SILVACO

TonyPlot

User's Manual

Silvaco, Inc.

4701 Patrick Henry Drive, Bldg. 2

Santa Clara, CA 95054

Phone: (408) 567-1000

Web: www.silvaco.com

February 16, 2018

Notice

The information contained in this document is subject to change without notice.

Silvaco, Inc. MAKES NO WARRANTY OF ANY KIND WITH REGARD TO THIS MATERIAL, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTY OF FITNESS FOR A PARTICULAR PURPOSE.

Silvaco, Inc. shall not be held liable for errors contained herein or for incidental or consequential damages in connection with the furnishing, performance, or use of this material.

This document contains proprietary information, which is protected by copyright laws of the United States. All rights are reserved. No part of this document may be photocopied, reproduced, or translated into another language without the prior written consent of Silvaco Inc.

AccuCell, AccuCore, Athena, Athena 1D, Atlas, Blaze, C-Interpreter, Catalyst AD, Catalyst DA, Clarity RLC, Clever, Clever Interconnect, Custom IC CAD, DeckBuild, DevEdit, DevEdit 3D, Device 3D, DRC Assist, Elite, Exact, Expert, Expert C++, Expert 200, ExpertViews, Ferro, Gateway, Gateway 200, Giga, Giga 3D, Guardian, Guardian DRC, Guardian LVS, Guardian NET, Harmony, Hipex, Hipex C, Hipex NET, Hipex RC, HyperFault, Interconnect Modeling, IWorkBench, Laser, LED, LED 3D, Lisa, Luminous, Luminous 3D, Magnetic, Magnetic 3D, MaskViews, MC Etch & Depo, MC Device, MC Implant, Mercury, MixedMode, MixedMode XL, MultiCore, Noise, OLED, Optolith, Organic Display, Organic Solar, OTFT, Quantum, Quantum 3D, Quest, RealTime DRC, REM

2D, REM 3D, SEdit, SMovie, S-Pisces, SSuprem 3, SSuprem 4, SDDL, SFLM, SIPC, SiC, Silvaco, Silvaco Management Console, SMAN, Silvaco Relational Database, Silos, Simulation Standard, SmartSpice, SmartSpice 200, SmartSpice API, SmartSpice Debugger, SmartSpice Embedded, SmartSpice Interpreter, SmartSpice Optimizer, SmartSpice RadHard, SmartSpice Reliability, SmartSpice Rubberband, SmartSpice RF, SmartView, SolverLib, Spayn, SpiceServer, Spider, Stellar, TCAD Driven CAD, TCAD Omni, TCAD Omni Utility, TCAD & EDA Omni Utility, TFT, TFT 3D, Thermal 3D, TonyPlot, TonyPlot 3D, TurboLint, Universal Token, Universal Utility Token, Utmost III, Utmost III Bipolar, Utmost III Diode, Utmost III GaAs, Utmost III HBT, Utmost III JFET, Utmost III MOS, Utmost III MultiCore, Utmost III SOI, Utmost III TFT, Utmost III VBIC, Utmost IV, Utmost IV Acquisition Module, Utmost IV Model Check Module, Utmost IV Optimization Module, Utmost IV Script Module, VCSEL, Verilog-A, Victory, Victory Cell, Victory Device, Victory Device Single Event Effects, Victory Process, Victory Process Advanced Diffusion & Oxidation, Victory Process Monte Carlo Implant, Victory Process Physical Etch & Deposit, Victory Stress, Virtual Wafer Fab, VWF, VWF Automation Tools, VWF Interactive Tools, and Vyper are trademarks of Silvaco, Inc.

All other trademarks mentioned in this manual are the property of their respective owners.

Copyright © 1984 - 2018, Silvaco, Inc.

Table of Contents

1. Introducing TonyPlot

1.1 What is TonyPlot

TonyPlot (version 5) is a graphical post processing tool for use with all Silvaco TCAD simulators and is an integral part of the TCAD Interactive Tools. TonyPlot can operate stand-alone or along with other TCAD Interactive Tools, such as DeckBuild, VWF, or SPDB.

Users of earlier versions of TonyPlot or TonyPlot3D will find some of the functionality familiar. This version has new some capabilities and a new user interface. The interfaces are easier to use and the resulting display has improved. This version provides:

- Multiple file loading
- Plot comparison and overlay
- Movie function
- HP4145 emulator
- Cross section profile integration
- Poisson Solver
- Improved user interface and ease of use
- Improved cutline definition and creation
- Faster drawing
- New Structure file features
- User definable setup parameters and set files

1.2 Examining Data

TonyPlot can be used to examine several data files all at once, each in its own plot window. You can combine these plot windows by “overlaying” the data sets so you can make direct comparisons. Plots can be interactively added, deleted and duplicated, overlaid and separated.

Not only does TonyPlot allows you to display any data file produced by Silvaco tools, it also provides extensive “tools” for examining these plots and the associated data. For example, you can take cut-line or cut-plane slices through structures, integrate a curve to calculate an area, or perform simple electrical simulations on 1D devices.

TonyPlot allows you to rescale, zoom, and pan plots. You can also add grids, customize axes, and draw arbitrary labels on the data. All titles, marks, labels, ranges and so on are automatically set to useful defaults but can all be explicitly set whenever necessary. The appearance of all plots in TonyPlot can be customized. There are many “preferences” that can be tailored to either suit your needs or the requirements to a particular set of data. See Preferences for more information.

TonyPlot can also be used to study the output from the Silvaco process and device simulators Atlas, Athena, Clever, Victory Process, Victory Device, Victory Mesh and Victory Cell. TonyPlot can display regression models, response surfaces, scatter plots, histograms, SPC charts, pie charts, and more.

1.3 Online Help

You can access TonyPlot's online user's manual. If the manual does not answer a particular problem, then contact Silvaco (support@silvaco.com). See Section 2.2.7 Help Menu for more information.

1.4 Terminology

The following terminology is used throughout TonyPlot:

Table 1-1 TonyPlot Terminology

Terms	Definition
Structure	This is the set of data contained within a structure file (e.g., doping, geometry, or bias points). One structure usually requires one plot but sometimes two. The same structure can be reused in many plots.
Plot	This is one drawing. A plot can be of one or more structures but cannot exist without a structure. Several plots can show the same structure, which allows the data to be simultaneously examined from two different angles. TonyPlot can display many kinds of plots, each with its own distinct preferences.
Selected Plot	Not all plots need be operated on at once. A subset of plots is defined by selecting required plots. Selected plots are indicated by an orange triangle in the top right corner of the subwindow.
View	This is the collection of plots in the main window. This term refers to all plots, selected or not.
Subwindow	This is the area where a plot is shown. Each plot has a unique subwindow, and each subwindow can only show one plot. You can modify the arrangement of subwindows within the view.
Display	Each plot has a display setting, which is the set of parameters that defines how to draw the structures in the plot.

2. Using TonyPlot

2.1 Starting TonyPlot

TonyPlot can be started independently from a UNIX or Windows command prompt, or from other simulator tools such as DeckBuild or SPDB. In the case of the simulator tools, starting TonyPlot is accomplished by selecting the respective command button. The TonyPlot Base Window (Figure 2-1) appears if files are not immediately loaded when started.

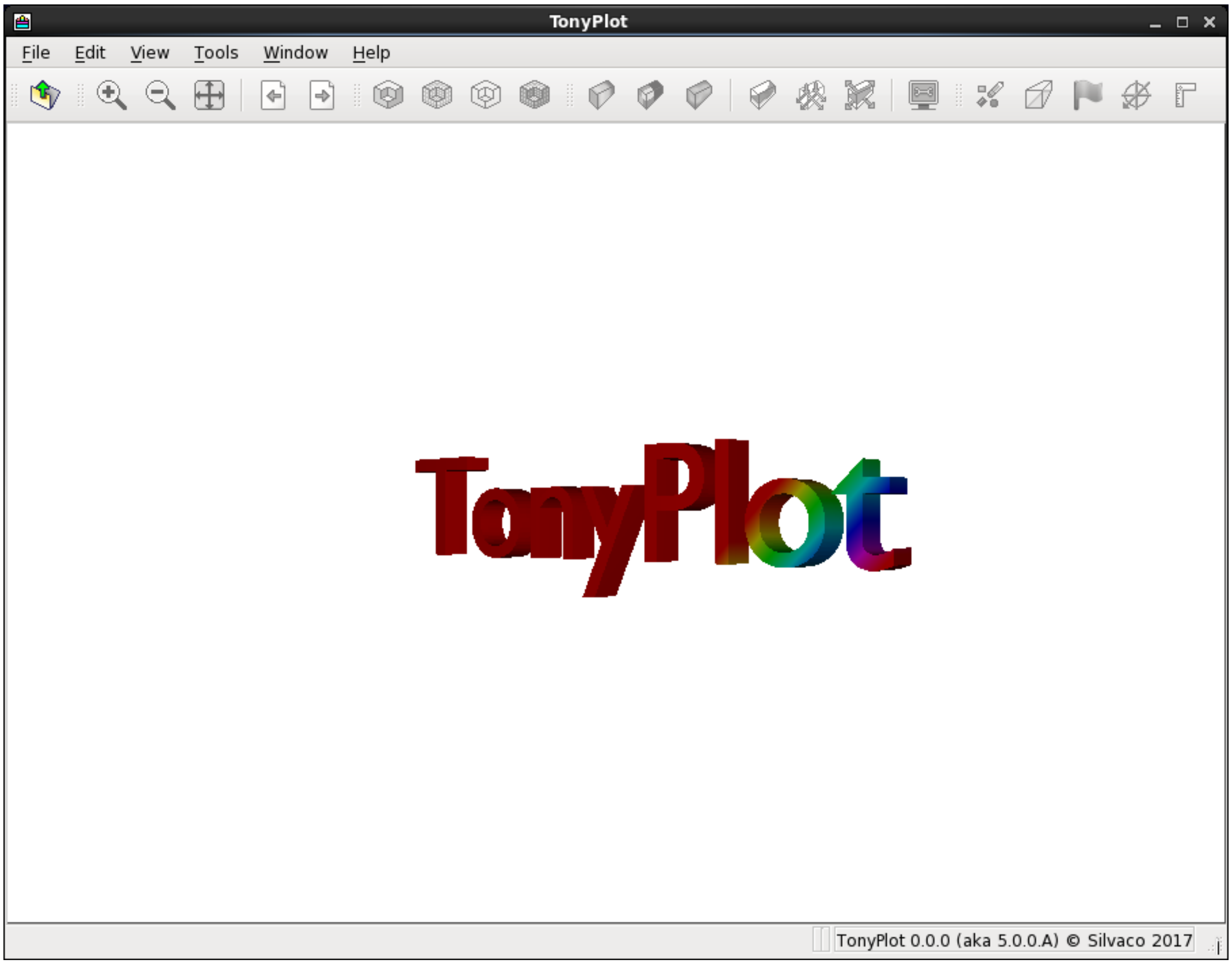


Figure 2-1 TonyPlot Base Window

To start TonyPlot at the UNIX or Windows command prompt enter:

```
% tonyplot
```

This starts TonyPlot with no data file loaded and with all options set to their default values. To load files at the same time TonyPlot starts, just specify the filenames on the command line. TonyPlot figures out the types of files and displays the data accordingly.

2.1.1 Command Line Options

When started from a command prompt, TonyPlot supports the following command line options:

Table 2-1 Command Line Options

Command Line Option	Description
`-ccd`	Allows the overlay of results of the DeckBuild `extract max.conc.file` output on 2D mesh plots. This is used in the CCD analysis to determine the potential maximum/minimum.
`-help`	Displays a list of all command line options that TonyPlot recognizes.
`-mtitle <maintitle>`	Overrides the default plot main title and set it to maintitle instead. Use single quotes around `<maintitle>` if you wish to use spaces. All files loaded so far (all that precede this option on the command line) are affected.
`-ttitle <subtitle>`	Overrides the default plot subtitle and set it to subtitle instead. Use single quotes around `<subtitle>` if you wish to use spaces. All files loaded so far (all that precede this option on the command line) are affected
`-nosplash`	Disables the TonyPlot splash screen.
`-nosipc`	Disables inter-process communication.
`-overlay`	Instructs TonyPlot to overlay all files after the overlay flag.
`-set <file>`	Instructs TonyPlot to load the set filename and restore the display to the condition that TonyPlot was in when that set file was created. The set file is applied to all files loaded at that point. In other words, all files that preceded this option on the command lines. Files given after this option on the command line are not affected.
`-glinfo`	Displays information about the OpenGL graphics context.
`-measured <file>`	Loads measured data from <file>.
`-jpg <file>`	Renders all plots to <file> in JPEG format.
`-png <file>`	Renders all plots to <file> in PNG format.

Command Line Option	Description
`-geom <width>x<height>`	Sets the plot area geometry to **width** x **height.**
`-noexit`	Causes TonyPlot to not exit after use of the -jpg or -png flags.
`-cuboid_level <level>`	Plot mesh boxes for the specified level.
`-resolution_level <level>`	Plot the Cartesian mesh inside the mesh boxes.
`-resolution_style [1|2|4]`	Set the mesh boxes line style.
`-cut <file>`	Export a cutplane to <file> (3D only).
`-plane [XY|XZ|YZ]`	Set the cutplane to specified coordinate plane (3D only).
`-start <value>`	Sets the cutplane start value (3D only).
`-stop <value>`	Sets the cutplane stop value (3D only).
`-step <value>`	Sets the cutplane step value (3D only).
`-project_vectors [true|false]`	Project the cutplane vector quantities (3D only).

2.1.1.1 Loading Examples

To load two Atlas log files `temp340.log` and `temp450.log` and display the graphs overlaid in a single plot, enter:

```
% tonyplot -overlay temp340.log temp450.log
```

To load a structure file called `meshX.str` and set its display to a previous set up stored in `mx.setx` and then load a file containing IV data in user data format, enter:

```
% tonyplot meshX.str -set mx.setx iv.data
```

Note: TonyPlot version 5 and later uses the newer .setx XML file format, but is still able to read the older, legacy .set files. See section 13.2 Set Files for more information.

2.2 The Base Window

The TonyPlot Base Window contains the area where all plots are displayed. When there are no plots to display (e.g., when no files have been loaded), a banner page will appear. Along the top of the Base Window is the main menu bar that has the basic control menus for TonyPlot, and the toolbar that displays icons for commonly used actions. Each of these menus are further explained in the following sections.

2.2.1 File Menu

This menu allows you to access file control operations. These operations consist of loading structures, overlaying structures, saving data, set file control, image export, printing, production mode, structure information, and exiting from TonyPlot.

Table 2-2 File Menu

Item	Description
Open…	This action is used to load structures and data files.
Overlay…	This can be used to overlay structures onto the current plot.
Save As…	This allows you to create data files from plots. TonyPlot supports saving from Cross Section plots, XY GRAPH plots, and 3D Mesh plots.
Load Set File…	Loads a **Set File** from disk. The **Set File** specifies user applied attributes, such as contour drawing for a given plot.
Save Set File…	Saves a **Set File** to disk. The **Set File** specifies user applied attributes, such as contour drawing for a given plot.
Export Image…	The Export Image dialog is used to save plots to the system clipboard, or to image files on disk.
	This allows you to print the currently selected plots, one per page. You can print on a large variety of paper sizes at full printer resolution.

Item	Description
Print…	
Production Mode	When used with VWF, TonyPlot provides powerful Production Mode capabilities. Once you select this option, these features remain active.
Info	This displays information about the current plot, such as bias info and region info. If no plot is currently displayed, TonyPlot will display information about the OpenGL display context provided by the graphics card driver.
Recent Files	This displays a list of recently opened files. Select a file from the list to load it.
Exit	This removes all plots, structures and subwindows, and quits TonyPlot.

2.2.2 Edit Menu

The **Edit** menu provides control over the current view using operations that affect groups of plots as described below. Some items on this menu may be inactive (grayed out). This occurs when the operation is not applicable to the current group of selected plots.

Table 2-3 Edit Menu

Item	Description
Undo	This will undo the last action taken, reverting the plot to its previous state.
Redo	This will redo the last action that was undone.

Item	Description
Undo All	This will undo all previous actions, leaving the plot the same as when it was first loaded.
Redo All	This will redo all edits that had previously been undone.
Make Overlay	This will overlay all currently selected plots. Note that the plots must be of the same type to overlay, such as all Mesh 2D, or all XY Graph.
Split Overlay	This will split the currently selected overlay plots back into their original base plots.
Plot Difference	This requires you to have at least two compatible plots selected. When chosen, this option will create a new plot of the differences in the selected plots.
Delete	This can be used to delete certain selected objects from the scene, such as Rulers or Isosurfaces. Applies to 3D Mesh plots only.
Hide	Sets the visibility of the selected region to hidden. Applies to 3D Mesh plots only.
Show	Sets the visibility of the selected region to shown. Applies to 3D Mesh plots only.
Opaque	Sets the opacity of the selected region to opaque. Applies to 3D Mesh plots only.
Transparent	Sets the opacity of the selected region to transparent. Applies to 3D Mesh plots only.

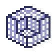
Item	Description
Functions…	Functions allow you to further customize the output that can be produced and to extend the amount of data that can be plotted without needing further simulation runs and large data files. TonyPlot allows you to create functions for use with XY Graph or 2D Mesh plots. See Section 2.8 Functions for more information.
Shift Coordinates…	This function gives you the ability to shift one or more curves in an XY Graph plot. You can shift the curve on both the X and Y axes.
Transform 2D Mesh…	This function allows you to perform mathematical operations between multiple selected 2D Mesh quantities. The meshes must be identical to perform an operation.
Transform XYG…	This function allows you to perform mathematical operations between multiple selected XY Graph curves.
Preferences…	The Preferences dialog is used to customize and manage TonyPlot's many preference settings. See Preferences for more information.

2.2.3 View Menu

The **View** menu provides control over selected plots. The items on the menu can operate on many selected plots of the same type. If plots of different types are selected, only those with the same type as the first selected plot are affected.

Table 2-4 View Menu

Item	Description
Zoom Full	Selecting this action returns the structure to its initial location and size.

Item	Description
 Zoom In	Selecting this action performs an incremental zoom in on the structure.
 Zoom Out	Selecting this action performs an incremental zoom out on the structure.
Normalize	Selecting this action performs anisotropic scaling of the axes such that the axis aligned bounding box of the structure becomes a unit cube.
 Annotation…	This allows you to specify plot parameters that are independent on the data and which do not fall into the category of display settings. For example, titles and axis ranges.
 Level Names…	This allows you to change the names assigned to overlay plots. See Overlays for details about overlays.
Show From	The Show From option has six sub-options: Top, Bottom, Front, Back, Left, and Right. Each of these sub-options moves the view point to display the structure from the selected position.
 Solid	Show regions as solid (filled) areas.
 Meshed	Show only the mesh lines of regions.
 Edges	Show only the sharp edges of regions.
 Solid and Meshed	This is a combination of Solid and Meshed modes.

