SIEMENS EDA

Calibre® WORKbench™: RET Flow Tool (RFT) v2.0 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 RFT v2.0 Environment

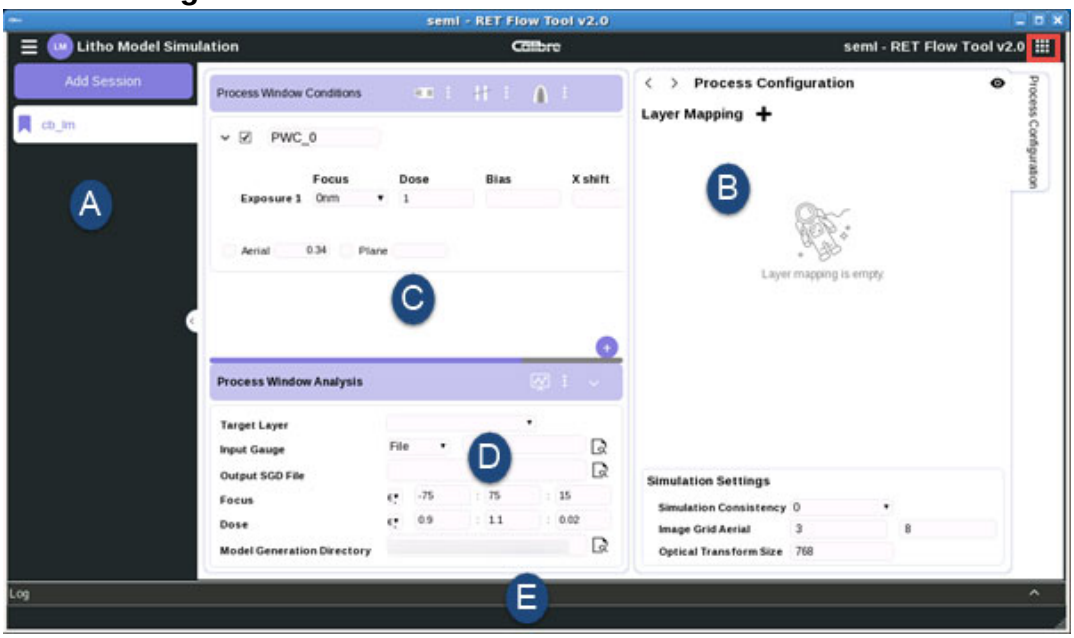
The Calibre® RET Flow Tool (RFT) v2.0 is available for Calibre® WORKbench™ releases 2019.3 or higher. The Calibre RFT v2.0 environment is a central access point for selected Calibre RET lithography tools, and is designed for the recipe developer to assist in debugging.

Calibre RFT v2.0 GUI

The Calibre RFT v2.0 GUI (accessed through Calibre WORKbench by selecting **Litho > RET Flow Tool v2.0**) uses dynamic components and lists to show only active and relevant information.

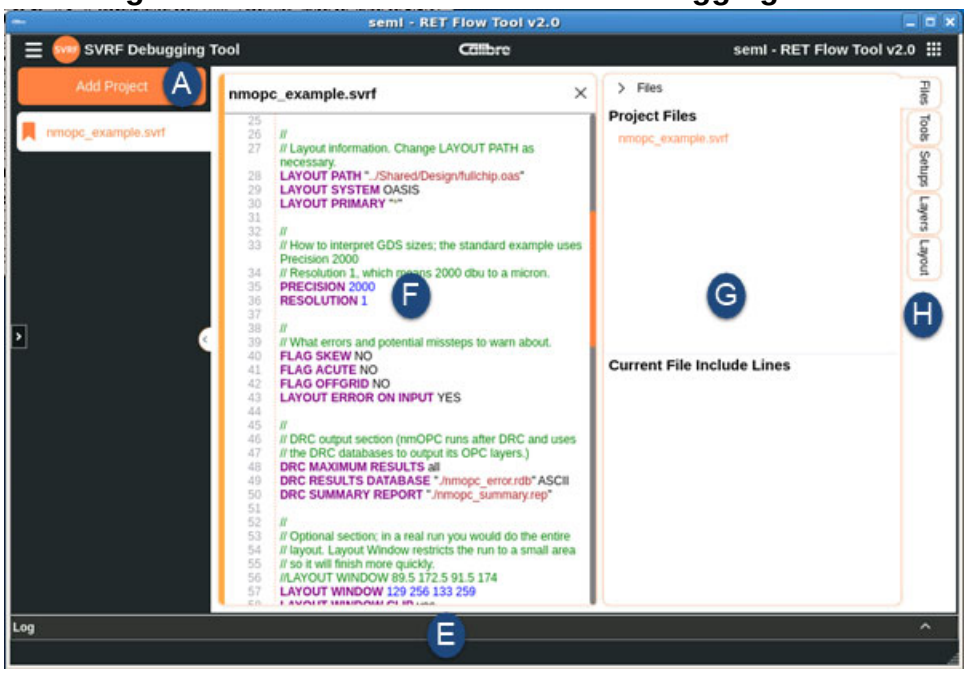
In Simulation Mode (Litho Model and Sparse Simulation modes), using the tool requires you to configure a session and process configuration before creating one or more process conditions.

Figure 1-1. Calibre RFT v2.0 GUI - Simulation Mode



In Debugging Mode (SVRF Debugging Tool and Setup Session modes), you open and edit existing SVRF, Calibre® nmOPC™, and Calibre® OPCverify™ scripts.

Figure 1-2. Calibre RFT v2.0 GUI - Debugging Mode



Reference	GUI Element	Description
A	Session Management	Shows the list of sessions active in the GUI. Use the modal selector in the upper right hand corner menu (red box in the top figure) to select a mode: 
B	Process Management	Sets or changes process component information. Models and layer definitions are set in this area.
C	Process Window Conditions	Lists defined process window conditions. Operations (such as print image, intensity cutline, and image map) are run on the list.
D	Process Window Analysis	Generates a process window from a litho model, and displays the results in the PW Plot tool. This operation can be performed without an OPC recipe.
E	Log	Shows operation results.
F	Source Editor	Shows SVRF, Calibre nmOPC, or Calibre OPCverify source files when loaded in Debug Mode.
G	Source Browser	Shows components for the file in the Source Editor based on the Info Controls. Hovering over an item may reveal additional controls.
H	Browser Controls	Selects which group type is visible in the Source Browser.

Calibre RFT v2.0 Fundamental Concepts

There are several fundamental concepts that are used by the Calibre RFT v2.0 environment.

- **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 related *Lithomodel* file. The *Lithomodel* file also describes a set of process and mask conditions.
- **Process Window Analysis** — Generates a super gauge data file (*.sgd*) that includes the CD information at all gauges and all conditions. The PW Plot tool also uses this information.
- **Process Window Condition** — Calibre RFT v2.0 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. You can save these settings in a session file. Changes to fields in Calibre RFT v2.0 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 RFT v2.0 Prerequisites

The Calibre RET Flow Tool v2.0 is part of Calibre WORKbench. You must be able to run Calibre WORKbench (referred to generically as “a viewer”). Calibre RET Flow Tool v2.0 does not require a separate license.

In addition, for easy use of the Calibre RET Flow Tool v2.0, the following are recommended:

- A design file to run simulations on.
- Familiarity with Calibre viewers. This manual does not describe the viewer interface outside of Calibre RET Flow Tool v2.0.
- For tuning tool flows and OPC corrections, familiarity with SVRF and the languages used by the particular tools.

Verifying Models With Simulation Flow

One of the primary uses for simulation is verifying that the process window covered by the optical model produces reasonable results for a particular layout.

Prerequisites

- A layout in GDS or OASIS format
- A litho model or global litho model to be verified

Procedure

1. From a console prompt, invoke Calibre WORKbench and load your layout file:

   ```
   calibrewb your_layout_file
   ```

2. Select an area of the layout you are interested in, because the simulation tools can take excessive amounts of time when simulating over a full chip layout, depending on the complexity of the chip.
3. Open the Calibre RFT v2.0 GUI (**Litho > RET Flow Tool v2.0**).
4. Add a session (litho model or sparse) and map layers.
5. Configure at least one process window condition.
6. Depending on the type of session you created, you can configure the tool settings available for that session.
7. Run tools that are available for your session.

Examples

The step-by-step example demonstrates the flow using only a set of models and a design to quickly run an aerial image simulation.

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.

1. From a console prompt, invoke Calibre WORKbench and load the *fullchip.oas* file, located in the *Shared/Design* directory.

   ```
   calibrewb fullchip.oas
   ```

2. Turn off all layers but the poly layer (layer 9). Zoom the viewer window in to some visible geometry.
3. Open the RET Flow Tool by selecting **Litho > RET Flow Tool v2.0**.
4. Click **Add Session** and select **Litho Model** from the list of buttons.

   Navigate to the litho model in *Module5/Modeling/Models/dense_lm* and select it.

5. Select the layer mapping for the simulation by clicking the + button next to **Layer Mapping**, then select the mask from the litho model mask list (Mask 0, Layer 0 is the only option available in this example).
6. Click the Select design layer field and assign the WORKbench Layer 9 - Poly to it.
7. Test the run by clicking the Print Image button.

Related Topics

Generating a Print Image

Running an Intensity Cutline Simulation

Creating a Contour Map

Running a Process Window Analysis

Debugging Calibre Rule Files Flow

Use the Calibre RFT v2.0 tool to browse and debug Calibre Rule files. Its component browser lets you search by components, edit the file and environment variables, and save the results as a setup file.

Prerequisites

- A previously-created SVRF rule file, Calibre nmOPC runscript, or Calibre OPCverify runscript.
- The corresponding layout file output from running the rule file or script.

Procedure

1. Change directories to the location of your SVRF or Calibre nmOPC/OPCverify files.
2. Invoke Calibre WORKbench, and run Calibre RFT v2.0 (Litho > RET Flow Tool v2.0)
3. Select one of the Debugging Modes using the grid button in the upper right corner:
   - **SVRF Debugging Tool** — For SVRF files.
   - **Setup Session** — For Calibre nmOPC or Calibre OPCverify setup command files.
4. Hover the mouse over the **Add Project** button (SVRF Debugging Tool) or **Add Session** button (Setup Session).
5. Load a file into the editor.
6. Use the right side browser controls to work with the displayed file.

Related Topics

SVRF Debugging Tool Session

Setup Session

Chapter 2
Using Sessions

The Calibre RFT v2.0 environment works with sessions, which pair process models with user-defined process conditions so that simulations can be run on them.

This release has four types of session:

- **Litho Model Session —** Uses the newer *Lithomodel* process model storage paradigm. Simulates using the dense OPC engine.
- **Sparse Session —** Uses the older separate model paradigm and simulates using the sparse OPC engine.
- **SVRF Debugging Tool —** Uses a SVRF rule file in browse mode to quickly find information in large rule files.
- **Setup Session —** Uses Calibre nmOPC or Calibre OPCverify setup session files. Files are editable and executable.

Litho Model Simulation Session

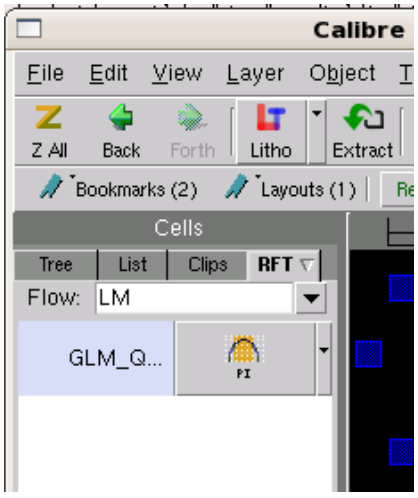
The Litho Model session type is designed to work with the current generation of Calibre tools. It supports normal litho models and EUV type (global) litho models.

