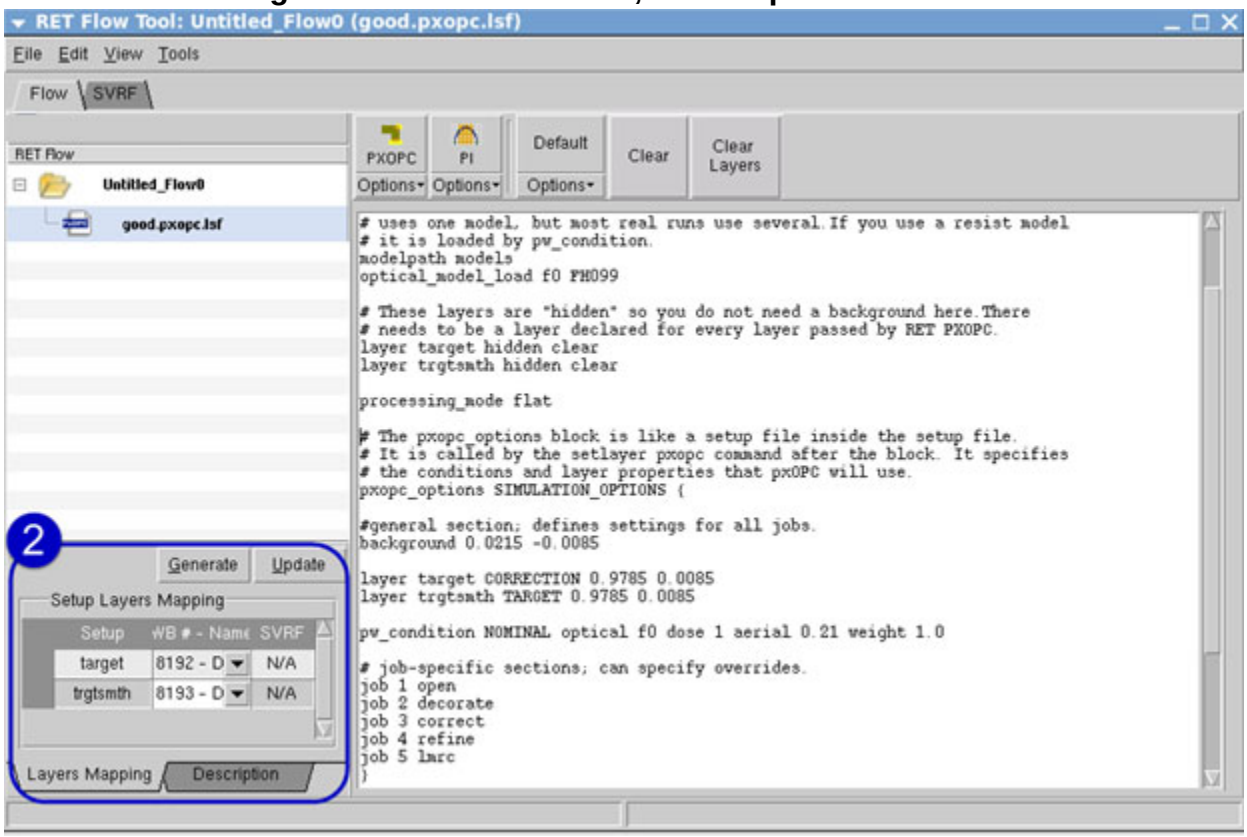
Prerequisites

- A valid litho model or optical model.
- A layout loaded in the main viewer window.

Procedure

1. Follow the steps in "Creating a Session" to create a Calibre pxOPC session.
    - If you used one of the methods that loaded an existing file, your RET Flow Tool should look similar to Figure 3-8.
    - If the session text entry pane is empty, you can:
        - Click **Options** under the Default button, enable the jobs you would like to use, and click **Default**.
        - Use the **Copy** and **Paste** options from the **Edit** menu .
        - Open the right-mouse context menu inside the text pane to copy directly from a Calibre pxOPC rule file.
        - Right-click the pxOPC session in the RET Flow tree and choose **Open Setup**.

Figure 3-8. RET Flow Tool, Calibre pxOPC Session



2. Assign layers in the Setup Layers Mapping list. “Setup” lists the layers from the setup file. “WB # - Name” shows the layout layer names.
3. In the text area, make any additional changes to the setup file such as adding output layers.
4. Click **Update**, above the Setup Layers Mapping list.
5. Click **PXOPC**.

Results

The status bar in Calibre WORKbench shows “pxopc...”. When the operation has completed, Calibre WORKbench shows the new layers. Layers are added starting at the layer number set in the **Options** area under **PXOPC**.

Repeat steps 3, 4, and 5 until you are satisfied with the results. Solutions to common layout issues are discussed in “Calibre pxOPC Configuration and Tuning” in the *Calibre pxOPC User’s and Reference Manual*.

Save the recipe by right-clicking on the session line in the RET Flow tree and selecting **Save Setup File**.

Examples

The set of Jobs Options shown in Figure 3-9 generate the setup file shown after. The different job types are described in the “job” reference page in the *Calibre pxOPC User’s and Reference Manual*. The “Reshape” option creates an Open job.

Notice that the setup file created this way includes only the nominal process window condition. For the sorts of processes in which ILT is used, there are rarely fewer than three conditions. These need to be added in the text area of the PxOPC session.

Figure 3-9. Options Pane Under Default Button (pxOPC)

Note

In the following code, some lines wrap because of the page width. Because this is a verbatim copy, line continuation characters (\) were not added. If you copy the file for your own uses, you will need to add line continuations for the example to run.

```
# ************************************************************************ #
# ** This file is automatically generated by Calibre Workbench Litho Flow ** #
# ** Tool. It represents pxOPC setup file for doing inverse               ** #
# ** Lithography operations.                                              ** #
# ************************************************************************ #

# Calibre WORKbench version v2016.3_7.0007 x86-64 Linux Thu Jul 14 13:40:34 PDT
2016

# The directory(ies) having the input models.
modelpath .:models:/user/name/.calibrewb_workspace/models

# Loading the input models.
optical_model_load OPTICAL_1 opticalD0

# The mask(s) background.
background dark

# The mask(s) layers.
layer M1_1

tilemicrons 15
progress_meter on

# PXOPC options block "OPTIONS_BLOCK_1".
pxopc_options OPTIONS_BLOCK_1 {
  debug_level 0

  # The mask(s) background.
  background dark

  # The mask(s) layers.
  layer M1_1 correction clear mask 0

  # Smooth target layers.
  layer SMOOTH_M1_1 target clear mask 0

  # Process Window Conditions.
  pw_condition nominal optical OPTICAL_1 dose 1.0 aerial 0.3

  # The initialization layer.
  init M1_1

  # The jobs definitions section.
  job 1 open
  pw_select nominal
  iterations 8
  rate 1.0

  job 2 decorate
  pw_select nominal
  iterations 8
  scatter_belt 0.25

  job 3 correct
  pw_select nominal
  iterations 20
```

```
  job 4 refine
  pw_select nominal
  iterations 10

  job 5 lmrc
  pw_select nominal
  mrc_min_internal 0.015
  mrc_min_external 0.015
  mrc_min_edge 0.015

}

# The Curve target of the PXOPC operation.
setlayer SMOOTH_M1_1 = curve M1_1 order 6 cpdist 0.05 maxcp 4 maxdist 0.05
smooth 0.05 midpt 1 jog_tol 0.010 jog_adj 0.01

# PXOPC setlayer statements for the layer "M1_1".
setlayer PXOPC_M1_1_lmrc = pxopc M1_1 SMOOTH_M1_1 MAP M1_1 OPTIONS
OPTIONS_BLOCK_1
```

Calibre pxOPC Session Operations

A Calibre pxOPC session contains a few of the common GUI controls.

The **Cutline**, **FInfo**, **EPE**, and **PMap** operations in the toolbar of the main Calibre WORKbench window are not supported with the PxOPC session.

Table 3-5. Calibre pxOPC Session Operations

Operation	Control	Location	Description
See the *Calibre pxOPC User's and Reference Manual*	**PXOPC**	RET Flow Tool toolbar	Runs Calibre pxOPC
"Generating a Print Image" on page 105	**PI**	RET Flow Tool toolbar	Simulate OPC effects on geometries on opc layers that lie within a selected area.
Create a template setup file	**Default**	RET Flow Tool toolbar	Replaces the current contents of the editing area with a template using the settings from the **Options** area.

Calibre nmSRAF in the RET Flow Tool (cnsraf and mbsraf Sessions)

When you select nmSRAF as one of your sessions, you access the Calibre nmSRAF tool. This session uses a Calibre nmSRAF-based setup file and contains settings for simulation tools.

Configuring a Calibre nmSRAF Session

A Calibre nmSRAF session must be correctly loaded and configured before you can use it to perform SRAF treatment on your design.

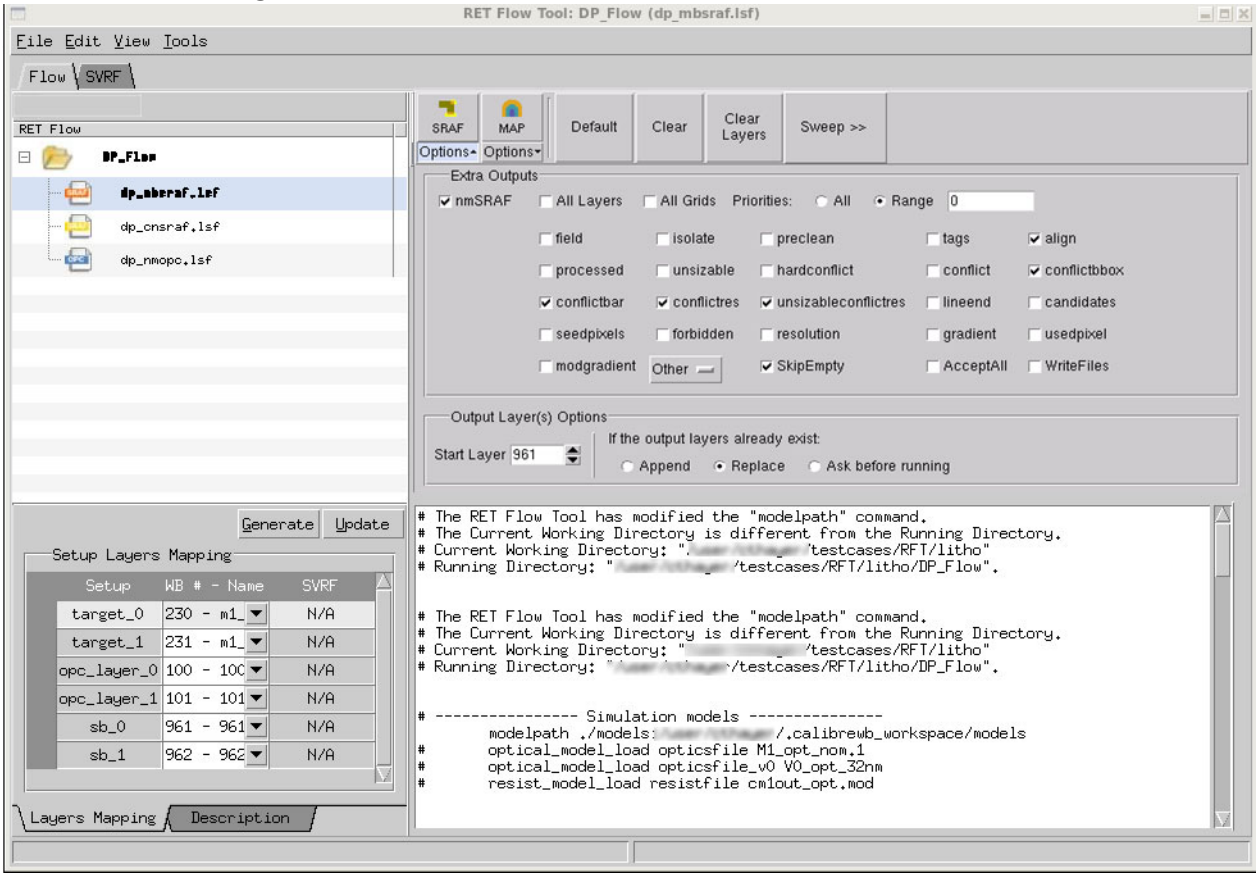
Prerequisites

- A known, working Calibre nmSRAF command set located in one of:
  - An SVRF file containing the Calibre nmSRAF command file inline, with operational cnsraf or mbsraf syntax
  - A transcript from a previous Calibre nmSRAF treatment
- The relevant design layout loaded into Calibre WORKbench

Procedure

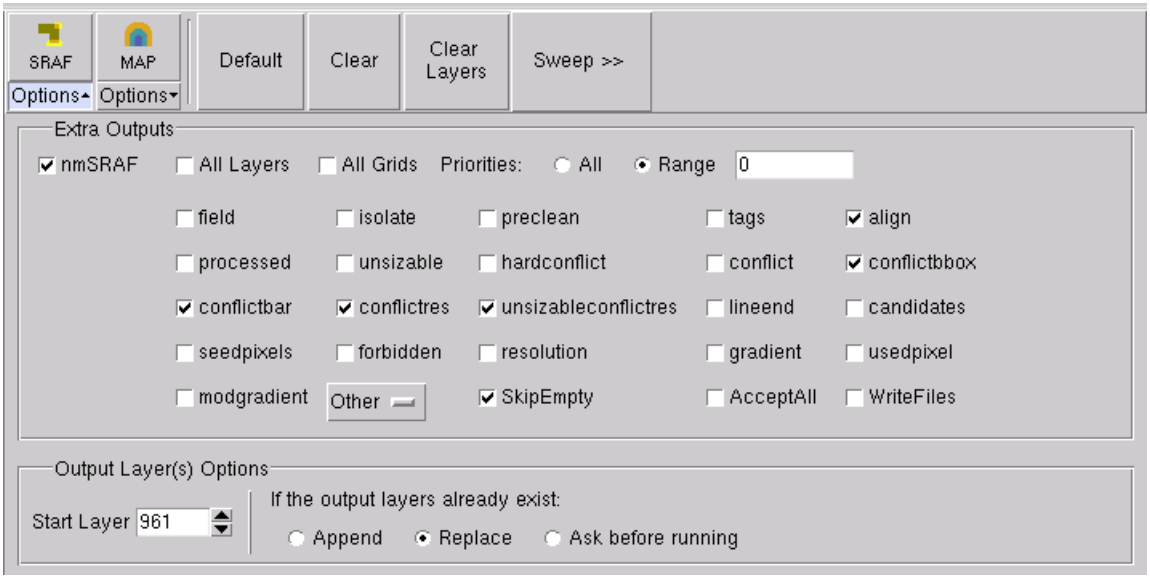
1. Follow the steps in "Creating a Session" on page 38 to create a Calibre nmSRAF session.
   - If you used one of the methods that loaded an existing file, your RET Flow Tool should look similar to Figure 3-10.
   - If the session text entry pane is empty, you can:
     - Use the **Copy** and **Paste** options from the **Edit** menu.
     - Open the right-mouse context menu inside the text pane to copy directly from a rule file.
     - Right-click the session in the RET Flow tree and choose **Open Setup**.

Figure 3-10. RET Flow Tool, Calibre nmSRAF Session



2. Assign layers in the Setup Layers Mapping list from the active layout list to the listed layer outputs.
   - If you leave a layer set as Dummy Layer, it is considered to be hidden. Only layers assigned from the list are considered visible.
   - If layers are incorrectly mapped, running the tool may give unexpected results.
3. Open the SRAF Options panel and select checkboxes as needed.

Figure 3-11. SRAF Options



4. Test Calibre nmSRAF by clicking the **SRAF** button.

Results

Calibre nmSRAF runs the script, and one or more result layers are added to the active layout in the viewer main window.

Calibre nmSRAF Session Operations

A Calibre nmSRAF session contains GUI controls that run tools using the conditions defined in the session.

The operations and where their controls are located are summarized in Table 3-6.

Table 3-6. Calibre nmSRAF Session Operations

Operation	Control	Location	Description
See the *Calibre nmSRAF User's and Reference Manual*	**SRAF**	RET Flow Tool toolbar	Run Calibre nmSRAF.
"Creating an Aerial Image Contour Map" on page 101	**MAP**	RET Flow Tool toolbar	Create a contour map of the light intensity at the surface of the resist.
"Cutline Operations" on page 107	**Cutline**	Calibre WORKbench toolbar	Compare one or more simulations along cutlines.
"Running Tests on Multiple Values With the Sweeping Tool" on page 131	**Sweep**	RET Flow Tool toolbar	Examine multiple values for specified command parameters.

Running Calibre nmSRAF Within the Calibre RET Flow Tool

This procedure explores the steps for executing Calibre nmSRAF for dimensional- and model-based SRAF generation (cnsraf and mbsraf, respectively) on a double-patterning design.

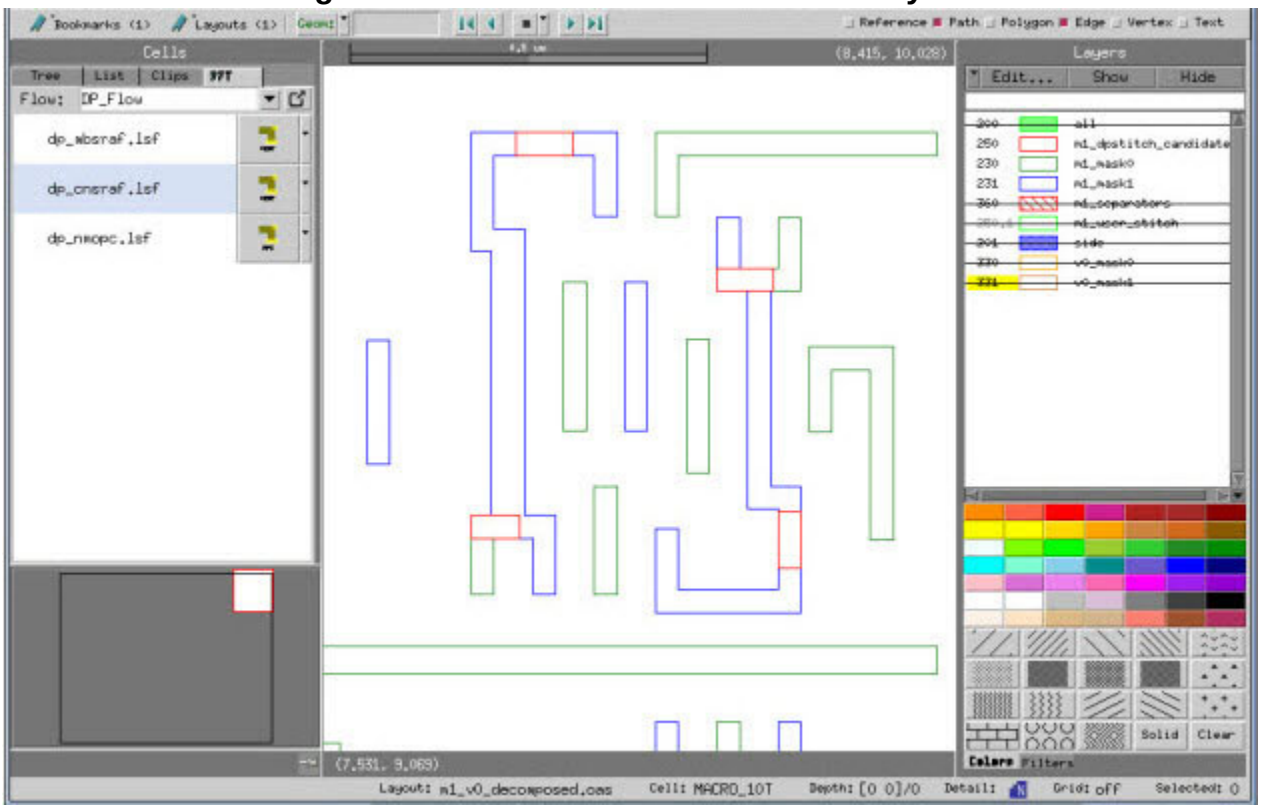
Prerequisites

- You must have previously loaded a Calibre nmSRAF session.
- You must have an existing layout.

Procedure

1. Zoom in to the area of the layout you want to run Calibre nmSRAF on.

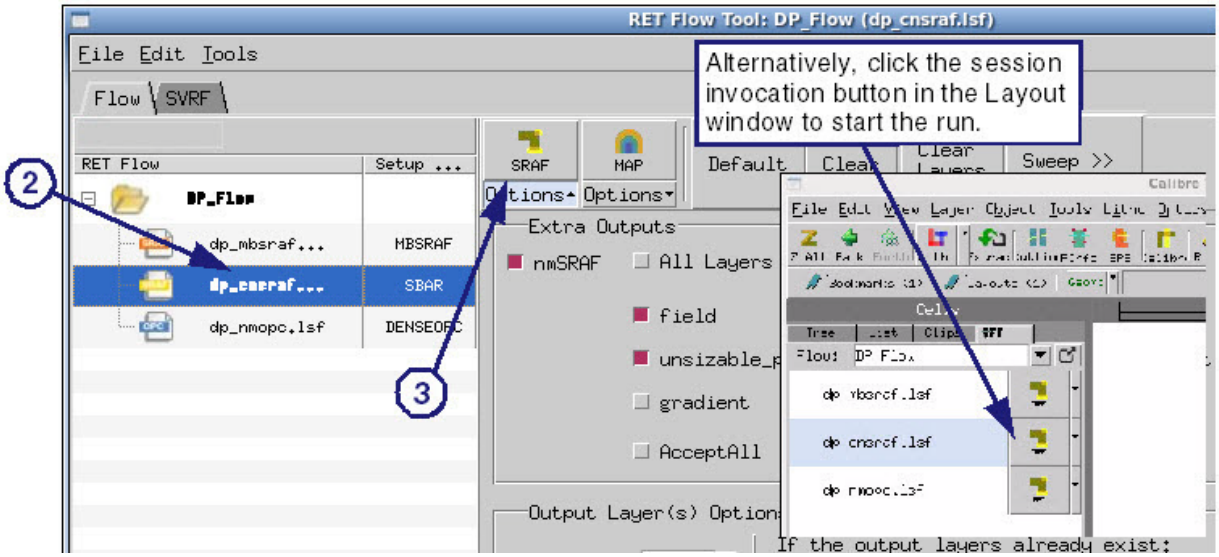
Figure 3-12. Zoom in to an Area of the Layout



2. In the RET Flow Tool, select the session you want to run.
3. Press the **SRAF** button (or alternatively click the associated button of the session you want to run in the Layout window) as demonstrated in the following figure.

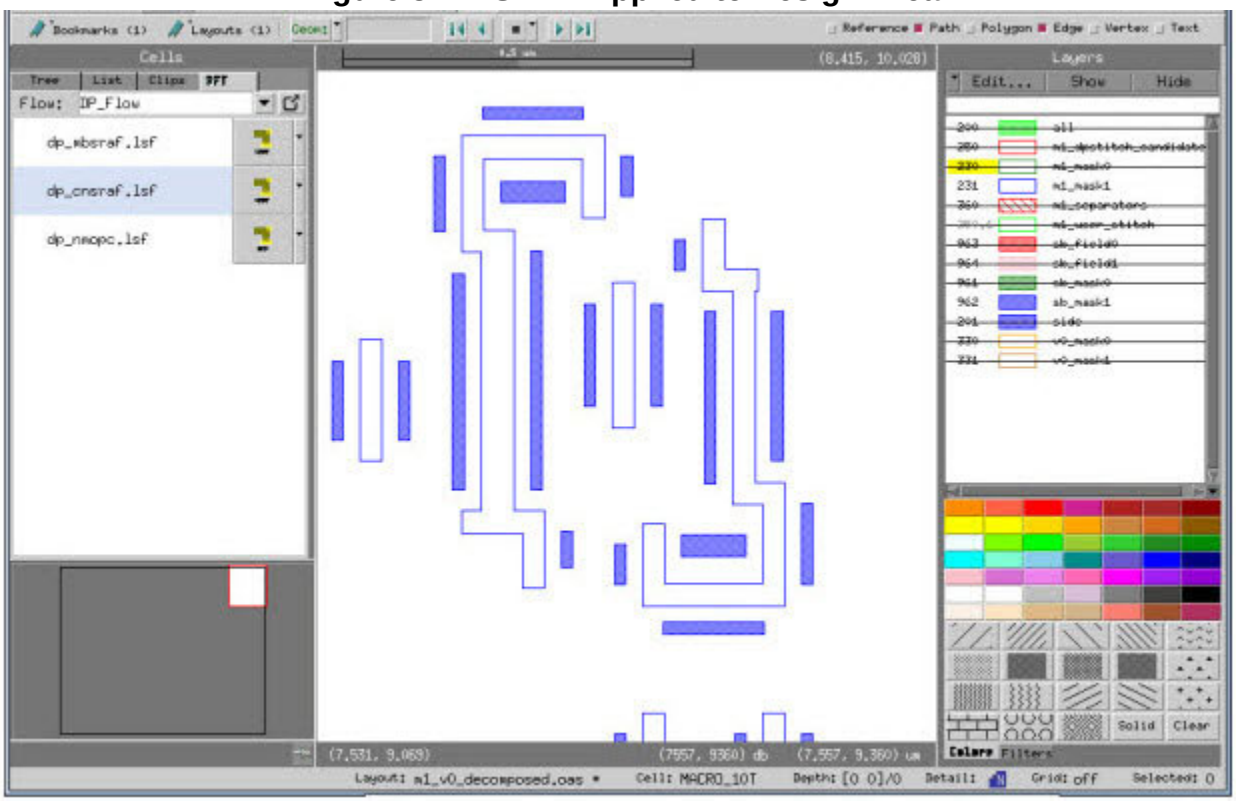
   The session begins running.

Figure 3-13. Start a Calibre nmSRAF Session



When the session finishes, SRAFs are placed in the area of the design you zoomed into.

Figure 3-14. SRAF Applied to Design Area



Results

You have executed Calibre nmSRAF within the RET Flow Tool and generated SRAFs on an area of the layout.

Generating SRAFs with cnsraf and mbsraf

Generating SRAFs with cnsraf and mbsraf can be done with the RET Flow Tool.

The cnsraf Extra Outputs section of the cnsraf session provides different types of layer information to output various iterations of SRAF generation, including gradient maps, debug layers, and all SRAFs.