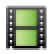
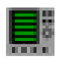
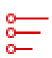
Item	Description
Point Cloud	Show only the vertices of regions.
Materials	Show regions using the color assigned to the region material.
Regions	Show regions using a color assigned to each region in the data.
Contours	Shows regions in colors representing a selected Quantity value.
Isosurfaces	Shows surfaces that represent points of a constant value within the data.
Rays	Shows ray traces within the data.
Vectors	Shows arrows representing vector quantities with the data.
Display…	Opens the Display dialog for 1D/2D Plots or 3D Mesh Plots. See 2D Mesh Plot Display or 3D Mesh Plot Display for more information.
Next Page	Moves focus to the next plot subwindow.
Previous Page	Moves focus to the previous plot subwindow.

2.2.4 Tools Menu

As well as displaying information contained in structures, TonyPlot can examine that information in a variety of ways. Each method of examining the data is called a **Tool**. The **Tools** menu shows all the tools available. The **Tools** menu may show some unavailable items. This is because the tool cannot be applied to the current set of selected plots. For example, the **HP4145 Emulator** tool only works with **Graph** plots so it is unavailable if no **Graph** plot exists and is selected. The following provides brief descriptions of the tools.

Table 2-5 Tools Menu

Item	Description
Cut Line…	The **Cut Line** tool is used to create 1D cross section plots from arbitrary positions within a 2D structure.
Cut Plane…	The **Cut Plane** tool is used to create 2D slices through a 3D structure. The resulting slices can be viewed as 2D mesh plots (see 2D Mesh Plot Display), or used as inputs to a 2D device simulator such as Atlas.
Object Browser…	The **Object Browser** provides a quick way of setting many different properties that define how a plot is drawn. Selecting this item will toggle the **Object Browser** state between shown and hidden.
Markers…	The **Markers** tool is used to create labels or markers on a 3D mesh plot. A marker consists of a small dot at a specified (X, Y, Z) coordinate, a line, and a text label.
Probe…	The **Probe** tool can be used to examine the geometry and impurity data of a 2D or 3D Mesh.
Ruler…	The **Ruler** tool can be used on any type of plot. It provides coordinate geometry information of any line drawn over a structure.
Labels…	Labels are used to add arbitrary notes and informative text to any plot. These labels can be drawn with leader arrows or can be free standing.

Item	Description
 Movie…	The **Movie** tool allows you to combine a group of plots into an animated sequence, which can be viewed in playback like a slideshow.
 HP-4145…	The HP4145 Emulator is available for any **Graph** plot. Only one plot, however, can be used with the emulator at any one time.
 Integrate…	The **Integrate** tool is used to measure the area under a single plot curve or the area between two curves.
 Poisson Solver…	The **Poisson Solver** performs an electrical simulation with the 1D structure, and calculates profiles for a set of electrical quantities.
 Tracers…	Tracers are used to illustrate the path of vector fields within 2D Mesh structures. They are drawn as small markers, which can be positioned anywhere inside a vector field, and are then animated by TonyPlot to show field strength and direction.

2.2.5 Production Menu

This menu becomes available when you use TonyPlot with VWF and you enable the **Production Mode**. It contains two groups of options. The first group is a list of all the main **Production Mode** features. The second group allows direct access to some parameter editing dialogs used in **Production Mode**. See Legacy VWF Production Mode for a full description of these advanced topics.

Table 2-6 Production Menu

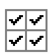
Item	Description
Interactive…	This allows interactive user control over regression model parameters to study the effects on the response surface.

Item	Description
Failure Analysis…	**Failure Analysis** predicts the most likely cause of failure in a production situation, given the characteristics of the input parameters and the failed condition.
Calibration…	The **Calibration** tool is used to calibrate model parameters. This tool will vary model parameters to best fit a set of measured data points.
Synthesis…	After a set of outputs required from a production environment is given, this feature calculates the best set of inputs that should be used to achieve that goal.
Yield Analysis…	**Yield Analysis** is used to predict the characteristics of output yield from a production situation from known experimental data.
Input Parameter Ranges…	As RSM data is loaded into TonyPlot, a record is kept of the greatest range of each input parameter.The range and nominal value of any input can be changed by using the Input Parameter Ranges dialog.
Input Distributions…	As RSMs are loaded into TonyPlot, a default distribution is assigned to each one. Using this data, TonyPlot can simulate real-life input values by sampling data with the given distribution parameters. To alter the distributions for any input, use the Input Distribution dialog.
SPC Limits…	Each output parameter used in Production mode has a set of Statistical Process Control (SPC) parameters, which can be used to monitor the value of some measured value. The **SPC Limits** dialog can be used to add or modify SPC Limit values for any output.
Experimental Results…	Each output parameter modeled by an RSM has an associated experimental value that was measured when the process input parameters were set to their nominal values. **Experimental Results** is used if no experimental value is given for an output, or if one needs to be changed.
Measured Data…	Loads a file containing measured point information.
Optimizer Setup…	The **Synthesis** mode of the Production dialog provides two synthesis methods. One method is the Levenberg-Marquardt optimizer. The parameters for the Levenberg-Marquardt optimizer can be changed in the Optimizer Setup dialog.
ASA Setup…	The parameters for Adaptive Simulated Annealing (ASA) can be changed in the ASA Setup dialog.

2.2.6 Window Menu

The **Window** menu provides access to window related functions, such as duplicating, closing, and tile management.

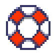
Table 2-7 Window Menu

Item	Description
 Duplicate	Creates duplicate copies of the selected plots.
 Close	Closes the selected plots and discards the data from memory.
 Select All	Selects all plots.
Tile Vertically	Arrange the plot subwindows as vertical tiles.
Tile Horizontally	Arrange the plot subwindows as horizontal tiles.
Stack	Stack the plot subwindows to maximize the viewing area for the first plot.
Thumb Stack	Stack the plot subwindows and display thumbnail images for navigation between plots.

2.2.7 Help Menu

The **Help** menu provides access to the online assistance available in TonyPlot.

Table 2-8 Help Menu

Item	Description
 TonyPlot Help	This shows the latest TonyPlot user's manual in PDF format.
 Release Notes	This shows the latest TonyPlot release notes in PDF format.
 About TonyPlot	This shows a dialog notice displaying the version number of the program and its component libraries.

2.3 Plot Control

Each plot has an associated group of parameters, collectively known as a display setting, that determine how to draw the plot in its subwindow. These parameters are independent from the structure(s) that are represented in the plot, so you can apply them to a number of other structures to make data comparison easier. A plot is controlled using a combination of the following techniques.

- Plot Selection
- Pointer Zooming
- Key Commands
- Command Stream
- Plot Menu

2.3.1 Plot Selection

To control plots and change their display parameters, select one or more plots. When the view consists of only one plot, it is always selected. When the view consists of multiple plots, you must select those to be used. To select a single plot, click on the plot using either the left or middle mouse button. To select additional plots, hold the Ctrl key while clicking either the left or middle mouse buttons. Alternatively, all the plots can be selected at once by selecting **Window→Select All**.

When multiple plots are selected, the plots that are currently selected are indicated with an orange triangular earmark in the top right corner of the plot. The first selected plot, or focus plot, is indicated by a thin blue border around the plot.

2.3.2 Pointer Zooming (1D/2D)

To zoom in on a specific portion of a plot, you can use a method known as pointer zooming. By dragging the pointer across the plot, a dynamic box can be drawn around the area of interest. If you hold down the Shift key while dragging the mouse, you can also move the start point of the rectangle.

When you release the mouse button, the plot will be redrawn so that the area within the box fills the whole subwindow.

Note: All selected plots of the same type are zoomed in by the same relative coordinate of the zoom rectangle.

2.3.3 Key Commands

Key commands are available for some plot control. Point at a plot with the mouse and press one of the following keys:

- **B** (bias) display bias information for structure from device simulation.
- **G** (grid toggle) turns the axis grid on or off.
- **H** (help) display a list of these key commands.
- **J** (junk data) prints a list of all “junk data” attached to the plot structures.
- **R** (region data) prints all attached region data for all structures in the plot.
- **V** (value label) adds a spot height value label to contour plot (RSM 2D plots only)

2.3.4 3D RSM & VWF Plot Control

3D RSM and VWF plots can be rotated and scaled, but cannot be zoomed.

To rotate an RSM or VWF 3D plot, hold down the left mouse button and drag the pointer left and right over the plot. A wire-frame bounding cube will be drawn around the plot and rotates as the mouse is moved. Position this cube to the desired viewing angle, and release the mouse button. The plot is redrawn from the new view point.

To scale an RSM or VWF 3D plot, hold down the Shift key and the left mouse button. Then, drag the pointer up and down over the plot. A wire-frame bounding cube is drawn around the plot and grows or shrinks as the mouse is moved. Scale this cube to the desired size, and release the mouse button. The plot is redrawn at the new size.

2.4 Legends

Data is of little use without a legend to explain its meaning. TonyPlot uses legends in a variety of situations to explain how the information in a plot is being represented. Each legend is drawn in a default position in a plot that does not overlap with any other legend. You can change these default positions, if necessary. You can also drag legends to any location in the window. In an overlay plot, a legend applies to all levels, since the data displayed on each level is the same. An overlay plot also has its own “level” legend to indicate the different levels in the plot.

2.4.1 Legend Types

- **Contours**: This indicates the value range for a given color when using filled contours. This legend also indicates the value of a color when using line contours. Up to three of these legends may appear at once, since a 2D plot may contain up to three sets of contours. Contour legends show only whole integer values when the quantity is contoured on a log scale.
- **Regions**: This can be displayed on 2D mesh plots and Cross section plots. The region legend shows region-to-color relationships. Usually, the regions are represented by the materials that they are made of and the legend shows these materials and their colors.
- **Graphs**: Whenever graph lines are drawn, there is a graph legend. This indicates the different quantities shown by each graph line. These appear, for example, in XY Graph plots and Cross Section plots.
- **Vectors**: This indicates the size of arrows and magnitudes that they represent: largest, smallest, and halfway between. You can judge the magnitude of other arrows by comparing to this legend.
- **Levels**: To distinguish data between levels in an overlay plot, a level legend appears whenever there is more than one level. These show the level-to-color relationship for that plot. By default, overlay levels are distinguished by the name of the file that was used to create each level. You can change these names to any other name.

2.4.2 Positioning Legend Boxes

There are two ways to position legend boxes. The first method uses the **Legend Options** category of the Preferences dialog. This shows a list of all the types of legends mentioned above. There are six possible default locations. Optionally, you may use the middle mouse button to drag the legend to the desired location.

The default locations are set up in such a way that there are no overlapping legends. When choosing new default locations, be aware of the possibility of overlapping legend boxes. The settings can be saved as the default settings to be used between sessions.

The second method is to position legend boxes manually. To do this, point to a legend box on a selected plot. Then, drag the mouse while holding down the middle button. This allows you to drag the legend box to any position in the window.

2.4.3 Legend Drawing Styles

There are some TonyPlot preferences that control how to draw legends. These are described in Preferences but are briefly described here.

- **Legend type**: This can be transparent or opaque. Transparent legends allow you to see the plot underneath. Opaque legends overwrite the plot underneath them.

- **Function label**: When a function is plotted, it can be indicated on the appropriate legend with either its name or its definition. For example, if **Function 1** has been set up as `log(current/10)` and you choose **Name**, the legend will display “Function 1**”**. If you choose **Definition**, the legend will display “log(current/10)”.

2.5 Annotation

The **Annotation** dialog (Figure 2-2) allows you to specify plot parameters that are independent on the data, which do not fall into the category of display settings. These are features such as titles and axis ranges. The titles, axis labels and footers can use Greek characters and math symbols via special tags. See Appendix C Greek and Math Characters for more information. The features of a plot that are independent from the type of plot are called annotation features. To access this dialog, select **View→Annotation...**.

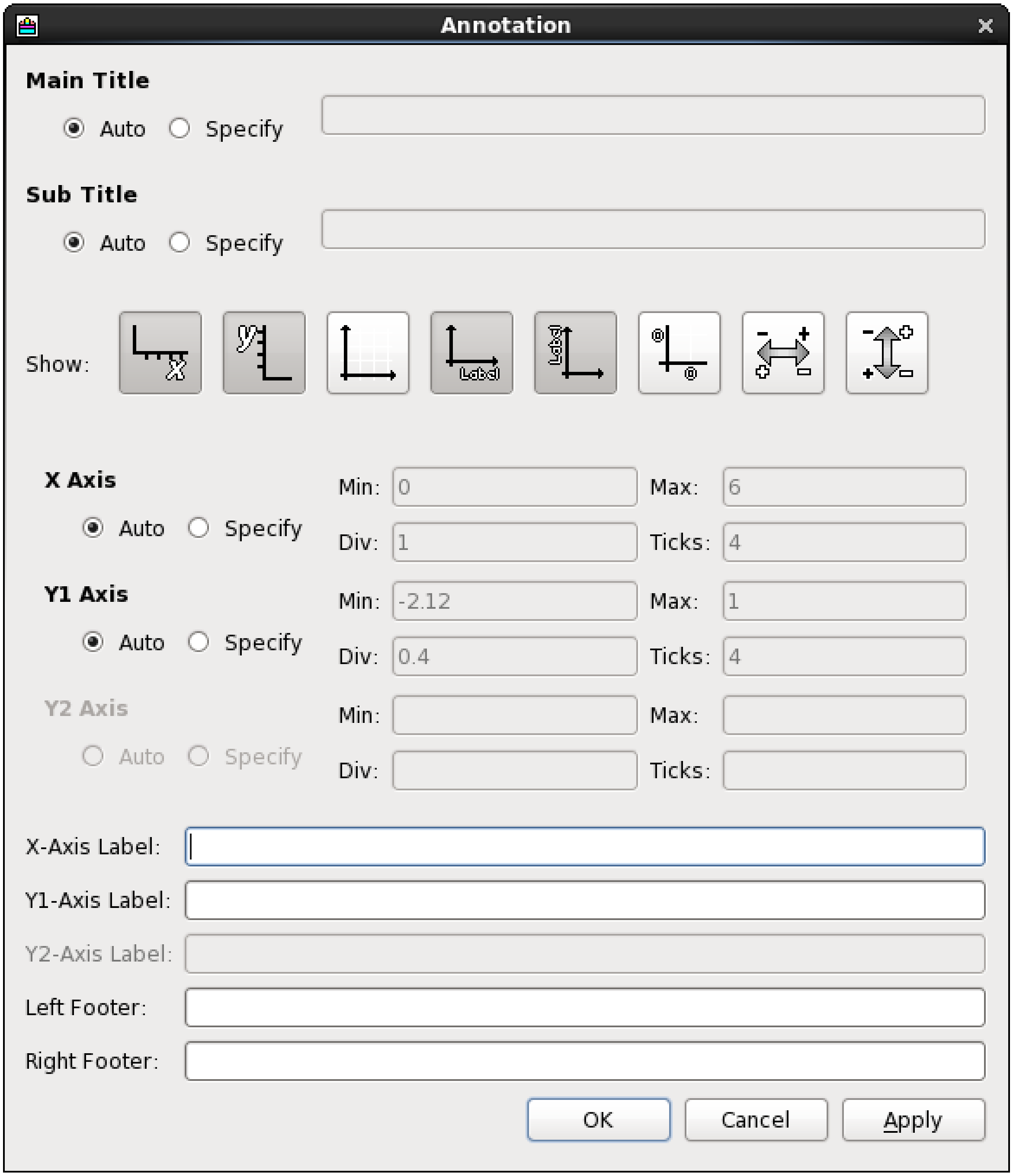


Figure 2-2 Annotation Dialog

2.5.1 Titles

Each 1D or 2D plot has two titles. TonyPlot assigns these titles by default. You can change them if required. The current main title and current subtitle are displayed in their respective text fields on the dialog. You can change the titles without having to redraw the plot(s) completely by changing the title and pressing the **Apply** button. This leaves all other items as is and only the titles are redrawn on the selected set of plots.

Note: If you select **Specify**, you can change the titles. If **Auto** is selected, titles cannot be changed. This allows you to change more than one plot with the Annotation dialog without the titles on all plots ending up the same.

2.5.2 Show

This item controls features that appear around the edge of the plot. The icons represent, from left to right, X axis ticks and numbers, Y axis ticks and numbers, grid, X axis label, Y axis label, and zero lines. The large buttons can be used to invert the plot. For example, reverse the positive and negative directions of the X or Y axis.

2.5.3 Range

The default ranges on the X and Y axes are calculated to ensure that all of the data from all structures in the plot can be seen. You can change these values. Selecting **Specify** rather than **Auto** enables the axis control items, allowing you to enter the minimum and maximum values, as well as the division and number of ticks per division.

When the axis ranges are specifically set and applied to multiple selected plots, all plots are scaled the same. This allows easy plot comparison of similar data.

2.5.4 Statistics Plots

Some statistics plots do not have the regular range controls as described above. Instead, the controls allow you to enter user-specified “bin values”, which are used when calculating data distributions.

When you select a statistics plot and summon the **Annotation** plot, the range controls include a choice to allow you to select **Auto** bins (automatically determined by TonyPlot according to the data range) or to specify the bin values. When you choose **Specify**, use the **min** and **max** text fields to enter the minimum and maximum bin values. Then, enter the **Number** of bins to be used between these limits. TonyPlot adds each bin value to the scrolling list when you press the **Apply** button.

2.5.5 Axis Labels

The X axis and Y axis labels can be modified. These are only updated on the plot when the **Apply** button is clicked on. Once an axis label has been set this way, it is always shown, even if the quantity represented on the axis is changed. To return to the normal axis label, erase the user-specified label from this field and click on **Apply** again.

Note: Cross Section plots generated by the Cutline tool will display an automatic x axis label if none is entered manually. This automatic title is provided by TonyPlot according to the type of cutline x axis desired (see Preferences).

2.5.6 Footers

Any 1D or 2D plot can have footer subtitles in the left or right or both corners. No footers are drawn by default. You can add them with these text fields in this dialog.

2.5.7 Special Characters

Titles, axis labels and footers of 1D and 2D plots may all contain “special characters” if needed. These allow alternative letters and symbols to be drawn, such as Greek letters, and superscript numbers.

2.6 Exporting Images

The Export Image dialog (Figure 2-3) is used to save plots to the system clipboard, or to image files on disk. It allows you to export all currently selected plots, all plots (discrete) or all plots (composite). If you have more than one plot selected, or have selected “All plots (discrete)” in the Export drop-down menu, image output files will have an index attached to them, such as `export-1.png`, `export-2.png`, `export-3.png`, depending on the number of plots selected.