The following operations are available for a litho model session once you configure it:

Table 2-1. Litho Model Simulation Operations

Operation	Control	Location	Description
"Generating a Print Image" on page 41	**Print Image**	Process Window Conditions pane	Simulate OPC effects on geometries on opc layers that lie partially or fully within a selected area.
"Running an Intensity Cutline Simulation" on page 44	**Cutline**	Process Window Conditions pane	Compare one or more simulations along cutlines.
"Creating a Contour Map" on page 52	**Map**	Process Window Conditions pane	Create a contour map of the light intensity at the surface of the resist.
"Running a Process Window Analysis" on page 54	**Process Window Analysis**	Process Window Analysis pane	Generate a super gauge data (*.sgd*) file of the simulated gauges.

The sessions you create in LM mode also appear with relevant tool buttons in Calibre WORKbench in the Cells pane on the left.



Open a Litho Model Session

Create a Litho Model session using a litho model (LM) or global litho model (GLM), a setup file, or load a previously saved session.

Prerequisites

- A layout open in Calibre WORKbench
- A litho model or global litho model

Procedure

1. Open the Calibre RFT v2.0 GUI.
2. Click the mode selection button in the upper right corner (red box in the figure), then select **Litho Model Simulation** as the session type.

Figure 2-1. Open LM Simulation Session Using a Litho Model

3. Hover your cursor over **Add Session**. The list of available session loading commands appears. Select an item from the list:
   - **Litho Model** — Loads a litho model from the disk.
   - **Session File** — Loads a previously-saved litho model session file.
   - **Setup File** — Loads a previously-saved setup file.
4. Navigate to the appropriate file, then click **Choose**.

Results

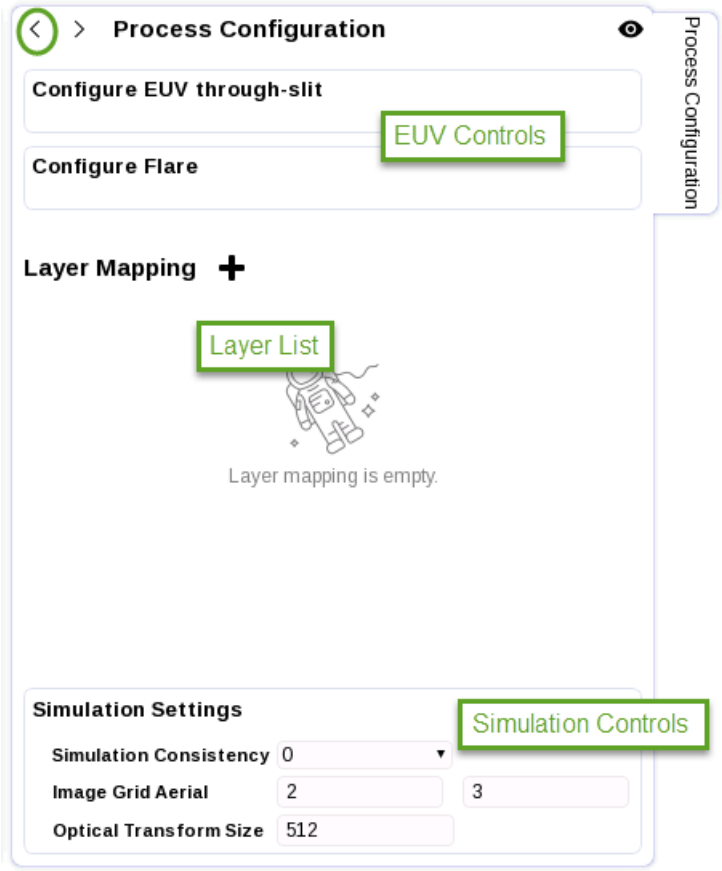
The selected item is loaded into the Calibre RFT v2.0 GUI. The new session is named using the name of the litho model or global litho model you loaded.

If you are loading a litho model, you must perform additional configuration tasks, as described in “Configuring the LM Process Management Section” on page 19.

Configuring the LM Process Management Section

When loading a litho model, the LM simulation session requires you to map layers and set up your simulation parameters in the Process Configuration section on the right side of the RFT v2.0 GUI. The available parameters depend on the type of model that was loaded.

Figure 2-2. Litho Model Process Configuration Panel



Prerequisites

- A layout open in Calibre WORKbench.
- An open Calibre RFT v2.0 LM simulation session with at least one model loaded. See "Open a Litho Model Session" on page 19.

Procedure

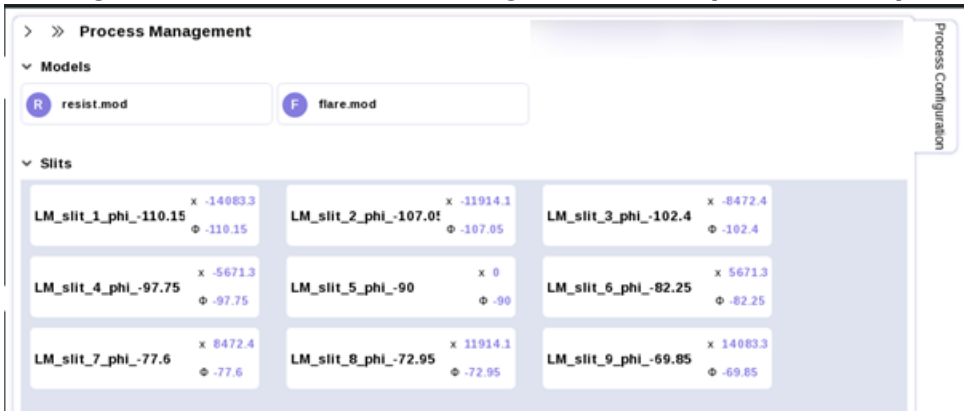
1. For litho models, the model configuration is done automatically when you load a litho model; the Process Model Configuration section is mostly present for viewing the model contents.

   - Clicking the arrow on the left (<) opens a flyout panel (Process Management) where you can view the process model contents and set the design layer used for the mask; however, this action is more easily performed in the Layer Mapping List.

     Note

     The Mask panel that lets you set the design layer for EUV models only appears if you click one of the Slits listings; setting the design layer in the Mask panel sets the layer for all slits.

Figure 2-3. LM Process Management View (EUV Shown)



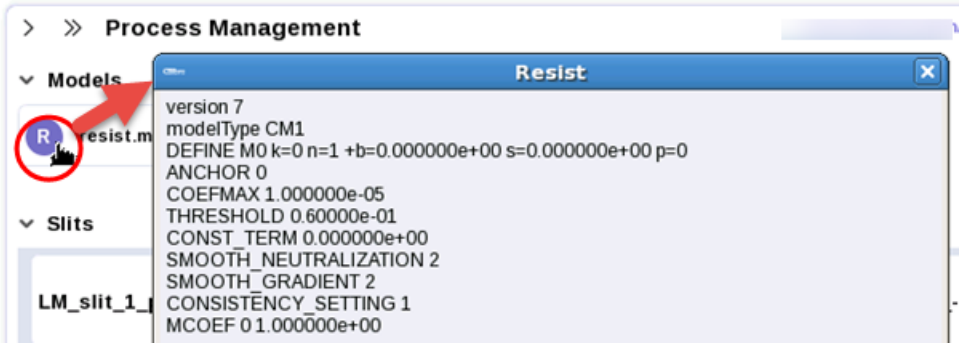
Clicking the arrow on the left (>) of this flyout panel returns to the Process Configuration mode.

- Clicking the eye symbol on the right displays the process model contents in a more compact format. Clicking the X in the upper right of compact display mode returns to the Layer Mapping List view.

Note

Certain icons in the expanded views (the **R** in the resist model line, the **F** in the flare model line, and the various defocus models) can be selected. Click a model button to open up a window showing the model definition file.

Figure 2-4. Example: Resist Model Content Review



2. If your litho model is EUV, the two EUV / GLM control bars (**Configure EUV through-slit** and **Configure Flare**) are visible. Hovering over the bar expands the control.

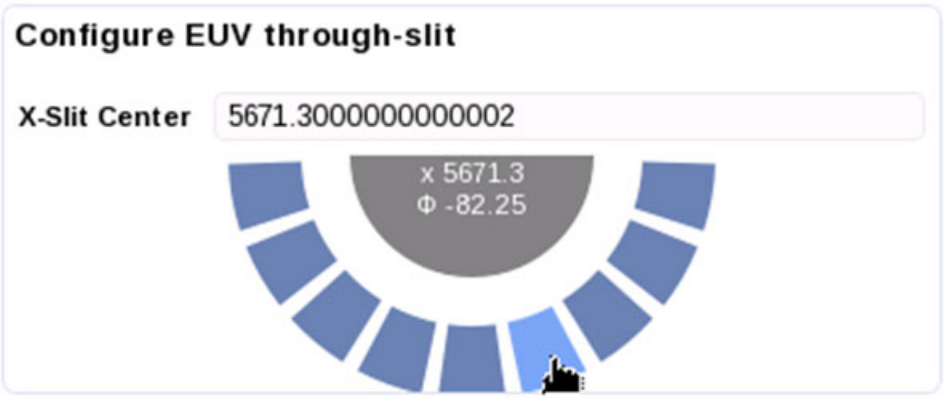
Note

DUV models do not feature these control bars; non-EUV users can skip to step 3.

- **Configure EUV through-slit** — Optional. Changes the X-slit Center setting.

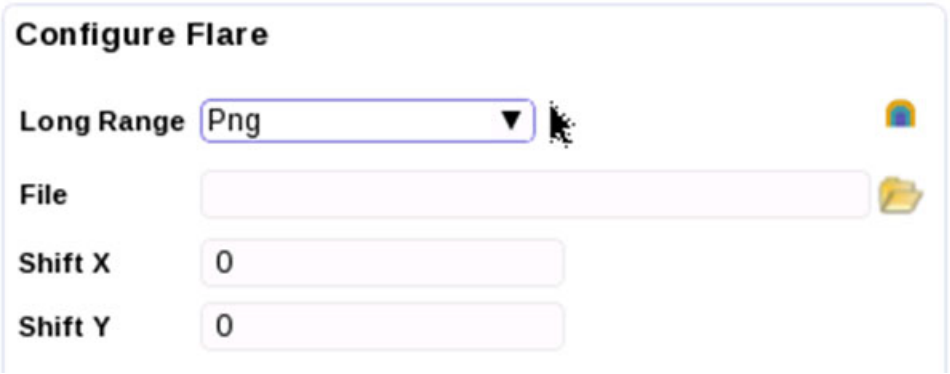
While you can enter a value manually in the X-Slit Center field, clicking the dark blue half-circle opens a selection wheel; hovering the cursor over each choice displays the x and phi setting for that slit. Clicking a button sets the selected X and phi to be the new center slit.

Figure 2-5. Expanded Configure EUV through-slit Panel



- **Configure Flare** — Optional. Sets the flare model type and any additional information needed.