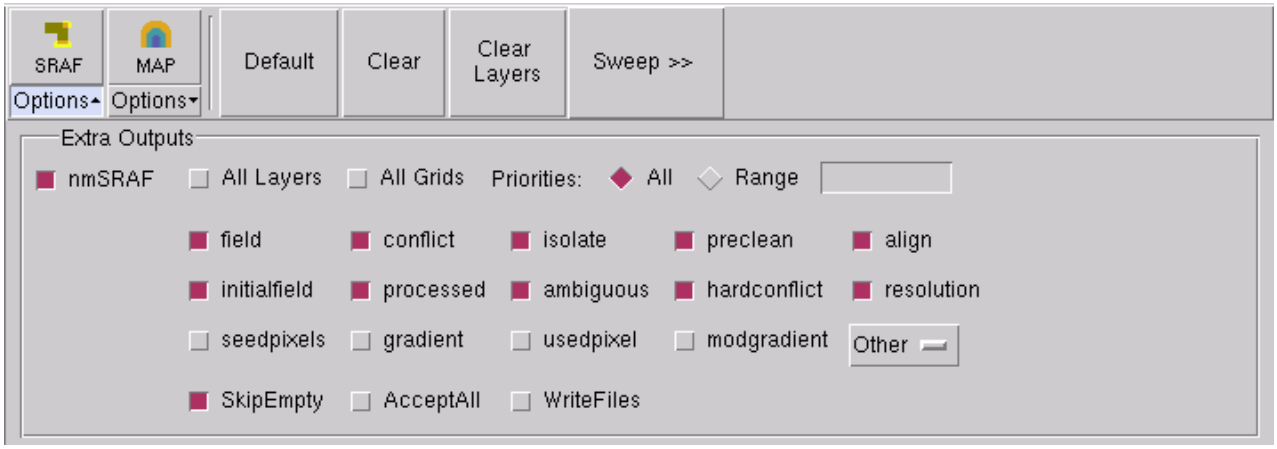
Only the viewable region in the layout window is simulated and written to the selected layers.

Prerequisites

- You must have loaded the flow containing sessions for cnsraf and mbsraf.

Procedure

1. Select the cnsraf session.
2. In the cnsraf session, under the SRAF **Options** button, select the **nmSRAF** checkbox.

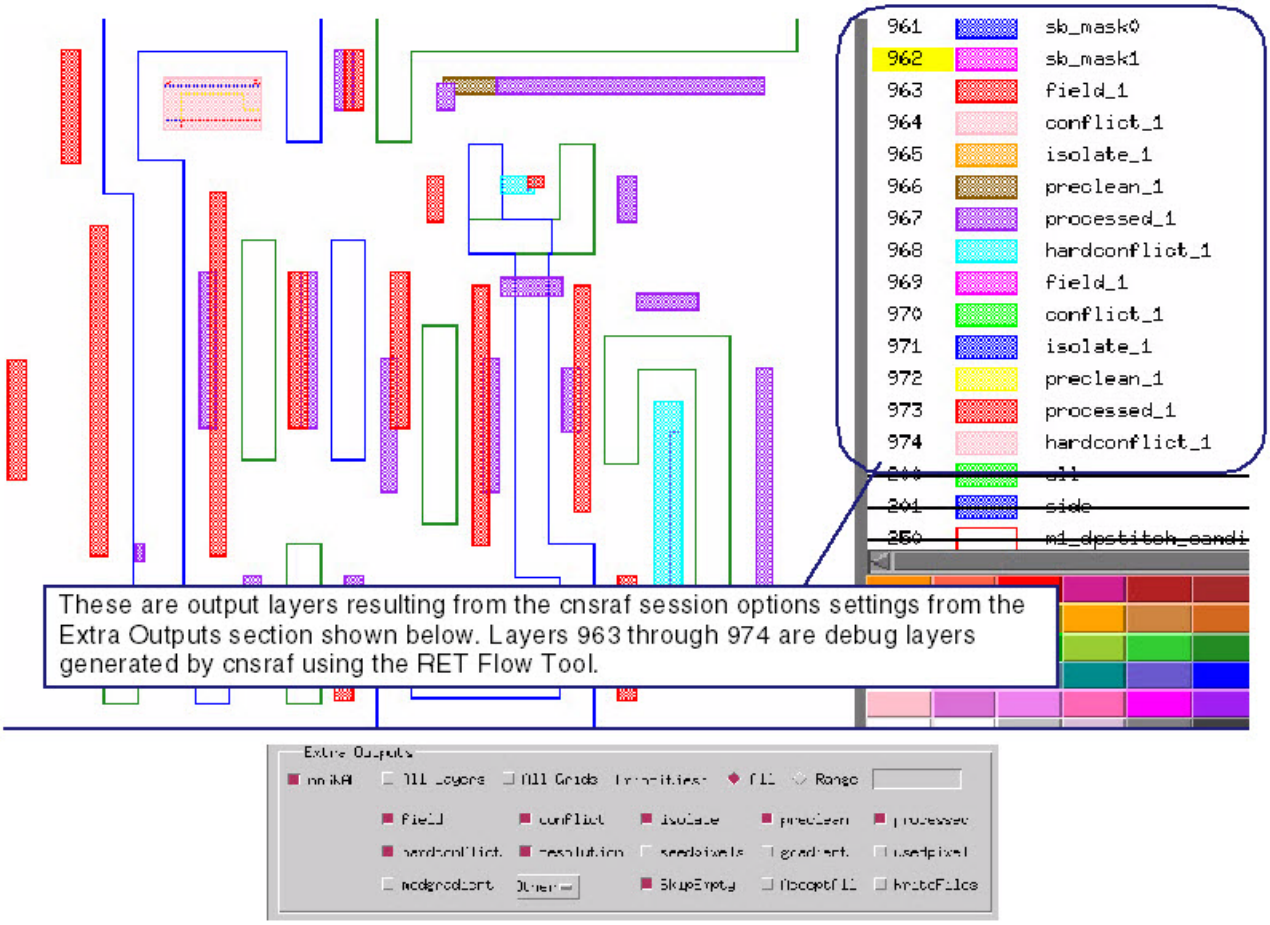


   This picks various default layers.

3. Click the **SRAF** button.

   Calibre cnSRAF runs. The SRAFs appear in the layout window.

Figure 3-15. Output SRAF and Layers from cnsraf

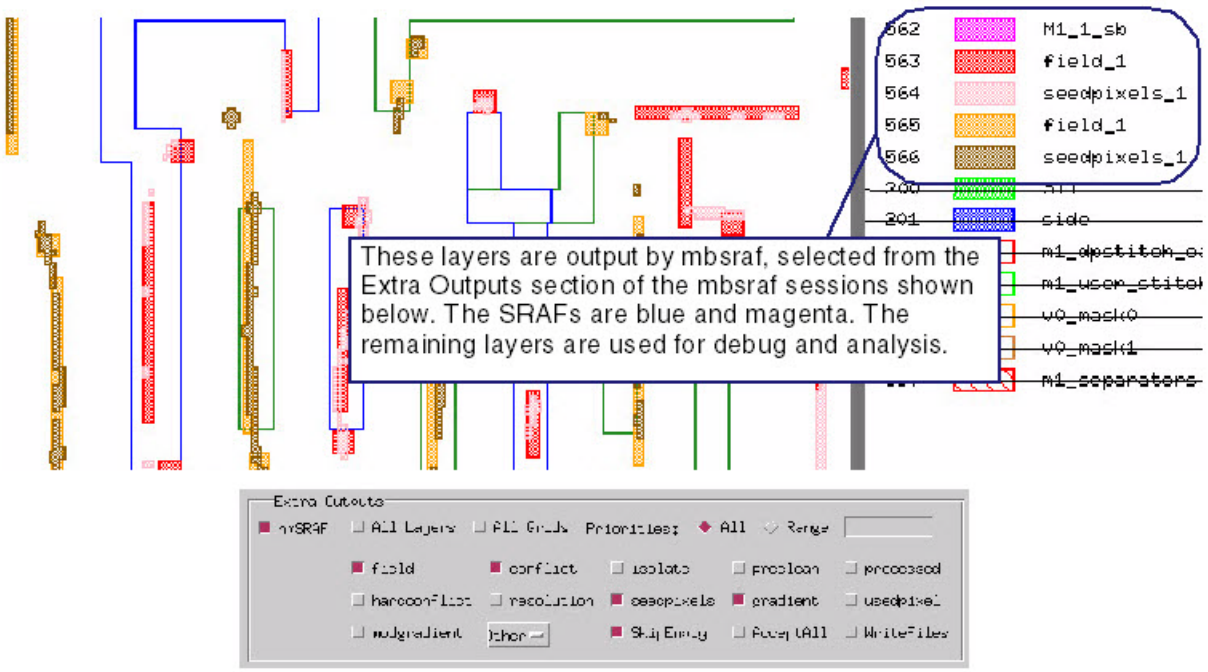


4. Select the MBSRAF session.
5. In the MBSRAF session, select the **nmSRAF** button.

   This picks various default layers and maps.
6. Click the **SRAF** button.

   Calibre mbSRAF runs. The SRAFs appear in the layout window.

Figure 3-16. Output SRAF and Layers from mbsraf



Results

Calibre generates model-based SRAFs on output layers as well as selected gradient maps for a double-patterning design.

Displaying Gradient Maps for SRAFs

This section guides you through the steps to display the gradient maps superimposed on your layout using Calibre WORKbench.

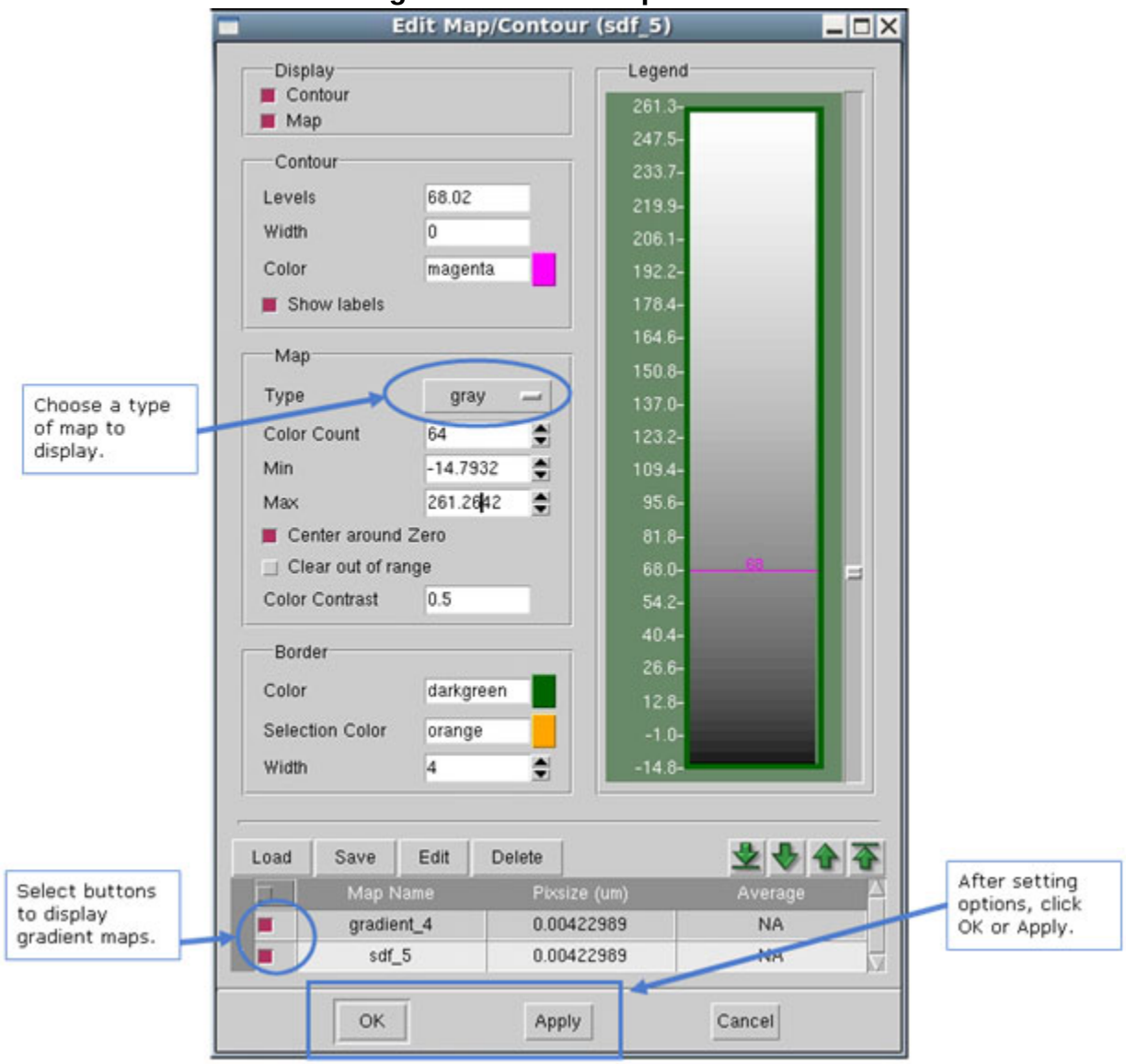
Prerequisites

- You must have previously performed SRAF, gradient, and sdf map generation.

Procedure

1. In the Calibre WORKbench layout window double-click on the gradient map in the layout window (alternatively select **Edit > Maps/Contours**). This invokes the Edit Map/Contour dialog box shown in Figure 3-17.

Figure 3-17. Edit Map/Contour



2. Turn off the **gradient** map and leave the **sdf** map on.
3. Click **Apply** in the Edit Map/Contour dialog box. The display changes in Calibre WORKbench to display the sdf map overlay only.
4. Turn the **gradient** map back on for the next procedure.
5. Click **Apply** or **OK** in the Edit Map/Contour dialog box. The display resets in Calibre WORKbench to re-display the gradient map.

Results

You have modified the display of the layout's gradient and sdf maps.

Drawing Intensity Cutlines

This section guides you through the steps to draw a cutline and display the resulting aerial intensity plot using Calibre WORKbench.

Prerequisites

- You must generate gradient maps using mbsraf with the RET Flow Tool as described in "Displaying Gradient Maps for SRAFs" on page 64.

Procedure

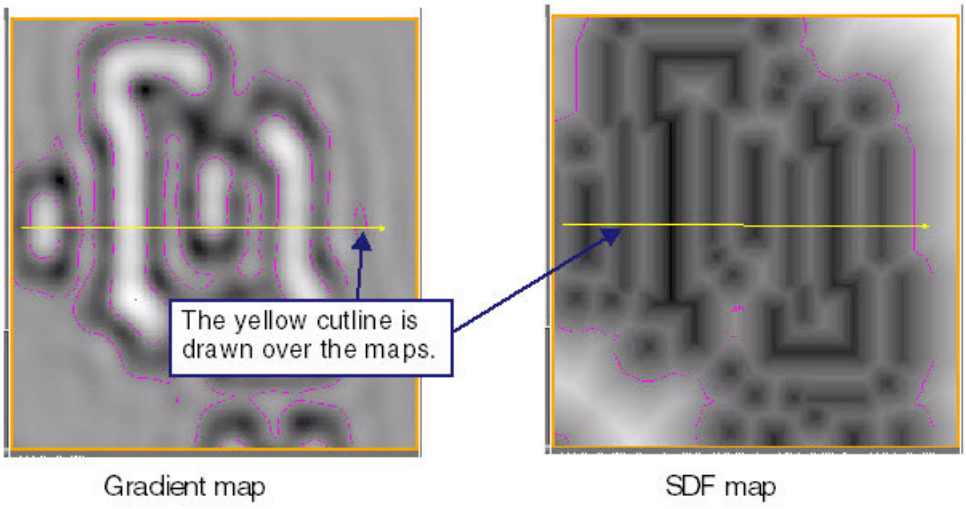
1. Select the map by clicking on it.

   The map border turns orange.

2. In the WORKbench icon bar, select **Cutline**.

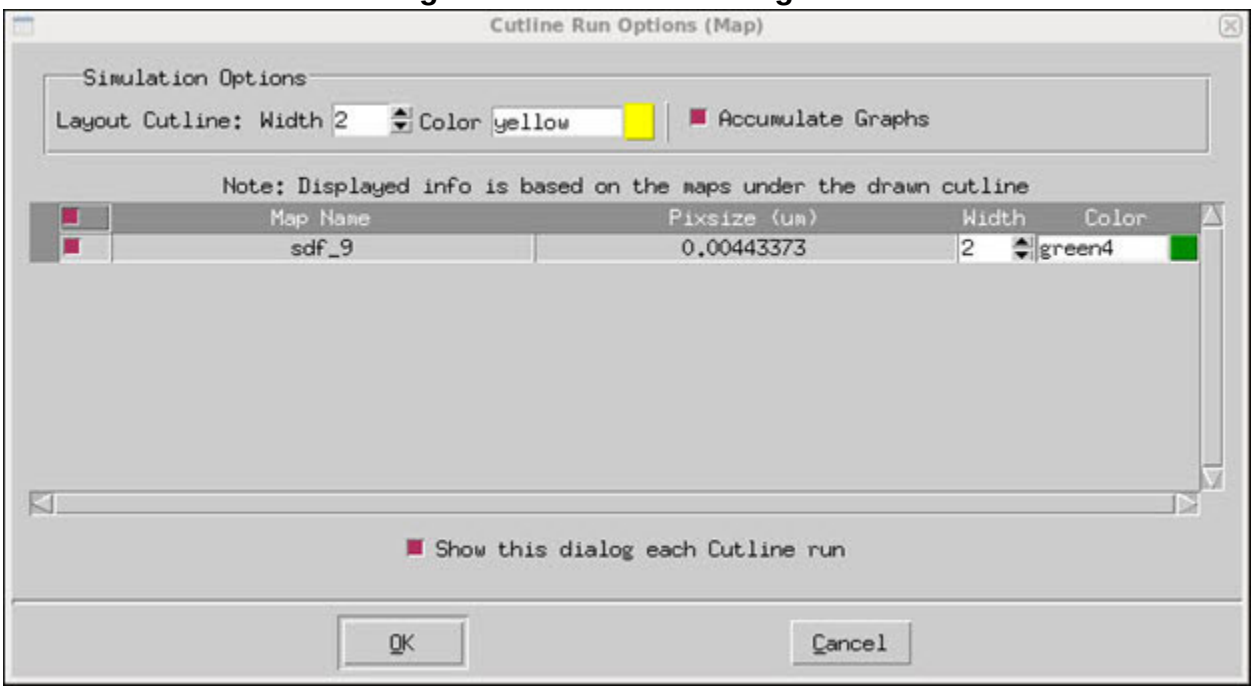
3. Draw a cutline through the map by clicking once for the cutline start and once for the cutline end. This is shown in Figure 3-18 for the gradient and SDF maps.

Figure 3-18. Drawing a Cutline Through Maps



After clicking the cutline end, the Cutline Run Options dialog box appears.

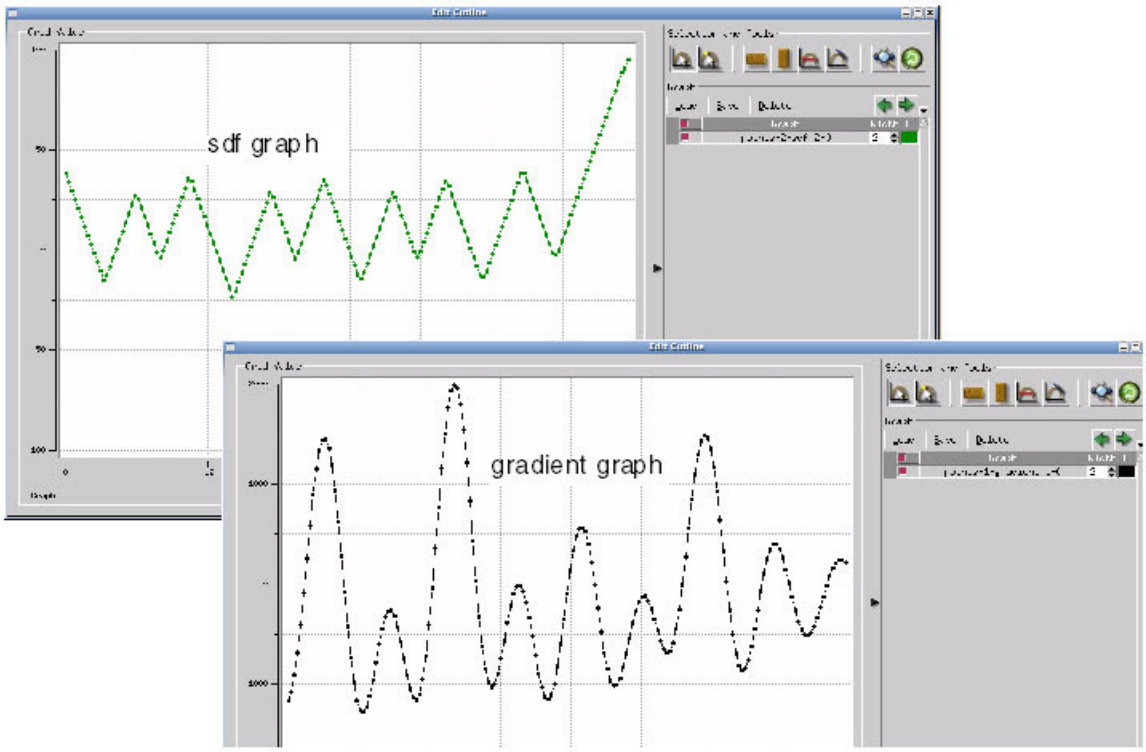
Figure 3-19. Cutline Dialog Box



4. Select the graph to display and click **OK** in the Cutline Run Options dialog box.

   The Edit Cutline dialog box appears displaying the cutline graph. The graphs show the cutline distances on the X-axis and signed distance function and aerial intensity, respectively, on the Y-axis.

Figure 3-20. Aerial Intensity Graphs



Results

You have drawn cutlines over sdf and gradient maps and displayed their respective graphs for a portion of a double-patterning layout.

Calibre nmBias in the RET Flow Tool (nmBIAS Session)

When you select nmBIAS as one of your sessions, you access the Calibre nmBias tool. This session uses a Calibre nmBias-based setup file and contains settings for simulation tools.

Configuring a Calibre nmBias Session

A Calibre nmBias session must be correctly loaded and configured before you can use it to perform biasing on your design.

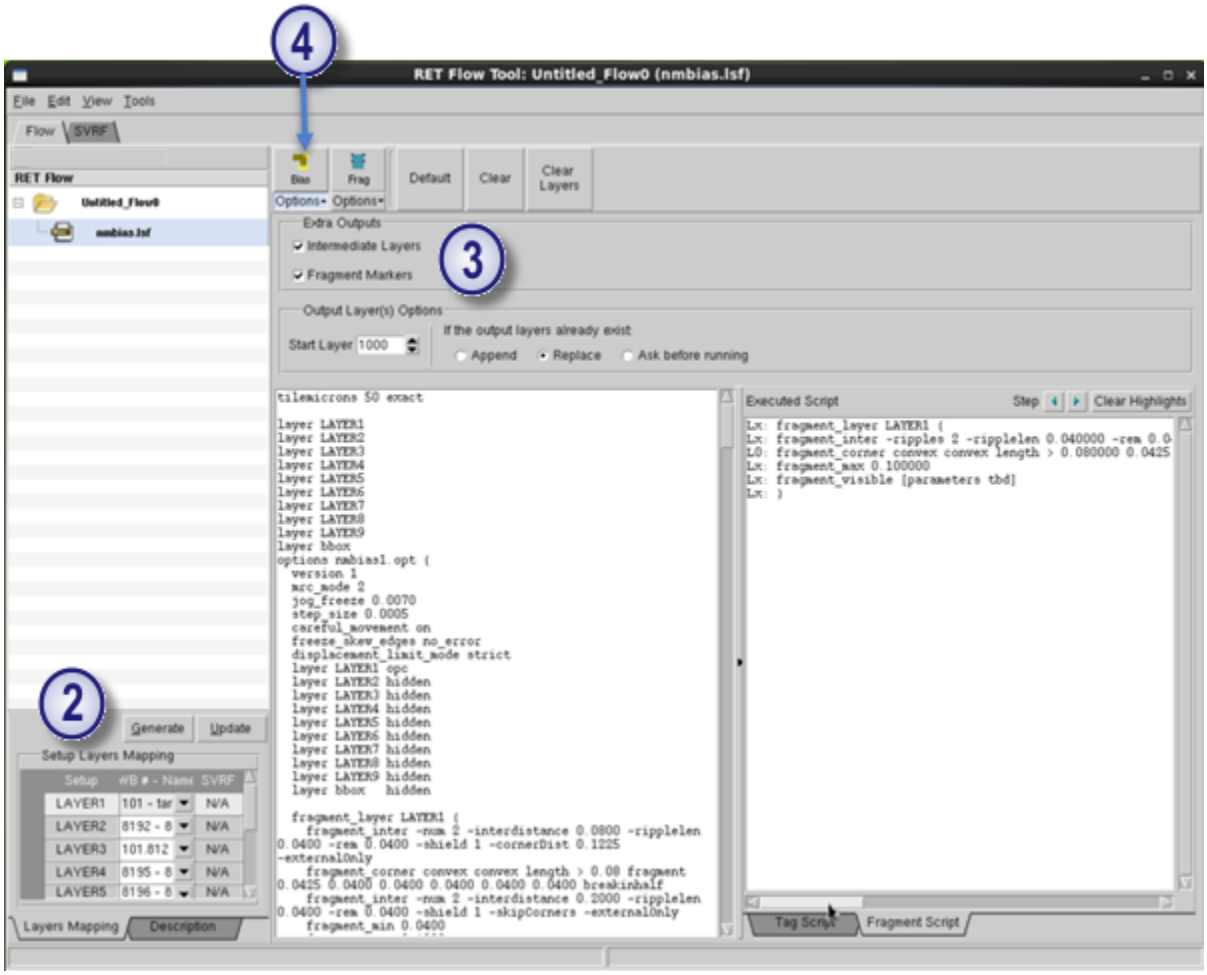
Prerequisites

- A known, working Calibre nmBias command set located in an SVRF file with an operational BIASRULE Tcl scripting command
- The relevant design layout loaded into Calibre WORKbench

Procedure

1. Follow the steps in "Creating a Session" on page 38 to create a Calibre nmBias session.
   - If you used one of the methods that loaded an existing file, skip to the next step.
   - If the session text entry pane is empty:
     - Use **Copy** and **Paste** from the **Edit** menu.
     - Open the right-mouse context menu inside the session text entry pane to copy directly from a rule file.
     - Right-click the session in the RET Flow tree and choose **Open Setup**.

Figure 3-21. RET Flow Tool, Calibre nmBias Session



2. Assign layers in the Setup Layers Mapping list from the active layout list.

   - If you leave a layer set as Dummy Layer, it is considered to be hidden. Only layers assigned from the list are considered visible.
   - If layers are incorrectly mapped, running the tool may give unexpected results.