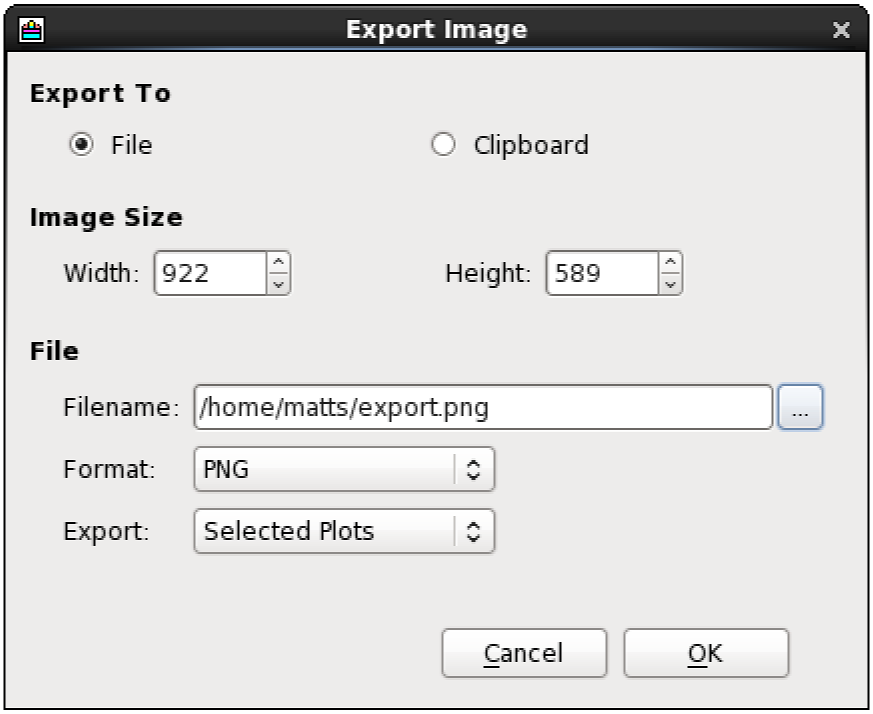


Figure 2-3 Export Image Dialog

2.7 Tools

As well as displaying information contained in structures, TonyPlot can examine that information in a variety of ways. Each method of examining the data is called a **Tool**.

2.7.1 Object Browser

The **Object Browser** provides a quick way of setting many different properties that define how a plot is drawn. Selecting **Tools→Object Browser…** will toggle the **Object Browser** state between shown and hidden. Hiding the **Object Browser** will allow the plot subwindows to occupy more screen area.

The **Object Browser** displays the configurable objects of a plot as a hierarchical tree. When an object is selected, that object's properties are displayed as **Property**, **Value** pairs in the table below. Each **Value** can be edited, and any changes are immediately updated in the plot drawing.

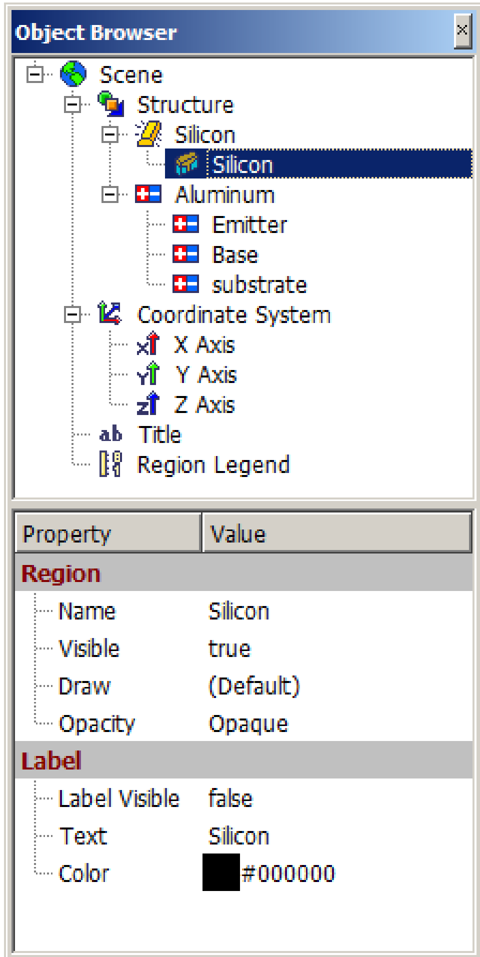


Figure 2-4 Object Browser

2.7.2 Cut Line

The **Cut Line** tool is used on 2D Mesh plots only. It is used to create 1D cross section plots from arbitrary positions within a 2D structure. The **Cut Line** tool also has the ability to move cut lines through a device and watch the profile move and shift as the cut line position updates. The **Cut Line** tool also allows these sequences to be combined into a **Movie**.

The **Cut Line** tool dialog (Figure 2-5) consists of the following items:

Create: The top section provides different choices for creating cut lines. These choices are free, vertical, horizontal, chained, interface, and keyboard. Each of these is described below.

- **Select**: The center portion allows you to choose any cut plane created for manipulation. This includes shifting and movie making of the cutline.
- **Movie**: The movie section (displayed when the **Make movie** button is clicked on) is used to make a movie from a cut line by repeatedly moving its position.

Shift Position: You can move a cut plane once created by using these controls, which are displayed when you click on this button.

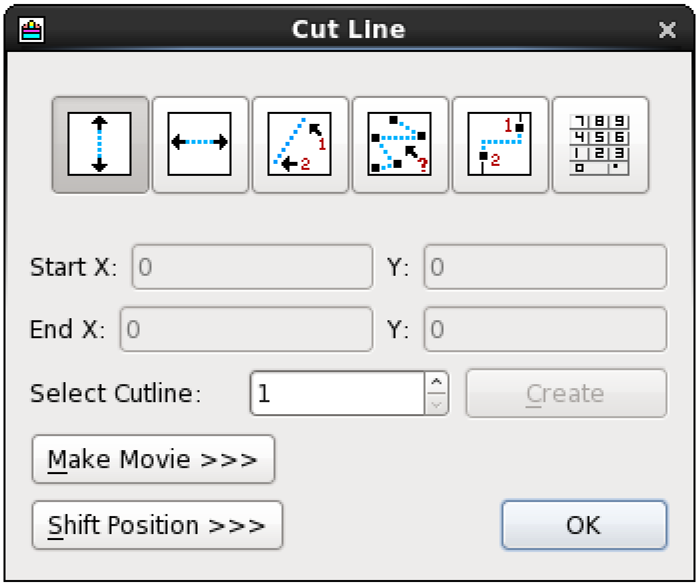


Figure 2-5 Cut Line Dialog

2.7.2.1 Creating a Cutline

To create a cut line, select one of the **Create** options and follow the actions specific to the mode to define the cross-section.

Note: While dragging, holding down the Shift key causes the start point to move as well as the end point.

- **Free**: To define a free cut line, drag the mouse over the source plot to define a line through the mesh. Releasing the mouse button creates the cross section.
- **Vertical**: Same as free but dragging is restricted to a vertical direction. Release the mouse button to create the cross section.
- **Horizontal**: Same as free but dragging is restricted to a horizontal direction. Release the mouse button to create the cross section.
- **Chained**: To create a chained cut line, click on the mouse left button in various places to create a polygon of chain line segments. To erase to last point placed, use Shift + Left Click. Press the Return key to create the cross section.
- **Interface:** To create this type of cut line, click on the mouse left button to place two points on any interface (region boundary). Use Shift + Left Click to erase to last point placed. Press Return to see the portion of the interface along which the cross section is calculated.

If the portion is wrong, press to try other routes. When ready to create to cross section, press the Return key once more.

- **Keyboard**: To create a new cut line with exact start and end coordinates, enter the start and end points into the text fields supplied and click on **Create**. This creates the new cross section from the line so defined.

2.7.2.2 Creating From Multiple Plots

If you select more than one mesh plot when a creating cut line, a cross section will be calculated for each of them, and will generate the appropriate number of new plots. If you make a cut line from an overlaid mesh plot, the cut line plot generated is also an overlay plot with each level showing the cut line profile from each level in the mesh plot.

2.7.2.3 Cross Section

The cross section created displays the profile of the quantity that was contoured on the mesh, or shows a default profile if no contours were drawn. You can control this new plot just like any other cross section plot. All the quantities that were present in the mesh are also available in the cross section.

Default titles show the mesh data file from which the cut was made, and show the start and end positions of the line.

Interface cut lines create an overlay plot from a single mesh plot. Each level in the overlay represents profiles from one of the materials present at the interface. For example, an interface cut line taken along an oxide/silicon interface creates an overlay cross section with one level showing profiles in oxide and the other level showing profiles in silicon.

Any other type of cut line produces an overlay cross section if the source plot was an overlay plot. In this case, the new cross section contains one level for each level in the mesh plot. For example, if two meshes alpha and beta are overlaid and a cut line taken, the new cross section plot contains two levels. The first level contains profiles from alpha. The second level shows profiles from beta.

2.7.2.4 Deleting

To delete a cut line, delete the cross-section plot. This removes the cutline from the mesh from which it was created if still present in the view.

2.7.2.5 Shifting

To shift any created cut line (except interface cut lines), click on the **Shift Position** button. By using the directional arrows on the **Cut Line** tool dialog, the cut line position is moved up, down, left or right.

The amount moved is shown in the **Delta** text fields, which can be modified. To use the shift feature, select the mesh plot that contains the cut line.

2.7.2.6 Movies

You can create a cut line movie automatically from the **Cut Line Tool** dialog. It is created simply by moving the cut line position many times and sequencing the resulting cross sections. To create a movie, define the step size and number of steps, and define whether to move the cross section horizontally or vertically. Note that this does not move the actual position of the cut line on the plot as shifting does. To use the movie feature, select the mesh plot that contains the cut line.

2.7.3 Cut Plane

The cutplane tool is used to create 2D slices through a 3D structure. The resulting slices can be viewed as 2D mesh plots (see 2D Mesh Plot Display), or used as inputs to a 2D device simulator such as Atlas.

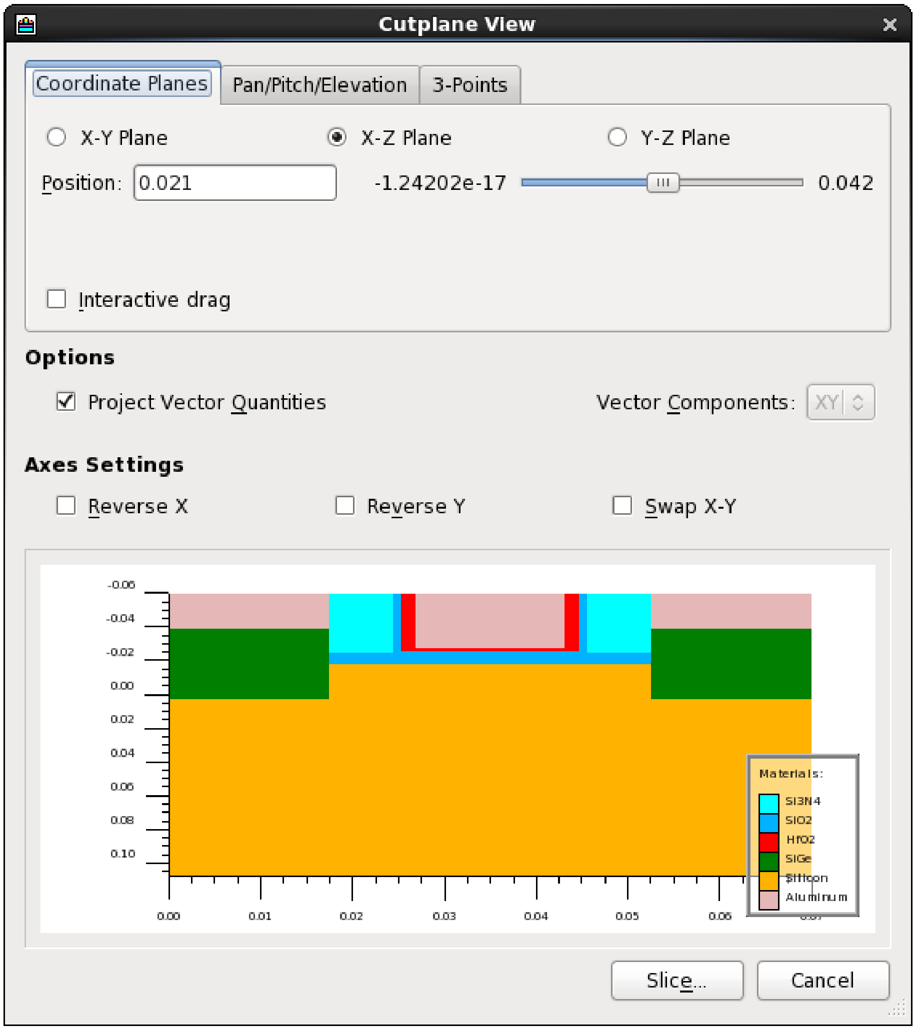


Figure 2-6 Cutplane Dialog

2.7.3.1 Creating a Cutplane

The Cutplane Dialog is shown in Figure 2-6. Using the tabbed controls, the slicing plane can be expressed as a position on one of the coordinate planes, by pan-pitch-elevation, or by specifying three points on the plane.

- **Coordinate Planes**: Allows slicing in the XY, XZ and YZ planes. The position line-edit and slider controls specify the absolute position along the axis normal to the chosen plane.
- **Pan/Pitch/Elevation**: The pan and pitch angles are used to orient a normal to the plane, and the elevation specifies a relative distance along the normal.
- **3-Points**: Three points can be used to specify the slicing plane. The entered coordinates must be unique and must not be collinear.

While positioning the cutplane, you can monitor the positioning continuously as it changes or update it after it has been moved. To do this, toggle the **Interactive Drag**. When **Interactive**

Drag is checked, the cutplane preview will update continuously when the slider controls are adjusted.

Note: When a structure has many mesh vertices, selecting **Interactive Drag** can cause the controls to lag and become unresponsive while TonyPlot repeatedly re-computes the 2D slice.

2.7.3.2 Options

TonyPlot allows you to control how 3D vector quantities are sliced onto the 2D cutplane. When **Project Vector Quantities** is checked, vector quantities are mapped onto the XY plane.

The top half of the dialog shows the various settings to adjust the cutplane, while the bottom half shows a preview of the extracted cutplane.

2.7.3.3 Axes Settings

The orientation of the axes on the 2D cutplane can be adjusted using the **Axes Settings.** Checking **Reverse X** inverts the X-axis. Likewise, **Reverse Y** inverts the Y-axis, and **Swap X-Y** swaps the X- and Y-axes.

2.7.3.4 Export Slice

Clicking the **Slice…** button brings up the Export Slice dialog shown in Figure 2-7. Using the **Export Settings**, the 2D cutplane slice can be displayed in a new plot subwindow and/or saved to a file on disk.

The controls in the **Slice Settings** area optionally allow multiple, uniformly spaced, cutplane slices to be created in a single action. The resulting slices will all be parallel to the slice defined in the Cutplane Dialog.

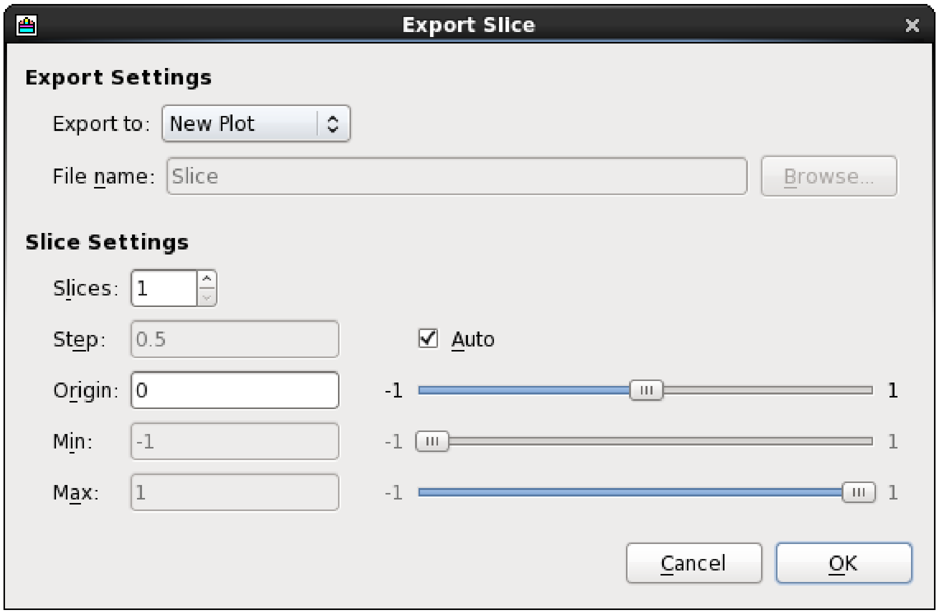


Figure 2-7 Export Slice Dialog

2.7.4 Markers

The **Markers** tool is used to create labels or markers on a 3D mesh plot. A marker consists of a small dot at a specified (X, Y, Z) coordinate, a line, and a text label. The positions of the line end point and the label are determined automatically by projecting a ray from the center of the structure, through the marker coordinate.

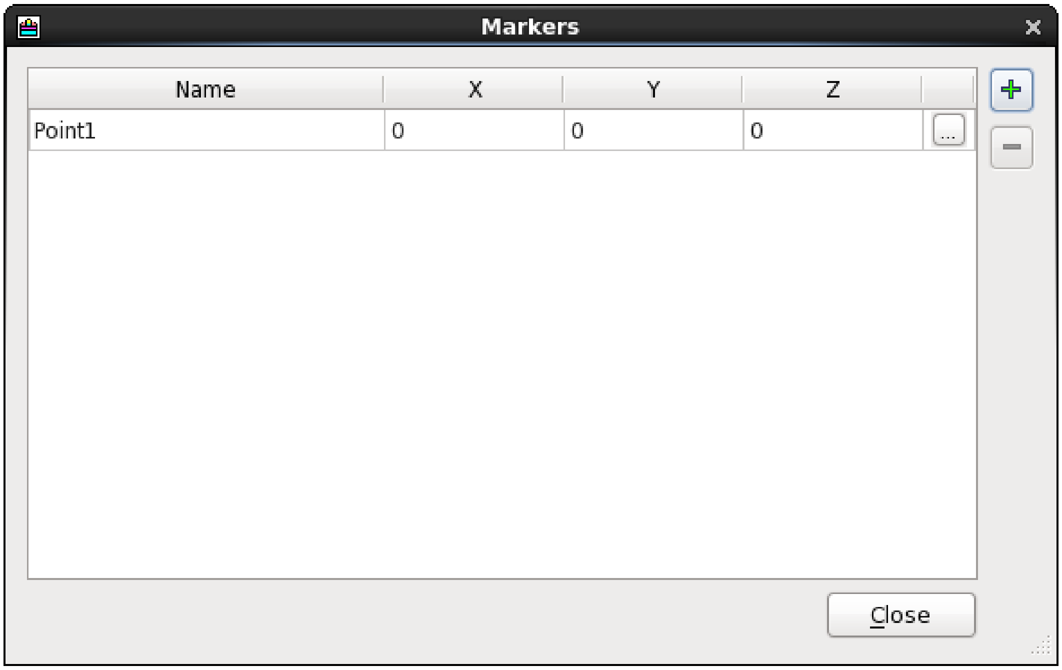


Figure 2-8 Markers Dialog

2.7.5 Probe

The Probe tool allows you to probe information about any point within a 2D or 3D Mesh.