Figure 2-6. Expanded Configure Flare Panel

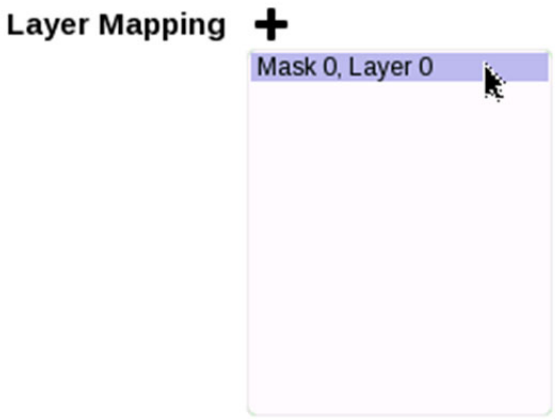


Changing the Long Range type changes the controls in the panel.

- **None** — Ignores the flare model for process window conditions.
- **PNG** — Enables the use of a flare map file with X and Y offsets. This type also adds a flare map simulation button to the panel.
- **Constant** — Enables setting a constant flare value.

3. Map your design layers in the Layer Mapping List.
   a. Click the Layer Mapping + button, then choose from the list of the mask and layer definitions in the loaded litho model.

Figure 2-7. Select Mask/Layer Pair



b. In the entry that appears, map the mask/layer pair to the correct design layer (this should be the simulation layer).

Note

You must specify at least one layer mapping for print image operations. You can map multiple layers for “Mask 0 Layer 0.”

4. (Optional) Modify the Simulation Settings. These are the simulation settings the Calibre OPCverify engine uses.

Note

Changing these settings is recommended for advanced users only.

- **Simulation Consistency** can be set to 0, 1, 2, or euv. The default value is 0. If you set a value other than 0, Calibre WORKbench uses its own internal values and ignores the **Image Grid Aerial** and **Optical Transform Size** settings.

Table 2-2. Simulation Consistency Non-Zero Setting Overrides

Value	Image Grid Aerial Setting	Optical Transform Size Setting
0 (default)	• EUV: 2 3 • DUV: 3 8	• EUV: 512 • DUV: 768
1 (improves output consistency)	3 8	384
2 (faster simulation time)	1 2	512
euv	1 1	1024

- The **Image Grid Aerial** parameter specifies the numerator and denominator values for the resist computation grid as the upsampling factor relative to the Nyquist grid. For the EUV process, the default values are 2 for the numerator and 3 for the denominator. For the DUV process, the default values are 3 for the numerator and 8 for the denominator.
- The **Optical Transform Size** default value is 512 for the EUV process, and 768 for the DUV process. It is used to specify the number of Nyquist simulation pixels in each simulation frame.

Results

Changes to the Process Configuration panels are applied immediately. The new settings are used the next time you perform any tool operation.

Saving a Litho Model Simulation Session

You can save the litho model (LM) simulation setup or session status. This enables you to resume your simulation in another session or share with others. Saving your session as a setup file uses Calibre OPCverify commands.

Prerequisites

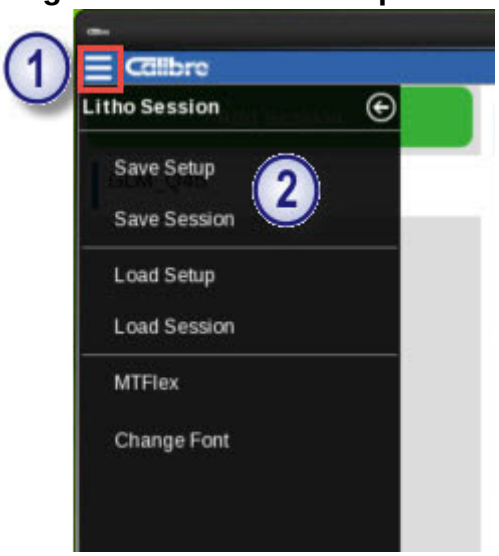
- A layout open in Calibre WORKbench
- An open Calibre RFT v2.0 LM simulation session

Procedure

1. Open the tool menu by clicking the menu button next to the "Calibre" logo at the top left of the Calibre RFT v2.0 screen.

   See the red box in Figure 2-8.

Figure 2-8. Tool Menu Options



2. Select either **Save Setup** or **Save Session**.

Note

Saving a setup file is only possible after all layers have been mapped.

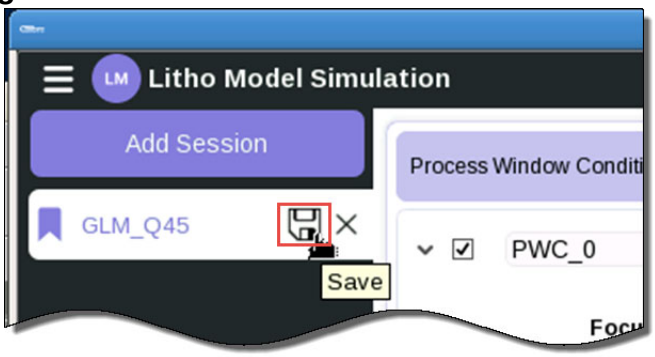
3. Navigate to the location you want to save your file, name it, then click **Save**.

Tip

You can also save a session by hovering your cursor over the LM simulation session name, clicking the **Save session** icon, then naming the file. This only applies to session saving, not to setup file saving.

See the red box in Figure 2-9.

Figure 2-9. Alternative Save LM Session Method



Results

The LM simulation session or setup file is saved in ASCII format.

Sparse Simulation Session

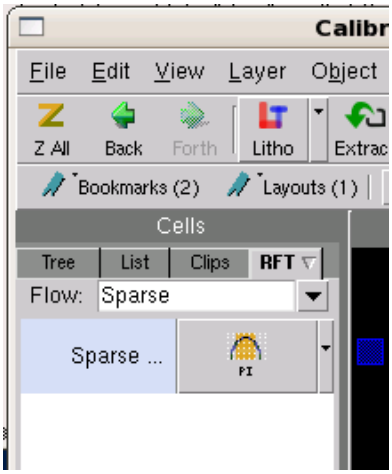
The Sparse simulation session enables you to individually load optical and resist models and define process conditions. This session corresponds to the legacy sparse OPC tool operations.

Currently, only one operation is supported in Sparse sessions.

Table 2-3. Sparse Simulation Operations

Operation	Control	Location	Description
"Generating a Print Image" on page 41	**Print Image**	Process Window Conditions pane	Simulate OPC effects on geometries on opc layers that lie partially or fully within a selected area.

The Sessions you create in Sparse mode also appear with relevant tool buttons in Calibre WORKbench in the Cells pane on the left.



Open a Sparse Session

Create a new Sparse session by using existing models, or load a previously saved session.

Prerequisites

- A layout open in Calibre WORKbench
- At least one resist model and one optical model

Procedure

1. Open the Calibre RFT v2.0 GUI.

2. Click the modal selection button in the upper right corner (red box in the figure). Choose **Sparse Simulation** as the session type.

Figure 2-10. Open Sparse Simulation Session

3. Hover your cursor over **Add Session**. Choose one of the following options:

   - **New** — If you do not have a saved session file, choose this option. The GUI automatically expands the Process Configuration panel on the right.

     Continue with the task, "Configuring the Sparse Process Management Section" on page 27.

   - **Session File** — If you have a previously-saved session file from Calibre RFT Flow Tool v2.0, choose this option to navigate to and load it.

Configuring the Sparse Process Management Section

Sparse sessions require you to manually load resist and optical models and map the mask layers to define processes, because they lack a litho model directory.

Note

Sparse Process Management starts with the Process Configuration panel open. This differs from the Litho Model Process Management task, where the process models are already set.

Prerequisites

- A layout open in Calibre WORKbench.
- An open Calibre RFT v2.0 Sparse simulation session. See "Open a Sparse Session" on page 26.

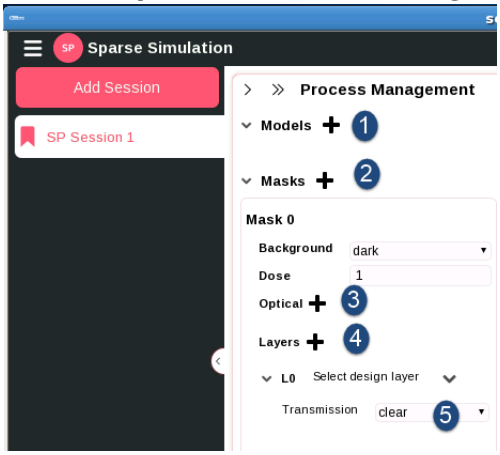
Procedure

1. Add the resist model. Click on the **+** button next to **Models**. Use the file navigator to select a previously-generated Calibre WORKbench resist model.

   Note

   Only one resist model is permitted in a sparse session. To change the model, click the recycle arrow.

Figure 2-11. Sparse Process Configuration



2. Define the mask background and dose. By default, Sparse sessions must contain at least one mask (shown as Mask 0). You can optionally click the + button next to **Masks** to add mask cards.

   All mask cards have **Background** and **Dose** fields.

   a. To change the default **Background**, click the adjacent field and select between:

      - **dark** — For use with negative tone masks
      - **clear** — For use with positive tone masks
      - **atten** — An additional field for **attenuation factor** is available with this choice
      - **complex** — Additional fields for **real** and **imaginary** are available with this choice

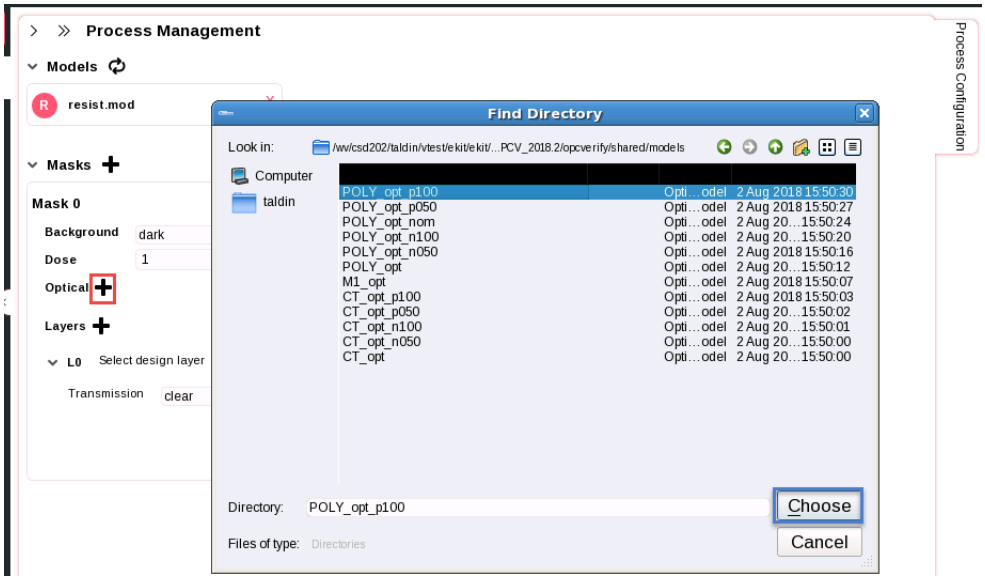
      The default **Background** is **dark**.

   b. To change the default **Dose** value, click the adjacent edit field and type the dose as a real number. The default **Dose** value is 1.

   Note

   You can click the red **X** in the top right corner of an added mask card to delete it. Mask 0 cannot be deleted.

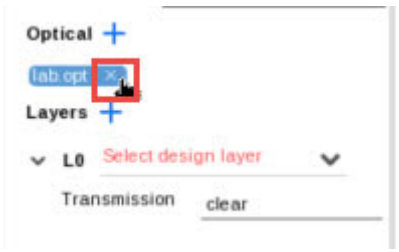
3. Click the + next to **Optical** to add one or more optical models.