3. Click the **Options** button under the **Bias** button and select **Fragment Markers** to enable fragment information selection from the layout.

   a. (Optional) Click the **Intermediate Layers** button to generate additional debug information.

      Using this option enables you to visualize the intermediate outputs of each of the BIASRULE and FRAGMENT_SMOOTHING statements. This may be useful while debugging your recipe.

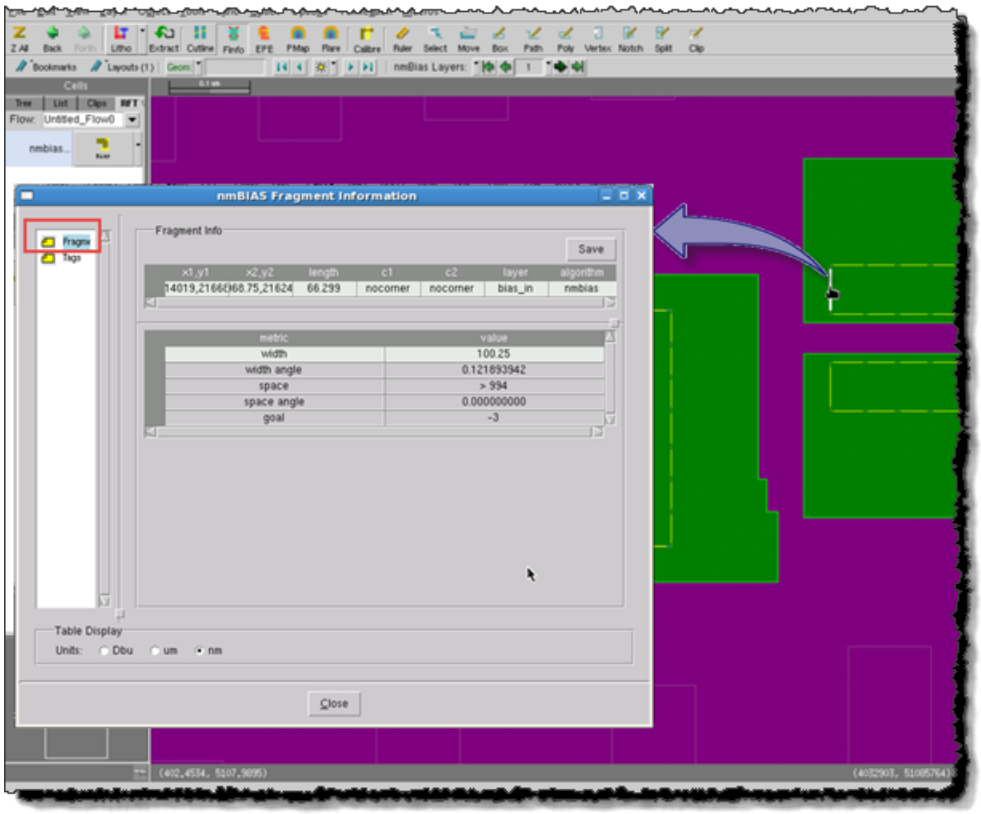
4. Click the **Bias** button.

   Biasing is performed only on fragments displayed within the layout window.

5. In the viewer toolbar, click **FInfo**. The cursor changes to a hand; click a fragment to open the Fragment Information window.

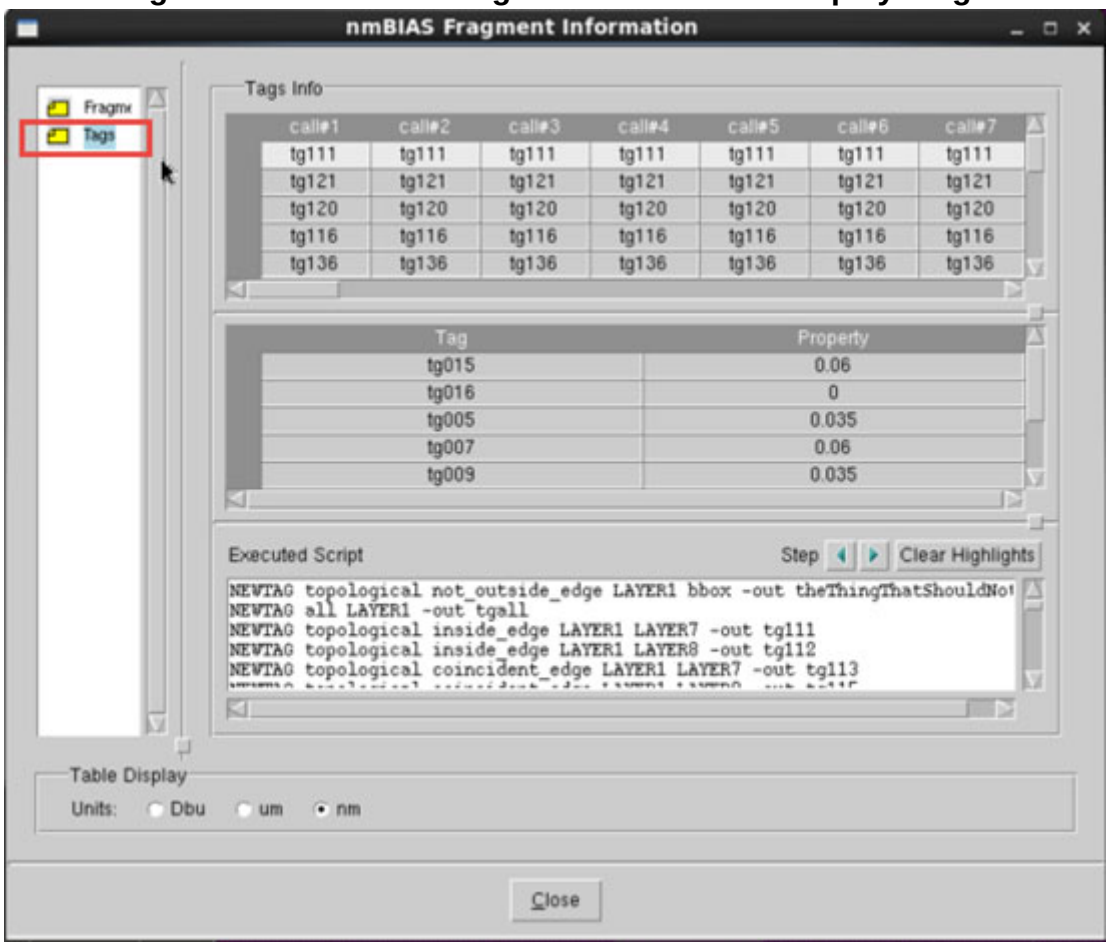
   The fragment is highlighted and the Calibre nmBIAS Fragment Information window is displayed.

Figure 3-22. nmBIAS Fragment Information Display: Fragment



Further tag information is available by selecting **Tags** in the upper left of the nmBIAS Fragment Information window. See the red box in Figure 3-23.

Figure 3-23. nmBIAS Fragment Information Display: Tags



Selecting an entry in the **Tags Info** table highlights the executed portion of the script.

Note

Annotated tags, as defined by the -annotated or -aout command options, can also be displayed in the Calibre nmBIAS Fragment Information window.

Results

Calibre nmBias runs the biasing script, outputs biased fragments, and adds associated result layers to the active layout in the layout window. This can include additional intermediate BIASRULE and FRAGMENT_SMOOTHING layers for detailed debug.

Calibre nmBias Session Operations

A Calibre nmBias session contains a number of GUI controls that run tools using the conditions defined in the session.

The operations and where their controls are located are summarized in Table 3-7.

Table 3-7. Calibre nmBias Session Operations

Operation	Control	Location	Description
See the *Calibre Rule-Based OPC User's and Reference Manual*	**Bias**	RET Flow Tool toolbar	Run Calibre nmBias.
"Fragment Generation" on page 116	**Frag**	RET Flow Tool toolbar	Calculate and show how edges are fragmented for OPC on the layout. Adjust the display properties of fragmentation points, mapping and measurement locations, and sites in a user-selected area.

Calibre OPCpro in the RET Flow Tool (OPCpro Session)

When you select an OPCpro session, you access the Calibre OPCpro tool. This session uses a Calibre OPCpro-based setup file and contains settings for simulation tools.

Configuring a Calibre OPCpro Session

A Calibre OPCpro session must be correctly loaded and configured before you can use it to perform OPC. Calibre OPCpro sessions show in the RET Flow Tool pane as session type OPC.

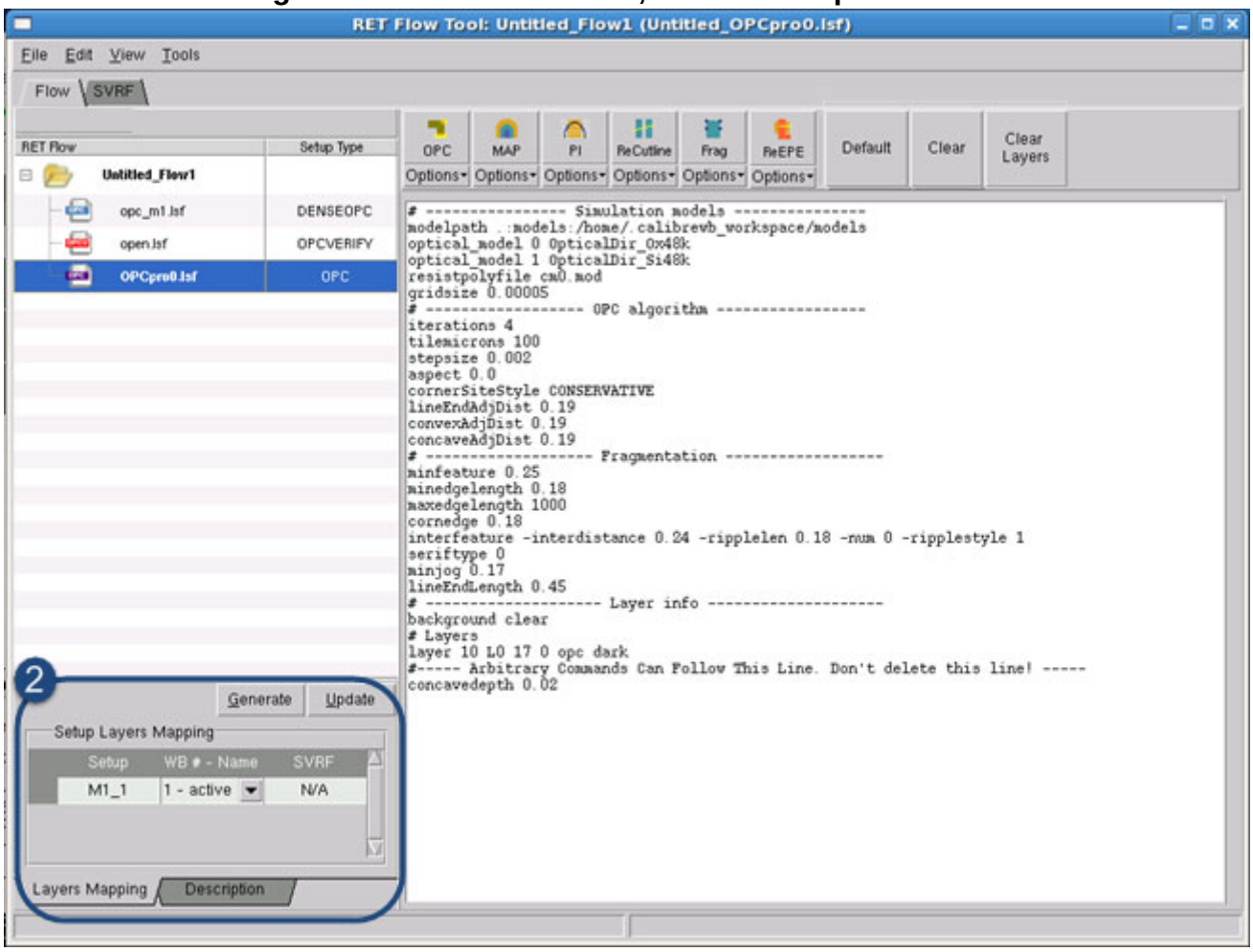
Prerequisites

- A known working Calibre OPCpro rule file in one of the following formats:
  - An SVRF file containing the Calibre OPCpro command file inline
  - A transcript from a Calibre OPCpro run
- The relevant design layout loaded into Calibre WORKbench

Procedure

1. Follow the steps in "Creating a Session" to create a Calibre OPCpro session.
   - If you used one of the methods that loaded an existing file, your RET Flow Tool should look similar to Figure 3-24.
   - If the session text entry pane is empty, you can:
     - Use the **Copy** and **Paste** options from the **Edit** menu .
     - Open the right-mouse context menu inside the text pane to copy directly from a Calibre OPCpro rule file.
     - Right-click the OPCpro session in the RET Flow tree and choose **Open Setup**.

Figure 3-24. RET Flow Tool, Calibre OPCpro Session



2. Assign layers in the Setup Layers Mapping list from the WB # - Name list to the listed layer outputs from the Calibre OPCpro file.

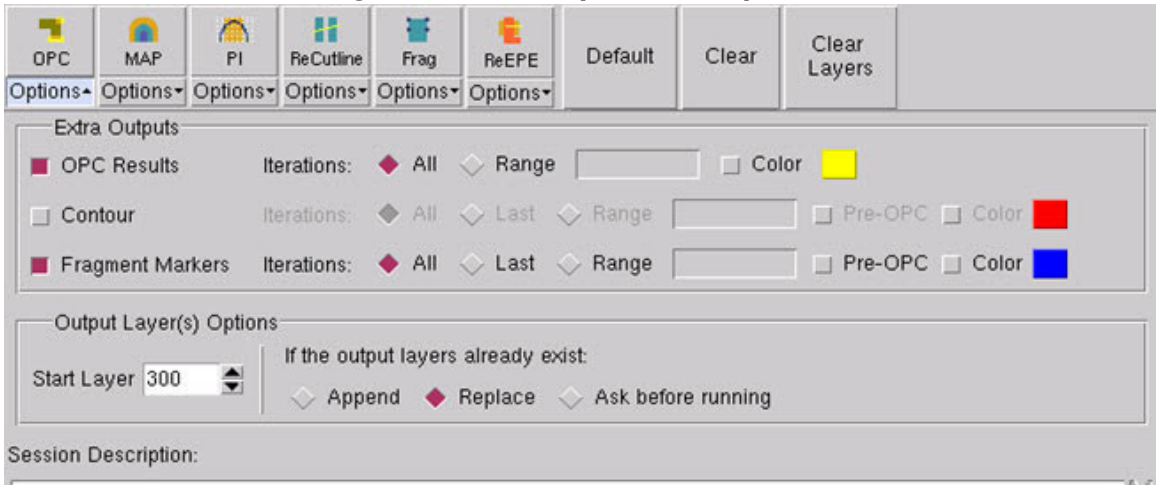
   - If you leave a layer set as Dummy Layer, it is considered to be hidden. Only layers assigned from the list are considered visible.
   - If layers are incorrectly mapped, running Calibre OPCpro may give unexpected results.

3. Under the **OPC** button, click **Options** to open the Output Options panel. Select checkboxes as needed.

   - **OPC Results** — Outputs the resulting OPC edge movements for the specified iterations. This is useful to track how edges move during OPC. It generates one consecutive output layer per iteration.
   - **Contour** — Outputs a contour based on the EPE measurements for the specified iterations. It generates one output layer per iteration. Selecting Pre-OPC includes pre-OPC contour information in the output.

- **Fragment Markers** — Outputs fragmentation information such as final EPE, position, length, type, feedback, displacement, target (goal) position, and the metric used to measure the EPE. It generates one consecutive output layer per iteration. This is required if you want to view fragmentation information in the post-OPC analysis phase. Selecting **Pre-OPC** includes pre-OPC fragment information in the output.
- **Color** — Changes the color of the output type.

Figure 3-25. OPCpro OPC Options



4. Test Calibre OPCpro by clicking the **OPC** button.

Results

Calibre OPCpro runs the script, and one or more OPC result layers are added to the active layout in the viewer main window.

Calibre OPCpro Session Operations

A Calibre OPCpro session contains a number of GUI controls that run tools using the conditions in defined in the session.

The operations and where their controls are located are summarized in Table 3-8.

Table 3-8. Calibre OPCpro Session Operations

Operation	Control	Location	Description
See the *Calibre OPCpro User's and Reference Manual*	**OPC**	RET Flow Tool toolbar	Run Calibre OPCpro.
"Creating an Aerial Image Contour Map" on page 101	**MAP**	RET Flow Tool toolbar	Create a contour map of the light intensity at the surface of the resist.

Table 3-8. Calibre OPCpro Session Operations (cont.)

Operation	Control	Location	Description
"Generating a Print Image" on page 105	**PI**	RET Flow Tool toolbar	Simulate OPC effects on geometries on opc layers that lie partially or fully within a selected area.
"Cutline Operations" on page 107	**Cutline** on the Calibre WORKbench icon bar	Calibre WORKbench toolbar	Compare one or more simulations along cutlines.
"Setting Cutline Properties" on page 108	**Re-Cutline**	RET Flow Tool toolbar	Change the display properties of cutlines.
"Fragment Generation" on page 116	**Frag**	RET Flow Tool toolbar	Calculate and show how edges are fragmented for OPC on the layout. Adjust the display properties of fragmentation points, mapping and measurement locations, and sites in a user-selected area.
"Measuring Edge Placement Errors" on page 77	**EPE**	Calibre WORKbench toolbar	Displays intensity measurements for a control site on an edge. **Tip:** EPE is unique to Calibre OPCpro mode.
	ReEPE	RET Flow Tool toolbar	Changes the display properties of EPE points.

Measuring Edge Placement Errors

The EPE tool measures edge placement errors by placing an OPCpro-style control site placed on the edge of a polygon.

Restrictions and Limitations

This tool is only available in Calibre OPCpro mode.

Prerequisites

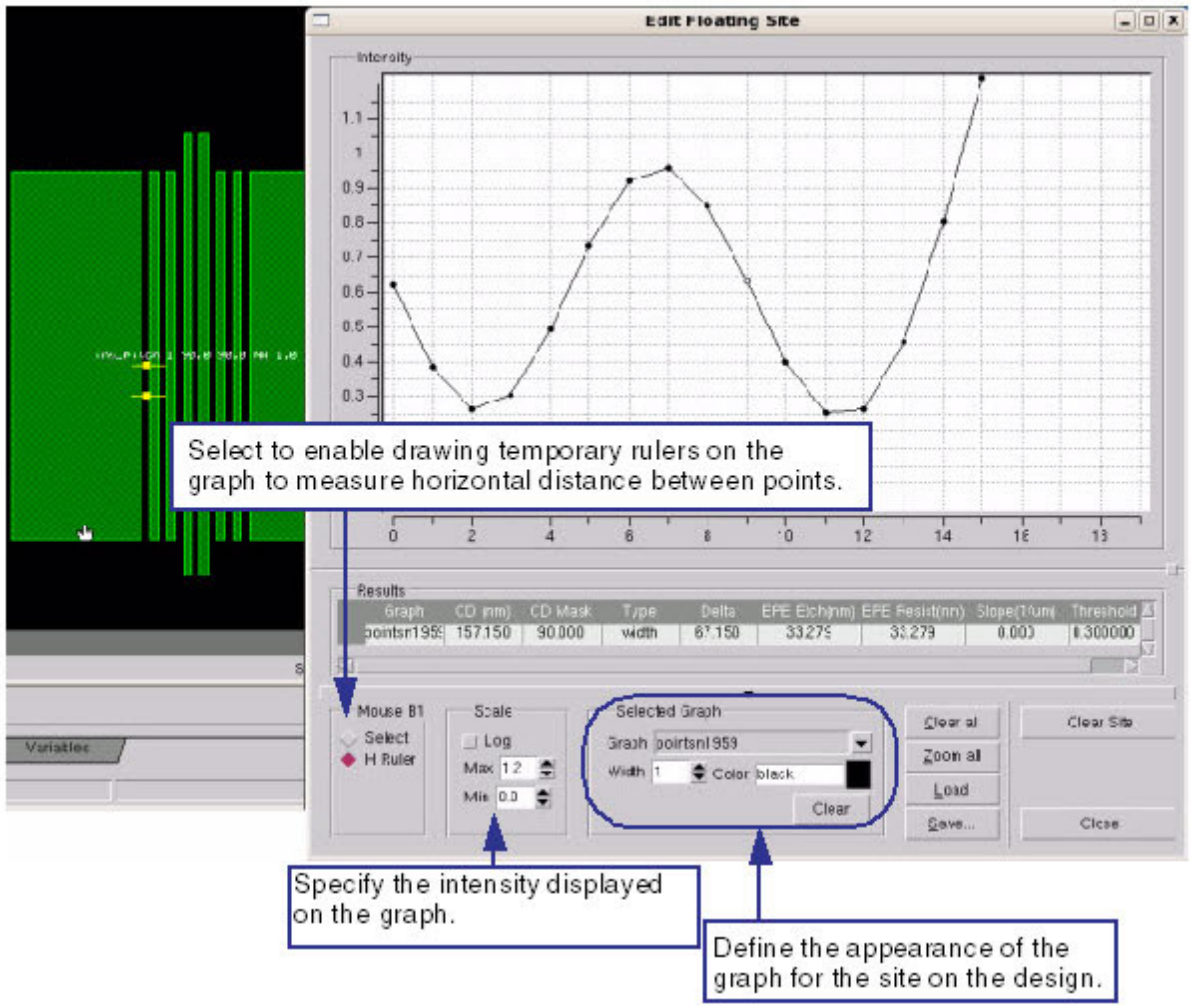
- A flow created in the RET Flow Tool, containing a properly configured OPCpro session. If you attempt to use the EPE tool before setting up an OPCpro session the viewer gives an error.

Procedure

1. Perform the steps in "Configuring a Calibre OPCpro Session" on page 74 to load the Calibre RET Flow Tool with a working Calibre OPCpro setup file.
2. In the Calibre WORKbench main window, click **EPE** to activate the tool, then click on an edge in the layout.

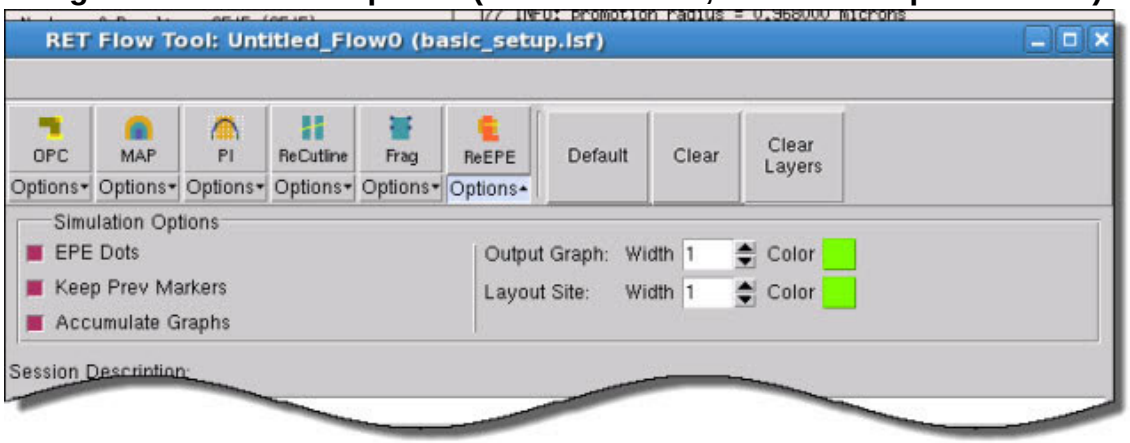
   The nearest edge to your mouse click displays the site used for that fragment, and the Edit Floating Site window appears.

Figure 3-26. Edit Floating Site Window (EPE Tool)



3. (Optional) Adjust the settings in the ReEPE Options pane in the RET Flow Tool.

Figure 3-27. ReEPE Options (RET Flow Tool, Calibre OPCpro Session)



Calibre nmMPC in the RET Flow Tool (nmMPC Session)

When you select nmMPC as one of your sessions, you access the Calibre nmMPC tool for mask-based corrections. This session uses a Calibre nmMPC-based setup file and contains settings for simulation tools.

Configuring a Calibre nmMPC Run

You must properly set up a Calibre nmMPC run in order to produce corrected results for your mask layout.

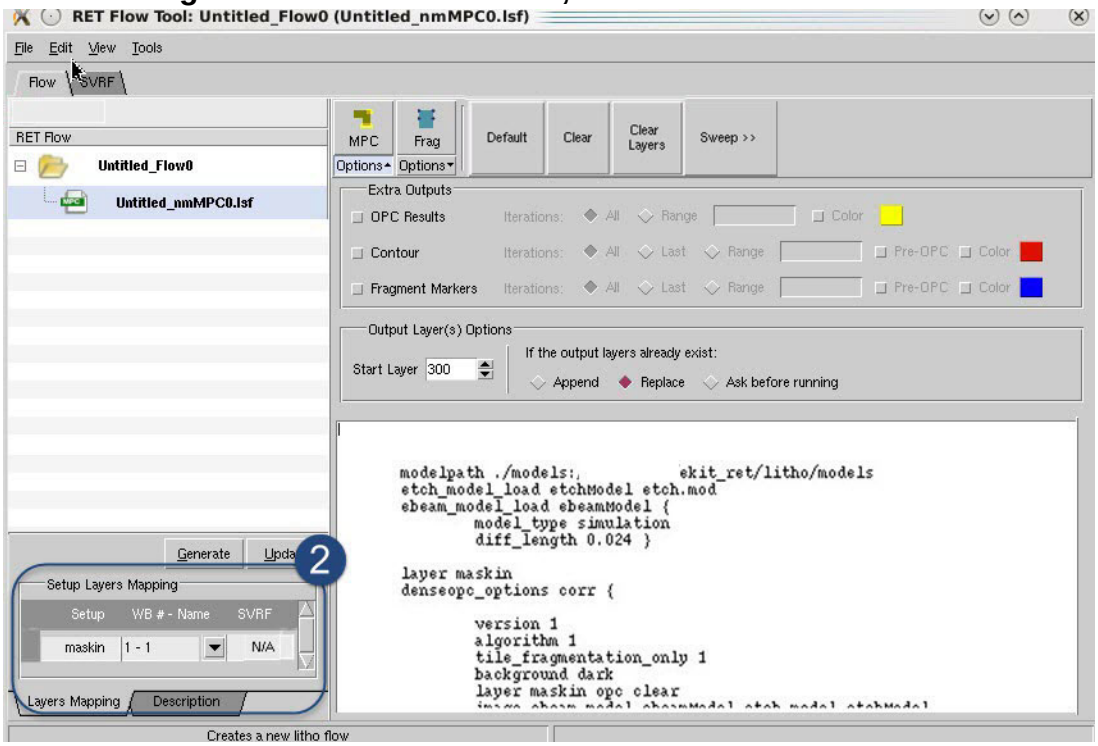
Prerequisites

- A valid e-beam model
- A valid etch model
- A mask layout loaded in the viewer main window
- A known working Calibre nmMPC SVRF file

Procedure

1. In the RET Flow Tool, select a session of type “nmMPC” or add a Calibre nmMPC session. Use the **File > Extract From SVRF** option to import the Calibre nmMPC file to the text pane of the window.
2. In the Setup Layers Mapping section, assign the layers in the layout to the layers defined in the Calibre nmMPC session.