2.7.5.1 2D Mesh

The Probe tool for 2D Meshes provides both geometry and impurity data and can be used to find specified structural features. This can be useful for debugging simulator output as well as for general use. To use the Probe, select one or more 2D Mesh plots and choose **Tools→Probe…**. Click anywhere within a structure to activate the probe. A crosshair marker indicated the last position clicked. Measurements are then displayed in the Probe dialog box as shown in Figure 2-9.

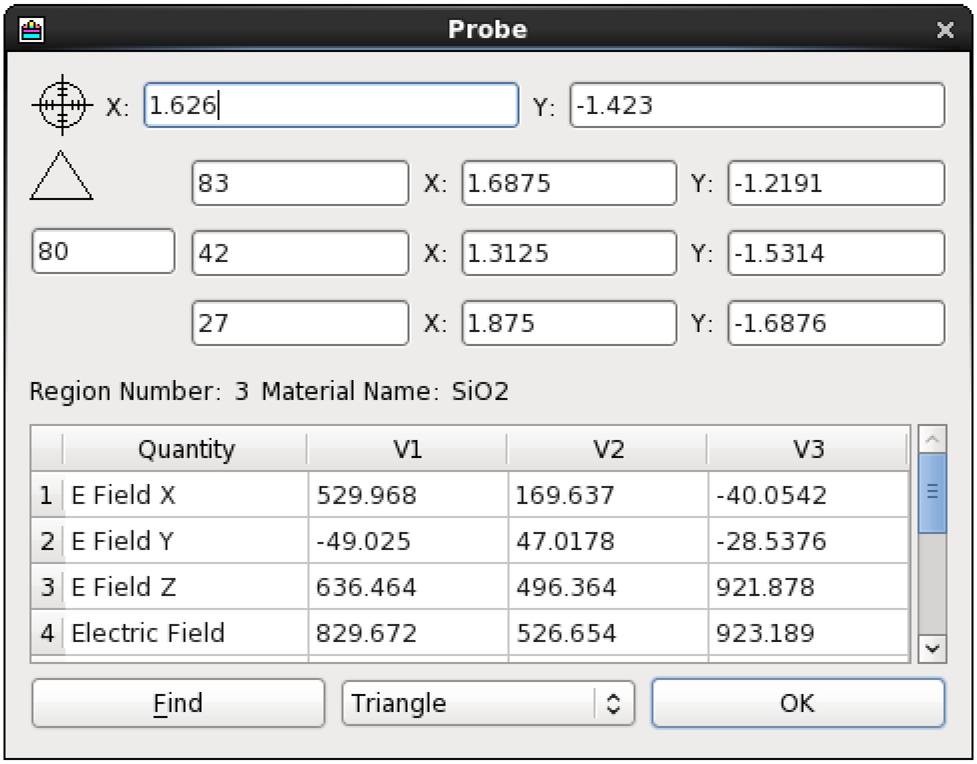


Figure 2-9 Probe Dialog

- **Probe Coordinates:** The panel at the top of the tool dialog box shows the position where the probe was last placed.
- **Geometry Info:** The second panel shows information about the triangle in which the probe was positioned. The internal index is given for the triangle itself and for each of its three vertices. The actual coordinates of the corners are also displayed.

2.7.5.2 3D Mesh

The Probe tool for 3D Meshes provides both geometry and impurity data. To probe a point, click on the **Probe** button in the Toolbar, or select **Tools→Probe...**, then click on the point you wish to examine. When you click on the structure, a probe marker sphere appears at the selected location on the structure. The Probe View (Figure 2-10) is then updated with the relevant information. If **Snap** is checked, the probe position is snapped to that of the mesh vertex nearest to the point that was clicked.

The Probe View will display the probe marker sphere and the picked element (tetrahedron, prism, triangle or quad), with node identifiers drawn at each of the vertices. For each node, quantity values are displayed. The exact position of the probe is also shown with its material and region identification. The ellipsis button on the Position line allows a probe position to be manually entered, for precise positioning at a point of interest.

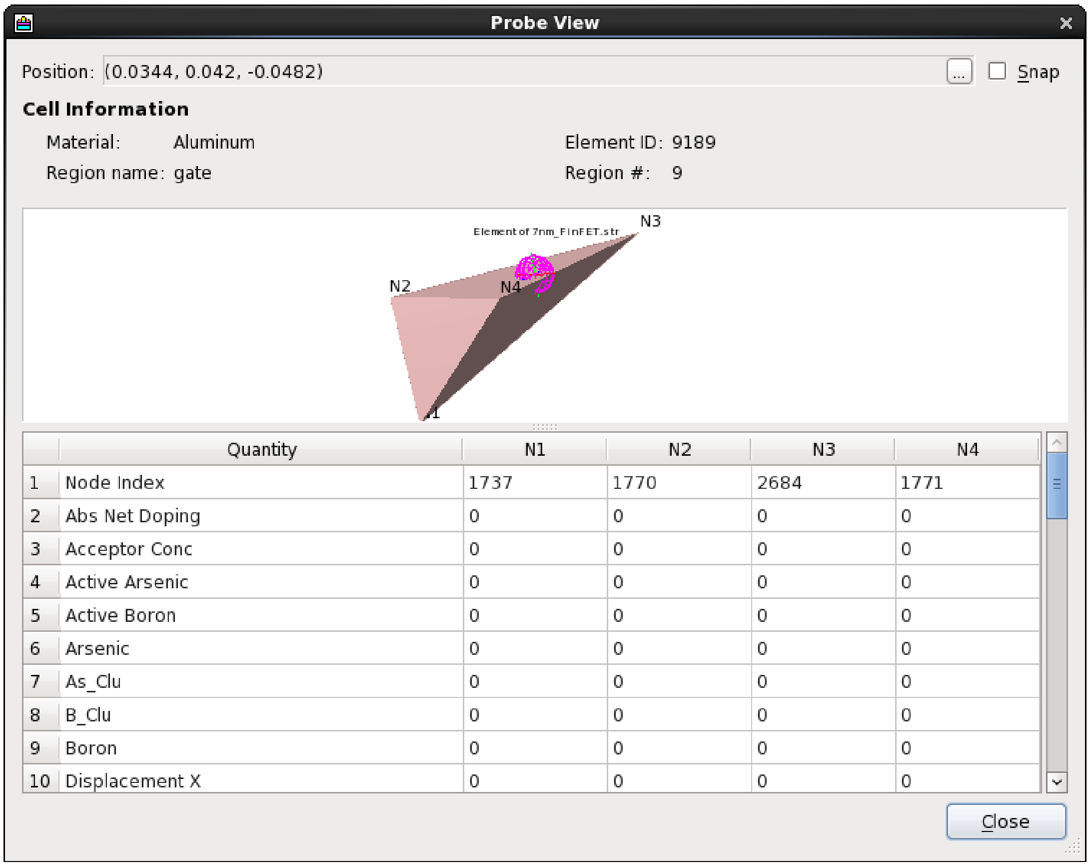


Figure 2-10 Probe View Dialog

2.7.6 Ruler

The **Ruler** tool can be used on any type of plot. It provides coordinate geometry information of any line drawn over a structure.

2.7.6.1 1D and 2D

To use the **Ruler** (Figure 2-11), select the plots in which measurements are to be taken and choose **Tools→Ruler...**. This item is only active when at least one plot is selected.

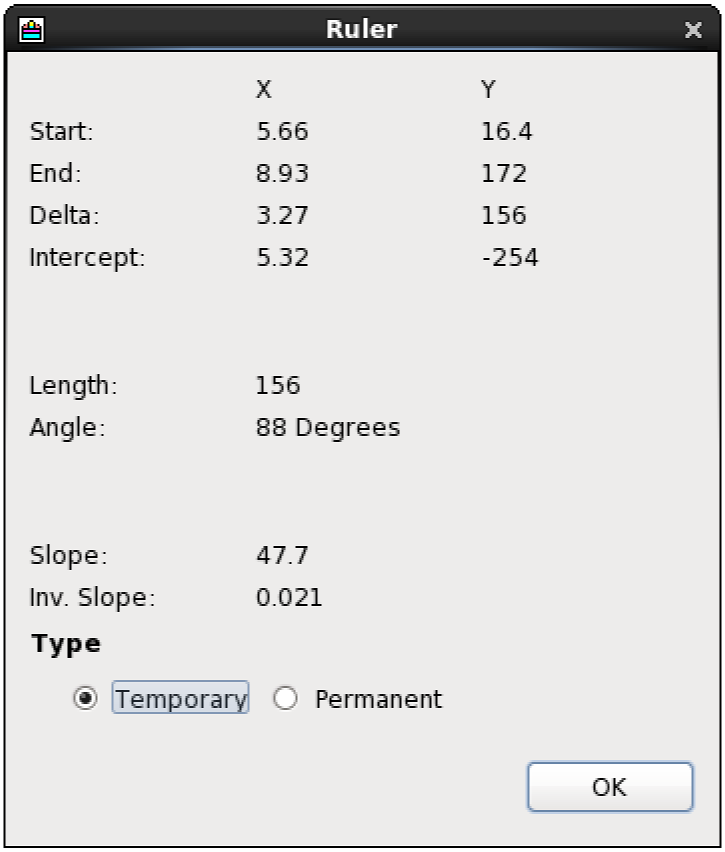


Figure 2-11 Ruler Dialog

The ruler position is defined by dragging the pointer across the plot to define a box and line. Holding the Shift key down while dragging the pointer moves the start and the end points of the ruler. The Ruler dialog shows the following information:

- **Start**: The coordinate of the start of the ruler.
- **End**: The coordinate of the end of the ruler.
- **Delta**: The vertical and horizontal distances between the start and end points.
- **Intercept**: The intercept point on the X and Y axes. The terms **X axis** and **Y axis** refer to the lines `y=0` and `x=0` respectively, and not the axes along plot edges.
- **Length**: The distance from the start point to the end point.
- **Angle**: The angle of the end point taken from the start point. 0 is towards positive X, -90 is negative Y, +90 towards positive Y, and ±180 is towards negative X.
- **Slope**: The gradient of the ruler line.
- **Inv. Slope**: The inverse gradient (1 / slope).
- **Type**: Two types of ruler are available. The default ruler is called a “temporary” ruler because once the mouse button is release, the lines are removed from the plot (but the values remain displayed in the **Tool** dialog). A “permanent” ruler, however, remains in the plot. The temporary ruler lines are drawn in the plot window, and some of the values from the dialog are also added at relevant places. This permanent ruler remains on the plot until you place the ruler once more. Switching the ruler type back to “temporary” also removes

a permanent ruler.

To return to normal use of the plot window, close the **Ruler** dialog.

2.7.6.2 3D

The **Ruler Dialog** for 3D Mesh Plots (Figure 2-12) is used to obtain information about quantities within a 3D mesh structure. To use the ruler, first select **Tools→Ruler...**. Press and hold the CTRL key and the left mouse button where you wish to start. Then, drag the mouse and release the button where you wish to stop.

The **Start** and **End** XYZ positions with the difference between each respective pair, **Delta**, displayed. The length of the line, **Length**, is also shown. Any quantity values present in the structure are also displayed for the **Start** and **End** points, along with their difference.

Rulers can be made permanent by setting the **Anchor ruler** switch. To anchor a ruler, check the **Anchor Ruler** box before you draw the ruler. A ruler identifier will be created. You can anchor multiple rulers in a structure, each of which will have a unique identifier. You can delete each ruler separately or all at once.

If **Snap to vertex** is checked, TonyPlot will snap the start and end points to the mesh vertex nearest to the mouse pointer when the ruler is placed. If **Snap to vertex** is unchecked, the start and end points will be interpolated.

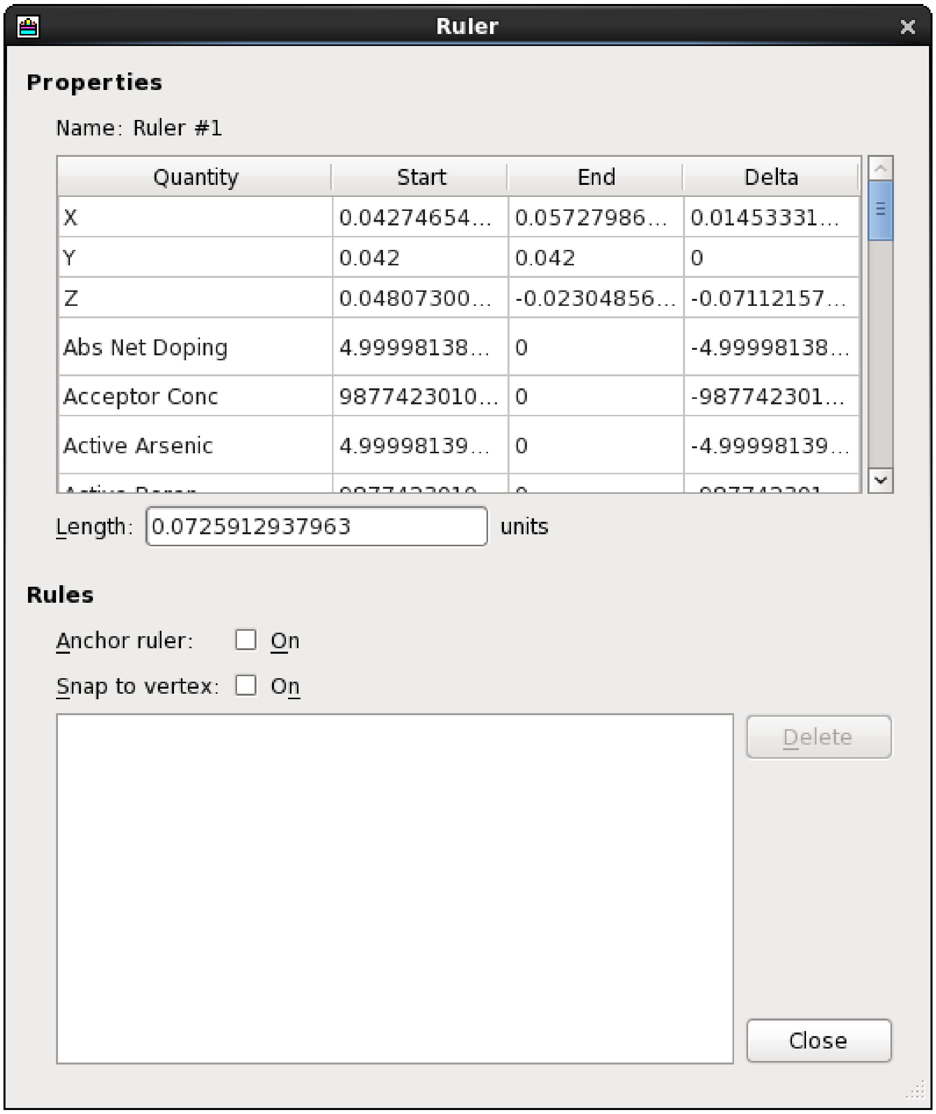


Figure 2-12 Ruler Dialog (3D)

2.7.7 Labels

Labels are used to add arbitrary notes and informative text to any 1D or 2D plot. For 3D plots, see the Markers tool. Labels can be drawn with leader arrows or can be free standing. The **Labels** dialog (Figure 2-13) is used to add, change, and delete these labels. To access this dialog, select **Plot→Labels**. As with the **Annotation** dialog, the Labels dialog is the same for all plot types. The difference, however, is that only the first selected plot is affected. Each plot has a list of labels associated with it. Each label has a position in the plot to which it belongs. The labels that belong to the selected plot are shown in the list on the Labels dialog. Labels can use Greek characters and math symbols via special tags. See Appendix C Greek and Math Characters for more information.

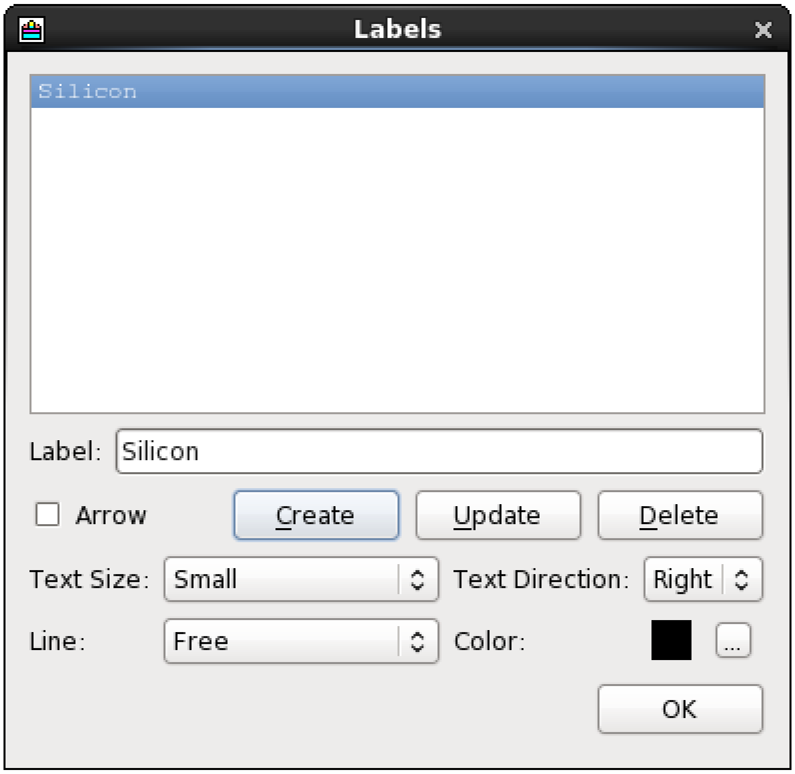


Figure 2-13 Labels Dialog

The following describes each of the items on the dialog.

- **List of labels**: This list contains all the labels that have been defined for the selected plot.
- **Label**: This shows the text of the selected label and is used to enter new text when creating or changing a label.
- **Arrow**: Checking this box will add an arrowhead to the end of the label leader line.
- **Create**: Clicking on this button creates a new label with the text shown in the **Label** text field. If the field is empty, the new label is created with its text set to **New label**. The label attributes are set from the state of the attribute items (e.g., arrow and size). You can have more than one label with identical text.
- **Update**: This replaces the selected label with new text or attributes or both. Use to change the label attributes, such as color and size.
- **Delete**: Clicking on this button deletes the label that is selected in the list. The label is removed from the plot if it has been placed.
- **Text Size**: This controls the size of the letters in the label. Three sizes are possible: **Small**, **Medium**, and **Large**.
- **Text Direction**: This determines the direction of the text. The normal choice is **Right**, which draws regular text. **Up** and **Down** draw text rotated by 90° upwards or downwards.
- **Line**: When placing a label with a leader line, the leader line can be forced to snap to angles of 45°. This is a “constrained” leader line. A “Free” leader line can be drawn at any

angle.