A dialog box opens enabling you to navigate to your optical model directory. Select the file and click **Choose**.

Note

You can click the **X** on an optical model to remove it.

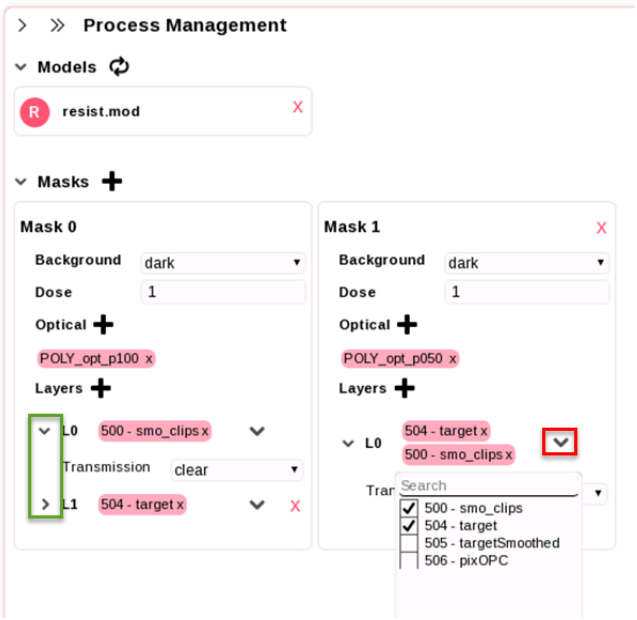


4. Map design layers. By default, Sparse sessions must contain at least one defined design layer mapping (**L0**).

Note

You can click the + button next to **Layers** to add more design layers.

   a. Click the down arrow icon to the right of a layer to expand the available design layer list, and select one or more layers to map.



Tip

There is also a search feature to identify layers of interest. Start typing a layer name to reduce the available choices.

b. (Optional) Use the arrows to the left of the layer name to expand and collapse the layer mapping details.

Note

Alternatively, you can map layout layers from the Sparse Process Configuration section. See "Adding Process Window Conditions" on page 32.

5. Click the field adjacent to **Transmission** to choose layer transmission values. Available options are:

   - **dark** — For use with negative tone masks
   - **clear** — For use with positive tone masks
   - **atten** — An additional field for **attenuation factor** is available with this choice
   - **complex** — Additional fields for **real** and **imaginary** are available with this choice

   The default **Transmission** value is **clear**.

Results

When you have completed Sparse Process Management Configuration, collapse the pane using the right arrow in the upper left corner.

Proceed to "Adding Process Window Conditions" on page 32.

Saving a Sparse Simulation Session

You can save the Sparse simulation setup or session status. This enables you to resume your simulation in a different session or share with others.

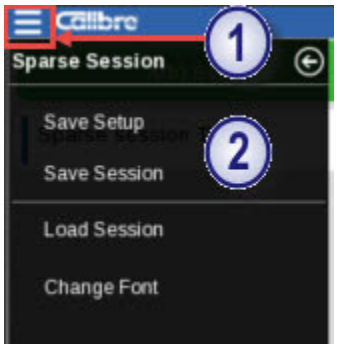
Prerequisites

- A layout open in Calibre WORKbench
- An open Calibre RFT v2.0 Sparse simulation session

Procedure

1. Open the tool side menu by clicking the menu button next to the "Calibre" logo at the top left of the Calibre RFT v2.0 screen.

Figure 2-12. Tool Side Menu Options



2. Select either **Save Setup** or **Save Session**.

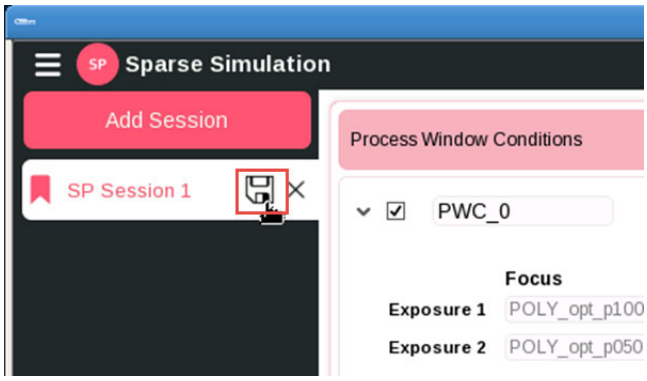
Note

Saving a setup file is only possible with a complete setup (one or more layers are mapped and one or more process window conditions are defined). Session files save the current state of the RFT v2.0 tool.

3. Navigate to the location you want to save your file, name it, then click **Save**.

Tip

You can also create a Sparse session file by hovering your mouse over the Sparse simulation session name and clicking the **Save session** icon.



Results

The Sparse simulation session or setup file containing OPCpro commands is saved in ASCII format and can be loaded in later sessions.

Adding Process Window Conditions

A single Process Window Condition (PWC) is created when you open a Litho Model or Sparse simulation. Adding additional PWCs causes the tool operations to run once for each defined PWC. This allows you to test multiple process scenarios with a single button press.

Prerequisites

- A layout open in Calibre WORKbench.
- An open Calibre RFT v2.0 simulation session.

Procedure

1. By default, the simulation session has one Process Window Condition (PWC). To add PWCs, click on the + button in the lower right of the Process Window Conditions pane.

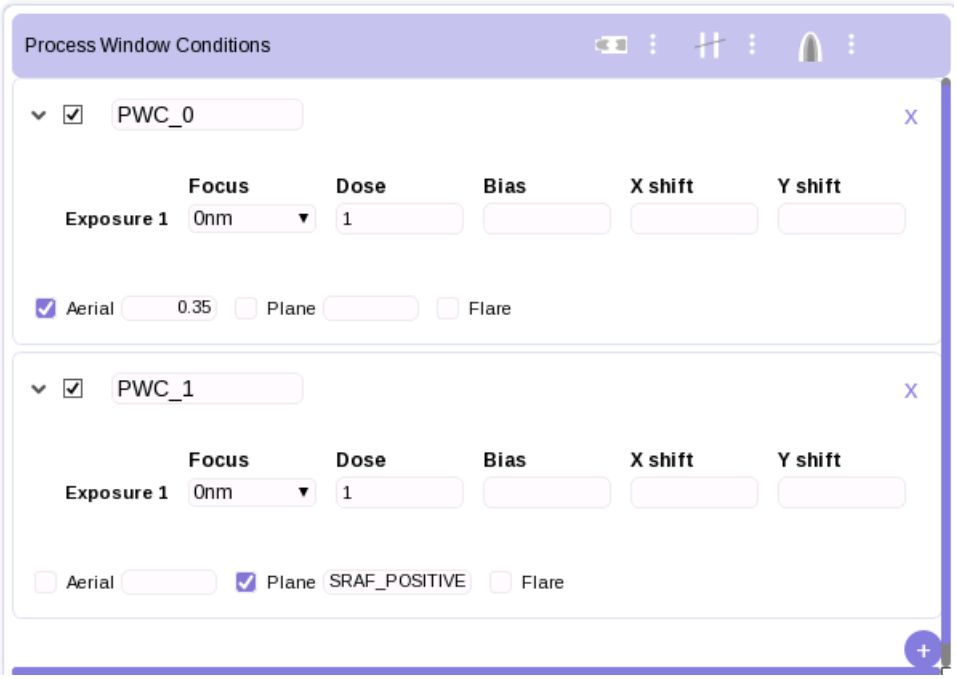
Figure 2-13. Process Window Configuration

Each PWC card has one **Exposure** per mask defined in the Process Configuration section. Each **Exposure** has default **Focus** and **Dose** field values corresponding to the mask definitions.

2. (Optional) For each PWC card, modify the PWC name and the **Exposure** values for **Focus** and **Dose**. Click in a field and type or select as appropriate to modify the value.
3. (Optional) Select the check box in the upper left of the PWC pane to include or exclude that PWC from the simulation operation.

4. (Optional) If you are using a Litho Model session, additional control buttons are available to toggle process exposure model usage:

Figure 2-14. Process Window Conditions (Litho Model Session)



- **Aerial** — Selecting this button ignores the resist model and uses a constant threshold resist (CTR) model for simulations. You can change the threshold value by typing in a value next to the Aerial text.
- **Plane** — Selecting this button performs a ZPlanes (3D resist imaging) simulation for the selected process condition. You must also enter an appropriate ZPlanes value in the field next to the Plane text and have a ZPlanes model available in your litho model.
- **Flare** — Selecting this button uses the flare model present in the loaded litho model for simulations.

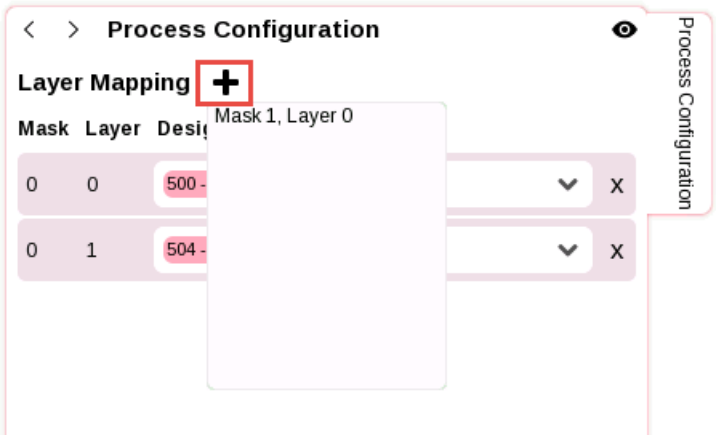
5. (Optional) Configure or reconfigure the mask layer mapping from the Process Configuration pane.

Note

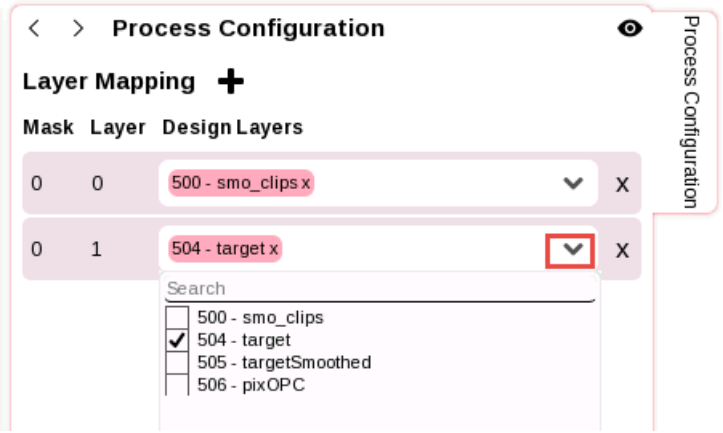
If you have already configured the mask layer mapping, you can usually skip this step; the controls are provided here for convenience.

   a. (Optional) To add a layer mapping, click the + button to the right of **Layer Mapping** in the Process Configuration pane. Choose from the available masks listed.

Figure 2-15. Process Configuration Layer Mapping



b. To change a layer mapping, click in the **Design Layers** field of the Process Configuration pane, then select a layer from the available choices.



Results

The process window conditions are set and simulations can be run on them using the toolbars, as described in the next chapter, “RET Flow Tool v2.0 Operations”.

SVRF Debugging Tool Session

The SVRF Debugging Tool is designed to let you browse SVRF rule files in read-only mode. It cannot be used to execute SVRF files in the current release.