Figure 3-28. RET Flow Tool, Calibre nmMPC Session



3. Click **Options** under the **MPC** button to expose the Extra Outputs pane.
4. In the Extra Outputs pane, set output options such as MPC iterations, contours, edge movement, and fragment information for the MPC run.
    - **OPC Results** — Outputs the resulting corrected edge movements for the specified iterations. This is useful to track how edges move during correction. It generates one consecutive output layer per iteration.
    - **Contour** — Outputs a contour based on the EPE measurements for the specified iterations. It generates one consecutive output layer per iteration. Selecting Pre-OPC includes pre-MPC contour information in the output.
    - **Fragment Markers** — Outputs fragmentation information such as final EPE, position, length, type, feedback, displacement, target (goal) position, and the metric used to measure the EPE. It generates one consecutive output layer per iteration. This is required if you want to view fragmentation information in the post-MPC analysis phase. Selecting Pre-OPC includes pre-correction fragment information in the output.
    - **Color** — Changes the color of the output type.
5. In the Output Layers Options section, set the initial output layer number. The output layer numbers increase by one for each new output layer per iteration produced.
6. Click the **MPC** button to run MPC.

Results

Output layers are generated depending on the settings in the Extra Outputs pane prior to the run. The output can be analyzed using a number of different tools and operations.

Calibre nmMPC Operations

Calibre WORKbench supports a number of core Calibre nmMPC operations. For instance, there are different types of analysis that can be performed on the results of an MPC run, as well as options to adjust how data is displayed.

The operations and where their controls are located are summarized in the following table.

Table 3-9. Calibre nmMPC Operations

Operation	Control	Location	Description
Mask Process Correction	**MPC**	Ret Flow Tool toolbar	See “Configuring a Calibre nmMPC Run” on page 80.
“Fragment Generation” on page 116	**Frag**	RET Flow Tool toolbar	Calculate and show how edges are fragmented for MPC on the layout. Adjust the display properties of fragmentation points, mapping and measurement locations, and sites in a user-selected area.

Calibre MPCverify in the RET Flow Tool (MPCverify Session)

When you select MPCverify as one of your sessions, you access the Calibre MPCverify tool. Sessions of type MPCVERIFY use a Calibre MPCverify-based setup file and contain settings for simulation tools.

Configuring a Calibre MPCverify Session

A Calibre MPCverify session must be correctly loaded and configured before you can use it to perform verification tasks.

Restrictions and Limitations

- image_sets are not recognized by the Calibre RET Flow Tool.

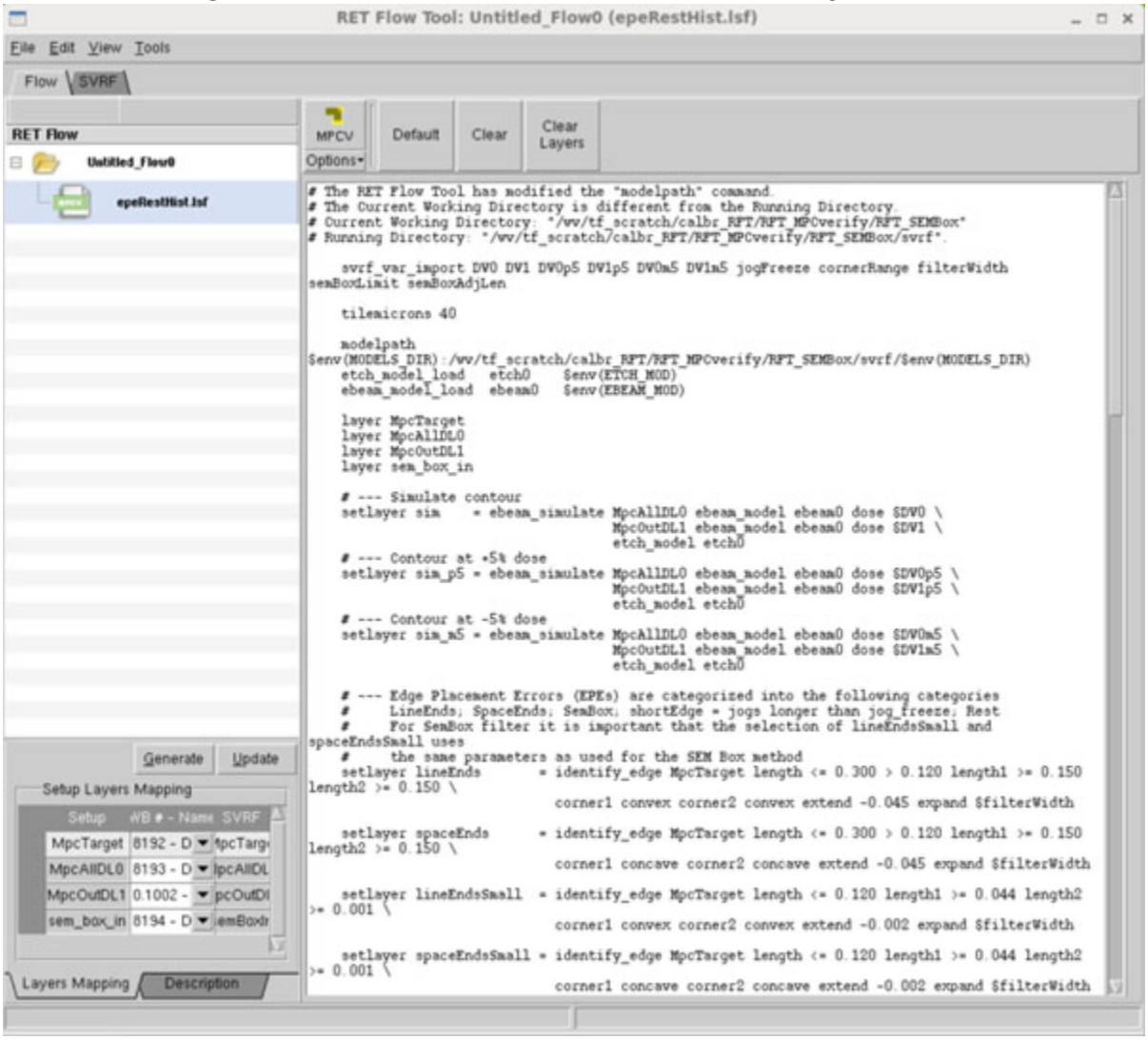
Prerequisites

- A known working Calibre MPCverify rule file in one of the following formats:
  - An SVRF file containing the Calibre MPCverify command file in-line
  - A transcript from a Calibre MPCverify run
- The relevant design layout loaded into Calibre WORKbench

Procedure

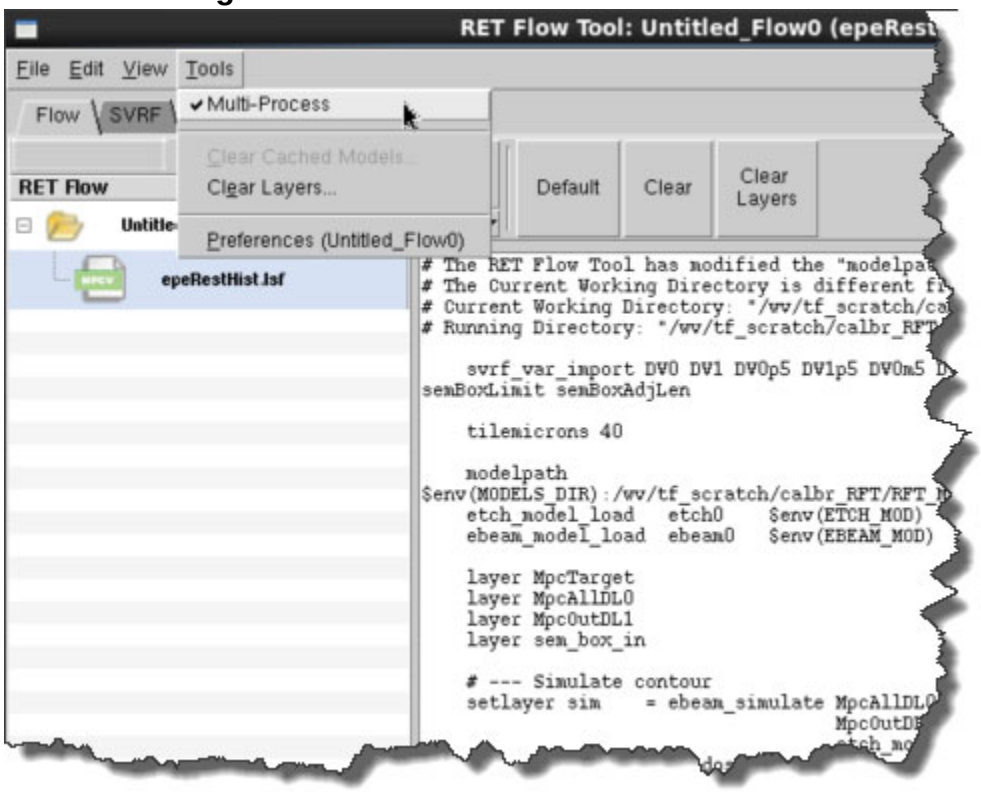
1. Follow the steps in "Creating a Session" to create a Calibre MPCverify session.
   - If you used one of the methods that loaded an existing file, your RET Flow Tool should look similar to Figure 3-29.
   - If the session text entry pane is empty, you can use the **Copy** and **Paste** options from the **Edit** menu or right-mouse context menu inside the text pane to copy directly from a Calibre MPCverify rule file.

Figure 3-29. RET Flow Tool, Calibre MPCverify Session



2. Adjust the layer mapping from the "Setup Layers Mapping" section in the bottom left part of the RET Flow Tool window.
3. (Optional) Run Calibre MPCverify in Multi-Process mode if you want to ensure the process is not affected by the parent Calibre WORKbench process.

Figure 3-30. Multi-Process Mode Selection



4. Test Calibre MPCverify by clicking the **MPCV** button.

Results

Calibre MPCverify runs the script, and any layers that created result markers or contours are added to the active layout in the viewer main window. Use the **Clear Layers** button if you want to clear the produced layers.

Calibre MPCverify Session Operations

A Calibre MPCverify session contains a single GUI control that runs the tool using the conditions defined in the session.

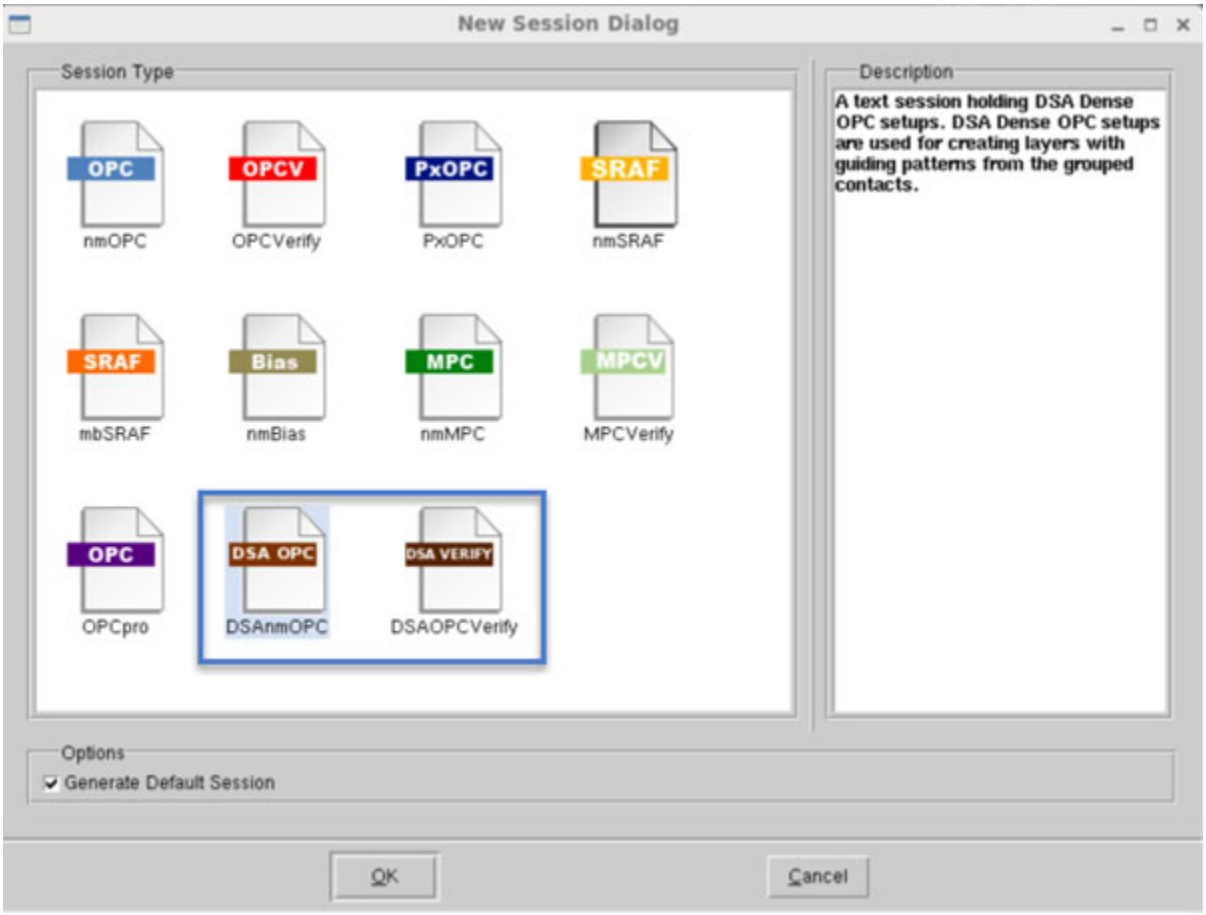
The operations and where their controls are located are summarized in Table 3-10.

Table 3-10. Calibre MPCverify Session Operations

Operation	Control	Location	Description
See the *Calibre MPCverify User's and Reference Manual*	**MPCv**	RET Flow Tool toolbar	Run Calibre MPCverify.

Calibre DSA in the RET Flow Tool

When you select **DSAnmOPC** or **DSAOPCVerify** as one of your sessions, you access the Calibre® DSA tools for creating directed self-assembly (DSA) models. These sessions use the Calibre RET Flow tool to manage Calibre DSA sessions.



Configuring a Calibre DSA nmOPC Run

You must load a Calibre DSA nmOPC file in the RET Flow Tool and configure it before you can produce results in the layout.

Prerequisites

- A known working Calibre DSA nmOPC file, post-synthesis
- The corresponding design file, loaded into Calibre WORKbench
- Calibre RET Flow tool invoked

Procedure

1. In the RET Flow Tool, select a session of type “DSAnmOPC” or add a DSA nmOPC session.
2. In the Output Layers Options section, assign a starting layer number for output layers. You may need to click the arrow next to the **DSA** button to open this flyout panel.
3. Select **File > Extract from SVRF** to open the Calibre DSA nmOPC file and load the script into the text window. (You can also copy and paste the file from a text editor.)
4. In the Setup Layers Mapping area, assign the layers in the DSA nmOPC script to the layers in the design file, as appropriate.
5. Click **DSA** to run the tool.

Results

The Calibre RET tool executes the script. The results are written to the specified layer number.

Configuring a Calibre DSA OPCverify Run

You must load a Calibre DSA OPCverify file in the RET Flow Tool and configure it before you can produce results in the layout.

Prerequisites

- A known working Calibre DSA OPCverify file
- The corresponding design file, loaded into Calibre WORKbench
- Calibre RET Flow tool invoked

Procedure

1. In the RET Flow Tool, select a session of type “DSAOPCverify” or add a DSA OPCverify session.
2. In the Output Layers Options section, assign a starting layer for output layers. You may need to click the arrow next to the **DSAV** button to open this flyout panel.
3. Select **File > Extract from SVRF** to open the Calibre DSA OPCverify file and load the script into the text window. (You can also copy and paste the file from a text editor.)
4. In the Setup Layers Mapping area, assign the layers in the DSA OPCverify script to the layers in the design file, as appropriate.
5. Click **DSAV** to run the tool.

Results

The Calibre RET tool executes the script. The results are written to the specified layer number.

Chapter 4
RET Flow Tool Operations

There are a number of operations that are shared across the sessions in the RET Flow Tool.

Setup File Operations

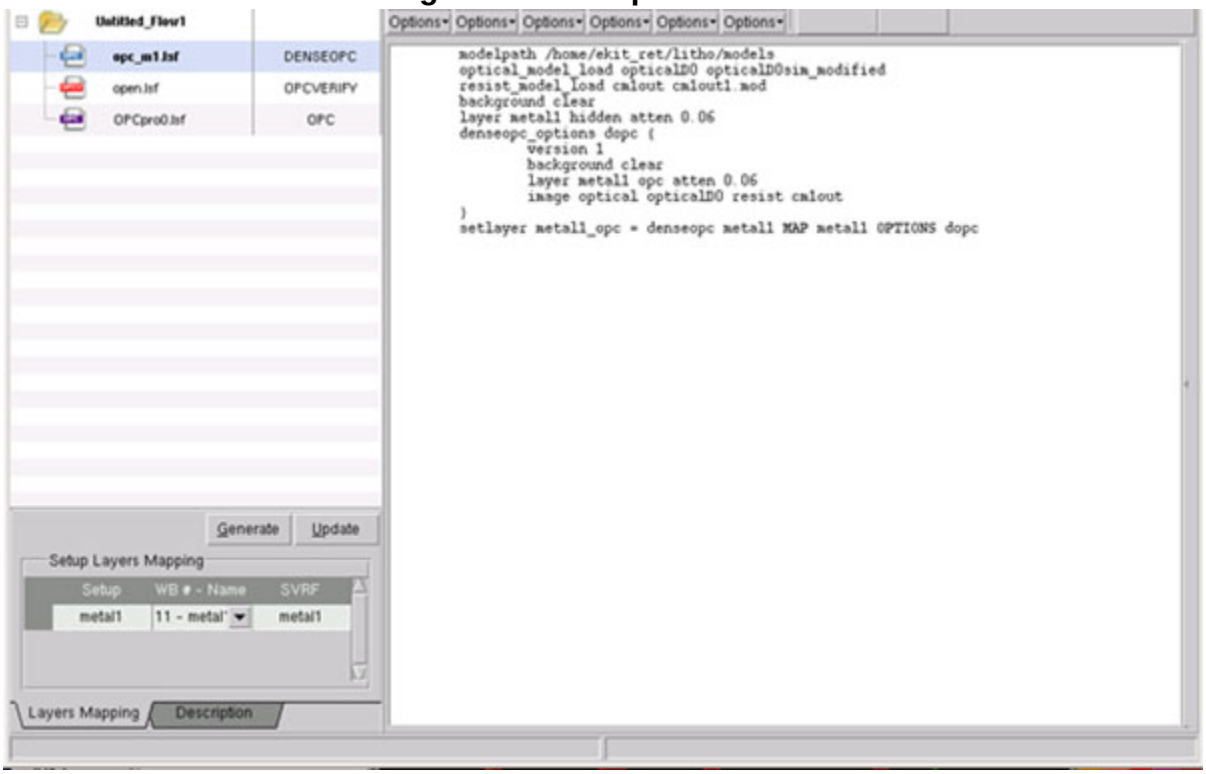
Calibre WORKbench and Calibre LITHOview use text files to configure important values and flags as input for multiple tools. Several tools in the RET Flow Tool have their own version of a setup file (*.setup* and *.in* extensions are typical). Each Calibre tool may have a setup file that is not cross-compatible with other Calibre tools.

The setup file editor is functionally the same for each tool that uses one. To create a new setup file, enter text into the main text entry field.

Using the Setup File Editor

The Setup File Editor has two functional areas: the main setup file display field and two tabs, **Layers Mapping** and **Description**, at the bottom left corner of the window. You can edit the setup file directly in the main display field.

Figure 4-1. Setup File Editor



Procedure

1. You can load, create, or import a setup file from a variety of sources. These include:

   - "Loading an Existing Setup File" on page 91
   - "Extracting a Setup File from a Transcript" on page 92
   - "Extracting a Setup File from SVRF" on page 93
   - "Starting from a Default Setup File" on page 97

   Once the setup file is loaded, you can edit the text directly in the main text display field.

   - Click **Re-Read** to re-load an existing loaded setup file.
   - Click **Update** if you want changes to the setup file to be reflected in the current run.

2. Use the **Setup Layers Mapping** field to map the layers in the RET Flow Tool. Setup files typically do not have layer numbers specified.

   If you want to reflect edits to the setup file in the SVRF used to generate derived layers, click **Generate**. These appear in the SVRF Layer Name column (and are saved in the session file).

3. Use the **Description** tab to enter any comments associated with your session.

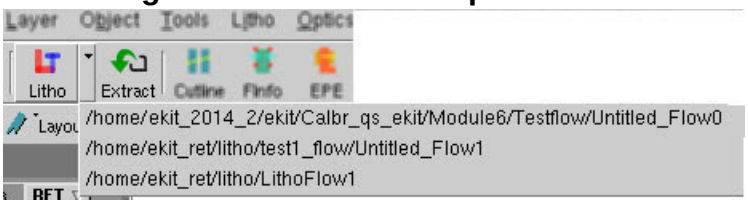
Loading an Existing Setup File

You can load a self-contained setup file into your current session from the **File** menu of the RET Flow Tool.

Procedure

1. On the RET Flow Tool window, select **File** > **Add Existing Setup** in the RET Flow Tool.
2. When the Open Setup File dialog box appears, select the setup file you wish to add, then click **Open**. The setup file is loaded as a new session in the RET Flow Tool.
3. You can alternatively select from a list of recently-loaded LITHO setup files by choosing from the pulldown list (the downward triangle) next to the **Litho** button.

Figure 4-2. Recent Setup Files List



Extracting a Setup File from a Transcript

You can extract the text of a setup file an output using the **File** menu of the RET Flow Tool. The RET Flow tool not only extracts the LITHO, RET, or MPC block, but also automatically loads it into the appropriate session tool. For example, if a Calibre OPCverify LITHO block is extracted, it is automatically added as an OPCverify session.

Procedure

Perform one of the following:

- To extract the entire contents from an existing transcript, either click **Extract** in the viewer, or select **File > Extract All from Transcript** from the RET Flow Tool.

  Figure 4-3. Extract Button in Calibre WORKbench

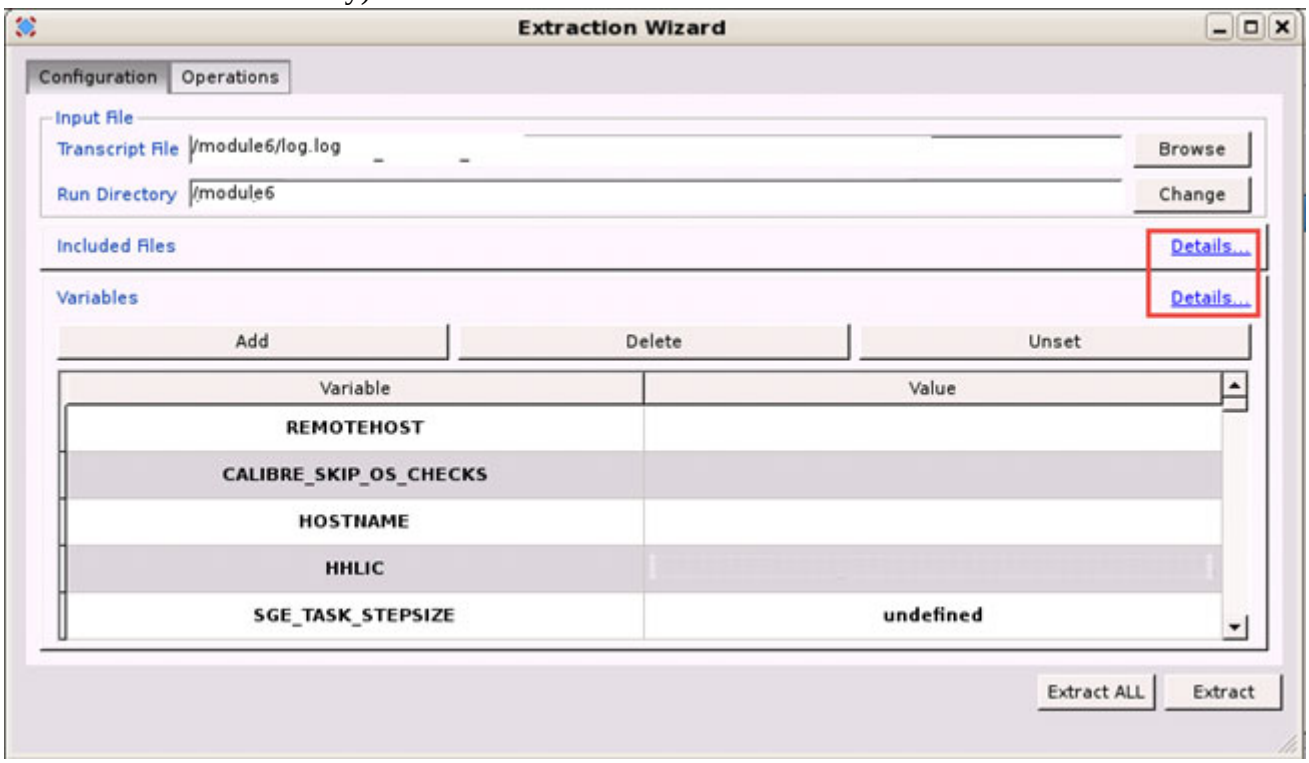
  When the Input Transcript File dialog box appears, specify the transcript for extraction and click **Open**.

- To extract a portion of code from an existing transcript, select **File > Extract from Transcript**.

  i. When the Extraction Wizard dialog box appears, specify the transcript file or run directory to locate the transcript.

  ii. (Optional) Expand the Included Files and Variables sections by clicking the associated Details link. Use these sections to verify your working environment is

configured properly (all required files are loaded, and all variables are set correctly).



Results

The litho setup files (LITHO FILE command blocks) appear in the **Operations** tab of the Extraction Wizard. You can select the specific setup file to extract as well as change the type of session to import to by selecting from the adjacent pulldown menu.