- **Color**: A color palette is provided for selection of the label color. This is used for both the text and the leader line.

2.7.7.1 Placing Labels

Labels are placed on the selected plot in one of two ways. One, by clicking to place a simple text-only label. Two, by dragging to place a label with leader line (see Figure 2-14).

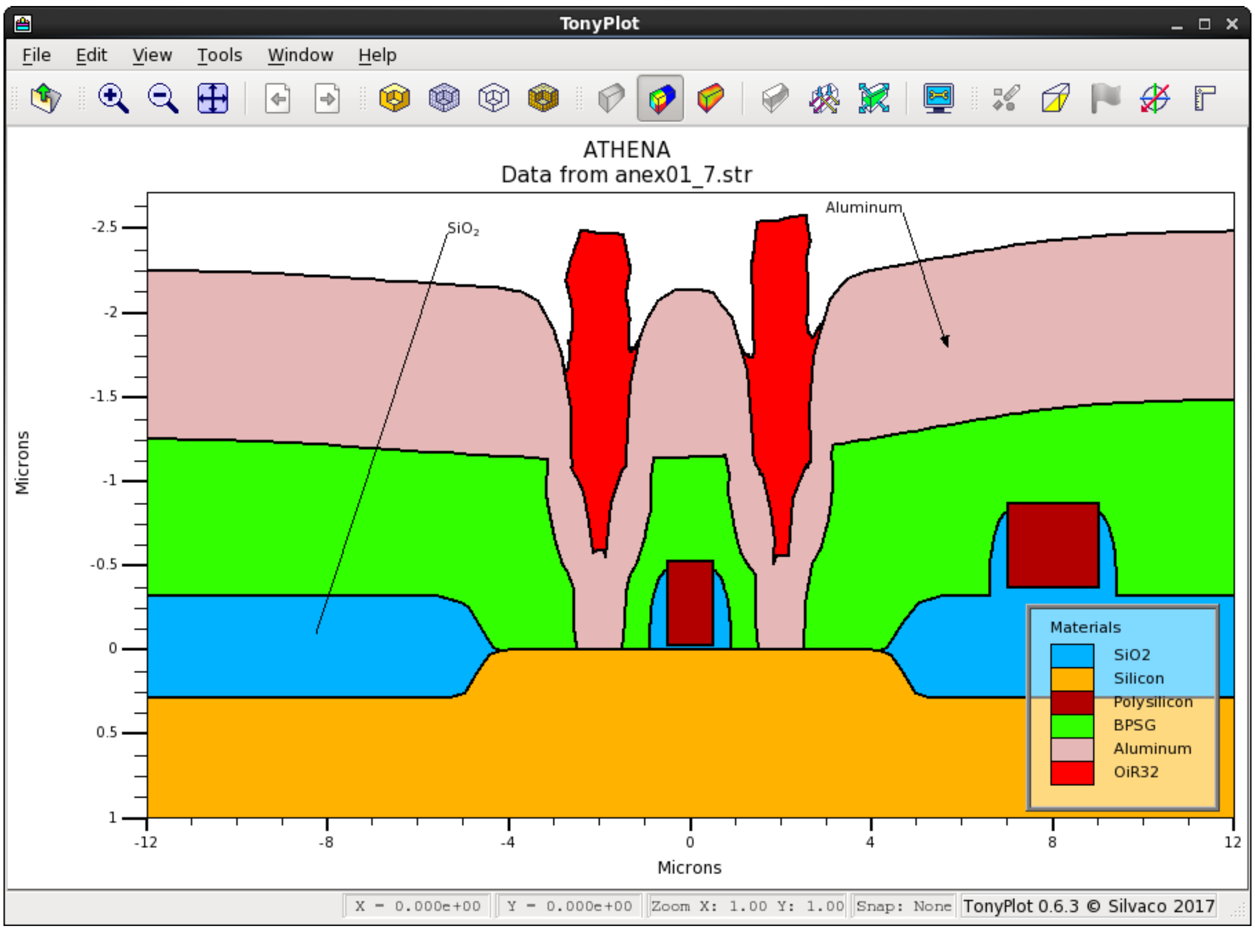


Figure 2-14 Plot with Labels Added

If a leader label is being placed, the start of the drag is where the text will appear, and the end of the drag will be the end of leader. In other words, the position to which the leader points. As with any drag operation, holding down the Shift key moves the start and the end points of the drag. When the drag is done, the label text is positioned correctly relative to the direction of the leader line. For example, if the leader points down and to the right, the text will be placed so that the leader starts from the bottom right corner of the text.

If you place a simple text-only label, the cursor indicates how the text is positioned by pointing to a corner. For example, if the cursor points up and right, the text will be placed so that the clicked point is in the bottom left of the text. You can change the cursor to obtain different alignments by pressing the P key on the keyboard. Four positions are available.

To move a label, repeat the placing procedure.

2.7.7.2 Special Labels

In some cases, TonyPlot generates labels automatically. If text appears on a plot, it is usually a label placed by TonyPlot that can be controlled with the regular label dialog as explained above. Some examples of special labels are:

- **Integration Tool**: This tool (see Integrate) adds a label to show the integrated x-range and area. Although placed in a default position, you can customize the label attributes with the Labels dialog.
- **2D RSM Plots**: Pressing the ‘V’ key in a 2D RSM plot adds a spot height label to the plot. The label can be moved but the height does not change, so the label value would then be invalid.
- **Electrode Names**: 2D Mesh structures from Athena or Atlas can contain electrode information. When electrode names are plotted, they appear as labels. By default, they are positioned over the appropriate electrode but can be moved if desired.

2.7.8 Movie

The **Movie** tool allows you to combine a group of plots into an animated sequence, which can be viewed in playback like a slideshow. To create a movie, you must select the slides. To do this, create a group of plots in the main TonyPlot view, select this group, and choose **Movie...**. This item is active when you select at least two plots. You will notice a delay while TonyPlot creates the movie sequence. When complete, the **Movie** dialog (Figure 2-15) appears, showing the first frame of the movie and a group of control items.

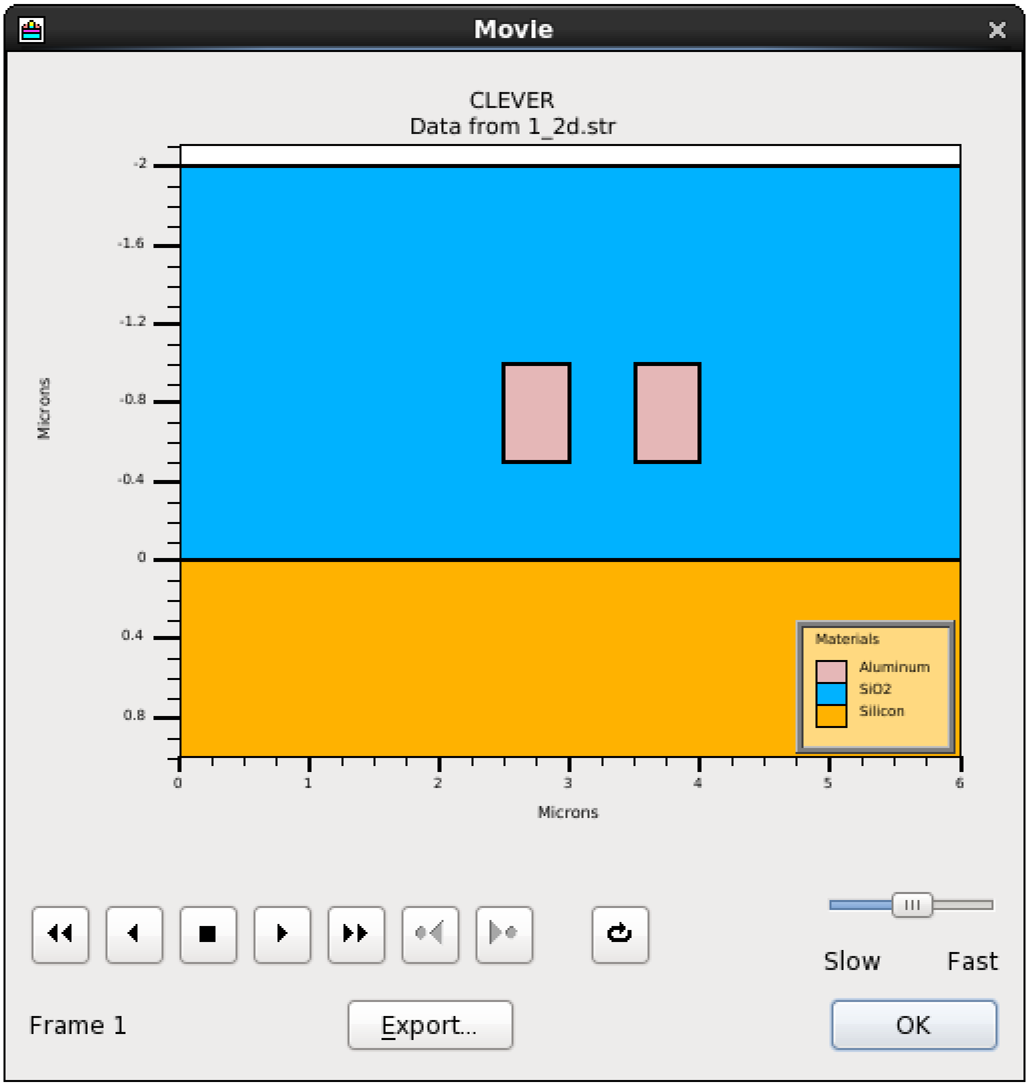


Figure 2-15 Movie Dialog

You can change the size of Movie dialog. This is explained in Preferences. The control items are:

- **Video controls**: The five play buttons perform the following functions: rewind to first frame, play backwards, stop at current frame, play forwards, and skip to last frame.
- **Repeat**: This is the item marked with a looping arrow. This repeats playback in an endless cycle in the direction determined by the play button pressed.
- **Speed**: Three playback speeds are available. To see playback speed, select the new speed and press a play button.
- **Export**: This item allows you to export the current movie as an animated GIF.

Note: Only one Movie tool can be displayed at once.

TonyPlot can create automatic movie sequences from cutlines without repetitive use of the **Cutline** and **Movie** tools.

2.7.9 HP4145

The **HP4145 Emulator** is available for any graph plot. Only one plot, however, can be used with the emulator at any one time. When you select this option (**Tools→HP4145**), the first selected graph plot changes to mimic the output of the HP4145. The **HP4145** dialog (Figure 2-16) appears containing controls that resemble the functions of the HP4145.

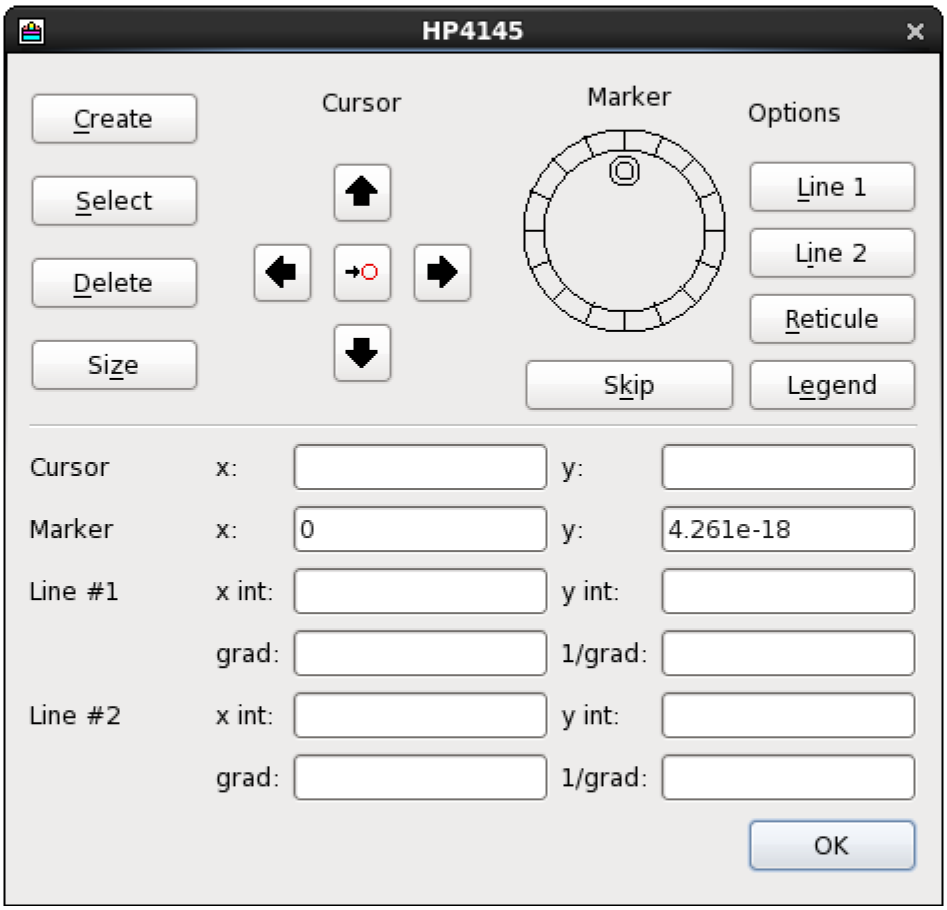


Figure 2-16 HP4145 Dialog

The controls of the **HP4145** dialog are:

- **Cursors**: The control section allows manipulation of cursors represented as plus-sign shaped crosshairs. The buttons allow you to create new cursors, to delete existing ones, to select between existing cursors (the current cursor is shown in bold), and to toggle the size of the current cursor between small and full screen. The four directional buttons move the current cursor. The central button moves the current cursor directly to the marker

- **Marker**: The marker can be moved along its current graph line by moving the dial. To move the dial counterclockwise, click at the left half of the button. To move the dial clockwise, clicking at the right half of the button. **Skip** moves the marker from one curve to the next, cycling back to the first curve when the last one has been reached.
- **Options**: Several options can be accessed from the middle panel. **Line 1** and **Line 2** toggles a line that joins the marker and cursor. Various geometry information about the lines is displayed on the dialog and on the legend. **Reticule** toggles the plot grid (the same grid that is shown using the **Annotation** dialog). **Legend** toggles the HP4145 legend. **OK** closes the HP4145 emulator and restores the plot to normal.
- **Information**: The lower panel gives position information for the marker and current cursor, and geometry information for both lines.

2.7.10 Integrate

The **Integrate** tool (Figure 2-17) allows you to measure the area under a single plot curve or the area between two curves. The X interval over which the area is calculated can be set by positioning marker lines at certain locations along the X axis. The **Integration** tool works with both **XY Graph** and **Cross Section** plots.

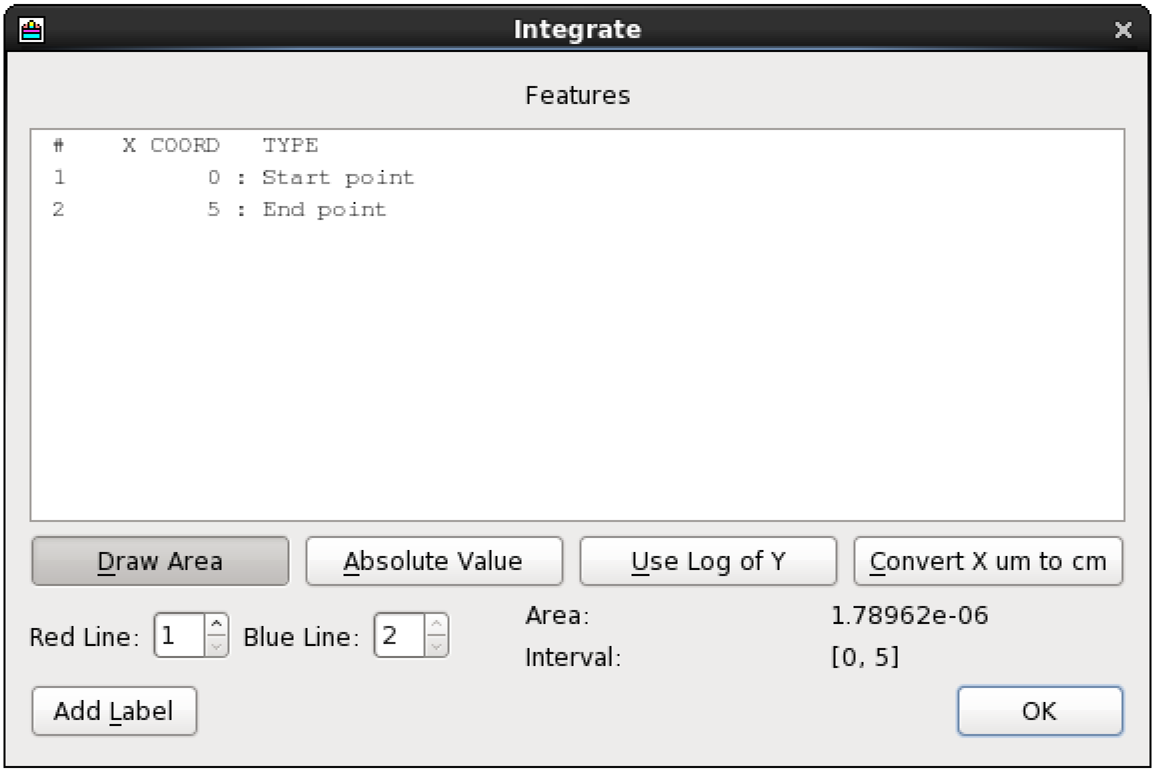


Figure 2-17 Integrate Dialog

2.7.10.1 Features

This list shows all the points of interest on the curve in the plot used. It shows all minima and maxima, as well as the start and end X values and positions of all material interfaces. The marker lines, which are used to define the X interval for the integration, can be moved to any of these features, using the arrow buttons or by using the mouse. You can also use the mouse pointer to add more features. See Plot Control for more information.

2.7.10.2 Options

You can toggle some options when using the **Integrate** tool. The following options appear in a line under the scrolling list.

- **Draw area** fills the area under/between the curve(s) with a hatched pattern when turned on. If turned off, no area is drawn, but it is still calculated.
- **Absolute value** uses positive areas only, taking the absolute value of all Y axis values. If turned off, areas below the Y=0 line have a negative area.
- **Use log of Y** calculates the area using log values of Y, rather than the true linear value. This option is independent of the method used to draw the Y-axis. In other words, you can draw a curve on a log Y-axis scale but calculate the area on a linear scale.
- **Convert X um to cm** converts from microns to centimeters. X-axis quantities are sometimes plotted in microns (e.g., cross section plots). Y-axis quantities are often given

in terms of cm or cm^3. Use this option to calculate the area with the X-axis values converted from microns to cm.