Open an SVRF Debugging Tool Session

Create a new SVRF Debugging Tool session by using an existing SVRF rule file, or load a previously saved session.

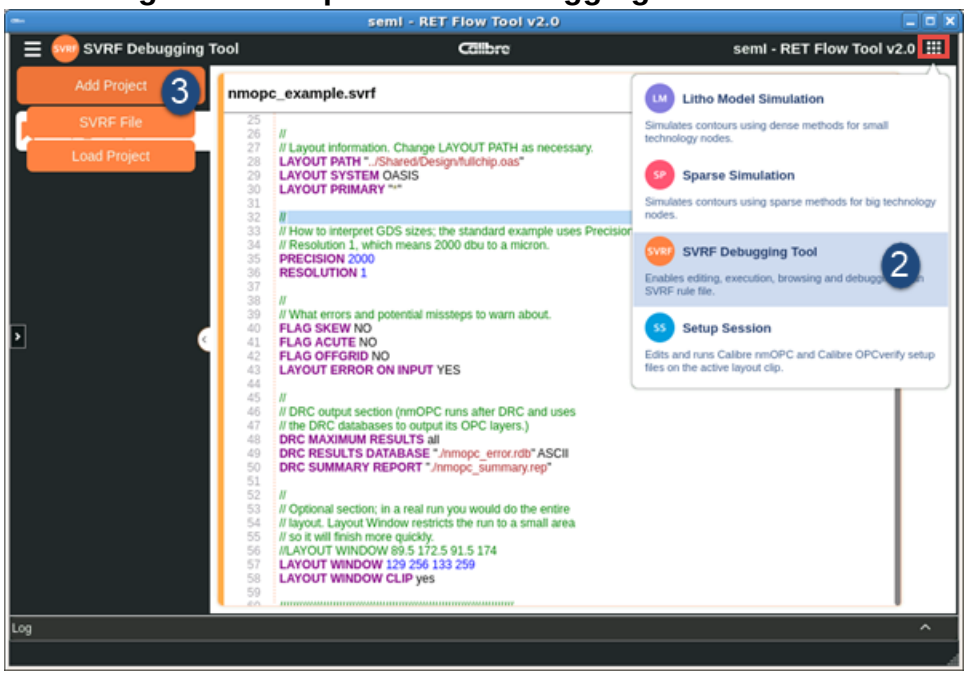
Prerequisites

- A layout open in Calibre WORKbench

Procedure

1. Open the Calibre RFT v2.0 GUI.
2. Click the modal selection button in the upper right corner (red box in the figure). Choose **SVRF Debugging Tool** as the session type.

Figure 2-16. Open SVRF Debugging Tool Session



3. Hover your cursor over **Add Project**. Choose one of the following options:

   SVRF File — Opens an existing SVRF file.

Load Project — Loads a previously-saved SVRF Debugging Tool project file.

Browsing With an SVRF Debugging Tool Session

SVRF Debugging Tool sessions are currently for browsing purposes. Use this tool to quickly navigate large SVRF rule files.

Note

This topic assumes you have an SVRF rule file open in Caibre RFT v2.0. (See "Open an SVRF Debugging Tool Session" on page 36 for more information.)

The right side of the SVRF Debugging Tool session has several tabs containing information about components in the loaded SVRF file:

- **Files** — Lists the project files and include files. Files are represented in purple in the SVRF file.
- **Tools** — Lists environment variables set for the session.
- **Setups** — Lists the setup files (LITHO FILE) statements in the SVRF file.
- **Layers** — Lists the layers (input and derived).
- **Layout** — Lists the layout(s) and RDB files referenced in the SVRF file. Hovering over the filename reveals clickable button to load the file. The Layout browser also shows the defined layout windows in the SVRF file.

Setup Session

The Setup Session is designed to let you load and run Calibre nmOPC and Calibre OPCverify sessions. It is intended to serve the function of the nmOPC and OPCverify sessions in the RET Flow Tool.

Open a Setup Session

Create a new Setup Session by opening an existing Calibre nmOPC or Calibre OPCverify setup command file, or by opening a saved session file.

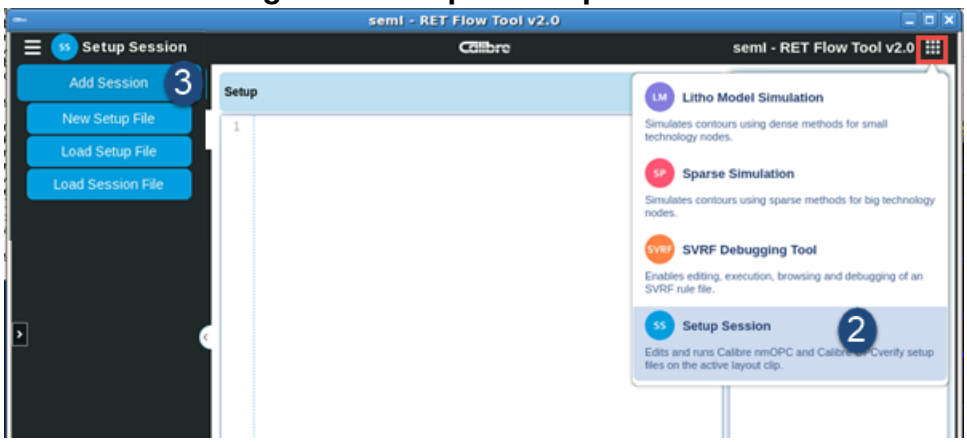
Prerequisites

- A layout open in Calibre WORKbench

Procedure

1. Open the Calibre RFT v2.0 GUI.
2. Click the model selector button in the upper right corner (red box in the figure). Choose **Setup Session** as the session type.

Figure 2-17. Open Setup Session



3. Hover your cursor over **Add Session**. Choose one of the following options:
   - **New Setup File** — If you do not have a setup command file, choose this option. This opens a blank page.
   - **Load Setup File** — If you have a setup command file available, choose this option to load it.
   - **Load Session File** — If you have a previously saved Setup Session file available, choose this option to load it.

Configuring a Setup Session

Setup Sessions run only on Calibre nmOPC and Calibre OPCverify setup command files. They must also have layers mapped in order to be executed.

Prerequisites

- Understanding of either Calibre nmOPC or Calibre OPCverify

  While the Setup Session performs syntax checking during execution, remedying issues requires knowledge of the relevant tool.

- A layout open in Calibre WORKbench
- An open Calibre RFt v2.0 Setup Session. See “Open a Setup Session” on page 38.

Procedure

1. Ensure that the setup command area has a valid setup command file.
   - If the setup session started out as empty, create a setup command file by typing commands or copying and pasting from an existing text file.
   - If the setup command started out as an SVRF file, remove SVRF commands until only the setup command file remains. This typically involves removing everything but the contents of the LITHO FILE command containing the setup command file.
2. In the right hand configuration panel, click the Refresh button (circular arrows) to load the list of input layers defined in the setup command file.
3. Assign layers from the layout to the input layers in the Layer Mapping list.
4. Click the Execute button at the top of the window to run the setup command file.

Results

If the setup command file returns output layers, new layers are added as the next available layer numbers.

You can save a setup session by hovering over the session name in the list on the left and clicking the disk icon.

Chapter 3
RET Flow Tool v2.0 Operations

There are a number of operations that are shared across sessions in the RET Flow Tool v2.0.

Generating a Print Image

The Print Image (PI) tool performs optical and resist simulations to generate image contours for a design file.

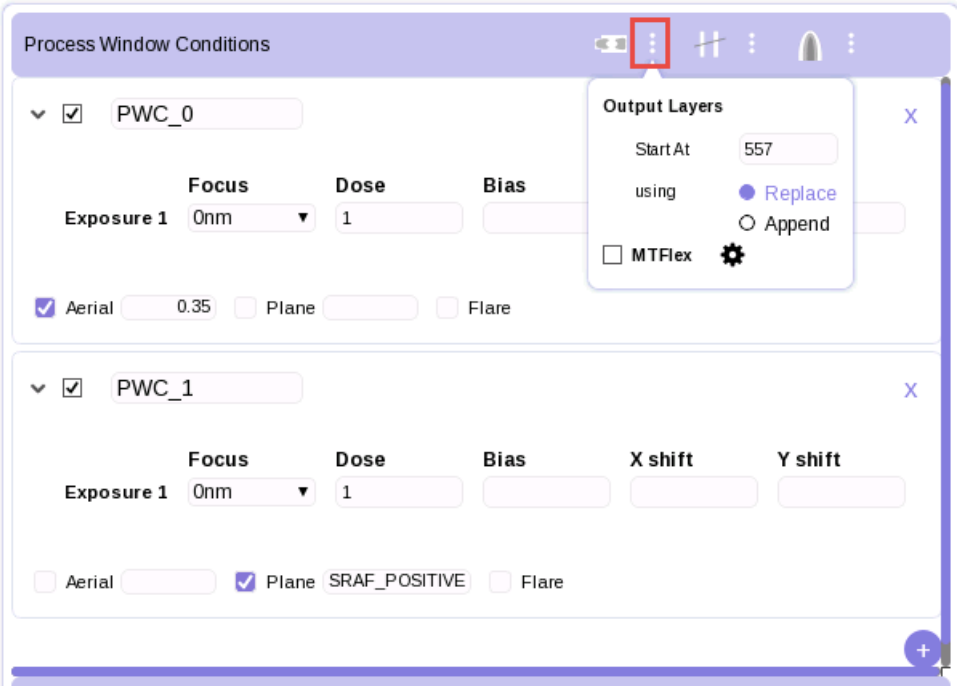
Prerequisites

- A layout open in Calibre WORKbench.
- An open Calibre RFT v2.0 LM or Sparse simulation session.
- At least one properly-configured Process Window Condition (PWC), as described in “Adding Process Window Conditions” on page 32.

Procedure

1. View and modify print image parameters in the Process Window Conditions area. Click the print image options icon.

Figure 3-1. PI Options



- **Start At** — Defines the starting layer number of the generated output layers. Each additional PWC you define is added to the next sequential layer.

  Note

  Choosing a Start At parameter that matches an existing layer causes the RET Flow Tool simulation to silently overwrite the layer. Ensure that you set Start At in such a way that you will not lose information.

- **Replace** or **Append** — The **Replace** option clears the layer's previous contents before running the simulation. This is the default.

  The **Append** option keeps an existing layer's contents, adding new simulation results to the layer.

- **MTFlex** — Opens a flyout panel to specify a remote host or remote file for use with Calibre MTFlex. This option is only available in Litho Model session PI simulations.