Extracting a Setup File from SVRF

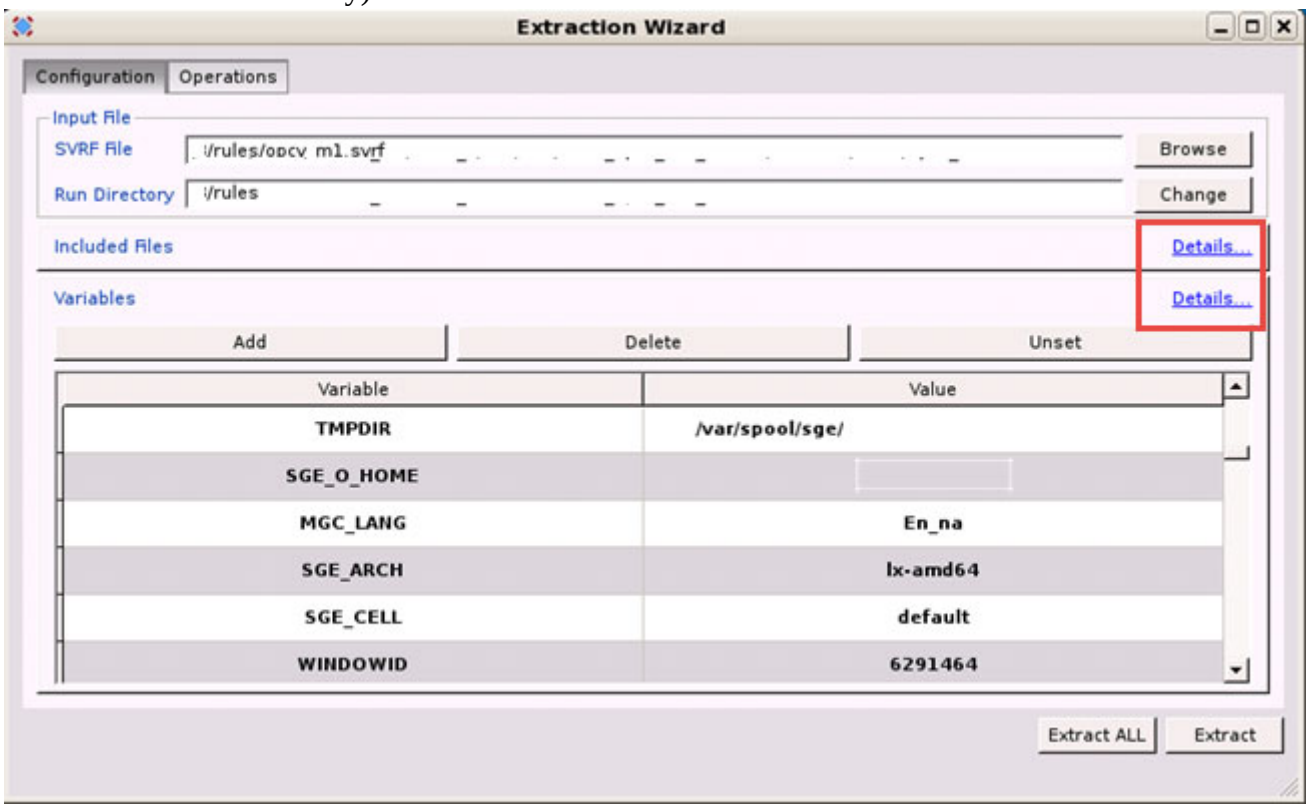
You can extract the text of a litho setup file from SVRF using the **File** menu of the RET Flow Tool. The RET Flow tool not only extracts the LITHO FILE block, but also automatically loads it into the appropriate session tool. For example, if a Calibre OPCverify LITHO block is extracted, it is automatically added as an OPCverify session.

Restrictions and Limitations

- The SVRF DISCONNECT statement is not supported when extracting litho setup files from SVRF rule files. Attempting to extract from an SVRF rule file that contains a DISCONNECT statement can have unexpected results, such as the file not loading properly.

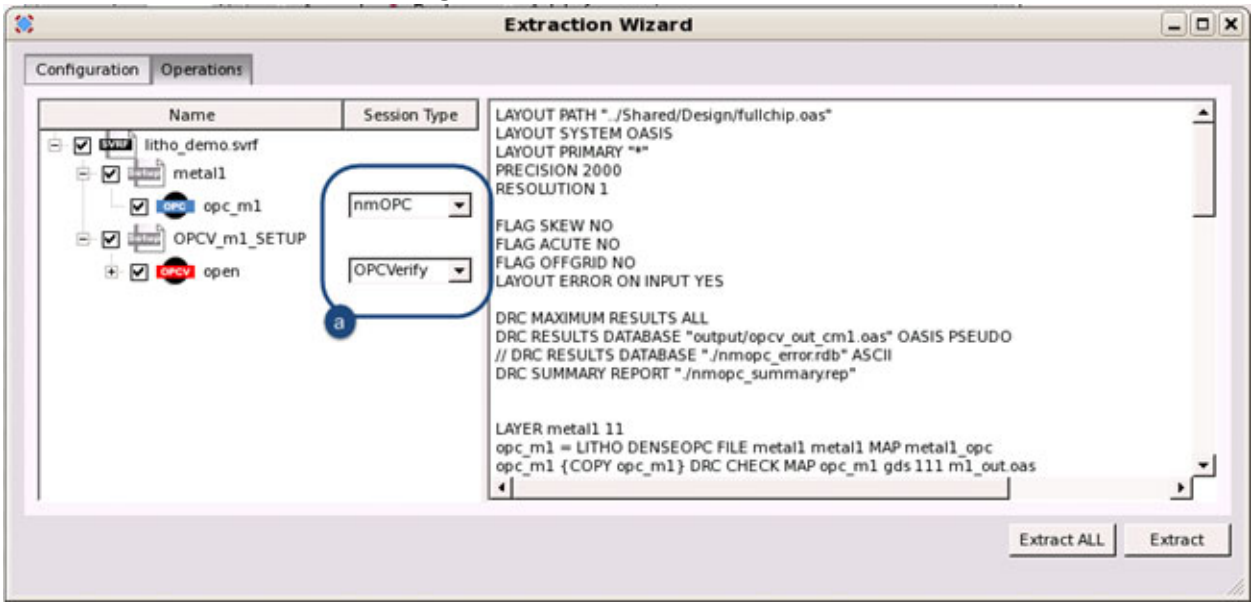
Procedure

1. In the RET Flow Tool window, perform one of the following:
   - To extract the entire contents from an existing SVRF file, either click **Extract** in the viewer, or select **File > Extract All from SVRF**. When the SVRF File dialog box appears, specify the SVRF for extraction and **Open**.
   - To extract a portion of code from an existing SVRF file, select **File > Extract from SVRF**.
     i. When the Extraction Wizard dialog box appears, specify the SVRF file or run directory to locate the SVRF file for extraction.
     ii. (Optional) Expand the Included Files and Variables sections by clicking the associated Details link. Use these sections to verify your working environment is configured properly (all required files are loaded, and all variables are set correctly).



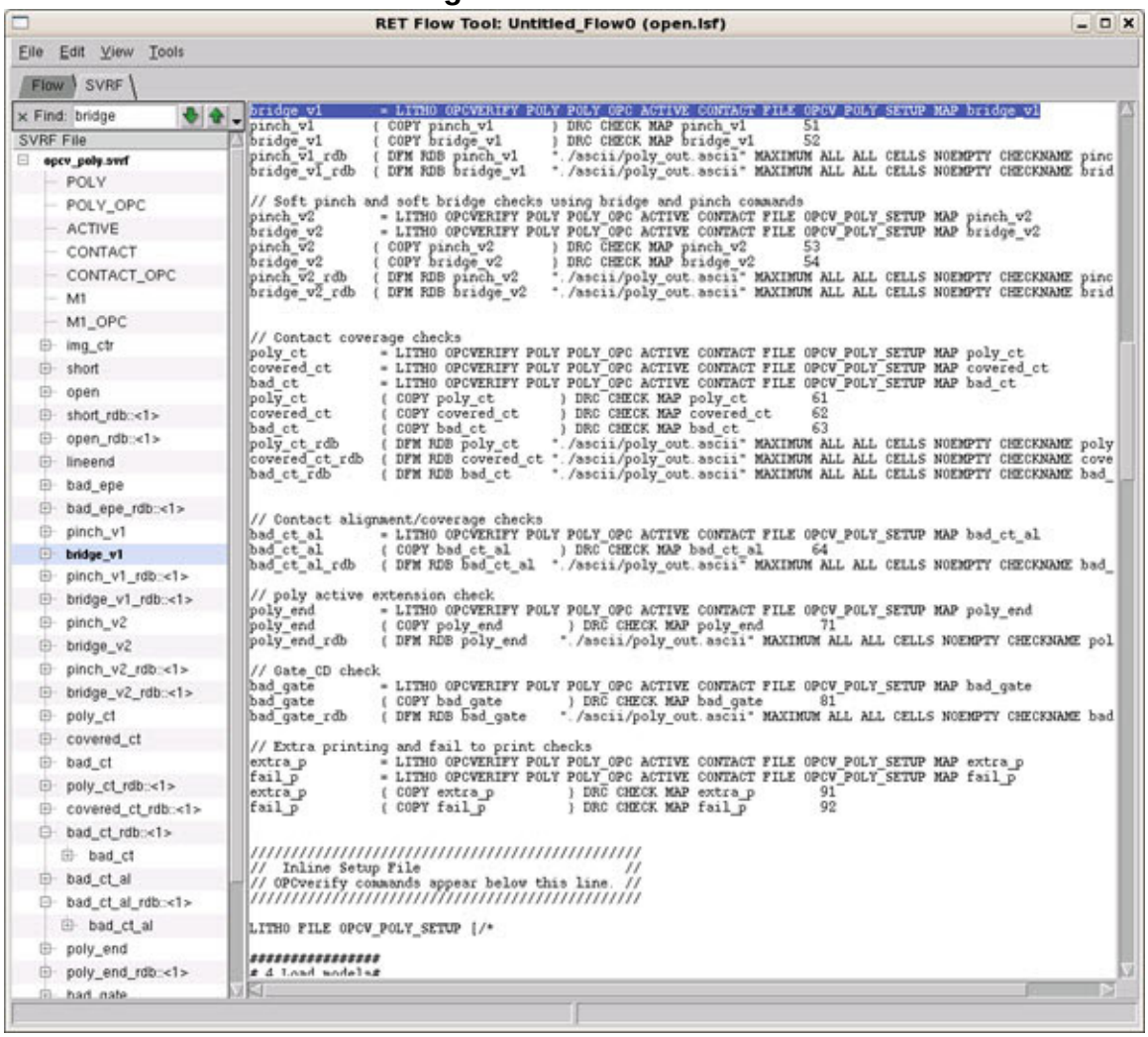
The litho setup files (LITHO FILE command blocks) appear in the **Operations** tab of the Extraction Wizard. You can select the specific setup file to extract as well as change the type of session to import to by selecting from the adjacent pulldown menu (**a** in Figure 4-4).

Figure 4-4. Extraction Wizard



2. View the contents of the imported SVRF by clicking the **SVRF** tab in the RET Flow Tool window.

Figure 4-5. SVRF Tab



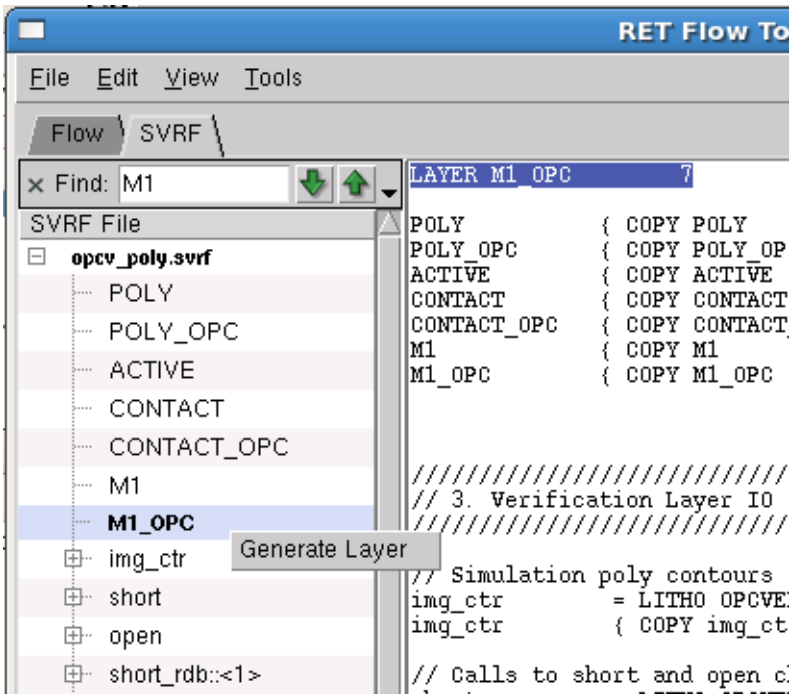
The **SVRF** tab displays the full contents of the SVRF rule file as well as the program hierarchy in the left pane.

3. To add derived layers (that is, layers generated by running the SVRF file):

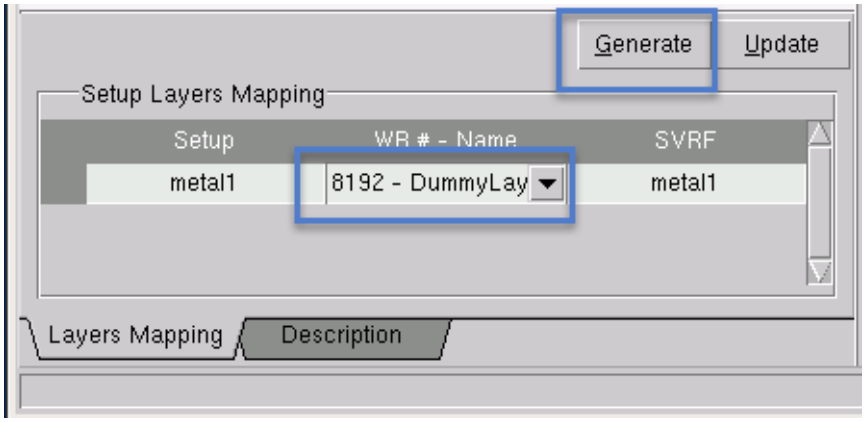
   - For a specific layer, right click on a layer in the **SVRF** tab and select **Generate Layer**; the RET Flow Tool only generates that layer.

Note

You can use the **SVRF** tab Find field to search the list of rule file layers by name. The dropdown arrow to the right of the field allows you optionally to match by whole words and case.



- For all derived layers, switch to the **Flow** tab and click the **Generate** button to have the RET Flow Tool generate any layers marked as “8192-Dummy Layer.”



Starting from a Default Setup File

If you want to create a new setup file, you can start from a default generated by Calibre WORKbench.

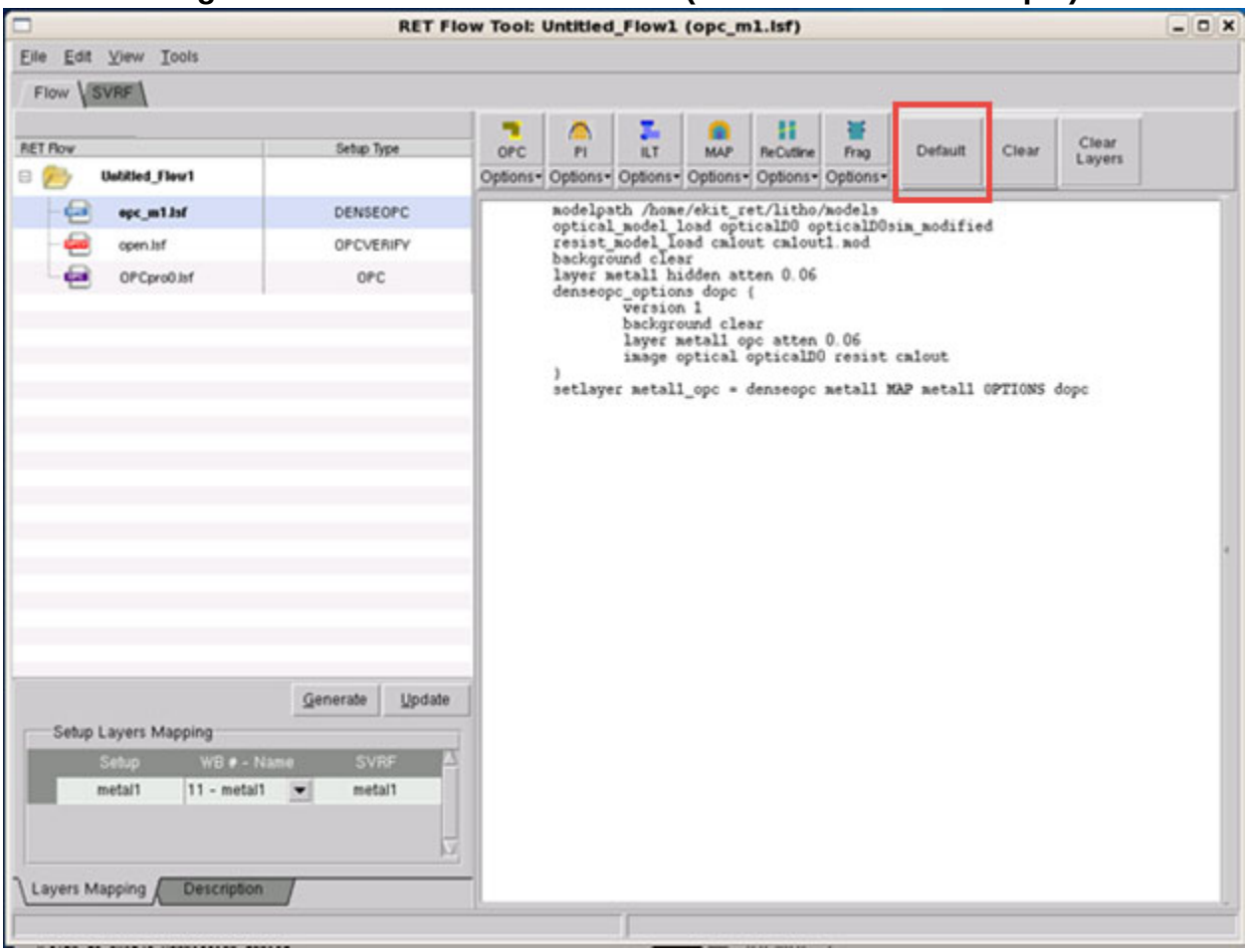
Procedure

1. Click the **Default** button in any session. This causes the viewer to automatically read data from the open layout and to construct a starting setup file based on the state of the GUI and the current working directory.

   Alternatively, when adding a new session (**File > Add New Session** in the RET Flow Tool) you can select Generate Default Session when choosing your session.

2. The RET Flow Tool writes the constructed setup file to the currently active mode.

Figure 4-6. RET Flow Tool Default (Calibre nmOPC Example)



Saving a Setup File

You can save your current setup file from the **File** menu of the RET Flow Tool.

Procedure

1. In the RET Flow Tool window, select **File > Save Setup File**.

2. When the Save Setup Text dialog box appears, navigate to the directory you want to save the file in, enter a name, then click **OK**. The setup file is saved in the specified directory.

Using Disk-Based Viewing of Layout Files

Viewing large OASIS layouts in Calibre WORKbench consumes less time and resources using disk-based viewing than attempting to load an entire layout using **File** > **Open Layout Files**. Disk-based viewing loads large layouts using a view-only, disk-resident, indexed file. An artificial top cell *disk_res_** is generated and the original design cell is placed beneath this new top cell.

Restrictions and Limitations

- When working with disk-based viewing mode in Calibre WORKbench, you cannot add or edit any layers to the original top cell. You also cannot add any design object (such as polygons or paths) to an already existing layer in the original top cell. Saved properties are not loaded and saving edits is not permitted. However, new layers and polygons can be added, though temporarily, to the *disk_res_** cell.

Procedure

1. Open Calibre WORKbench in any of the Litho modes (calibrewb and calibrewb -lv).
2. In Calibre WORKbench, select **File** > **View Oasis Layout**.
3. Open the RET Flow Tool from **Litho** > **RET Flow Tool**.
4. In the RET Flow Tool, open, extract, or create the input Litho flow, setup, session, or SVRF.
5. You can now configure the operation that you want to execute and press the run button for your session tool (for example, the **OPC** button). However, make sure that you are in the *diskres_** cell to generate the output polygons for the layout. The RET Flow Tool automatically switches to the same view in the *diskres_** cell for the operation that generates output polygons. No switching is performed for operations such as Map and Cutline that do not generate layout objects.

Specifying Models

Many sessions require various types of models that can be specified in a setup file.

Procedure

1. In your setup file, make sure you have one of the following:
   - A path to a valid litho model

- Valid paths to an optical model using the following:
  - **modelpath** — Specify the directory containing your optical model.
  - **optical_model_load** — Specify the directory containing your optical model file.

2. Make sure that you have mapped your layers from the setup file to the layers displayed in WORKbench.

Configuring an ILT Run

ILT uses Calibre nmOPC to perform inverse lithographic OPC correction. It is the most efficient way to run a nmOPC operation from the viewer.

Restrictions and Limitations

- ILT is only available in nmOPC sessions. It is not available in pxOPC sessions.
- It is strongly recommended to zoom in to only a few polygons. ILT runs perform complex calculations and take longer than other OPC runs.

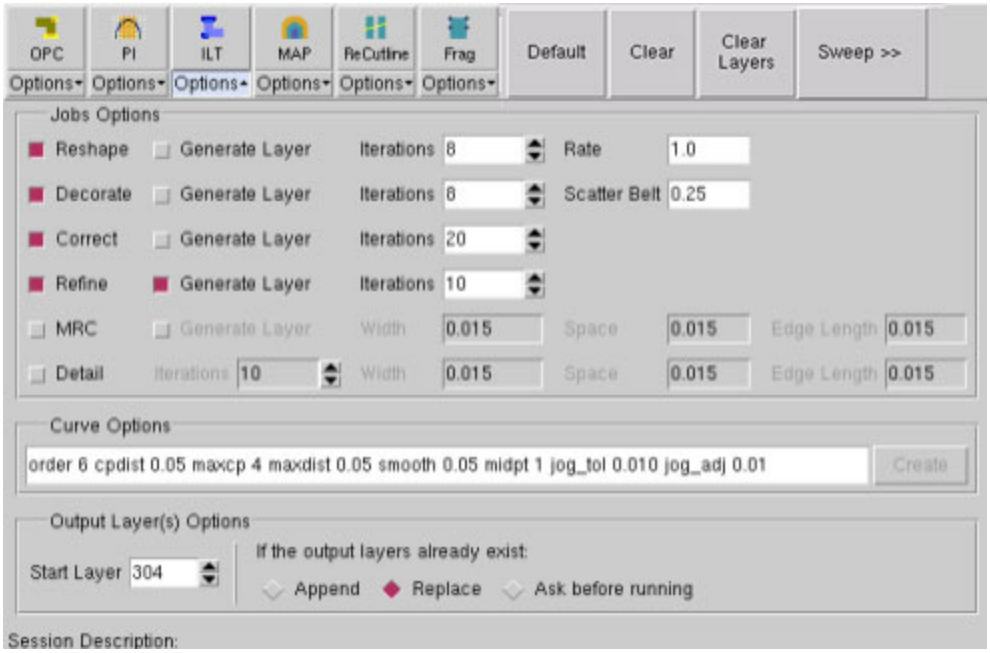
Prerequisites

- Layout file loaded in the viewer main window.
- An nmOPC session active.

Procedure

1. In the RET Flow Tool, click **Options** under the **ILT** button. Figure 4-7 shows the default values.

Figure 4-7. ILT Options



2. To see the output from the various phases, select Generate Layer. Unless you are familiar with Calibre pxOPC operation, do not change the default iterations, rate, or curve command.
3. Specify a Start Layer. Layers generated by the ILT run are added to the layout starting at the specified number.
4. Click **ILT**.

Related Topics

Calibre pxOPC Users and Reference Manual

Creating an Aerial Image Contour Map

The Map tool creates a contour map of the aerial image. This map is a representation of the light intensity at the specified defocus start value, normalized between 0 and 1, although it can be greater than 1. The Map tool sums transmission values for all overlapping polygons on multiple OPC layers.

Figure 4-8 shows the aerial image contour in magenta, and the output from the Map tool in gray.

Figure 4-8. Aerial Image Contour Map

Prerequisites

- Layout file loaded in the viewer.
- Output layers are specified (typically in the Setup Layers Mapping pane).

Procedure

1. (Optional) In the Calibre WORKbench toolbar click **Select** and draw a bounding box around an area. If you do not select an area, the entire visible part of the layout is used as the selection area.
2. In the RET Flow Tool, click **Options** under the **MAP** icon.
3. Specify parameters for the simulation. There are different parameters depending on your session type.

Table 4-1 lists the parameters available for Calibre nmOPC and Calibre OPCverify.

Table 4-1. Map Run Options Fields (OPC Session Types)

Field	Description
Map	Select the type of map. Aerial: Performs a dense aerial simulation. Resist: Performs a dense resist simulation. Flare: Performs a flare map simulation. For more information on generating flare maps, see the *Calibre WORKbench User's and Reference Manual*.
Pixsize, um	Specify the pixel size of the aerial map.
Clear previous results	Clears previous settings on run.
Convolve Output with Gaussian	Use a Gaussian kernel at the specified sigma size (default: 70 nm) on the output.

Table 4-2 lists the parameters available for Calibre cnSRAF and Calibre mbSRAF.

Table 4-2. Map Run Options Fields (SRAF Session Types)

Field	Description
Map	Specify options to produce a gradient map: • gradienttile • usedpixel • modgradient • sdf • aerialimage • ideal • gradientinfo • prioritylayer • tag • weightlayer • weight Refer to the *Calibre nmSRAF User's and Reference Manual* for information.
Clear previous results	Clears previous settings on run.

Table 4-3 lists the parameters available for Calibre OPCpro.