- **Results** continually displays the current area and interval on the right.
- **Line control** positions the two lines that specify the interval used for area calculation. These lines can be moved with the buttons marked with left and right arrows. The lines can be placed at any of the features that are shown in the list. You can move a line directly to any feature by choosing either **Move RED line to selected** or **Move BLUE line to selected** from the list. This causes the appropriate line to move to the feature currently selected in the list.
- **Add Label** creates a label in the plot with the integral information. Pressing the button multiple times updates the label with the latest **Real** and **Interval** values.

2.7.10.3 Using the Mouse and Pointer

The mouse pointer can be used to move either of the marker lines, and to add new features at any point along the X-axis. To move a line, click anywhere near one of the lines and drag the mouse. The line moves to the feature nearest to the pointer position. This method allows you to “pick up” a line, move it to a new position, and put it back down.

You can also create new features. To do this, hold down the Shift key and repeat the procedure above. This time, the marker line can be moved to any X location. When you release the mouse button, add a new feature at the current line position and move the line to it.

2.7.11 Poisson Solver

The Poisson Solver (Figure 2-18) performs an electrical simulation with the 1D structure, and calculates profiles for a set of electrical quantities.

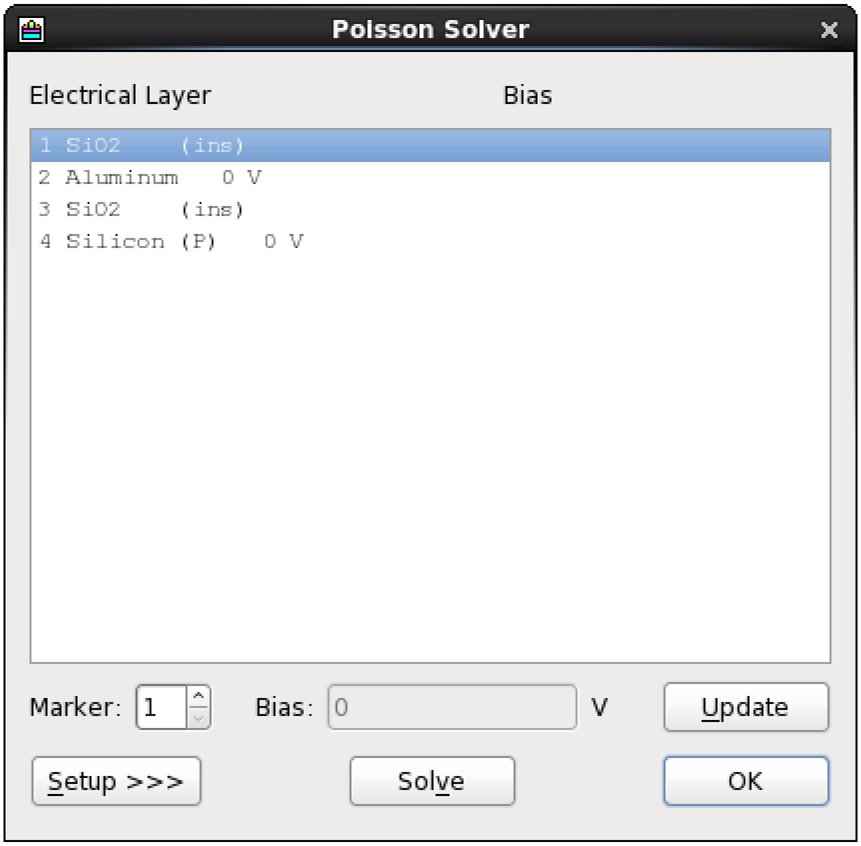


Figure 2-18 Poisson Solver Dialog

The following controls are available:

- **Layers and bias**: The **Poisson Solver** dialog shows a scrolling list of all electrical layers within a structure. The layers are areas of the same material. Silicon areas are divided up into n-type and p-type silicon. Along with each layer is shown a bias (in Volts), which is applied to that layer, when the solver is used.
- **Marker**: The **Marker** control can be used to move the marker arrow from one layer to the next. The marker arrow is used to select layers for applying an external bias.
- **Setup...**: Click on this button to open the **Poisson Solver** options panel. These options control the solver.
- **Solve**: Click on this button to perform the simulation with the current options and biases.

When the solution is complete, a plot of potential is displayed. Other solutions can be plotted by using the **Plot display** dialog. The following list of quantities solved is:

- Electron QFL
- Hole QFL
- Electron density
- Hole density
- Intrinsic conc. (nio)

- Potential
- Electron (e-) Mobility
- Hole (h+) Mobility
- Electric Field
- Electrical Conductivity

The **Poisson Solver** tool provides a built-in 1D electrical solver, which can be used to perform basic simulations of 1D structures. When it is used, the display of the first selected 1D plot shows all electrical “layers” in the structure and one profile (usually “net doping”). An arrow is drawn at the first layer.

2.7.11.1 Applying A Bias To A Layer

To apply an external bias to any layer, move the marker to any layer that is not an insulator. This can be done either with the left and right buttons on the dialog, or by using the mouse pointer to “drag” the arrow into a layer.

The current layer is selected in the scrolling list on the dialog. The bias can then be specified by typing the value into the field marked **Bias** on the dialog. Press **Return** to update the list. For p-type silicon, the bias is converted to a negative value automatically, and to a positive value for n-type. When you set all the desired biases, initiate the solver by clicking on the **Solve** button.

2.7.11.2 Setup Panel

To access the Poisson Solver Setup panel, click on the **Setup...** button on the **Poisson Solver** dialog. The following controls are available:

- **Display Solved Quantities**: This list shows all the quantities that the solver calculates. Only the ones selected, however, are displayed when the solution in complete. All other quantities can be accessed later from the **Plot Display** dialog. If other quantities are to be displayed automatically, choose them here. More than one can be selected. This does not affect which quantities are calculated, only the ones that are displayed by default.
- **Temperature**: This specifies the temperature to be used for the simulation or use the automatic default.
- **FE Mobility**: This activates the field effect mobility option for the simulation.
- **Work function**: Enter a specific work function with this option or use the default value.
- **SOI Device**: This simulates a device with Silicon On Insulator (SOI) structure layers.

2.7.12 Tracers

Tracers are used to illustrate the path of vector fields within 2D Mesh structures. They are drawn as small markers, which can be positioned anywhere inside a vector field, and are then animated by TonyPlot to show field strength and direction. The **Tracers** dialog (Figure 2-19) is used to control the positioning and animation of the markers.

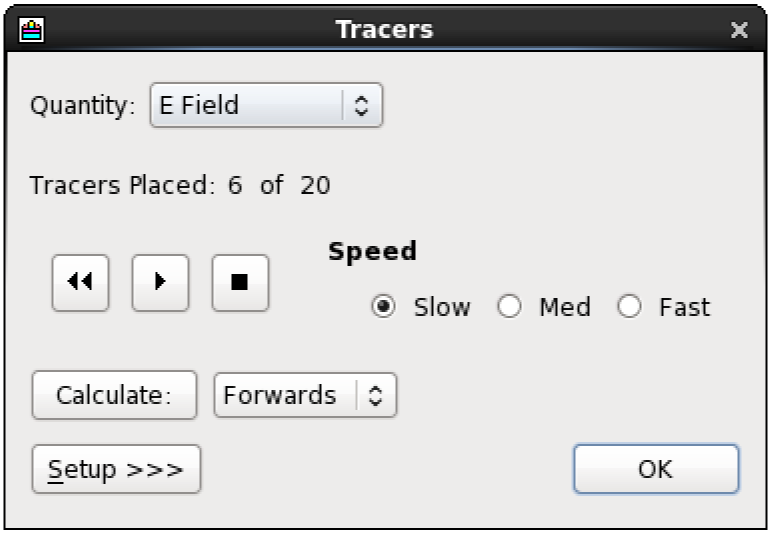


Figure 2-19 Tracers Dialog

- **Quantity**: This selects the vector quantity that the tracers should follow. These are the same quantities that are shown on the Vectors dialog (accessed from the Plot Display dialog), but the vectors do not have to be displayed for the tracers to work.
- **Tracers placed**: This indicates the number of tracers that have been placed on the plot, and the number of tracers available.
- **Animation control**: Three video-like controls are provided to control the animation of the markers. These controls do the following: return all markers to their starting points, starts the markers, and stops them at their current positions.
- **Speed**: There are three speeds available for the animation: **Slow**, **Medium**, and **Fast**. To change the speed while the markers are moving, press **Speed** again.
- **Calculate**: When you have placed all the markers you wish to animate, click on the **Calculate** button. TonyPlot then traces out the path of each marker. Progress is reported in the lower left corner of the main TonyPlot frame. When all the paths have been calculated, the markers can be animated. Markers can be placed anywhere within a vector field by clicking at the position where you want a marker to start. The counter on the dialog indicates how many have been placed. To remove a marker, press the Shift key and click near to the marker to remove. The one nearest the pointer is erased, and if its path has been drawn that too is erased.
- **Setup...**: Press this button to see the small panel of setup options available in the **Tracers** tool. See the following section.

2.7.12.1 Setup

Clicking on the **Setup...** button to reveal the options panel on the **Tracer** tool dialog

- **Color**: All markers placed use the currently selected color. Different markers can have different colors by changing the color for each marker placed. Tracer path lines are drawn in the same color as the marker that follows it.
- **Max. jump**: This value controls the “granularity” of the path calculation. Higher numbers reduce the calculation time but give only approximate paths with long jumps. Smaller numbers produce more accurate paths but take longer to calculate.
- **Lines**: When turned on, lines are drawn along the tracer paths as the paths are calculated. If turned off, the path is not shown but the tracers still follow the same route.
- **Cycle**: Three cycle modes are available which control the action of markers when they reach the ends of their paths. The first choice stops all tracers as soon as one tracer reached the end. The second choice stops each marker as it reaches the end of its own path. The third choice makes each tracer move in a loop, returning to its start point each time it reaches the end.

2.8 Functions

Functions allow you to further customize the output that can be produced and to extend the amount of data that can be plotted without needing further simulation runs and large data files. TonyPlot allows you to create functions for use with XY Graph or 2D Mesh plots. It also uses an advanced mathematical parser to calculate function results from math expressions.

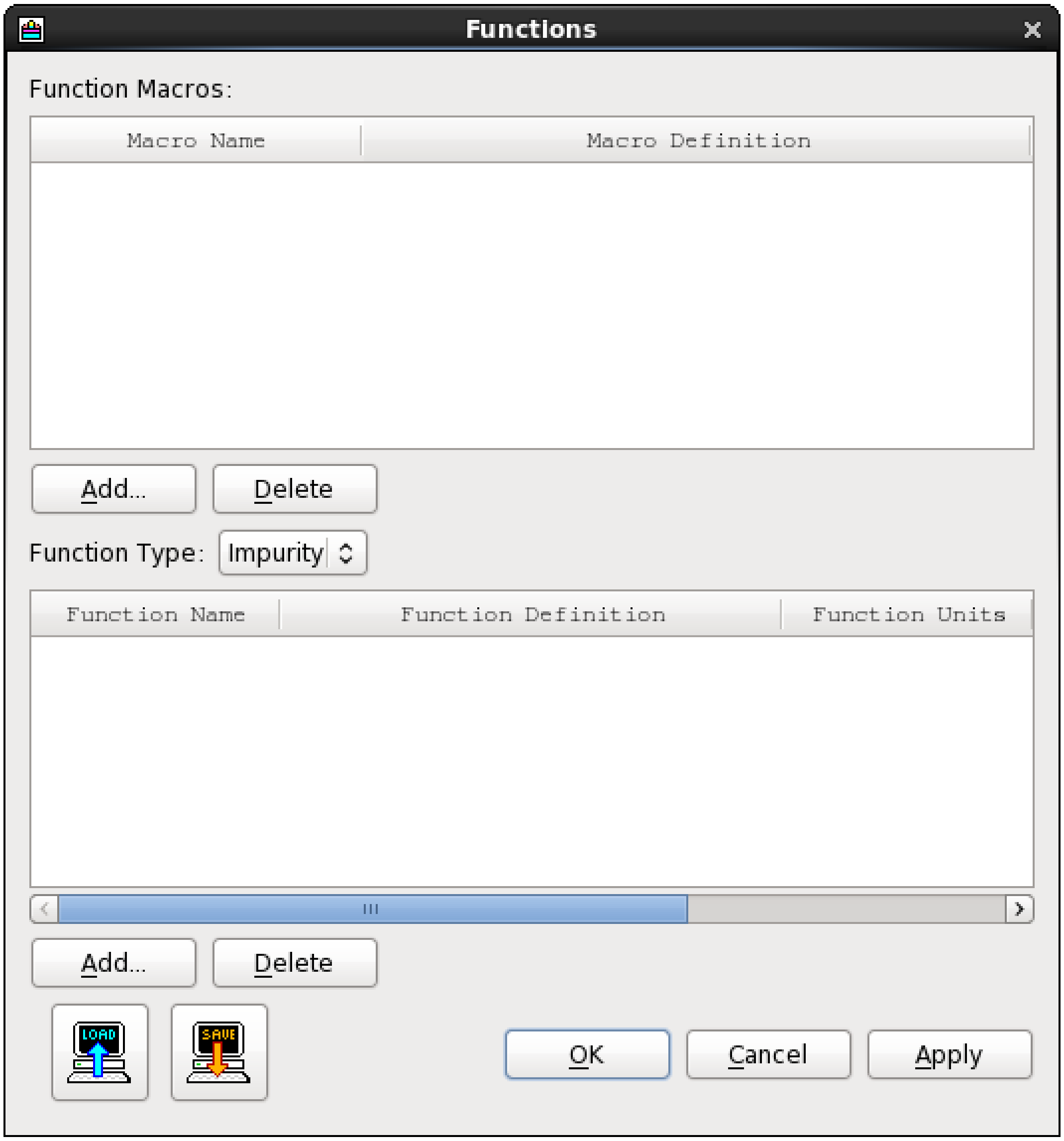


Figure 2-20 Functions Dialog

2.8.1 Use Of Functions

To use functions, first define a function in terms of quantity names (e.g. Boron, Drain bias, and Temperature), constants, and operators. Then, plot the function in the desired plot, by selecting the function name in the quantities list in either the XY Graph or Contours dialog. Using scientific notation in functions (e.g. le23) requires the mantissa to have a decimal point (e.g. 1.0e23), otherwise it does not work.

The Functions dialog can be displayed from the main Edit menu. It can also be displayed from the buttons marked **Functions...** that appears in the XY Graph Display dialog, and the Contours dialog.

2.8.2 Defining Functions

The Functions dialog is split into two sections. The top section allows function macros to be set up. The lower section is where the functions are defined. You can define two types of functions.

- Graph functions: These are used on XY Graph plots only and are defined in terms of graph (electrical) quantities.
- Impurity functions: These functions can be used in Mesh plots and Cross Section plots only and are defined in terms of impurities.

It is possible to nest functions by including the name of one function as a variable in another.

2.8.3 Plotting

When choosing a function to plot, TonyPlot evaluates the results of the function at each data point and stores these values in the data attached to each plot level. Then, the function can be drawn along with any other quantity also selected.

2.8.3.1 Example

Suppose a Master file contains values for the four dopant impurities boron, arsenic, phosphorus and antimony. We can use a function to compute the net doping by entering the following as Function 1.

```
boron – arsenic – phosphorus - antimony
```

Most plots of net doping, however, are shown on log scales (this is the default for TonyPlot) so you need to calculate the log of this sum. Make sure the total is positive beforehand, so use `abs`() to get the absolute value:

```
log10(abs(boron – arsenic – phosphorus - antimony))
```

The data for each dopant, however, is not useful below values of around 1e12. TonyPlot usually does not show values below this level. But with a function, it cannot tell that this is needed.

Therefore, use the `max`() operator to keep the function result within a useful range:

```
max(log10(abs(boron – arsenic – phosphorus - antimony)), 12)
```

Simplify this expression by splitting into two functions and nesting one inside the other, as follows:

```
Function 1 = boron - arsenic - phosphorous - antimony

Function 2 = max(log10(abs(Function 1)), 12)
```

Now plot Function 2 on a Mesh or Cross Section plot. It will show the profile of Net Doping (clipped at 1e12).

2.8.4 Function Macros

To allow further simplification of functions, the macro section is provided on the Functions dialog. It can be used to store common functions and identify them with an easy to remember name. You can use the following options to modify macros.

- Function Macros: This shows all the macro names currently known to TonyPlot.
- Macro Name: Shows the name of the macro.
- Macro Definition: Shows the definition of the macro.
- Add: To create a new macro, click this button, then enter a new name and definition for the macro. The new name appears in the list. If the name already exists, the old definition is replaced.
- Delete: This deletes the macro that is currently selected in the name list.

The macro names can then be used in any function as though the whole definition had been typed.

For example, in the Net doping example, we could add a macro called net_dop and in the definition window, enter:

```
max(log10(abs(boron - arsenic - phosphorus - antimony)), 12)
```

Then, we could define an Impurity Function to simply be net_dop.

This makes the function definitions easier to read and allows useful names to be added to the plot legend.

All macros can be saved to a defaults file for use any time TonyPlot is used. Two buttons on the dialog allow defaults to be saved and loaded at any time.

2.8.5 Function Syntax

Functions are constructed just like normal math expressions, but with names of quantities from data files used as variable names. Functions can be built with the following operators:

Table 2-9 Function Operators

`a + b`	`a` plus `b`
`a - b`	`a` minus `b`
`a / b`	`a` divided by `b`
`a * b`	`a` multiplied by `b`
`a ^ b`	`a` to the power of `b`
`-a`	negative `a`
`abs(a)`	absolute value of `a`
`log(a)`	natural log (base e) of `a`
`exp(a)`	inverse natural log of `a` (e ^ `a`)
`log10(a)`	log base 10 of `a`
`sqrt(a)`	square root of `a`
`sin(a)`	sine of `a` (`a` in radians)
`cos(a)`	cosine of `a` (`a` in radians)
`tan(a)`	tangent of `a` (`a` in radians)
`asin(a)`	arcsine of `a`
`acos(a)`	arccosine of `a`
`atan(a)`	arctangent of `a`
`sinh(a)`	hyperbolic sine of `a`
`cosh(a)`	hyperbolic cosine of `a`
`mag(a, b)`	magnitude of vector [`a`, `b`]
`hypot(a, b)`	hypotenuse of `a` and `b` (same as `mag(a, b)`)
`max(a, b)`	maximum of `a` and `b`

`min(a, b)`	minimum of `a` and `b`
`dydx(a, b)`	derivative of `a` with respect to `b`

Normal operator precedence is obeyed.

Expressions can use parentheses to change the operator precedence where needed.

If an expression contains an error, this is reported when the function is plotted. Invalid values are for the most part plotted as zero, except for `log()`, which uses the value predefined in the property called Log Zero. If a function does contain an error, TonyPlot displays a notice box informing you of the type of error encountered. A syntax error causes all function values to be zero. An evaluation error causes just the offending data points to be zero.