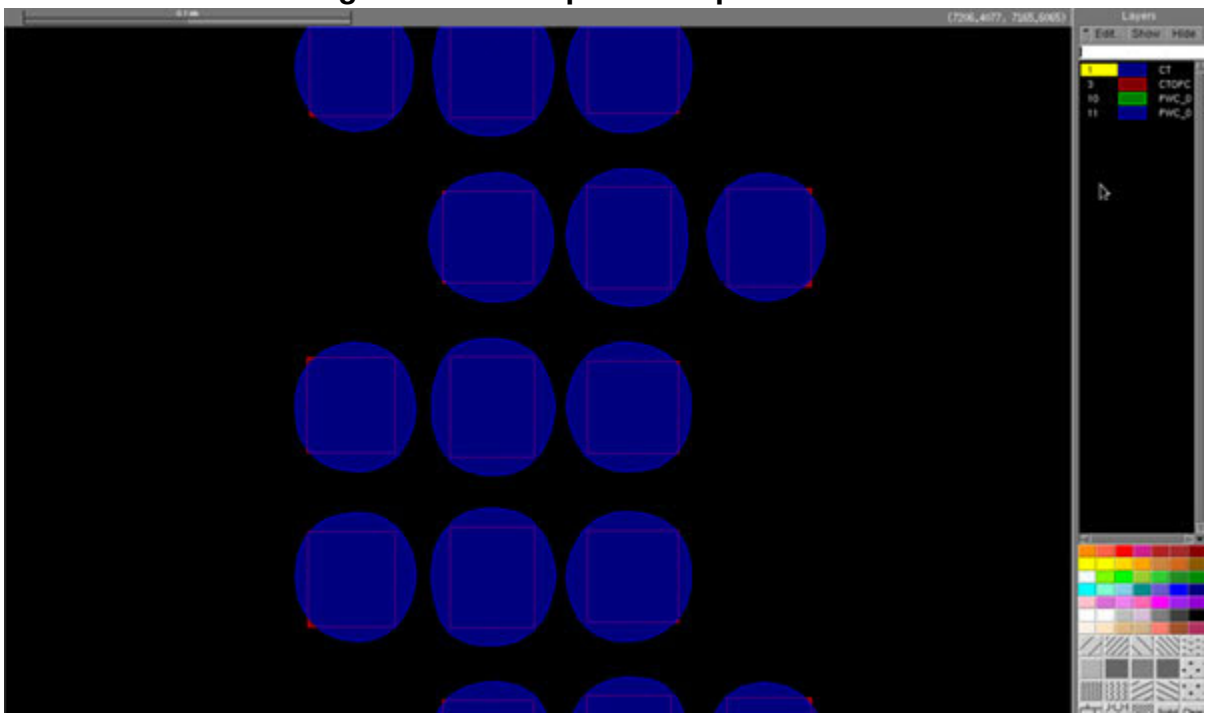
2. Ensure that the area of interest is showing or selected in the Calibre WORKbench layout display. Calibre WORKbench simulates the entire viewing window if nothing is selected, or simulates the interior of a selection box.
3. Run the simulation by clicking the Print Image icon.

Figure 3-2. Run the PI Simulation

Results

View the results of the print image simulation in the main Calibre WORKbench window. Contours represent the litho conditions that you specified.

Figure 3-3. Example PI Output Contour



If no results appear, try the following solutions:

- Check to see if the correct design layer has been selected. The layer selected should be the simulation layer.
- Check your background and layer transmission settings for the mask.
- If you are using Aerial image mode for a PWC, try changing the Aerial image threshold value.

Running an Intensity Cutline Simulation

You can generate an intensity curve for aerial or resist simulation modes, allowing you to compare intensity across the edges of interest for one or more simulations.

Restrictions and Limitations

- This operation is not available in Sparse sessions.

Prerequisites

- A layout open in Calibre WORKbench, zoomed in to the area of interest.
- An open Calibre RFT v2.0 LM simulation session. See “Open a Litho Model Session” on page 19.
- A previously-configured model in the Process Management area of Calibre RFT v2.0, as described in “Configuring the LM Process Management Section” on page 19.

Procedure

1. Configure your intensity curve cutline simulation run options. Click on the cutline options icon.

Figure 3-4. Intensity Curve Options

- **Type** — Specifies the type of cutline simulation to run.
  - **Aerial** — Dense aerial simulation.
  - **Resist** — Dense resist simulation.
  - **2D Resist** — Runs 2D resist profiling.

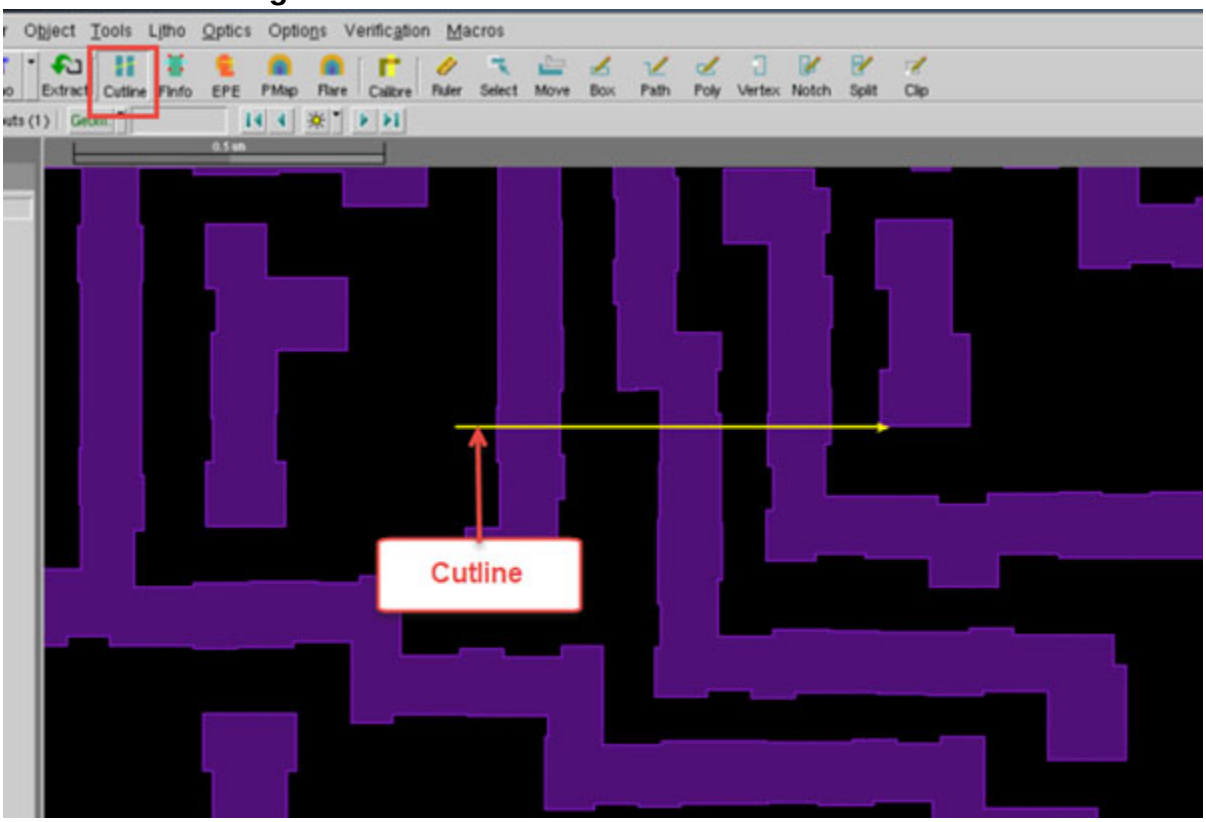
  Note

  2D Resist profiling cannot be run on normal optical models. It is only for 3D Resist models.

- **Pixsize** — Sets the simulation pixel size value of the scan in microns.
- **Accumulate Graphs** — Select this checkbox to add subsequent cutline simulations to the same graph output. By default, the Cutline dialog box is refreshed with a new cutline output each time you click the Cutline button in the RFT v2.0 GUI.

2. In Calibre WORKbench, select a layer, click the **Cutline** button, then draw a cutline arrow that crosses the edges of interest.

Figure 3-5. Draw Cutline in Calibre WORKbench



Note

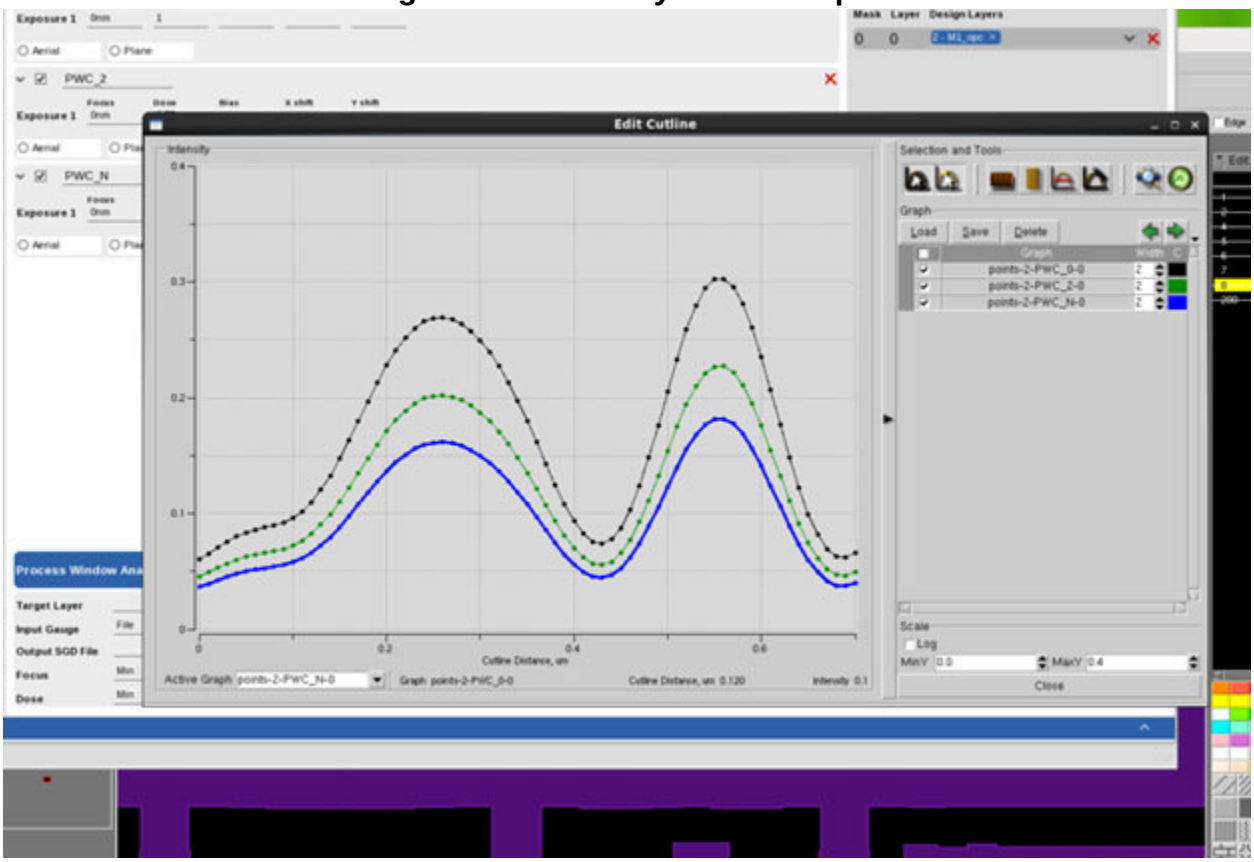
Calibre RFT v2.0 and Calibre RFT can both simulate the cutline operation. When both are running, Calibre RFT v2.0 settings take precedence.

3. Run the simulation by clicking the Cutline button.

Figure 3-6. Run the Cutline Simulation

After the simulation completes, an Edit Cutline window opens with the intensity curve of the cutline you specified in the layout.

Figure 3-7. Intensity Curve Output



4. (Optional) Adjust the cutline options as needed.