Table 4-3. Map Run Options Fields (Calibre OPCpro)

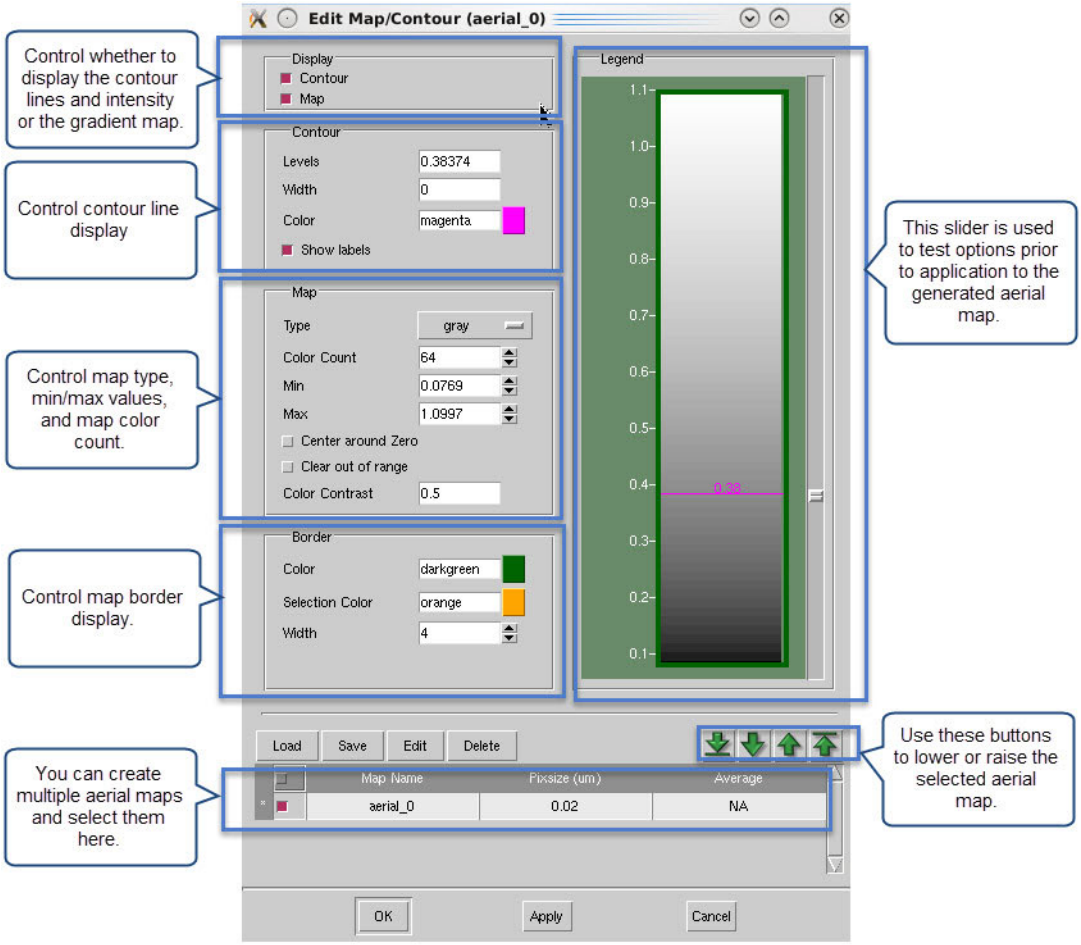
Field	Description
Map	Select the type of map. • Aerial: Perform a sparse aerial simulation. • Hopkins: Perform a Hopkins Fourier series simulation.
Pixsize, um	Specify the pixel size of the aerial map.
Check Visible Layers	Limits the run to only visible layers.
Clear previous results	Clears previous settings on run.
Convolve Output with Gaussian	Use a Gaussian kernel at the specified sigma size (default: 70 nm) on the output.

4. Click **OK** once you have finished making your selections.

   The Map tool runs, and produces aerial image output inside a green frame, as shown in Figure 4-8.

5. Double-click anywhere inside the green frame to raise the Edit Map/Contour dialog box.

Figure 4-9. Edit Map/Contour



The controls in this dialog box set the appearance of the aerial image map (gray pixels) and aerial image contour (magenta line). Adjusting the slider bar changes the simulation of the aerial image contour line dynamically in the layout view.

Generating a Print Image

The Print Image (PI) tool simulates geometries on OPC layers that lie fully or partially within a selected area.

Figure 4-10. PI Tool Output

Prerequisites

- Configured the run for your session as described in "Using Sessions" on page 37.
- Layout file loaded in the Calibre WORKbench main window.

Procedure

1. (Optional) In the Calibre WORKbench toolbar click **Select** and draw a bounding box around an area. If you do not select an area, the entire visible part of the layout is used as the selection area.
2. In the RET Flow Tool, click **Options** under the **PI** button.
3. Fill out the fields as shown in Table 4-4.

Table 4-4. PI Run Options Fields (OPC)

Item	Description
Sparse Mode Dense Mode (Calibre OPCpro only)	Select the simulation engine: Sparse or Dense.
Output Start Layer	Set the starting number for the layers to be output.
If the output layers already exist: Replace, Append, Ask First	Set how layers are output during subsequent runs of the tool.

4. Click **PI** once you have finished making your selections.

   The PrintImage tool runs, and produces output similar to that shown in Figure 4-10. It creates a new layer in your design corresponding to the Output Start Layer field value.

Cutline Operations

The Cutline tool lets you compare one or more simulations along cutlines you define. It sums transmission values for overlapping polygons on multiple OPC layers.

Drawing a Cutline

The Cutline tool is available with all session types. This task describes the steps to run the Cutline tool.

Prerequisites

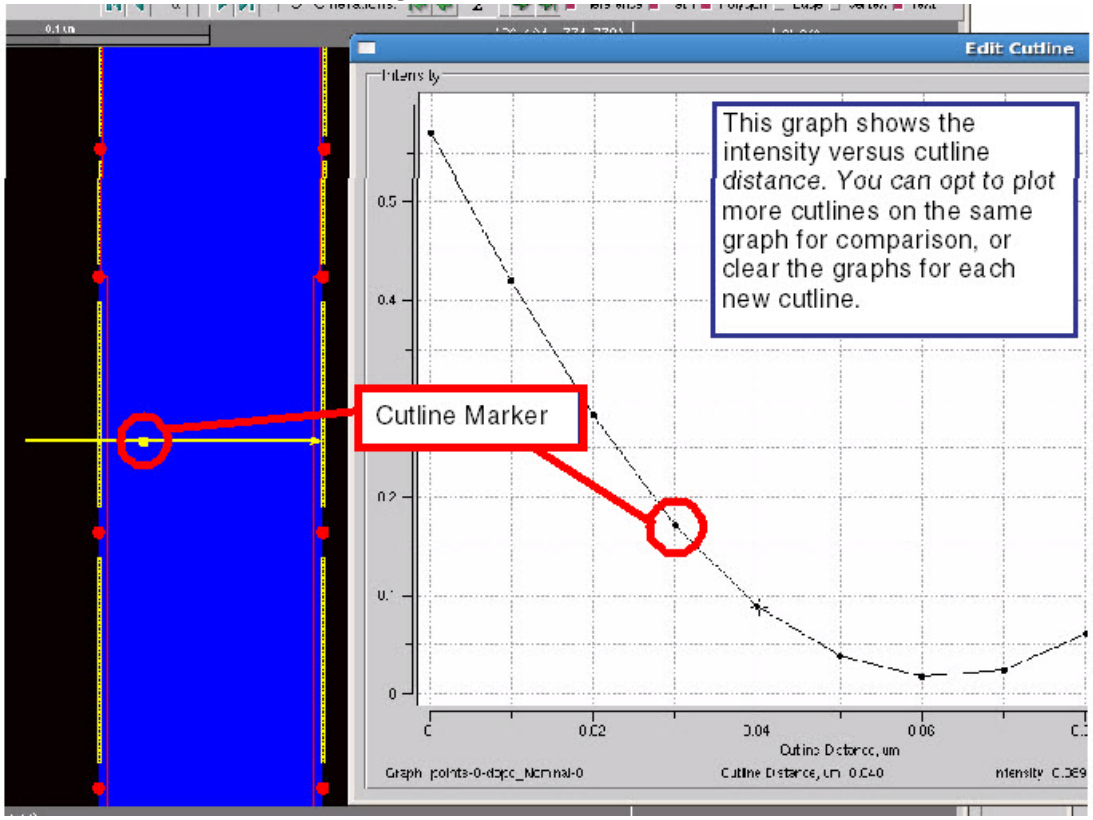
- Configured the run for your session as described in “Using Sessions” on page 37.
- Layout file loaded in the Calibre WORKbench main window.

Procedure

1. In the Calibre WORKbench toolbar, click **Cutline**.
2. In the layout window, click start and end points to create a cutline arrow that crosses edges of interest.

Results

Figure 4-11. Edit Cutline



The Edit Cutline tool appears with the results of the measurements. You can continue to select more cutline locations on the design window, or make adjustments to the cutline properties.

You can manipulate the intensity graph and the corresponding points with the following controls after clicking the **Mark Threshold and CD** icon:

- Click on the intensity graph to mark the corresponding point on the layout cutline.
- Hover over the intensity graph to show the corresponding point on the layout cutline.
- Ctrl-click the intensity graph to mark more than one point on the layout cutline.
- Press the Esc key or click on the graph panel outside the graph area to remove all the marked points.

Setting Cutline Properties

You can specify different simulation and display properties for your cutline by using **Re-Cutline** in the RET Flow Tool.

Prerequisites

- Configured the run for your session as described in “Using Sessions” on page 37.

- Loaded a layout file in the Calibre WORKbench main window.
- Drawn an initial cutline as described in "Cutline Operations" on page 107.

Procedure

1. In the RET Flow Tool, click **Options** under the **Re-Cutline** button.
2. Fill out the fields as shown in Table 4-5.

Table 4-5. Cutline Run Options Fields

Item	Description
Cutline Mode	Select whether the Cutline tool runs in 1D or 2D mode the next time you draw a cutline in the design window.
Pixsize, um	Select the simulation pixel size of the scan in microns from the pulldown menu.
2D Resist Options (Calibre OPCverify, Calibre nmOPC modes)	When 2D Resist is selected in Cutline mode, these options become active. • **Resist Image + Profile** / **Resist Profile** — Select the run mode for 2D resist profiling. • **Plane Start**, **End**, **Count** — Set from the corresponding parameters in nm from the topo optical model and the number of sample planes to measure. • **Threshold** — Set the corresponding threshold level from the CM1 resist model.
Simulation Options (Calibre OPCpro only)	Select the type of image. **Aerial** — Perform a dense aerial simulation. **Hopkins** — (Calibre OPCpro only) Perform a Hopkins simulation. **Resist** — Perform a dense resist simulation.
Output Graph (Calibre OPCpro only)	Set the appearance of the curve of the cutline graph in the Cutline window.
Layout Cutline	Set the appearance of the cutline marker in the display window.
Accumulate Graphs	Specify if the next cutline measurement replaces the current one (unchecked) or adds another curve to the graph. You can set the color and the width of the graph in this section in the Process Window conditions table.

Table 4-5. Cutline Run Options Fields (cont.)

Item	Description
Check Visible Layers (Calibre OPCpro only)	Checks for visible layers.

3. Click **Re-Cutline** once you have finished making your selections.

Determining the Slope of an Intensity Graph

The slope of an intensity graph provides useful information about the sharpness of an image. This information is available in the Cutline tool.

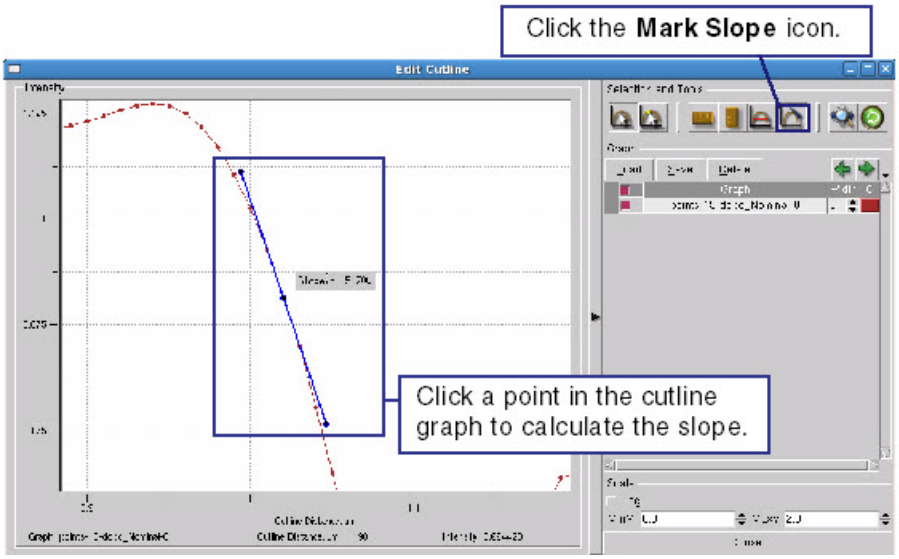
Prerequisites

- A cutline must be rendered in the Cutline tool (see "Drawing a Cutline" on page 107).

Procedure

1. In the Cutline tool, click the **Mark Slope** button.
2. Click a point on the cutline graph. This instructs Calibre to calculate and display the slope at the point selected.

Figure 4-12. Finding Slope in an Intensity Graph



Determining the Threshold and CD of an Intensity Graph

You can determine the CD and threshold value at any point in a cutline graph displayed in the Cutline tool.

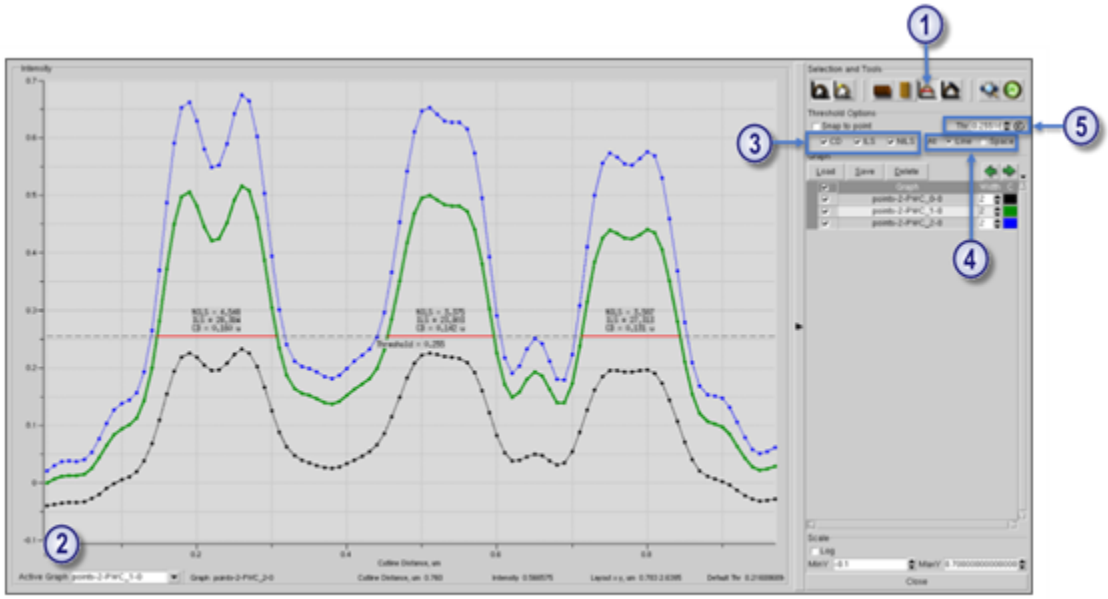
Prerequisites

- A cutline must be rendered in the Cutline tool (see “Drawing a Cutline” on page 107).

Procedure

1. Click the **Mark Threshold and CD** icon. See the following figure.

Figure 4-13. Edit Cutline GUI



2. Activate the graph you want to make measurements on. You can change the active graph using the **Active Graph** dropdown list below the plot.
3. If unchecked, click the critical dimension check button, **CD**. Optionally, click the image log slope and normalized image log slope check buttons, **ILS** and **NILS**.
4. Select whether to perform the computations on the layout feature locations or the space between the layout feature locations using the **At: Line** or **At: Space** radio buttons.

Note

At this time, the RET Flow Tool only performs calculations at the center of the graph, so selecting **At: Line** or **At: Space** has no effect.

5. Click your desired intensity threshold in the graph region of the Edit Cutline window. If you need to be more accurate, you can input the threshold value using the threshold box, **Thr**.

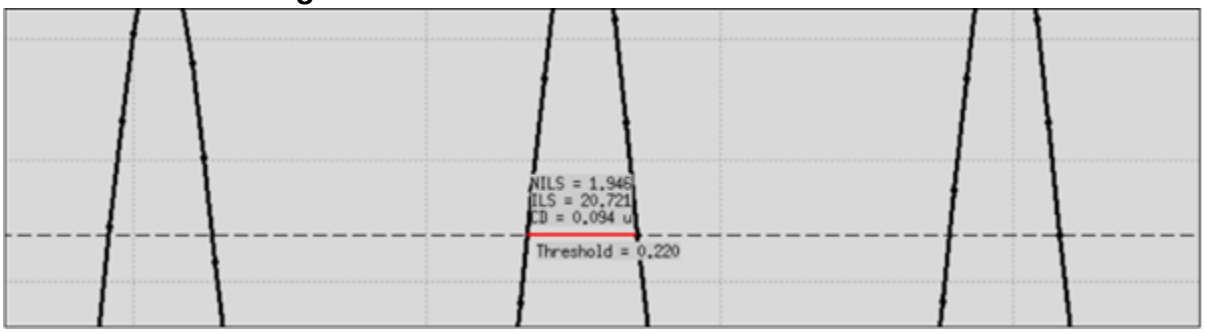
Tip

You can reset the threshold value to that specified by the PWC using the reset button to the right of the threshold box.

Results

A threshold line and CD markers are displayed. Figure 4-14 shows the calculated CD (and ILS and NILS).

Figure 4-14. User-Defined Threshold and CD



The measurements are computed at the center of the graph. No background transmission detection is performed.

Inspecting 2D Resist Profiles for 3D Resist Models

The Calibre RFT has an enhanced 2D resist cutline profiling functionality for users who have 3D resist-enabled models. The toploss resist model intensity view shows a heat map-based side view of the resist film underneath the top-down view of the resist film for 1D cutlines.

Restrictions and Limitations

- A minimum of 3GB RAM is required to run the 2D resist cutline simulation; the calculations required are memory intensive.
- 2D resist profiling cannot be done on normal optical models. It is only intended for use with 3D resist models.
- To use a mask model (including mask bias and cornerchop data), a litho model is required.

Prerequisites

- An optical model that is specially calibrated for toploss resist use. These optical models *must* use the optical model "imageplanes" keyword in place of the "defocuslevels" keyword, and should have been generated with the opticsgen -nolookuptables option.

  Note

  The optical model is described in "Optical Parameters File Format" in the *Calibre WORKbench User's and Reference Manual*.

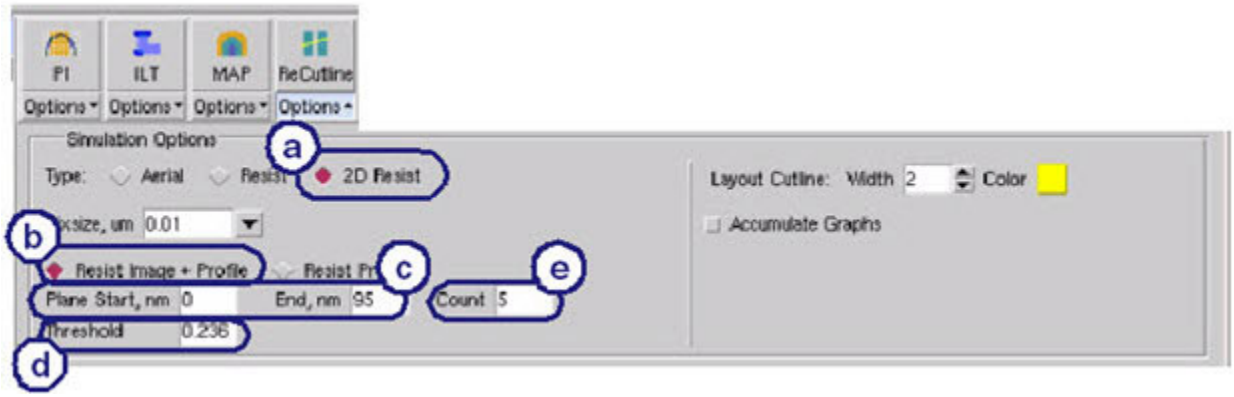
- A calibrated CM1 resist model. This CM1 model typically should be modelform 22 for PTD resists and modelform 44 for NTD resists. The setting for the RESIST_POLARITY field should be 1 for PTD and -1 for NTD.

- The design to test cutlines on loaded in Calibre WORKbench.
- If the design includes source maps, they must be located in the working directory.

Procedure

1. Set up a nmOPC or OPCverify session, including the following items in the session information:
   - The modelpath
   - A litho model that contains the following information:
     - The optical model that includes 3D resist information
     - The CM1 model
     - If your optical model includes DDM information, the DDM model
     - A mask definition with background and default dose information
2. In the Setup Layers Mapping field, change the WB# -Name dropdown to show the layer used for cutline measurements.
3. Switch to the layout viewer, click **Cutline**, and draw a regular cutline to establish an initial cutline location. The Cutline window appears with the 1D resist cutline.
4. In the Calibre RET Flow Tool, open up the **ReCutline** > **Options** panel. Configure the panel as shown in the figure.

Figure 4-15. 2D Resist Cutline Options Panel



   a. Switch the Type to 2D Resist.

   b. Set the Simulation Options to Resist Image + Profile.

   c. Change the values for Plane Start and End to match your film stack characteristics. This information can be found in your optical model.

   d. Change the Threshold to the calibrated threshold value. This information can be found in your CM1 model.

Tip

You can inspect the contents of your loaded models in the **Process** tab by clicking on a model line, then clicking the **View** button.

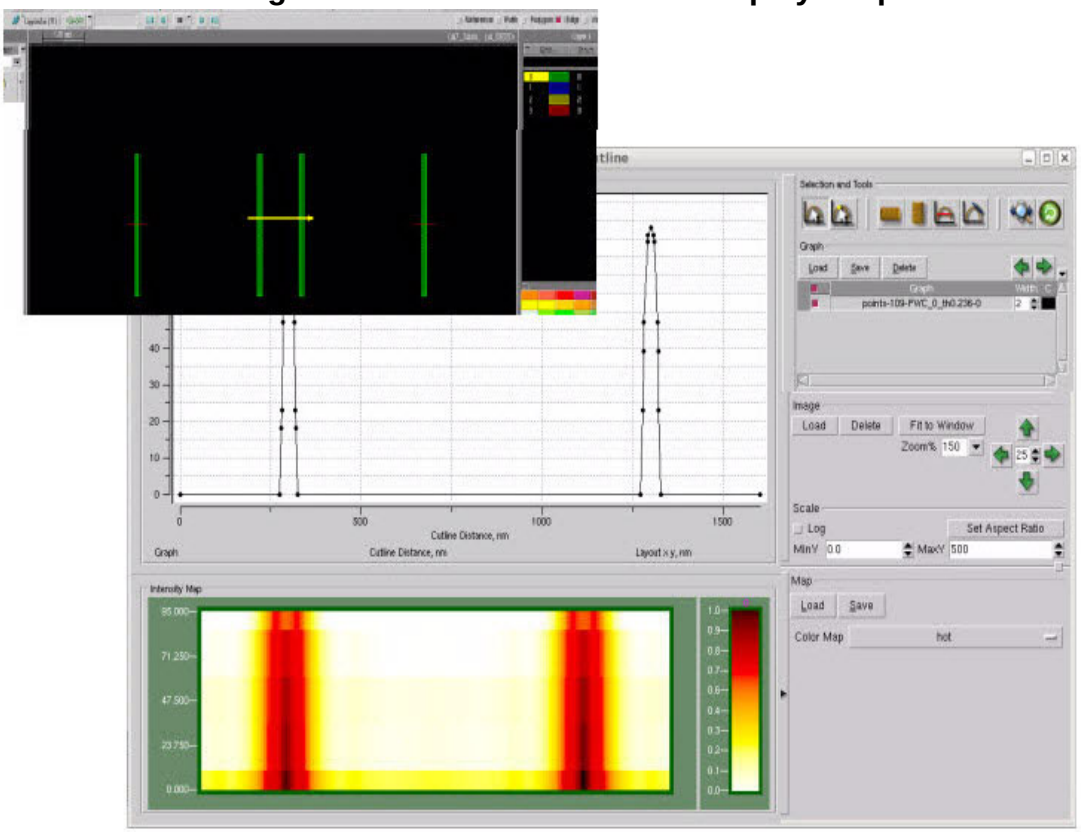
e. Change the Count field to an initial value lower than the default of 20 for faster processing on the initial run. 5 is typically a good value.

5. Click **ReCutline**.

Results

The console window for Calibre WORKbench shows the progress of the 2D cutline calculation. The initial run may take 10 to 20 minutes depending on the level you specified for Count in Step 5e. Subsequent cutline runs at other locations are faster, because the model is now cached.

Figure 4-16. 2D Resist Cutline Display Output

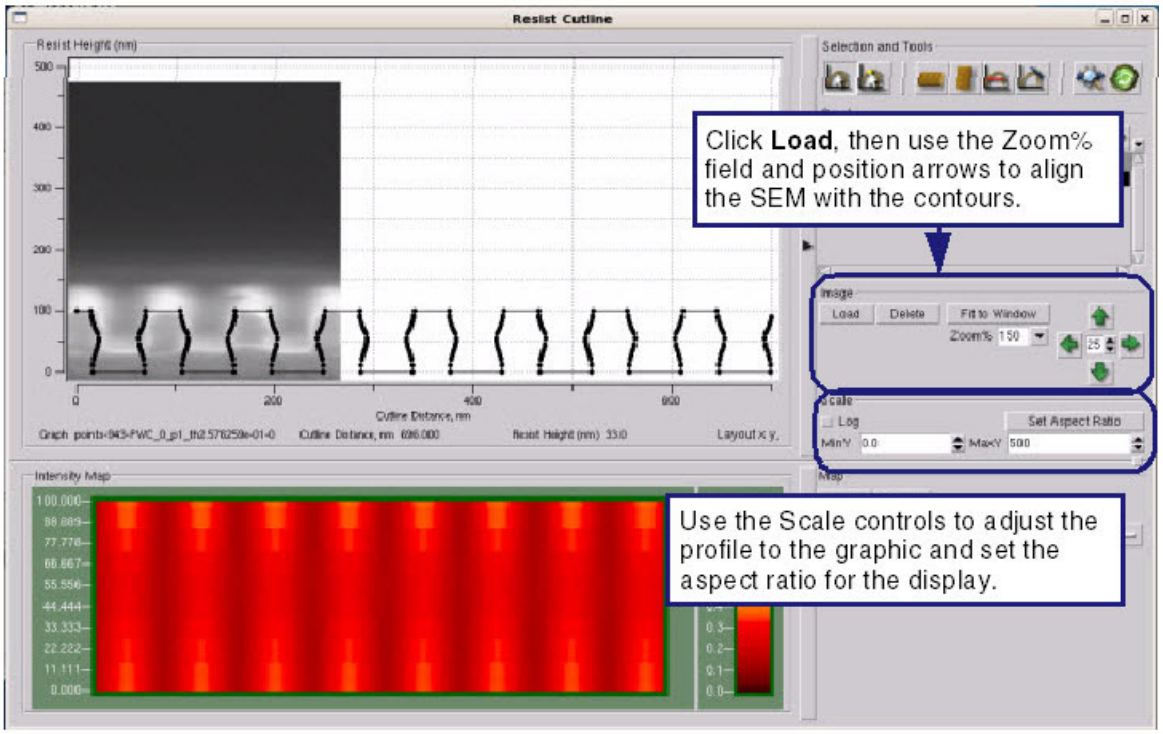


Tip

If Calibre WORKbench exits with a memory error, try lowering the Count value.