The derivative function `dydx` can take any two variables. It also accepts distance to represent the x value. For example, a vertical cutline `dydx(boron, distance)` would give the derivative of boron concentration against depth.

3. 2D Mesh Plot Display

3.1 Overview

If the **View→Display…** menu action is selected while a 2D Mesh plot is selected, the 2D Mesh Display dialog is shown (Figure 3-1). It shows the current display settings for the first selected 2D Mesh plot. When the settings on the display dialog are applied, all selected 2D Mesh plots are affected. In this way it is easy to apply global changes to similar plots in the view. The dialog shows the features that can be displayed in 2D Mesh plots.

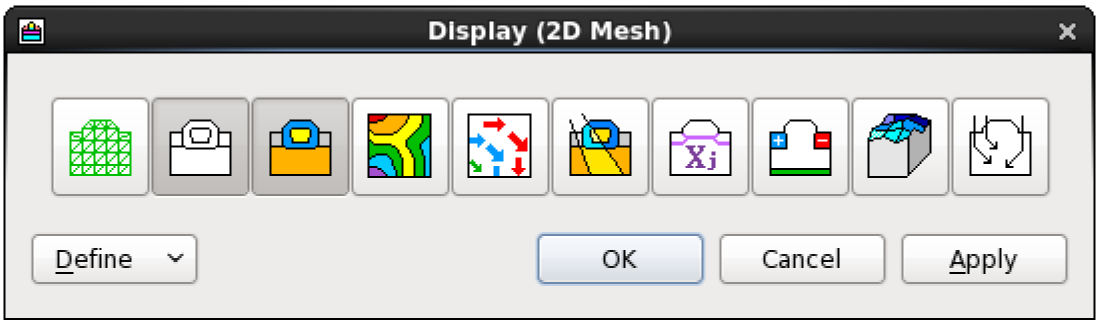
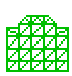
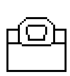
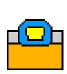


Figure 3-1 2D Mesh Display Dialog

The icons shown on the 2D Mesh Display dialog are described in Table 3-1.

Table 3-1 2D Mesh Display Modes

 Mesh	The triangular mesh used in the simulation.
 Edges	Sides of triangles classified as region edges.
 Regions	All material regions in the structure.

Contours	Color plotting of impurity values.
Vectors	Representation of vectored impurities.
Light	Light beam and ray information.
Junctions	Metallurgical junctions in the semiconductor regions.
Electrodes	Regions defined as being electrodes.
3D	Adds elevation to a plot so that 3D surface is plotted.
Lines	Adds lines dates to plots for ionization integrals or Monte Carlo ion implant.

Some of these features have further control dialogs, which can be accessed from the **Define** button. The features that have detailed control are: **Regions**, **Contours**, **Vectors**, **Light**, **Junctions**, **3D**, and **Lines**.

3.2 Regions

There are several ways that TonyPlot can display mesh regions. The **Regions** dialog (Figure 3-2), is accessed by pressing the Regions icon, and then selecting **Define→Regions**.

Figure 3-2 Regions Dialog

The first choice controls the way in which regions are drawn. The following options are:

- **Solid** fills the region area with color.
- **Lines** draws the region outlines only.
- **Lines/Points** draws the region outlines and marks the points defining the region border.

The second choice determines the parameter used when determining the regions color. The first two options are always available: **Material** and **Region**. If you choose **Material**, the color represents the material of the region (e.g., silicon or oxide) and the region legend shows the material names. If you choose **Region**, then each region has its own unique color and the legend shows the name of each region. If the data contains further region information (e.g., workfunction and phase), then these are also available, and the legend shows the values of these parameters in each region. Any regions that are one-dimensional (e.g., substrate electrodes) are drawn as thick lines, since they do not enclose a complete area.

3.3 Contours

Contouring is the most commonly used method for visualizing data on two dimensional meshes. The contouring facilities in TonyPlot provide sufficient control for obtaining any desired plot. Both contour plots and fringe plots (filled contours) are available with material naming and range control to limit the plot to a restricted subset of the data. Each plot can have up to three sets of contours displayed at once. This makes it possible to view more than one quantity simultaneously, either all filled but in different material regions, or all lines over all materials or any other combination. Of course, if all three sets are filled sets and all are plotted over the same materials, only the third set (the last one to be drawn) will be visible. If lines and filled sets are combined, the filled set should come before the line set.

TonyPlot selects a default quantity whenever possible. This allows contours to be plotted without the need to use the **Contours** dialog (Figure 3-3).

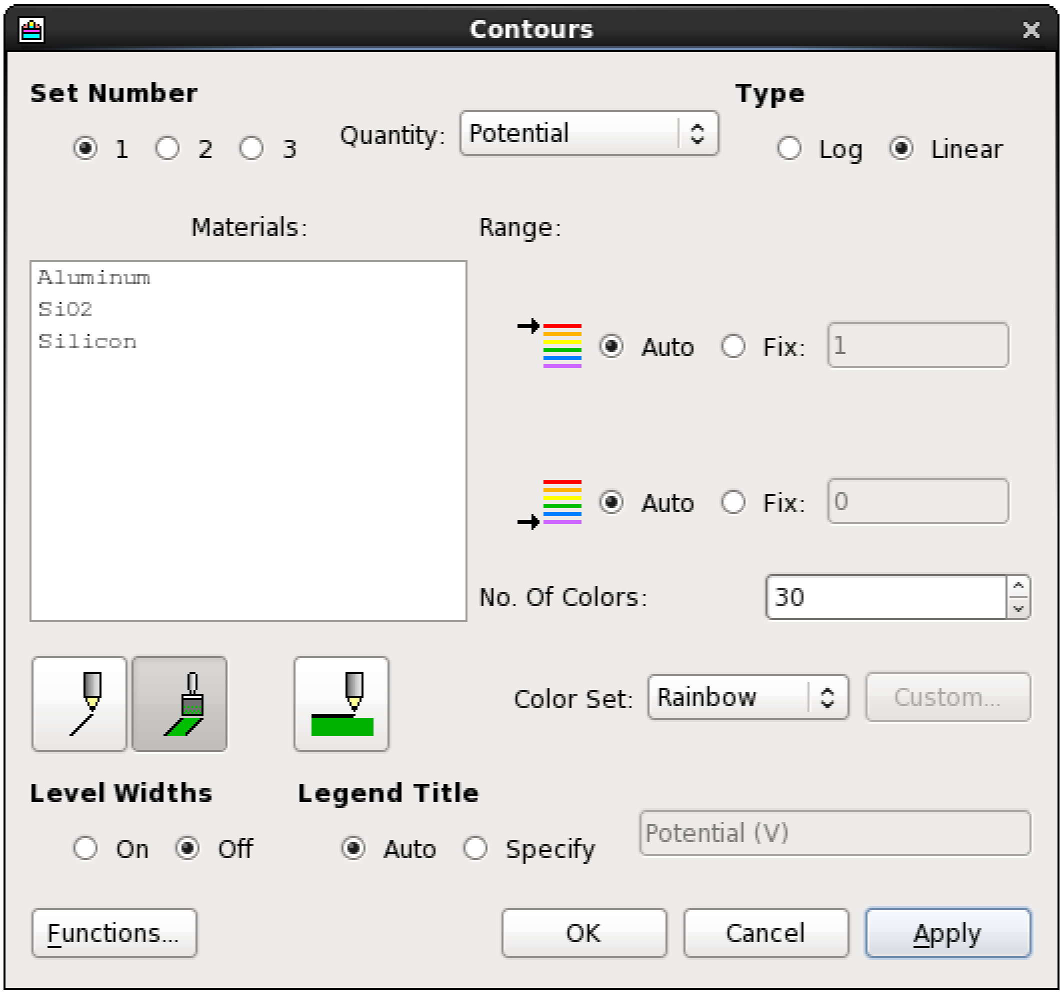


Figure 3-3 Contours Dialog

Just select contours from the **Mesh Display** dialog, and plot them by clicking on the **Apply** button. The **Contours** dialog appears if you select contours on the **2D Mesh** dialog and then select **Define→Contours**. The dialog is divided into subsections, as follows:

- **Set number**: This subsection shows which set is currently being edited. Set 2 will only be plotted if set 1 is plotted, and set 3 will only be plotted if set 2 is plotted. To plot a set, the quantity (see below) must be anything other than **None**.
- **Quantity**: Choose the quantity to be contoured. If the range items (see below) are set to **Auto**, the corresponding minimum and maximum text fields are updated to show the range of the new quantity. If **None** is selected, the current contour set will not be plotted. One of two functions may also be chosen. Functions are defined from the **Functions** dialog (see Section 2.8 Functions).
- **Materials**: The part of the structure on which contours are drawn can be limited to regions of a certain material. If no materials are selected in the list, this is treated as all being selected (the default). If you do not plot the contours on any material, set the **Quantity** to **None**.

- **Range**: The group of items on the right part of the dialog control the range of the data through which contours are plotted. The maximum (top item), minimum (middle item), and interval or number of steps (bottom item) can be set or left to automatic defaults. The default minimum and maximum values are the minimum and maximum values of the data in the structure(s). The default number of steps is the same as the number of colors in the current color set.
- **Drawing style**: Selecting the pencil creates line contours, while the paintbrush creates filled contours (fringe plot). If you plot filled contours, you can add optional outlines by selecting the **Outline** icon, which is to the right of the paintbrush. TonyPlot provides several color sets which can be used for contour plotting. If the contour range is determined by the number of steps (Num: selected), the number updates to match the number of colors available in the color set when one is selected.
- **Level widths**: If you set the drawing style to line contour, this option forces overlaid plots to use a different line width for each level.
- **Legend title**: The legend title can either be set automatically by TonyPlot (set to **Auto**), or manually entered to create a custom title (set to **Specify**). An automatic title consists of the name of the quantity plotted, with units if appropriate. A custom title is created by entering the desired text into the text field supplied. This title is used on the contour legend for this contour set.
- **Functions**: Click on this button to display the **Functions** dialog. This dialog can be used to define the functions of the original quantities that can be selected from the choice of **Quantities**.

3.4 Vectors

Vectors can be plotted for standard (the default) or user-defined vector quantities. TonyPlot automatically detects the standard quantities made of an X component and a Y component. They are shown in the **Quantity** pulldown menu. To create a vector made of unrelated X and Y quantities, select the **Custom** option.

Vectors are represented on the plot by arrows. The direction of the arrow shows orientation of the vector. The color or length of the arrow or both shows the magnitude of the vector. The **Vectors** dialog (Figure 3-4) will appear if you choose **Vectors** on the Mesh 2D display dialog.

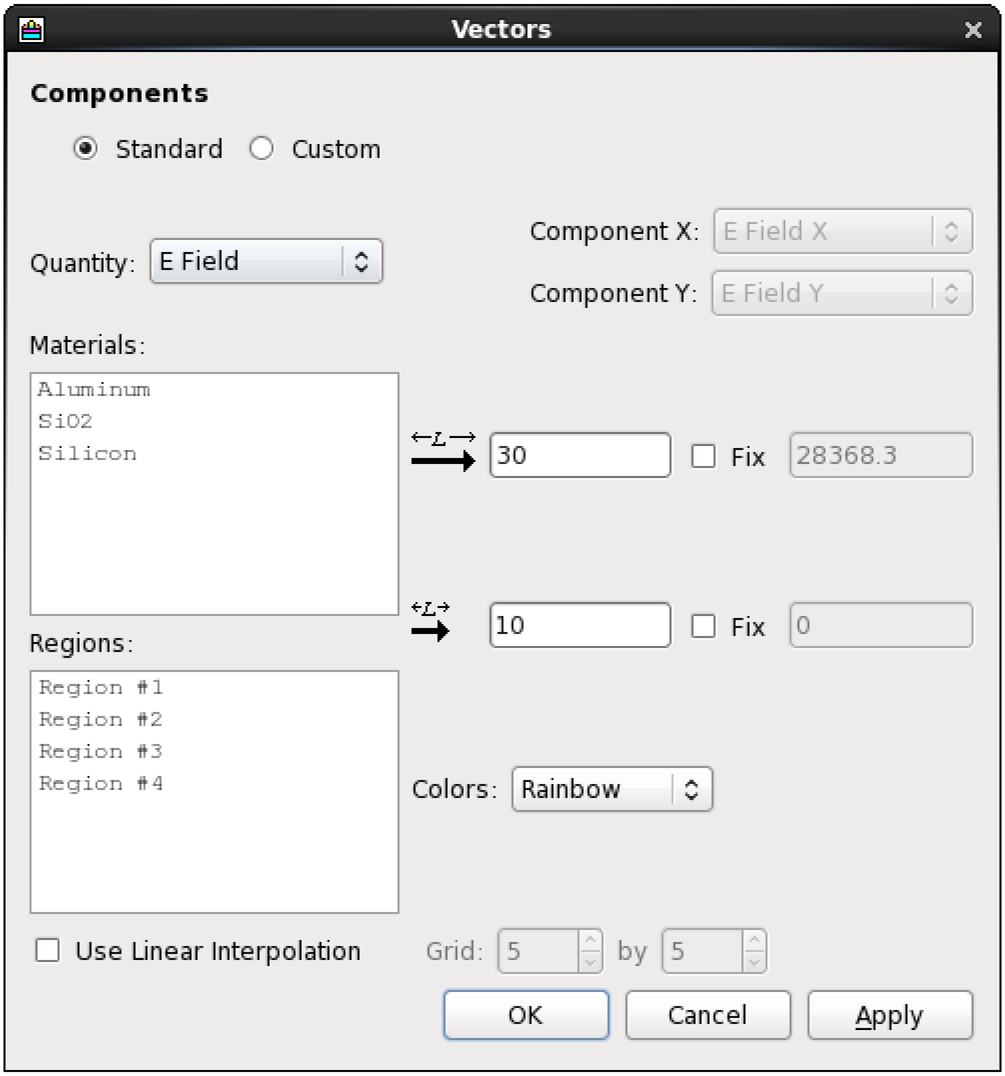


Figure 3-4 Vectors Dialog

The items on the dialog are as follows:

- **Components**: This option controls whether to draw standard or user-defined vector quantities.
- **Quantity**: All quantities of the structure that are standard vector quantities are shown in this pulldown menu. This menu is only active when the **Components** option is set to **Standard**. The list of quantities includes the two functions defined in the Functions dialog. See Section 2.8 Functions for more information.
- **Component X**: This menu is active only when the **Components** option is set to **Custom**. It is used to assign a quantity to the X component of the user-defined vector.
- **Component Y**: This menu is active only when the **Components** option is set to **Custom**. It is used to assign a quantity to the Y component of the user-defined vector.
- **Materials**: The part of the structure on which vectors are drawn can be limited to regions of a certain material. If no materials are selected in the list, then it is treated as all being

selected (the default). If you do not plot the vectors in any materials, set the **Quantity** or **Component X/Y** to **None**.

- **Range**: The range items on the right-hand side of the dialog control the sizes of the vector arrows drawn. The longest arrow matches the vector with the greatest magnitude, and the shortest arrow matches the smallest magnitude. If the smaller length is specified as zero, then the lengths of the draw arrow are directly proportional to the vector magnitude.
- **Regions**: The part of the structure on which vectors are drawn can be limited to the specified region. If no regions are selected in the list, then it is treated as all being selected (the default).
- **Colors**: This specifies the color sets for the arrows. These are the same color sets that are used on the Contours dialog.
- **Use Linear Interpolation**: If you check this option, you can interpolate vectors onto a cartesian grid. This can be useful if the vectors are particularly dense.
- **Functions**: This displays the Functions dialog used to define the two functions you can select in the **Quantity** pulldown menu.

3.5 Light Rays

If a structure contains light ray information, the **Light** dialog can be used to display that data in a number of ways. Light information consists of a number of **Beams**, and each beam is comprised of a series of **Rays**. A ray is a section of a beam between reflections and refractions. For example, if a beam originates from outside a structure, enters the structure (is refracted), travels to the bottom of the structure (gets reflected), and then moves back to the top of the structure, it consists of three rays. The **Light** dialog (Figure 3-5) appears if you choose **Light** on the 2D Mesh plot dialog.

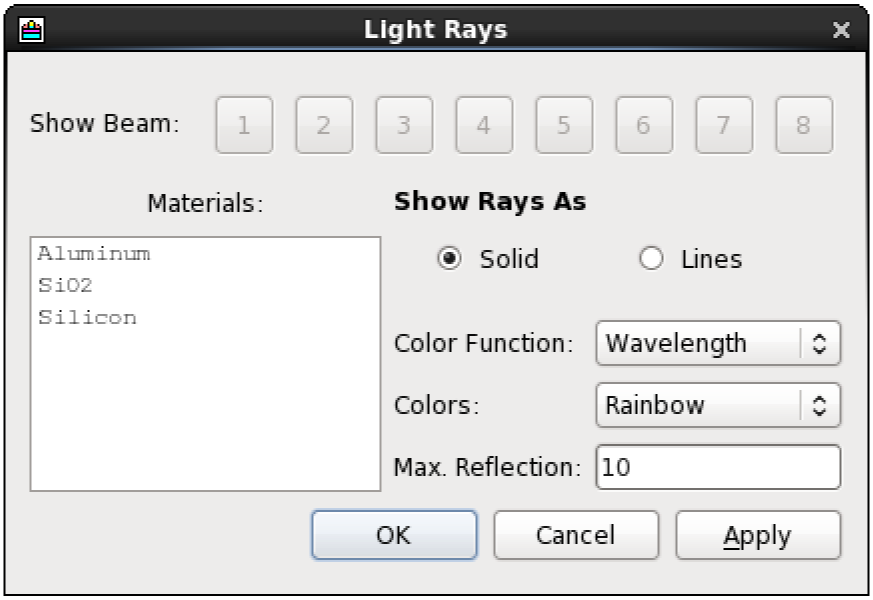


Figure 3-5 Light Rays Dialog

The items in the dialog are:

- **Beam**: When beams are present in the structure, they are shown as selectable numbers on this item. Any of the beams can be shown at once by selected the required beam numbers. If no light information is present in the structure, the option **None** is shown to show that there are no beams.
- **Materials**: The part of the structure on which light beams are drawn can be limited to regions of a certain material. If no materials are selected in the list, this is treated as all being selected (the default). If the beams should not be plotted on any material, deselect all beam numbers.
- **Show as**: There are two ways to show light beams. One way as lines that show the path of the beam. Another way solid areas that also show the width of the beam. Choose the required option with this item.
- **Color function**: The light beam rays can be colored in a variety of ways. **Wavelength** colors the rays to match the wavelength. Wavelengths in the ultra-violet region and shorter are shown as dark purple. Wavelength in the infra-red region and longer are shown as dark red. **Power** assigns a color from the chosen color set dependent on the beam intensity. **Beam number** assigns one color to each beam. **Reflection Index** assigns the color of a ray according to the number of times it has been reflected.
- **Colors**: This allows you to choose a color set. These are the same as those available on the **Contour** and **Vector** dialogs.
- **Maximum reflection**: This can be used to limit the number of rays drawn. Only rays that have been reflected this many times or less are drawn.