Figure 3-8. Edit Cutline Window

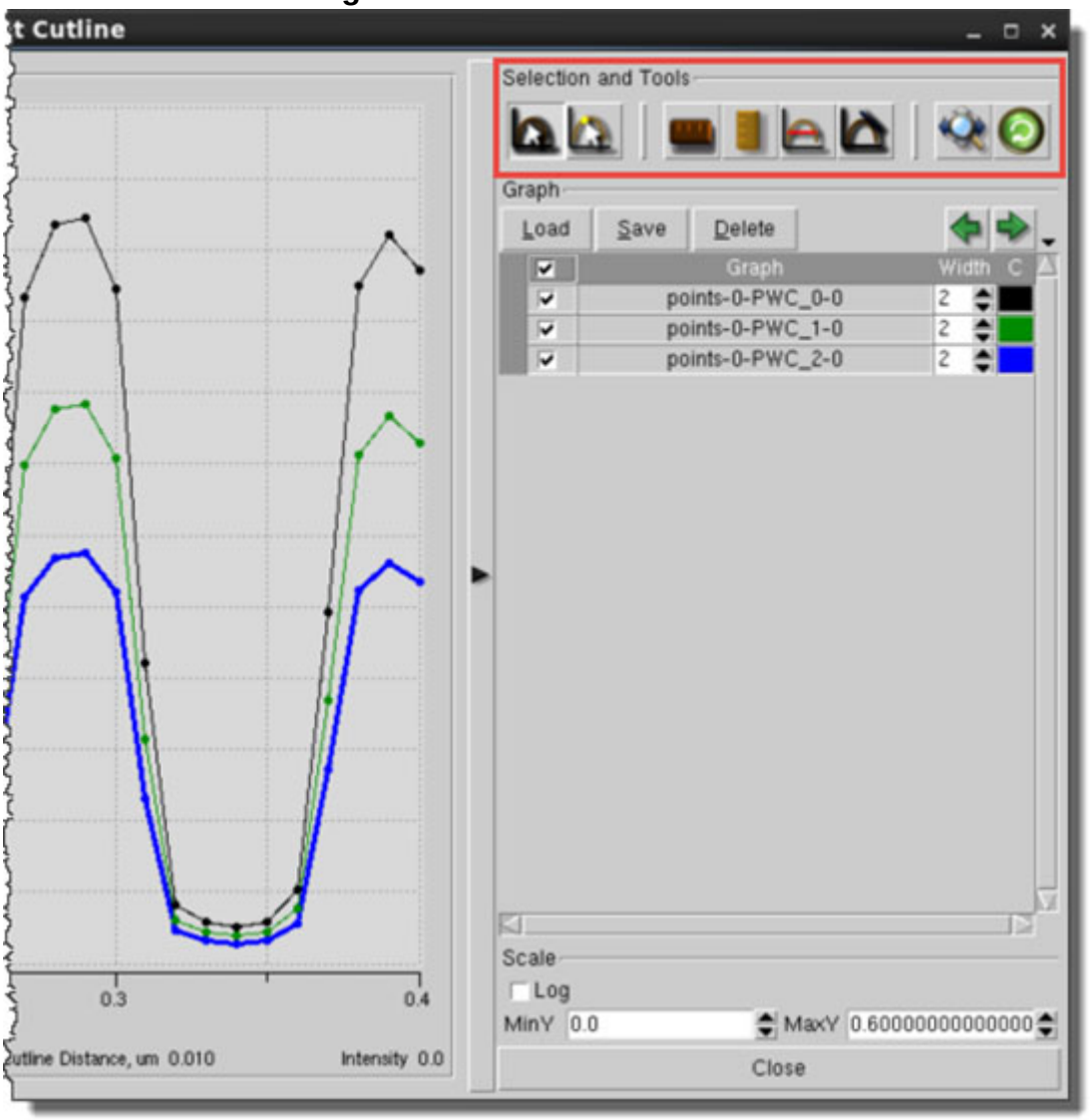


Table 3-1. Edit Cutline Window Options

Feature	Description
Select Graph	Enable to activate a graph. You can have multiple graphs but only one can be active at any time.
Select Graph Points	Enable to select a point in the graph. The corresponding position in the cutline is shown in the Calibre WORKbench layout view.
Horizontal Ruler	Enable horizontal ruler mode. When enabled, you can also choose to snap to point.
Vertical Ruler	Enable vertical ruler mode. When enabled, you can also choose to snap to point.

Table 3-1. Edit Cutline Window Options (cont.)

Feature	Description
Mark Threshold and CD	Similar functionality to Calibre RFT. See "Determining the Threshold and CD of an Intensity Graph" in the *Calibre WORKbench: RET Flow Tool User's Manual* for more details.
Mark Slope	Similar functionality to Calibre RFT. See "Determining the Slope of an Intensity Graph" in the *Calibre WORKbench: RET Flow Tool User's Manual* for more details.
Zoom All	Return to full graph view. You can zoom in by clicking the right mouse button and dragging from left to right. Zoom out by clicking the right mouse button and dragging from right to left.
Rerun Cutline Simulation	Rerun the simulation on a new cutline.

It is also possible to load, save, and delete graphs from the Edit Cutline window, and specify other parameters of interest.

Results

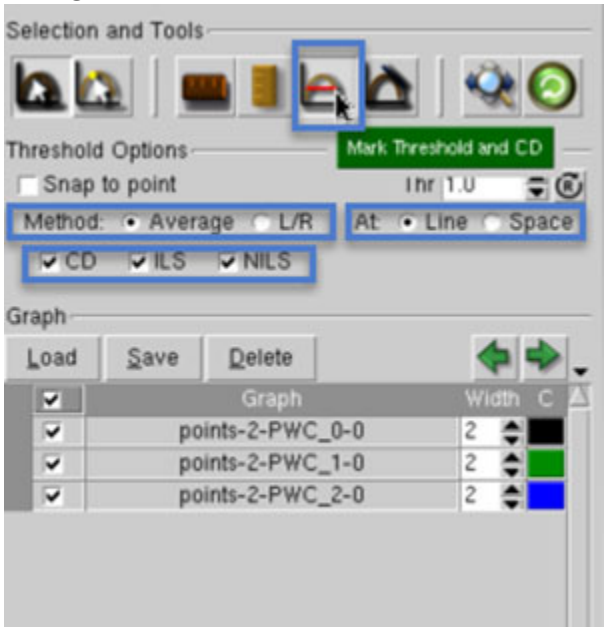
After the simulation completes, a new window opens with the intensity curve of the cutline you specified in the layout. There are additional functions within the Edit Cutline window which allow you to edit the intensity curve or rerun the simulation.

Examples

This example shows the steps to see the different data that the Edit Cutline window can display.

1. Click the **Mark Threshold and CD** button in the Edit Cutline window.

Figure 3-9. Mark Threshold and CD



2. Ensure that the following checkboxes are selected:
   - **CD** — Displays critical dimension data
   - **ILS** — Displays image log slope data
   - **NILS** — Displays normalized image log slope data
3. Because you selected **ILS** and **NILS**, you must choose where to make the measurements using the method controls. Select the **Method: L/R** radio button to display the left and right slope values where the threshold crosses the curve.

   The default setting is **Method: Average**, which averages the left and right slope values.
4. Select **At: Space** to perform the computations on the space between the layout feature locations.

   The default is to use the **At: Line**, which measures at the layout feature locations.
5. Click your desired intensity threshold in the graph region of the Edit Cutline window.

   Note

   If you need to be more accurate, you can adjust the threshold value using the threshold box, **Thr**.

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

   The graph changes to show the threshold you defined and the corresponding values for CD, ILS, and NILS. The values reported relate to the active graph. You can change the active graph using the **Active Graph** dropdown list below the plot.

Figure 3-10. User-Defined Threshold and CD for Space

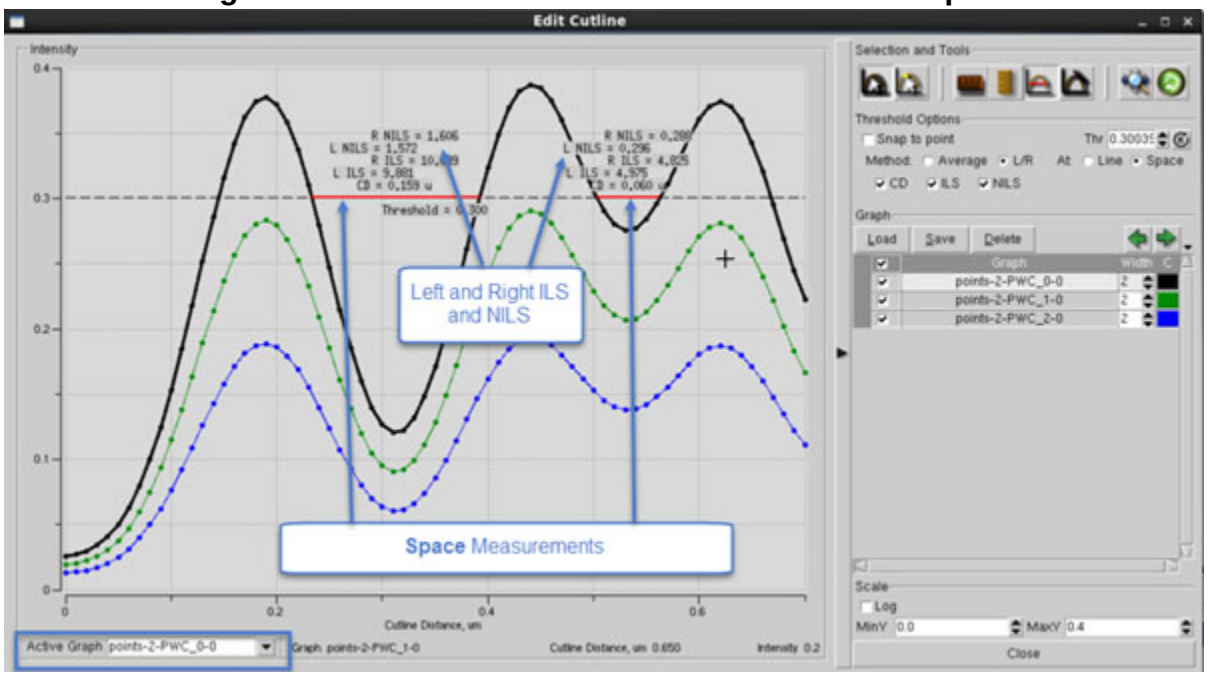
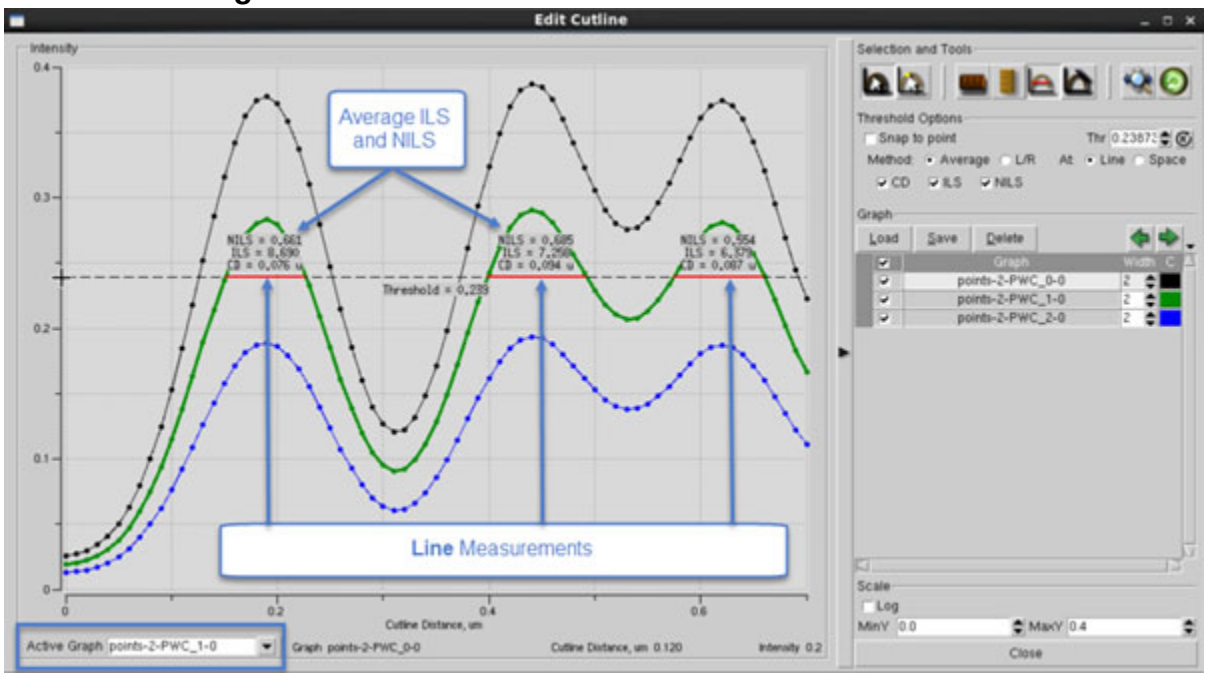


Figure 3-11. User-Defined Threshold and CD for Line



Creating a Contour Map

You can create a contour map representing the light intensity on the layout within a selected area.