If you have cross-section SEM images, you can view them in the Resist Cutline dialog box by clicking **Load** and importing a SEM image directly to the dialog box.

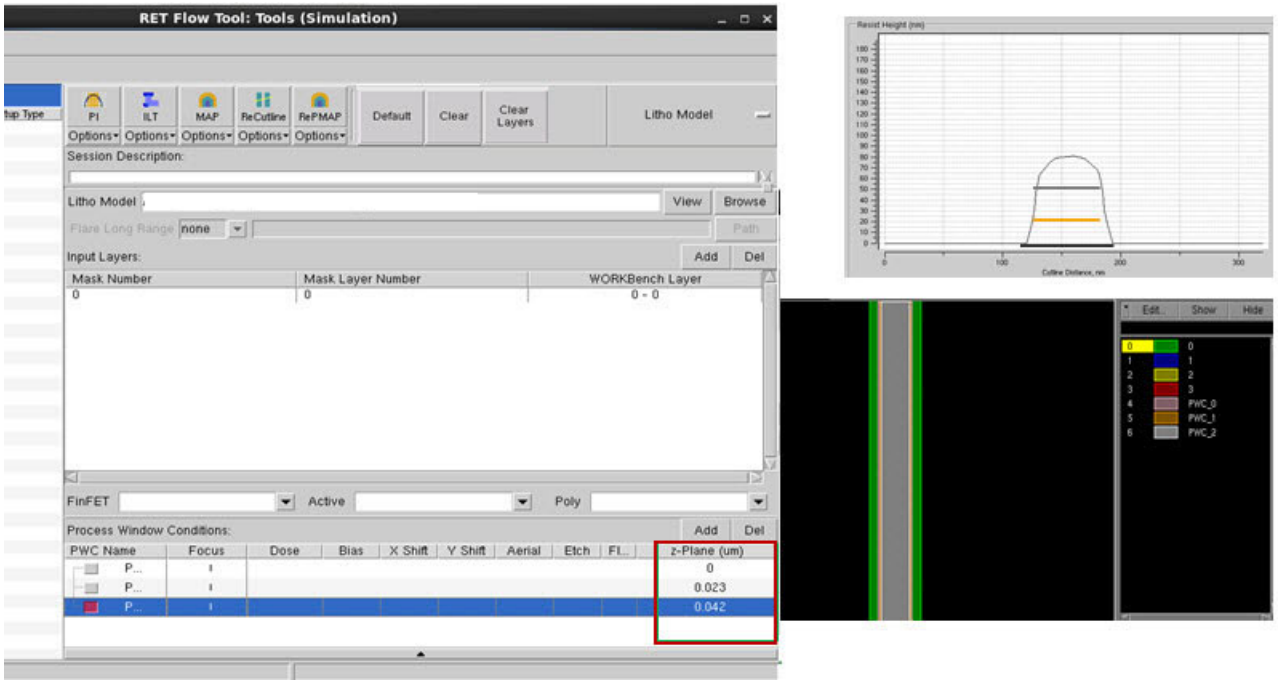
Figure 4-17. Cross-Section SEM Images in the Resist Cutline Dialog Box



If you use a litho model, you can use a 3D resist model to simulate the contour at any z-plane in a resist profile. You need the following:

- 3D Optical model with the “imageplanes” keyword.

Figure 4-18. Simulating Contours on the Z-Plane



Fragment Generation

The Frag tool calculates values and shows how edges are fragmented for OPC. When an OPC run is performed, each polygon edge is first divided into smaller fragments, and then each individual fragment is moved in order to apply correction to the simulation.

Displaying Fragments

Use the Frag tool to show how edges are fragmented in your layout.

Prerequisites

- Configured the run for your session as described in "Using Sessions" on page 37.
- Loaded layout file in the Calibre WORKbench main window.
- Performed an OPC run with **Fragment Markers** and **OPC Results** selected.

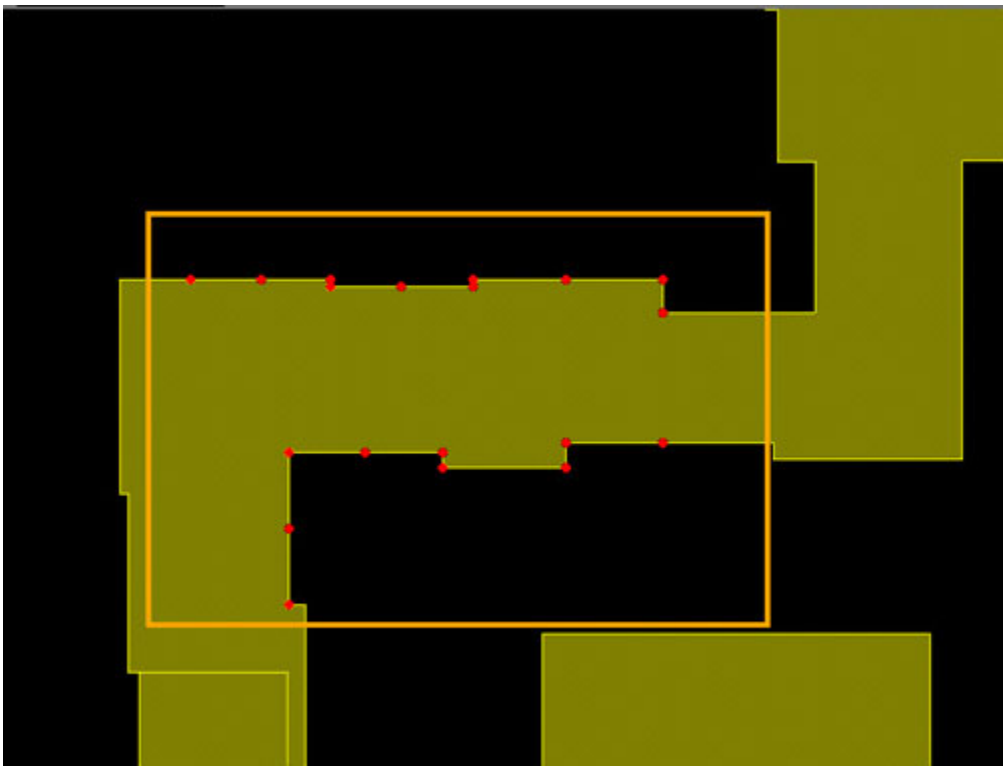
Procedure

1. (Optional) In Calibre WORKbench click **Select** and draw a bounding box around an area. If you do not select an area, the entire visible part of the layout is used as the selection area.
2. In the RET Flow Tool, click **Options** under the **Frag** button.
3. Fill out the fields as shown in Table 4-6.

Table 4-6. Frag Run Options Fields

Item	Description
Disable Tag Script Optimization (OPCpro session only)	This option displays all tags, whether they are used or not.
Show sites / measurement locations	Toggle the visibility of the site and measurement markers in the display window.
Clear previous results	Toggle whether previous Fragmentation tool markers are erased the next time you perform a Frag run.

4. Click **OK** once you have finished making your selections. The Frag tool runs and modifies the output in the viewer to show dots that are the endpoints of fragments in the selection window.



5. To see which fragmentation rules were applied to create the fragments, click a fragment and choose the **Edit > Fragmentation** menu option. Use the dialog box that appears to choose different groups of fragments to highlight.

Setting Fragment Display Properties

Once you have generated the fragments, you can set their general display properties.

Prerequisites

- Performed the steps described in “Displaying Fragments” on page 116.

Procedure

1. Click a fragment and choose the **Edit > Fragmentation** menu option. The Edit Fragmentation dialog box appears.
2. Click the **General** icon in the left side of the Edit Fragmentation dialog box.
3. In the Edit Fragmentation dialog box (General view), set the display properties for all fragments. The functional areas of the Edit Fragmentation dialog box (General view) are described in Table 4-7.

Figure 4-19. Edit Fragmentation Dialog Box (General)

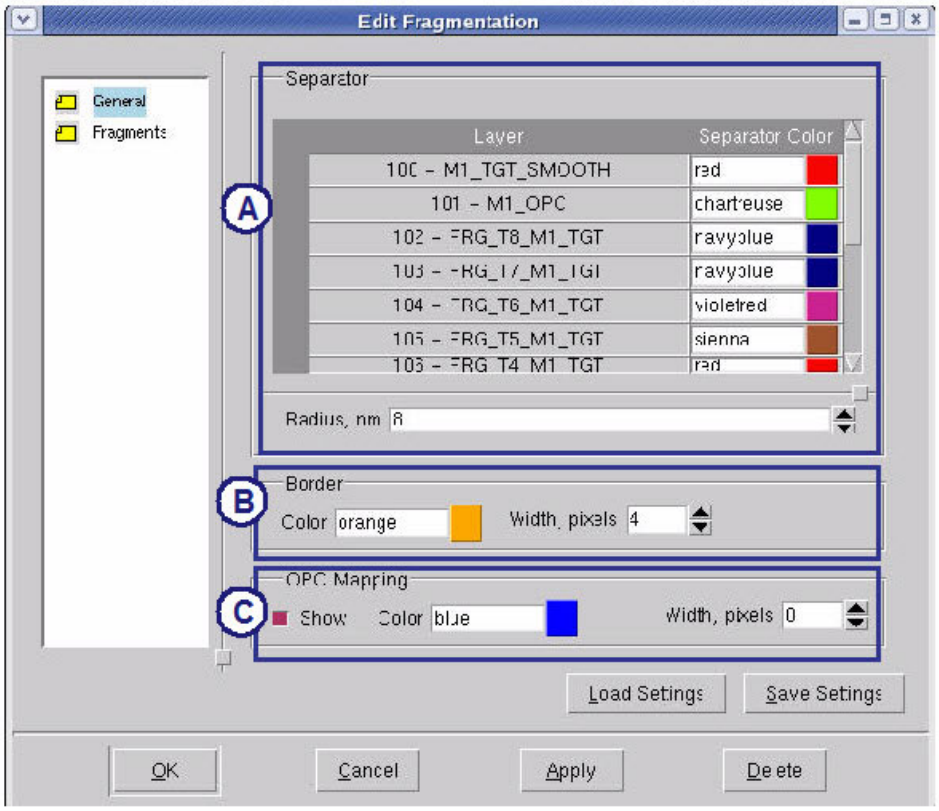


Table 4-7. Edit Fragmentation Dialog Box (General) Contents

	Item	Description
A	Separator	Assign the color and radius of separators (the dots that indicate the endpoints of a fragment).
B	Border	Select the color and width in pixels of the border lines.
C	OPC Mapping	Select the color and width in pixels of OPC Mapping separators. You can also turn off the mapping arrows.

Setting Display Properties for Tagged Fragment Groups

You can set the display properties for tagged groups of fragments, providing them with unique colors for easier identification on the layout.

Prerequisites

- Performed the steps described in “Displaying Fragments” on page 116.

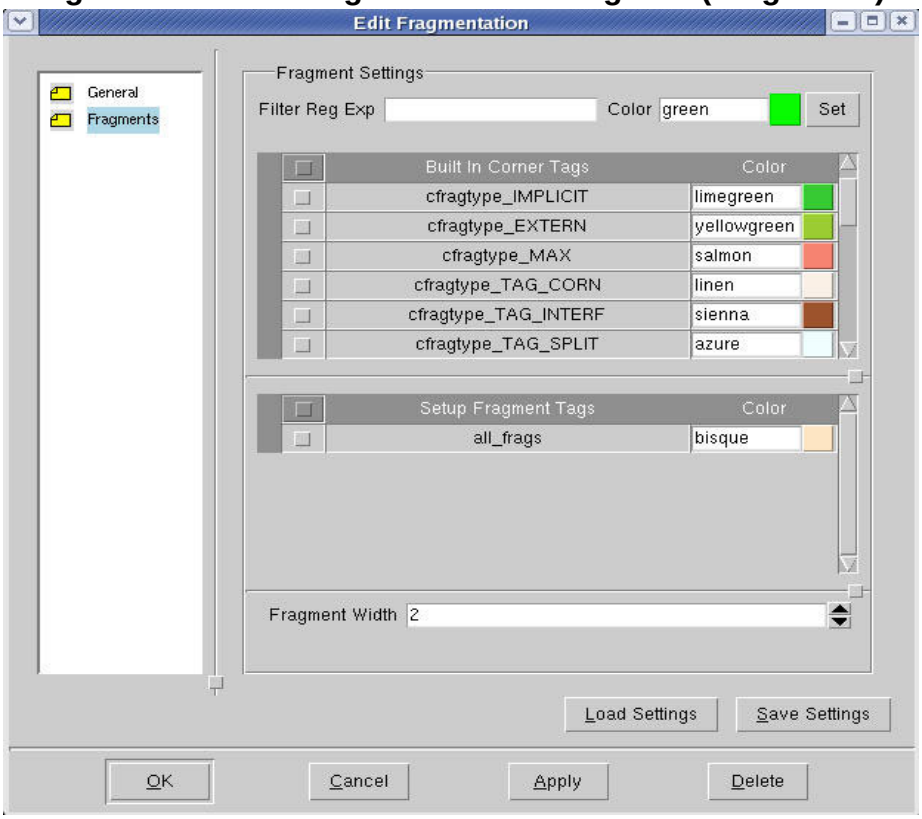
Procedure

1. Click a fragment and choose the **Edit > Fragmentation** menu option. The Edit Fragmentation dialog box appears.
2. Click the **Fragments** icon in the left side of the Edit Fragmentation dialog box.
3. In the Edit Fragmentation dialog box (Fragments view) (see Figure 4-20), set the set colors and widths for groups of tagged fragments.
4. Add tags to the list by selecting a fragment from one of the iterations of fragment layers.

   Although the Edit Fragmentation dialog box displays the entire list of built-in tags, if you select a tag that is not used by OPC, it will not highlight any fragments.

   This dialog box displays only the tags that are used by OPC in the region you select for fragmentation.

Figure 4-20. Edit Fragmentation Dialog Box (Fragments)



Fragment Information

The FInfo tool provides detailed EPE information on fragments.

Viewing Fragment Information

Each fragment in your layout contains information that can be useful for analysis. Use **FInfo** to display fragment information.

Prerequisites

- Configured the run for your session as described in the chapter, “Using Sessions”.
- Loaded layout file in the Calibre WORKbench main window.
- Completed the task “Fragment Generation” on page 116.

Procedure

1. In the viewer, select a layer containing a fragment iteration.
2. In the Calibre WORKbench toolbar, click **FInfo**.

3. The cursor changes to a hand; click a fragment to open the Fragment Information dialog box.
4. For dense OPC runs, in the Fragment Information dialog box, select the **Fragment** icon. For Calibre OPCpro setups, there are no icons but the information is otherwise the same.

   The Fragment view of the Fragment Information dialog box (see Figure 4-21) details the properties of a selected fragment. Additional information appears if PV Band PWOPC is used (see Figure 4-22).

Figure 4-21. Fragment Information Dialog Box (Fragment)

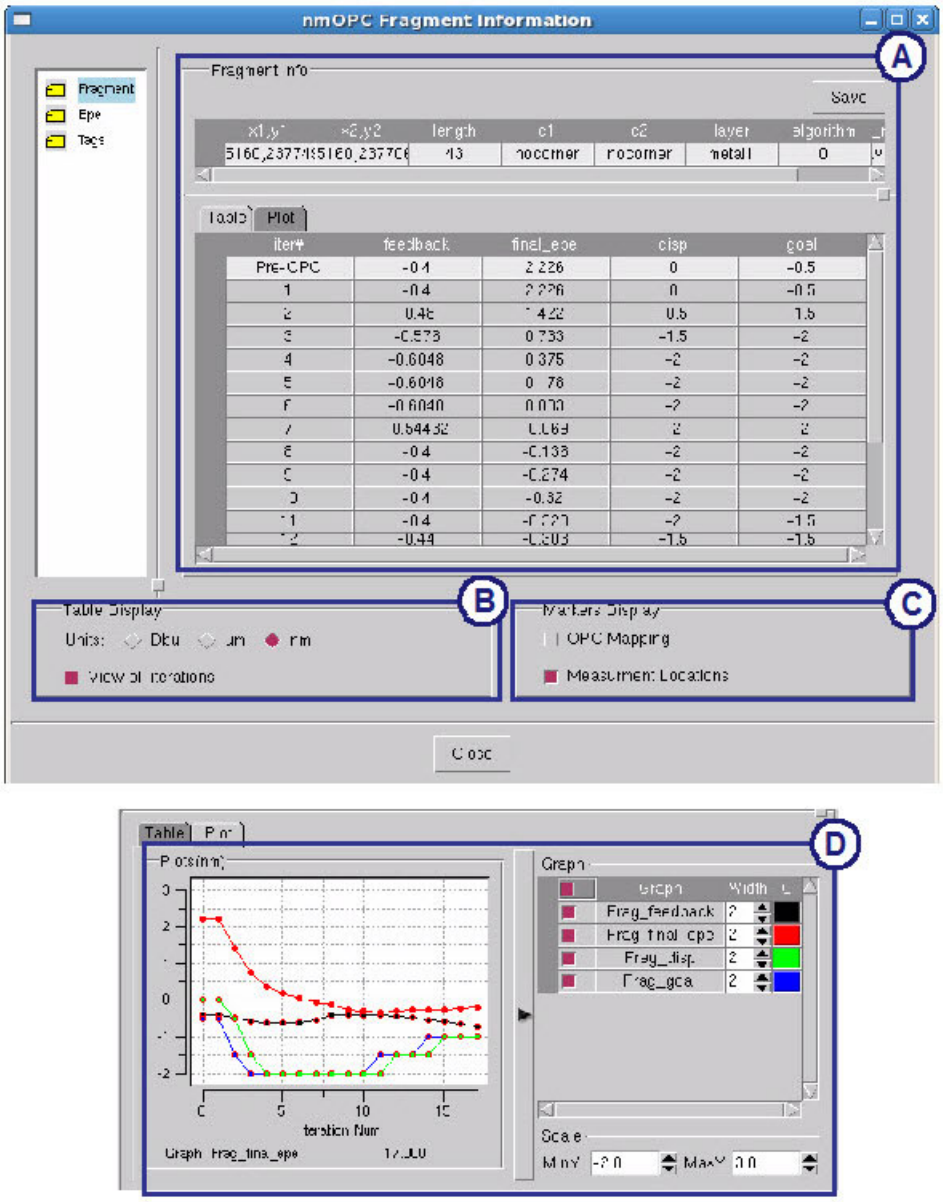


Figure 4-22. Fragment Information Dialog Box (Fragment) for PV Band PWOPC

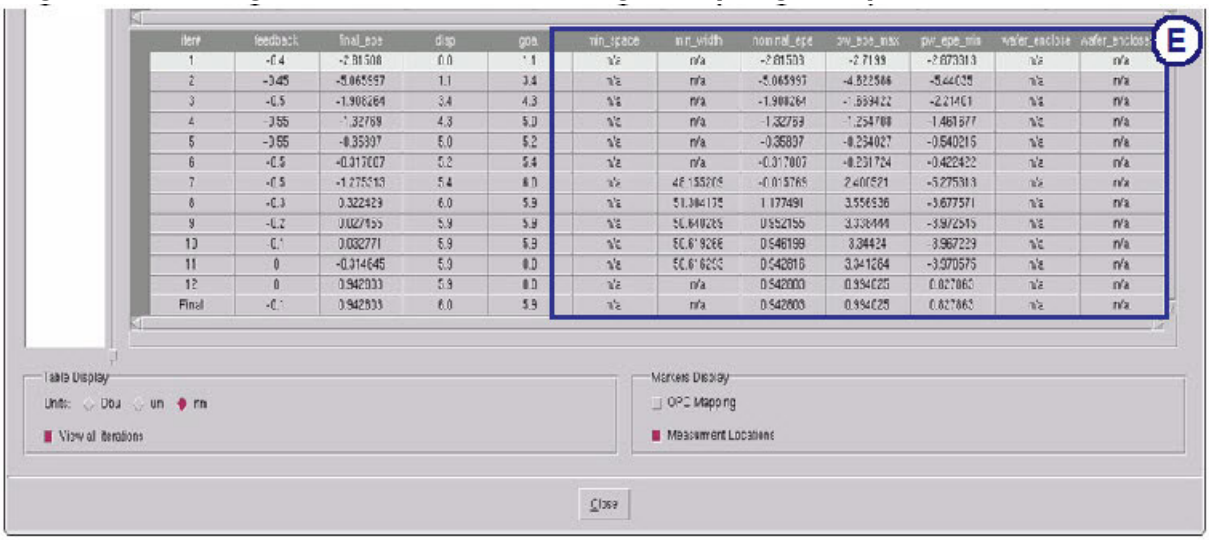


Table 4-8 describes the contents of this dialog box.

Table 4-8. Fragment Information (FInfo) Contents

	Item	Description
A	Fragment Info (**Table** tab)	Details the properties of a selected fragment, including final EPE, position, length, type, feedback, displacement, target (goal) position, and the metric used to measure the EPE. Click the **Save** button to save the fragment information to a CSV format file.
B	Display Units	Select display units in database units (dbus), microns (um), or nanometers (nm). You can view the information of the selected fragment for all iterations by enabling the **View all iterations** button.
C	Markers Display	You can select either of the following options: • **OPC Mapping** is for double-exposure masks, indicating where EPEs are remapped from a correction layer to the opc layer. • **Measurement Locations** indicate where the EPE measurements are taken.
D	Fragment Info (**Plot** tab)	Displays a plot graph of the selected EPEs (chosen in the Graph pane) versus iteration. You can change the scale of the plot by changing the MinY and MaxY settings.

Table 4-8. Fragment Information (FInfo) Contents (cont.)

	Item	Description
E	PVband PWOPC-specific information	Details fragment properties that are specific to PVband PWOPC, including: • Values for min_space and min_width • Values for nominal, minimum, and maximum values for process window EPEs • Settings for wafer_enclose and wafer_enclosedby

Viewing EPE Information for a Fragment

You can view the EPE information for a selected fragment in the Fragment Information dialog box.

Restrictions and Limitations

- Calibre OPCpro setups are not supported. View EPE information for Calibre OPCpro runs in the Fragment Info area of the FInfo dialog box.

Prerequisites

- Configured the run for your session as described in the chapter, "Using Sessions".
- Loaded a layout file in the viewer main window.
- Completed the task "Fragment Generation" on page 116.

Procedure

1. In the layout viewer, select a layer containing a fragment iteration.
2. In the Calibre WORKbench toolbar, click **FInfo**.

3. The cursor changes to a hand; click a fragment to open the Fragment Information dialog box.
4. In the Fragment Information dialog box, click the **EPE** icon.

   The EPE view of the Fragment Information dialog box (see Figure 4-23) details the EPE information about a selected fragment.

Figure 4-23. Fragment Information Dialog Box (EPE)

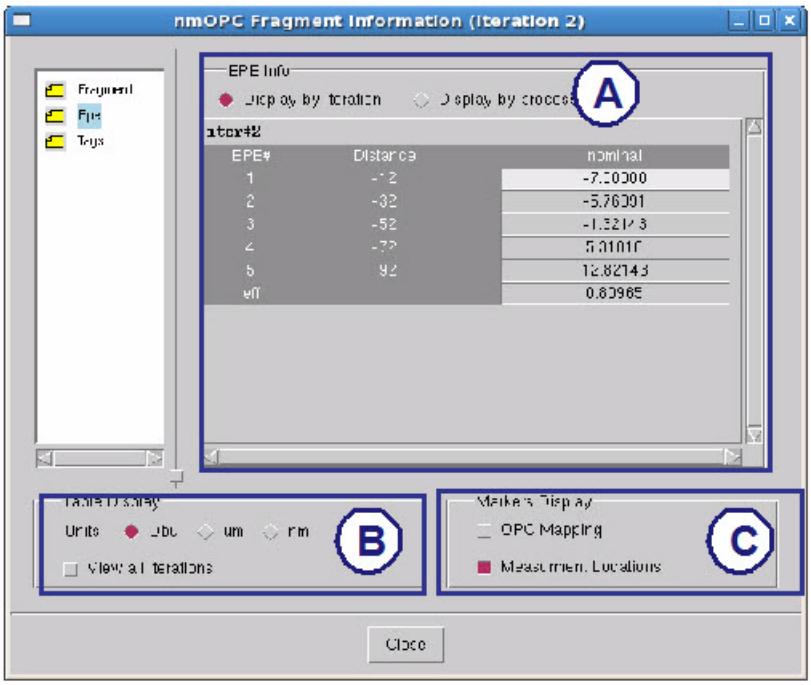


Table 4-9 describes the contents of this dialog box.

Table 4-9. Fragment Information (EPE) Contents

	Item	Description
A	EPE Info	Displays EPE information for each sampling point on the selected fragment. You can **Display by iteration** or **Display by process window**.
B	Display Units	Select display units in database units (dbus), microns (um), or nanometers (nm). You can view the information of the selected fragment for all iterations by enabling the **View all iterations** button.
C	Markers Display	You can select either of the following options: • **OPC Mapping** is for double-exposure masks, indicating where EPEs are remapped from a correction layer to the opc layer. • **Measurement Locations** indicate where the EPE measurements are taken.

Viewing Associated Tag Groups for a Fragment

You can find all the tag groups that your selected fragment belongs to through the Fragment Information dialog box.

Restrictions and Limitations

- Calibre OPCpro setups are not supported. View EPE information for Calibre OPCpro runs in the Fragment Info area of the FInfo dialog box.

Prerequisites

- Configured the run for your session as described in the "Using Sessions" chapter.
- Loaded a layout file in the viewer main window.
- Completed the task "Fragment Generation" on page 116.

Procedure

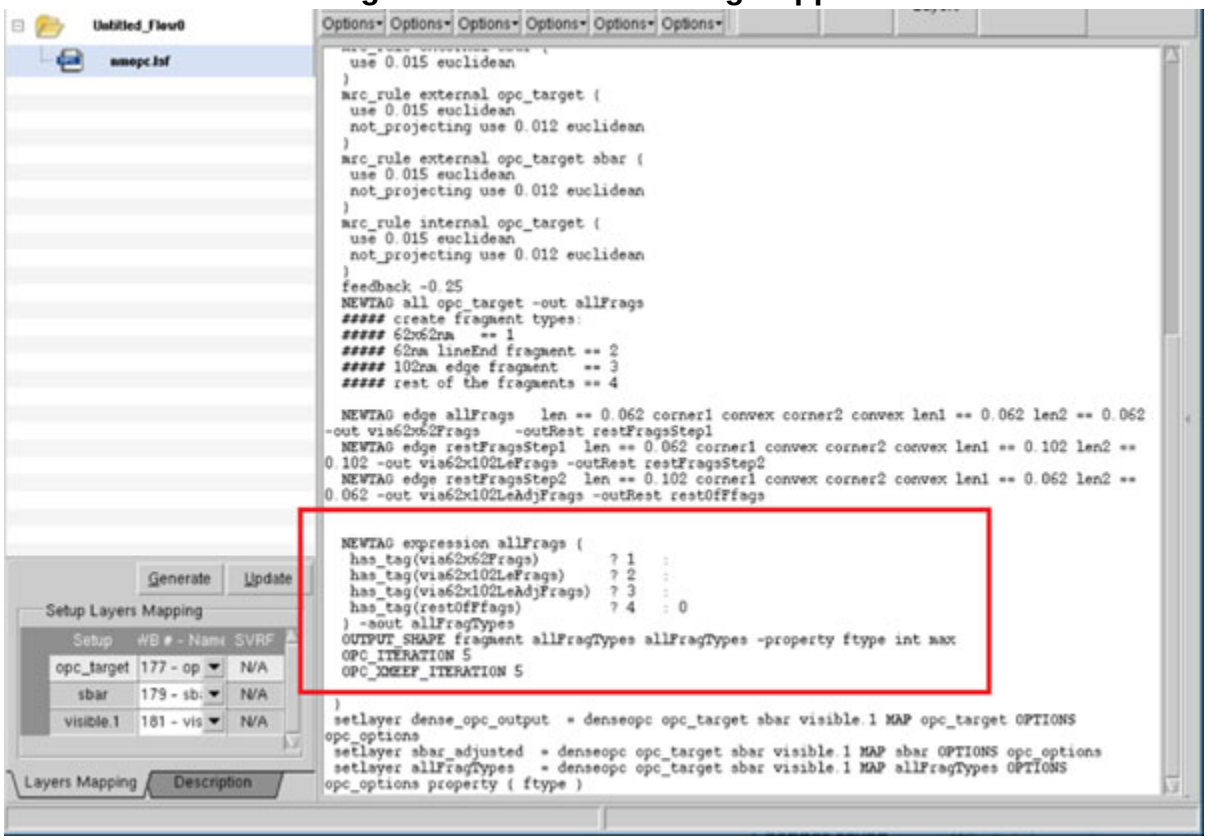
1. In the layout viewer, select a layer containing a fragment iteration.
2. In the viewer toolbar, click **FInfo**.

3. The cursor changes to a hand; click a fragment to open the Fragment Information dialog box.
4. In the Fragment Information dialog box, click the **Tags** icon.

Note

Annotated tags, as defined by the -annotated or -aout command options, can also be displayed in the Calibre nmBIAS and Calibre nmOPC Fragment Information windows. As an example, the annotated NEWTAG statement highlighted in Figure 4-24 (see the red box) can be viewed.

Figure 4-24. Annotated Tag Support



The Tags view of the Fragment Information dialog box details the tag information for the selected fragment. Figure 4-25 shows the four main sections:

Figure 4-25. Fragment Information Dialog Box (Tags)

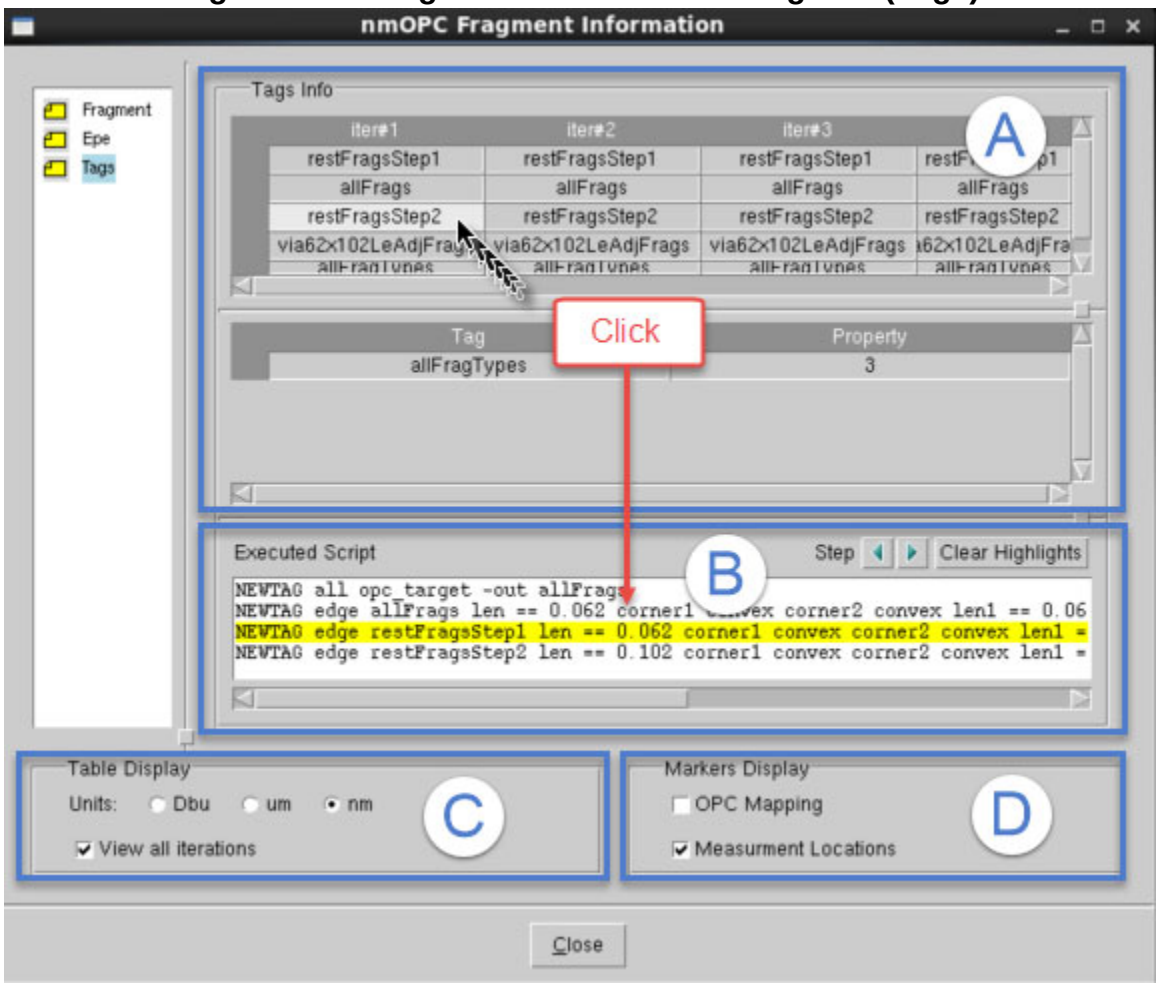


Table 4-10 describes the contents of this dialog box.

Table 4-10. Fragment Information (Tags) Contents

	Item	Description
A	Tags Info	Shows which tag groups the selected fragment belongs to, if any. Any tags you define appear only if you select the Fragment Markers for **All** iterations prior to running OPC.
B	Executed Tag Script	Highlights the tagging commands executed in the setup file. You can highlight and traverse the different fragments by clicking the **Highlight Frags** arrow keys, as well as clear the highlighting by clicking **Clear Highlights**.

Table 4-10. Fragment Information (Tags) Contents (cont.)

	Item	Description
C	Display Units	Select display units in database units (dbus), microns (um), or nanometers (nm). You can view the information of the selected fragment for all iterations by enabling the **View all iterations** button.
D	Markers Display	You can select either of the following options: • **OPC Mapping** is for double-exposure masks, indicating where EPEs are remapped from a correction layer to the opc layer. • **Measurement Locations** indicate where the EPE measurements are taken.

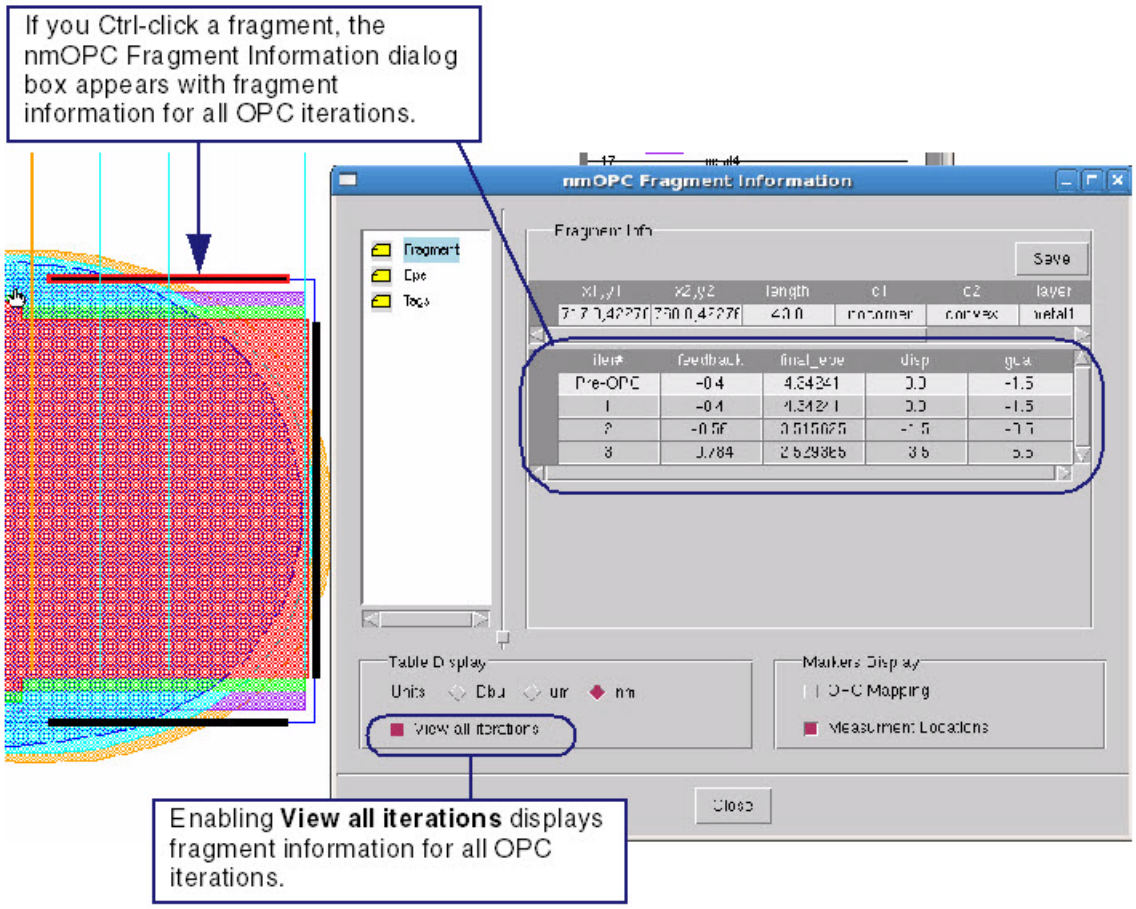
Displaying Fragment Information for Multiple Iterations

Calibre WORKbench can present side-by-side the OPC fragmentation information of selected iterations (for example, to inspect EPE trends).

Procedure

1. Run OPC from the RET Flow Tool with the **Fragment Markers** option enabled.
2. Make visible the layers containing the fragment marker layers corresponding to the desired OPC iterations.
3. Click the **FInfo** button.
4. To show information for all iterations that were output, click “View all iterations” in the Table Display area at the bottom of the Fragment Information dialog box, or ctrl-click the fragment (see Figure 4-26).

Figure 4-26. Fragment Information for All Iterations



Debugging Fragment Scripts

Use the RET Flow Tool fragment debugger to iteratively trace through a tagging script. The tool displays any fragments that a particular rule selects.

Restrictions and Limitations

- Fragment debugging is only supported in DENSEOPC sessions.

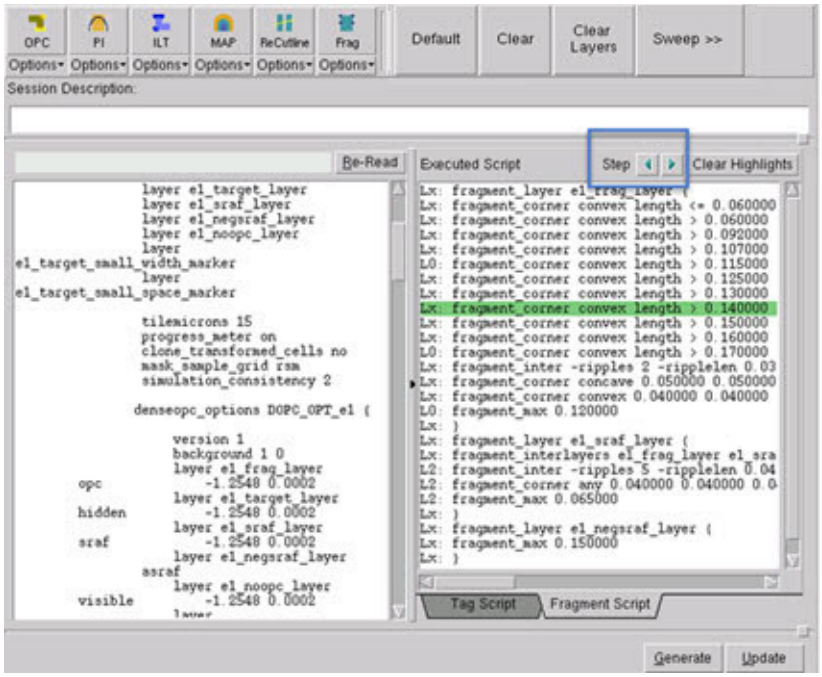
Prerequisites

- Configured the run for your session as described in “Using Sessions” on page 37.
- Loaded a layout file in the viewer main window.
- Completed the task “Displaying Fragments” on page 116.

Procedure

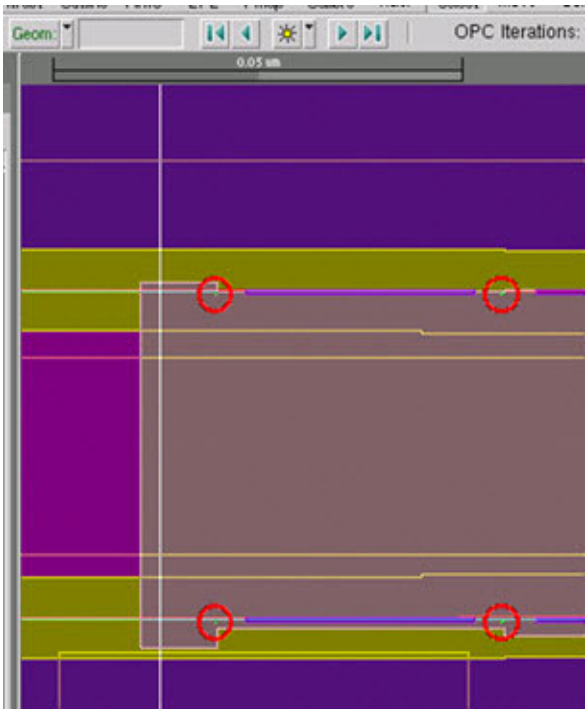
1. In the RET Flow Tool, click the black arrow on the right side of the display to open the **Tag Script** tab, then click the **Fragment Script** tab to open the fragment debugger.

2. In the fragment debugger, use the **Step** buttons to iterate through the fragment script.



Results

If a fragmentation rule created a fragment, stepping into that line in the fragmentation script highlights the fragment in the design file.



Running Tests on Multiple Values With the Sweeping Tool

Viewing the possible results of a Calibre nmOPC, Calibre MPC, Calibre nmSRAF, or Calibre mbSRAF run across multiple parameter values helps you see potential hotspots in your design. The Sweeping Tool generates a range of outcomes using combinations of the parameters you specify.

Prerequisites

- Calibre nmOPC, Calibre MPC, Calibre nmSRAF, or Calibre mbSRAF session file loaded into the RET Flow Tool
- The corresponding design file loaded into Calibre WORKbench
- A remote file for use with Calibre MTflex (optional)

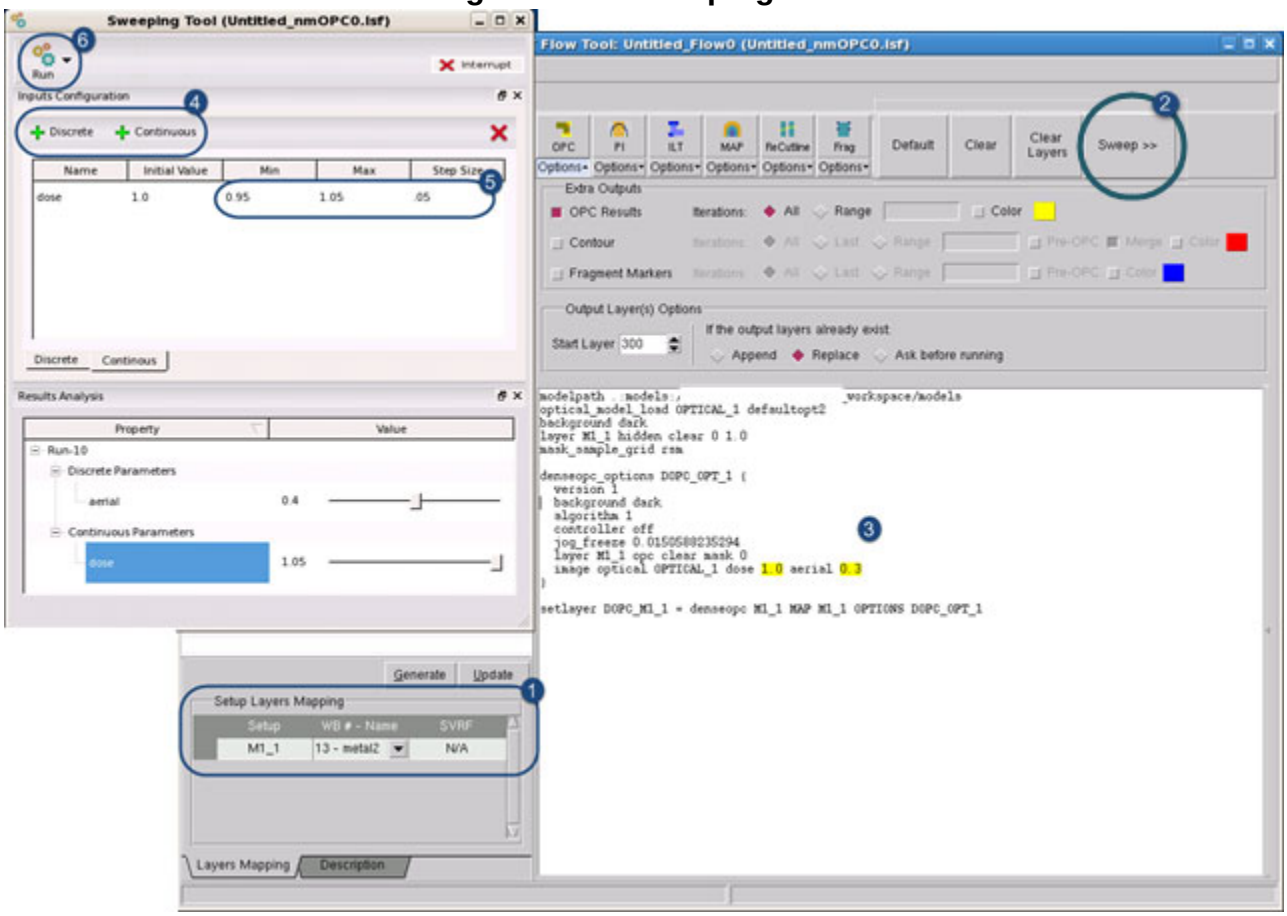
Note

To use the Sweeping Tool in TURBOflex mode, you must invoke Calibre WORKbench with the -remote argument.

Procedure

1. In the RET Flow Tool, ensure that the layers of interest are correctly mapped in the Setup Layers Mapping section. (If a layer is not mapped, or if its geometry is not visible in the design window, the sweeping tool does not generate any output.)
2. Click the **Sweep** button to raise the Sweeping Tool window, and position the window so that the RET Flow Tool window is visible.

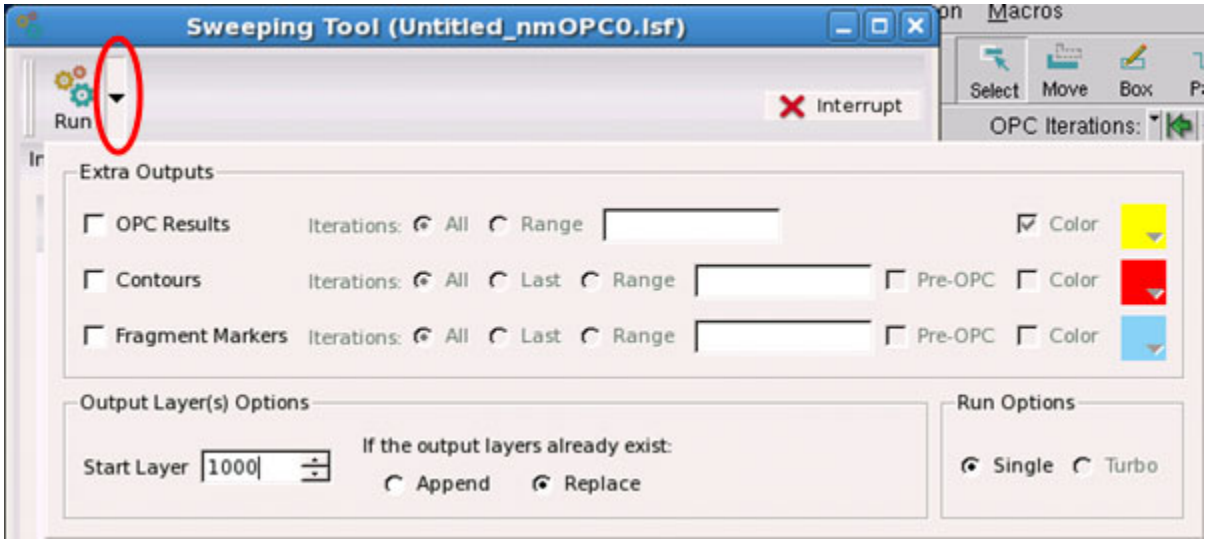
Figure 4-27. Sweeping Tool



3. In the text pane of the RET Flow Tool, double-click to highlight the value after a parameter. (Each highlight remains selected for every selected value.)
4. In the Sweeping Tool, click either **Discrete** or **Continuous** to add the parameter to the window.
   - **Discrete** — Creates runs based on a specific list of discrete values.
   - **Continuous** — Creates runs based on a list iterator (start ... end, step sequence).
5. Click the value's text field(s) to fill in the parameter values to sweep.
   - For Discrete parameters, click the Values List field and enter a list of values separated by semicolons.

- For Continuous parameters, click the Min, Max, and Step Size fields and enter a value range and a step iterator.

6. Click the arrow next to the **Run** button. The Sweeping Tool run controls appear.



Set the controls as needed, then click the **Run** button.

Note

Clicking **Interrupt** stops the sweeping tool run; no intermediate data is created.

Note

The Turbo option in the Run Options section is enabled only when you invoke Calibre WORKbench with the -remote argument and a remotefile. It runs the tool in distributed mode.

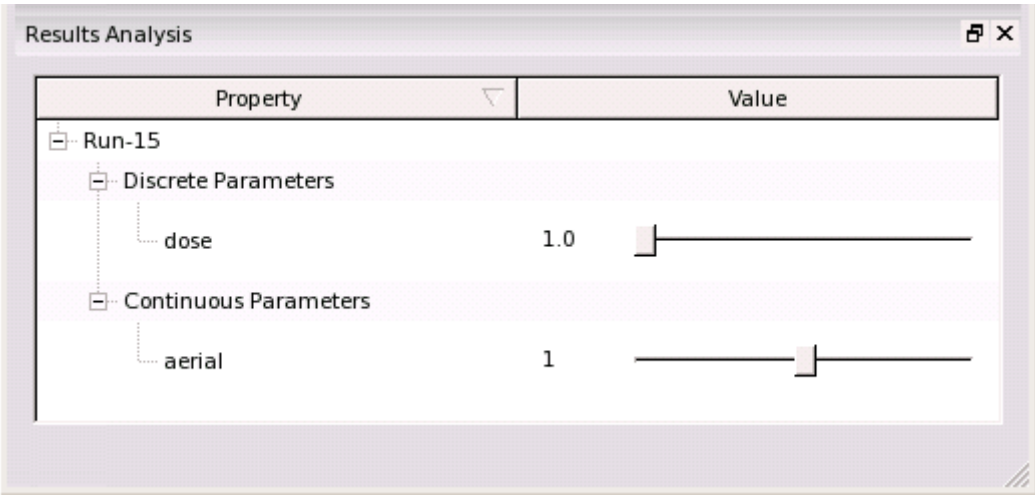
Results

The Sweeping Tool generates a list of all possible combinations of the values you specified, and runs the tool on each combination, adding layers to the design file using the following criteria:

- If the OPC Results, Contours, or Fragment Markers options were chosen in Step 6, each combination has a set of layers generated for the OPC, contour, or fragment operation.
- Parameters are appended to the name using the string syntax "*layer*" with a set of appended variations of "*_parameter*::*value*." For example, in Calibre nmOPC, sweeping a discrete dose value of 1.0 1.1 and a continuous aerial value of 0.95 to 1.05 step 0.05 on a layer M1_1 gives the following result layers:
  - DOPC_M1_1_dose::1.0_aerial::0.95
  - DOPC_M1_1_dose::1.0_aerial::1.0
  - DOPC_M1_1_dose::1.0_aerial::1.05

- DOPC_M1_1_dose::1.1_aerial::0.95
- DOPC_M1_1_dose::1.1_aerial::1.0
- DOPC_M1_1_dose::1.1_aerial::1.05

- Use the sliders in the Results Analysis part at the bottom of the Sweeping Tool window to show and hide only the layers for the selected parameter values. For example, with the settings shown in the following figure, only layers containing the string “dose::1.0_aerial::1.0” are set as visible.



Index

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.