Restrictions and Limitations

- This operation is not available in Sparse sessions.

Prerequisites

- A layout open in Calibre WORKbench. Zoom in to the area of interest for contour mapping.
- An open Calibre RFT v2.0 LM simulation session. See “Open a Litho Model Session” on page 19.
- A previously-configured model in the Process Management area of Calibre RFT v2.0, as described in “Configuring the LM Process Management Section” on page 19.

Procedure

1. Configure the run options for your contour map simulation. Click on the map options icon.

Figure 3-12. Contour Map Options

   a. Select the type of Map simulation to run:

      - **Aerial** — Runs an aerial image map simulation, using a CTR resist model in place of the resist model in the litho model.
      - **Resist** — Runs an image map with resist model information included.

   b. Set the Pixsize value to define the simulation pixel size of the scan in microns.

   c. (Optional) If you select Convolve Gaussian, the Gaussian kernel smooths the output map using a default kernel sigma of 70 nm. If Convolve Gaussian is not checked, the raw output is presented.

2. Run the simulation by clicking the Map icon.

Results

Contours are generated from the litho conditions you specified. View the results of the contour map simulation in the main Calibre WORKbench window.

Running a Process Window Analysis

The Process Window Analysis (PWA) operation is new for Calibre RFT v2.0. You can use it to generate a super gauge data (*.sgd*) file of the simulated gauges. You can view these in the PW Plot tool without using a litho setup file.

Restrictions and Limitations

- This operation is not available in Sparse sessions.

Prerequisites

- A layout open in Calibre WORKbench, zoomed in to the area of interest for the PWA operation.
- An open Calibre RFT v2.0 LM simulation session.
- Gauge data, either drawn on a layer in Calibre WORKbench or as a gauge file. The gauge file should be *.gg* format.
- A previously configured model in the Process Management area, as described in "Configuring the LM Process Management Section" on page 19.

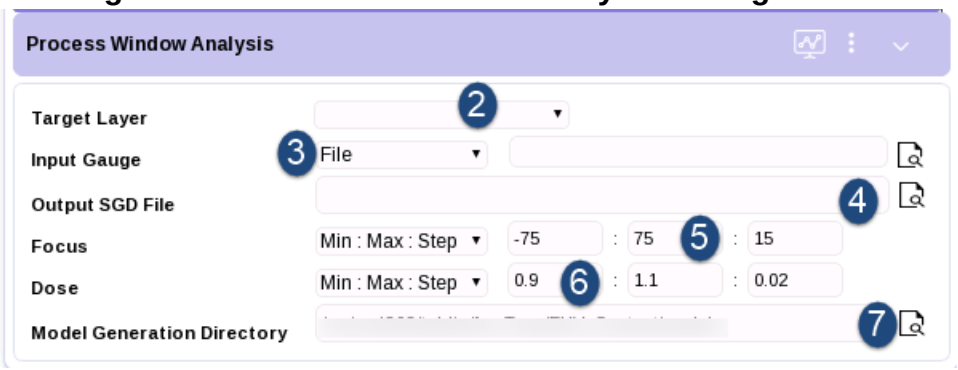
Procedure

1. In the Process Management area, set the Design Layer to the layer to be simulated (typically, this is the post-OPC layer).
2. In the Process Window Analysis panel, click to the right of Target Layer and select the target from the list of available layers (typically, this is the original drawn layer with the target CDs on it).

Note

The simulation and target layer cannot be the same layer.

Figure 3-13. Process Window Analysis Configuration



3. Select your input gauges by loading either a previously created gauge file (**File** selection) or gauges on a layer in the layout (**Layer** selection).
4. Set the Output SGD File by entering a name for the SGD file or using the file navigator.
5. Configure the Min, Max, and Step Focus values for the simulation.
6. Configure the Min, Max, and Step Dose values for the simulation.
7. Specify the litho model directory the PWA operation uses.

   The default directory is the current litho model directory. If you choose the default, any required missing optical model (focus) will be generated automatically in the litho model directory.

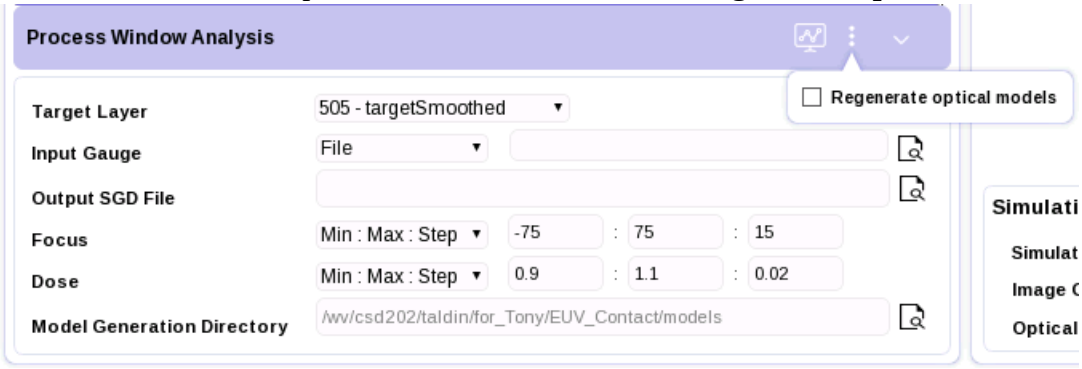
   Note

   The PWA operation generates new optical models for focus values that were not already present in the litho model directory. The PWA operation does not overwrite the optical models in the original litho model directory.

   To change the directory to safeguard the original litho model directory, choose one of the following options:

   - In the Model Generation Directory field, type the name of another directory or browse to an existing directory.

- Click the PWA options button and uncheck the **Regenerate optical models** box.



By default, if an optical model for a focus condition exists, it will not be regenerated. Use this option to force regeneration.

8. Click the indicated button in Figure 3-14 to run the PWA operation.

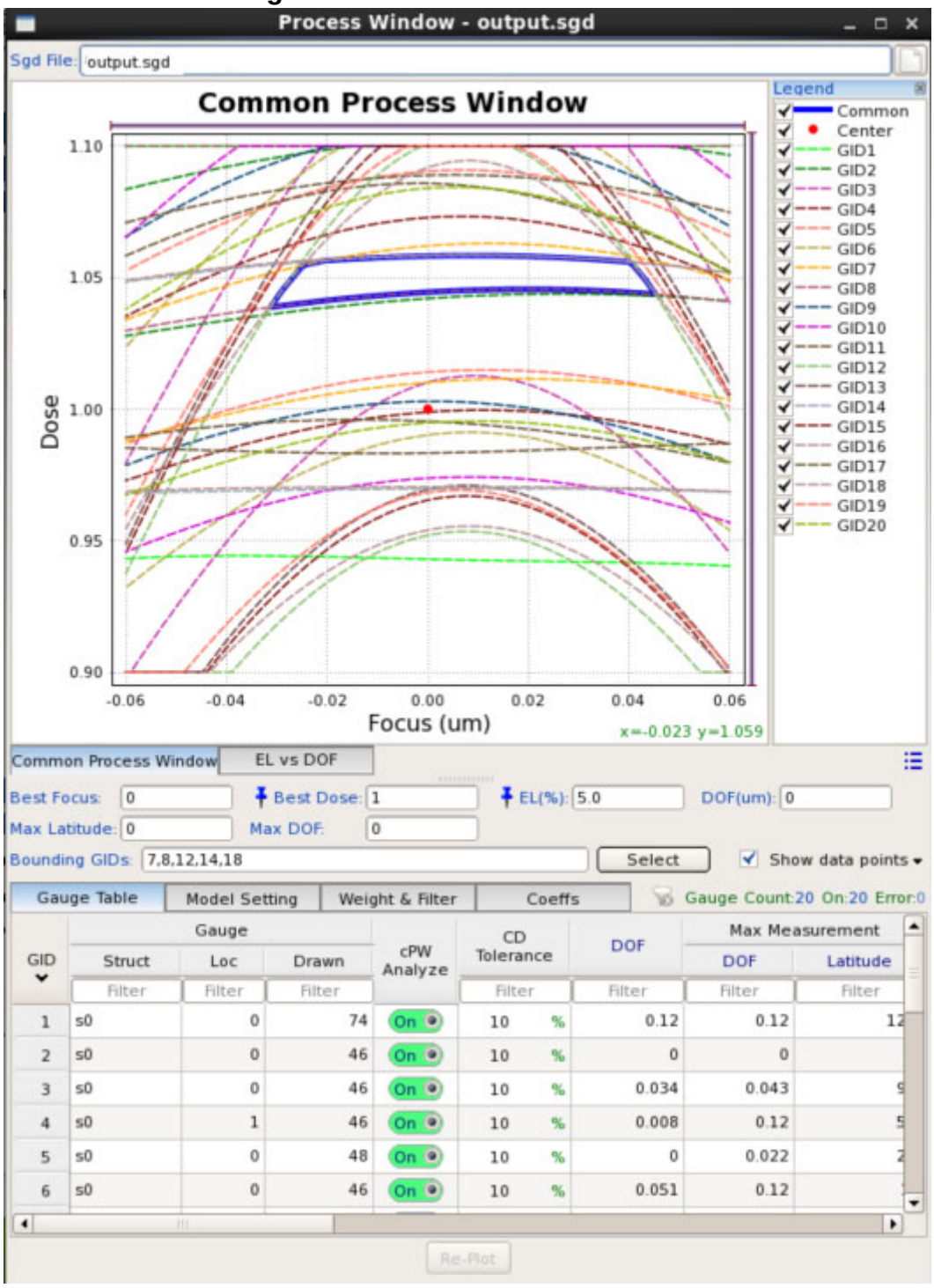
Figure 3-14. Start Process Window Analysis

You can view simulation progress in the Linux terminal from which Calibre WORKbench was invoked.

Results

After completing the PWA operation successfully, a super gauge data file is generated, and the PW Plots window opens with process window information contained in the output *.sgd* file.

Figure 3-15. Process Window Plot



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.