3.6 Junction

You can plot depletion region edges from device simulations (Figure 3-6) and metallurgical junctions. Depletion factor is the value of the ratio of majority carriers to doping used to determine the depletion region edge.

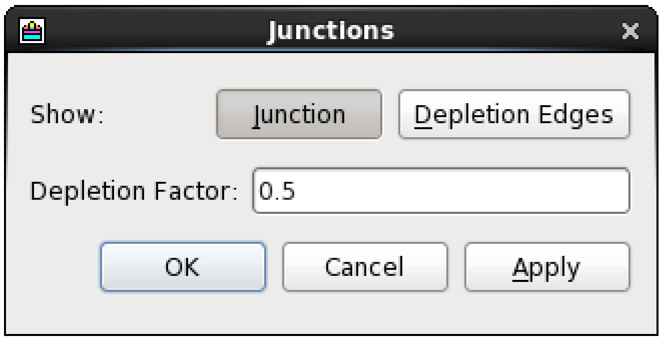


Figure 3-6 Junctions Dialog

3.7 3D

A 2D structure that can be contoured can also be elevated by choosing the **3D** option and defining some parameters in this dialog.

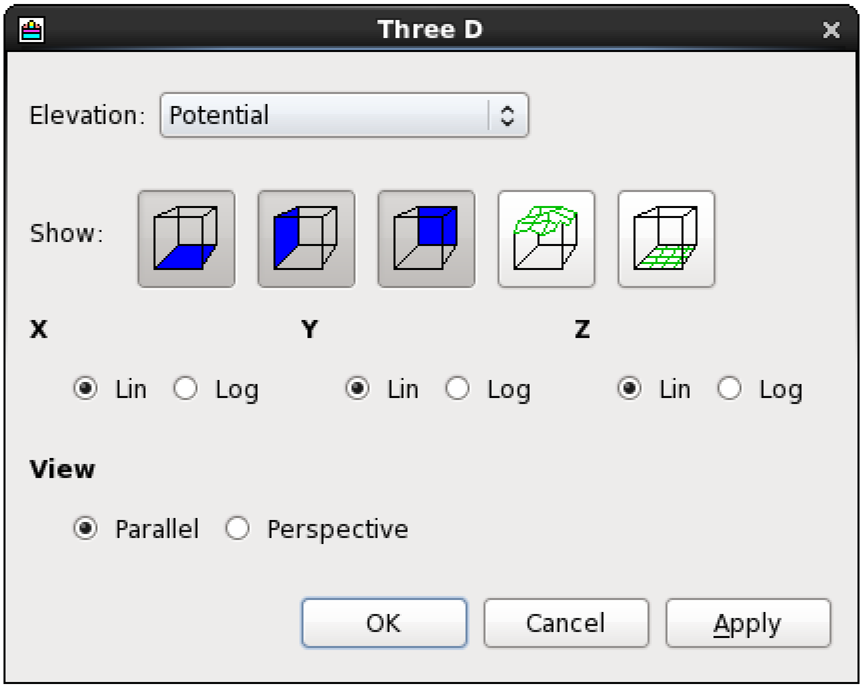


Figure 3-7 3D Dialog

- **Elevation**: Any quantity can be chosen as the elevation. The height of the surface at any point is proportional to the value of the elevation quantity.
- **Show**: There are number of optional items that can be drawn on a 3D plot, and each is described by a small icon on the **Show** item. Choose the ones desired from this list.
- **View**: The view projection can be one of two choices: **Parallel** or **Perspective**.
- **Log**: You can select whether to log each of the axes individually.

3.8 Lines

You can overlay lines onto a 2D plot using the Lines dialog (Figure 3-8). This is used to show static field lines on ion implant tracks.

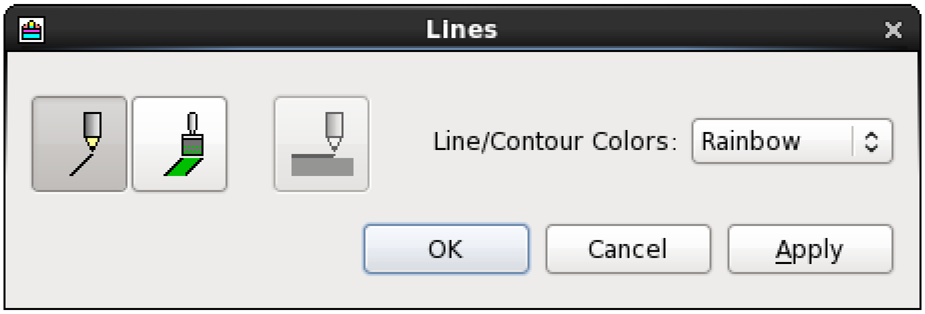
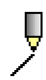
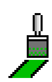
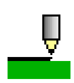


Figure 3-8 Lines Dialog

The icon buttons control the drawing of the Lines and pseudo Contours. The icon buttons are described in Table 3-2.

Table 3-2 Lines Dialog Icon Buttons

 Lines	Draw lines using the range of colors selected in the **Line/Contour Colors** control.
 Contours	Draw pseudo contours between the lines.
 Lines and Contours	Draw lines and contours.

3.9 Transforms

The Transform 2D Mesh dialog (Figure 3-9) allows mathematical operations to be performed between multiple selected 2D Mesh quantities. For example, you can add the voltages from two different structures.

Note: The meshes must be identical to perform an operation.

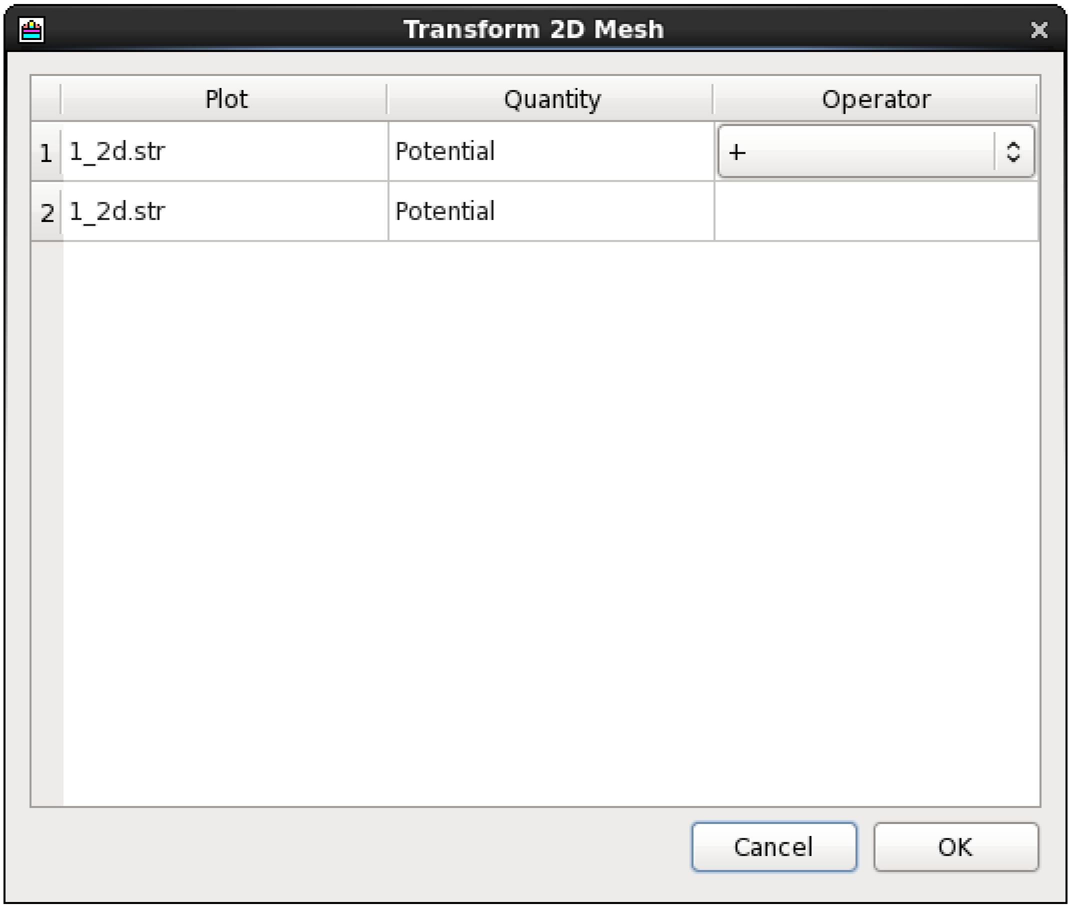


Figure 3-9 Transform 2D Mesh Dialog

4. 3D Mesh Plot Display

4.1 Overview

A variety of 3D Mesh Plot features are controlled using the 3D Display dialog. This is accessed by selecting **View→Display…** when a 3D Mesh Plot is selected.

4.2 Mesh Options

The Mesh Options dialog configures the appearance of the mesh of the structure. The available mesh options are:

- **Face**: Draws the mesh for the region surfaces only.
- **Element**: Draws the mesh for all region volume elements.
- **Volume**: Draws the mesh for all region volume elements, clipping the mesh data to be contained within a defined cylindrical volume.

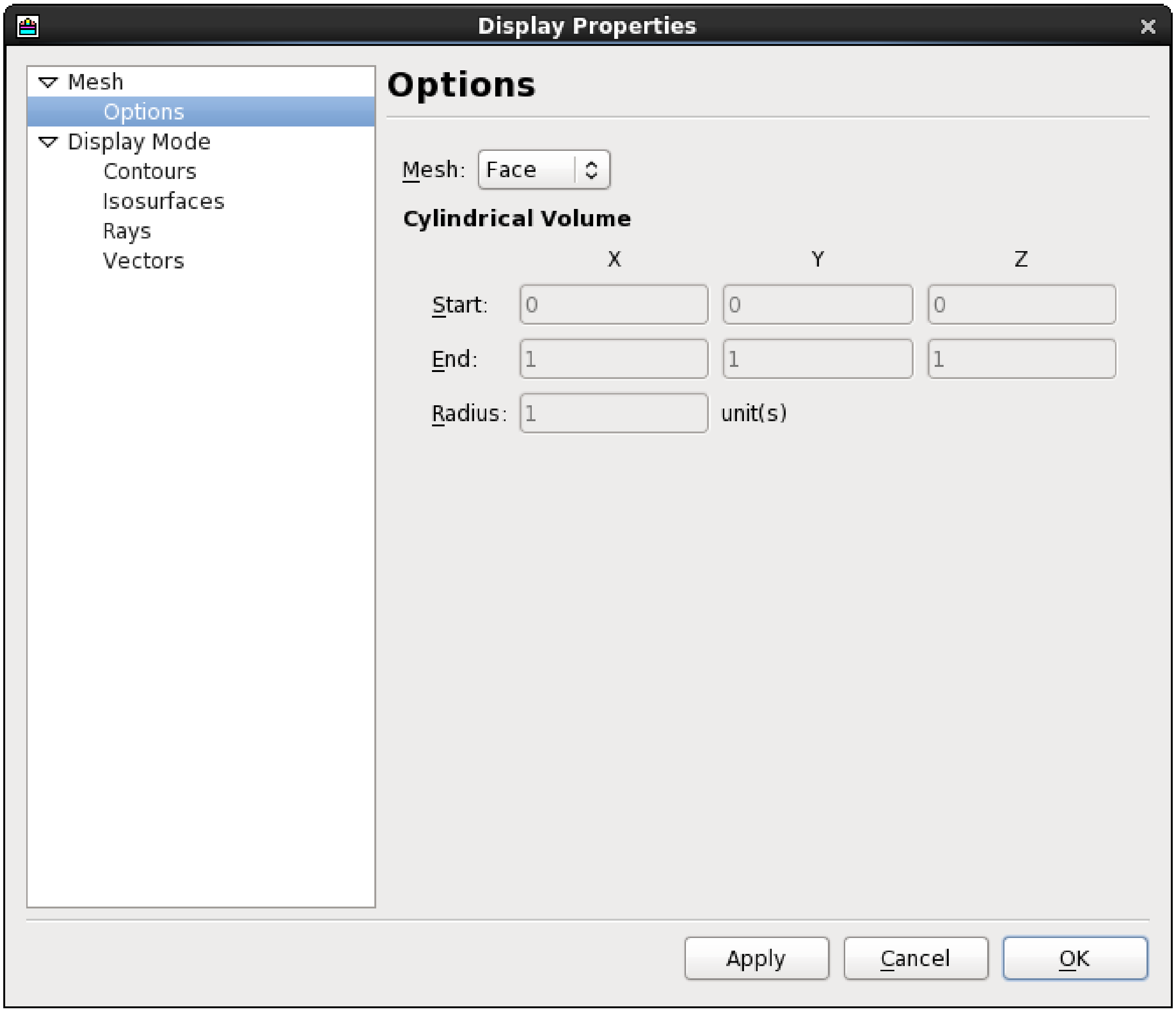


Figure 4-1 Mesh Options

4.3 Contours

The Contour Display Mode is shown in Figure 4-2. Contours are shown as a colorization of the exterior faces of the structure regions.

The **Quantity** option box holds all the quantities present in the data set. Choose one of these quantities to use for contouring.

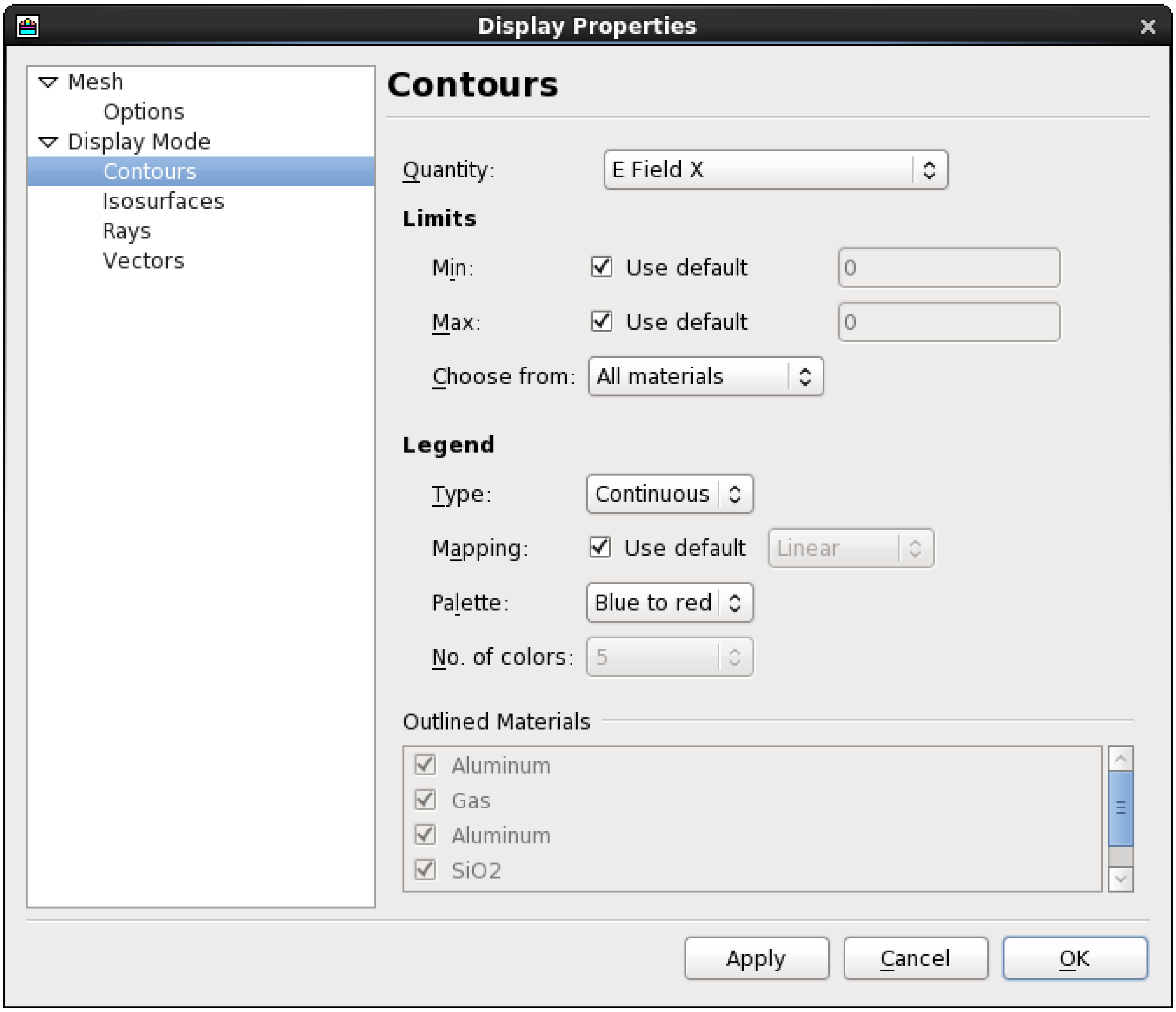


Figure 4-2 Display Mode: Contours

4.3.1 Limit Settings

The controls in the **Limits** section of the dialog set the data range used in the contours. If **Min** or **Max** is set to **Use default**, TonyPlot will extract the data range from the structure for the selected **Quantity**. You can use the **Choose From** control to change the materials used in the computation of the contours. The options are:

- **All Materials** All nodes in the structure are used to determine the data range.
- **Outline Materials** Only nodes in the checked materials in the **Outlined Materials** list are used.
- **Selected Objects** The data range is computed from the list of selected objects in the scene.

The limits can also be set manually by unchecking the **Use default** controls and specifying the

Min and **Max** values directly.

4.3.2 Legend Settings

The controls in the **Legend** section control how the contours are drawn on the region faces. When **Type** is set to **Continuous**, the contour colors will be smoothly interpolated between the reference colors of the legend. Setting **Type** to **Stepped** will produce a finite number of discrete color bands to use across the data range. The **Palette** and **No. of Colors** are also available to further change the appearance of the legend.

The quantity values can either be plotted with either a linear or logarithmic scale. When the **Mapping** is **Linear**, the values are directly mapped to the legend's colors. When the mapping is **Logarithmic**, the log10 is taken before the mapping occurs. If **Use default** is checked, TonyPlot will chose the mapping automatically based on the selected contour **Quantity**.

4.4 Isosurfaces

The Isosurface Display Mode (Figure 4-3) is used to show surfaces of constant value throughout a 3D structure. The **Quantity** controls which impurity is used in the computation. The value of an isosurface must be within the data range (Min, Max) of the selected **Quantity** and can be set using the **Value** text field, or using the slider control. To view an isosurface, you can either turn on the **Preview isosurface** or select the **Create** button. Once created, the isosurface will appear in the isosurface list. To delete one or more isosurfaces, select them in the list and then click **Delete**. You can use the **Draw mode** control to change the appearance of all the isosurfaces at once.

To obtain the junction surface for Net Doping, click the **Junction** button.

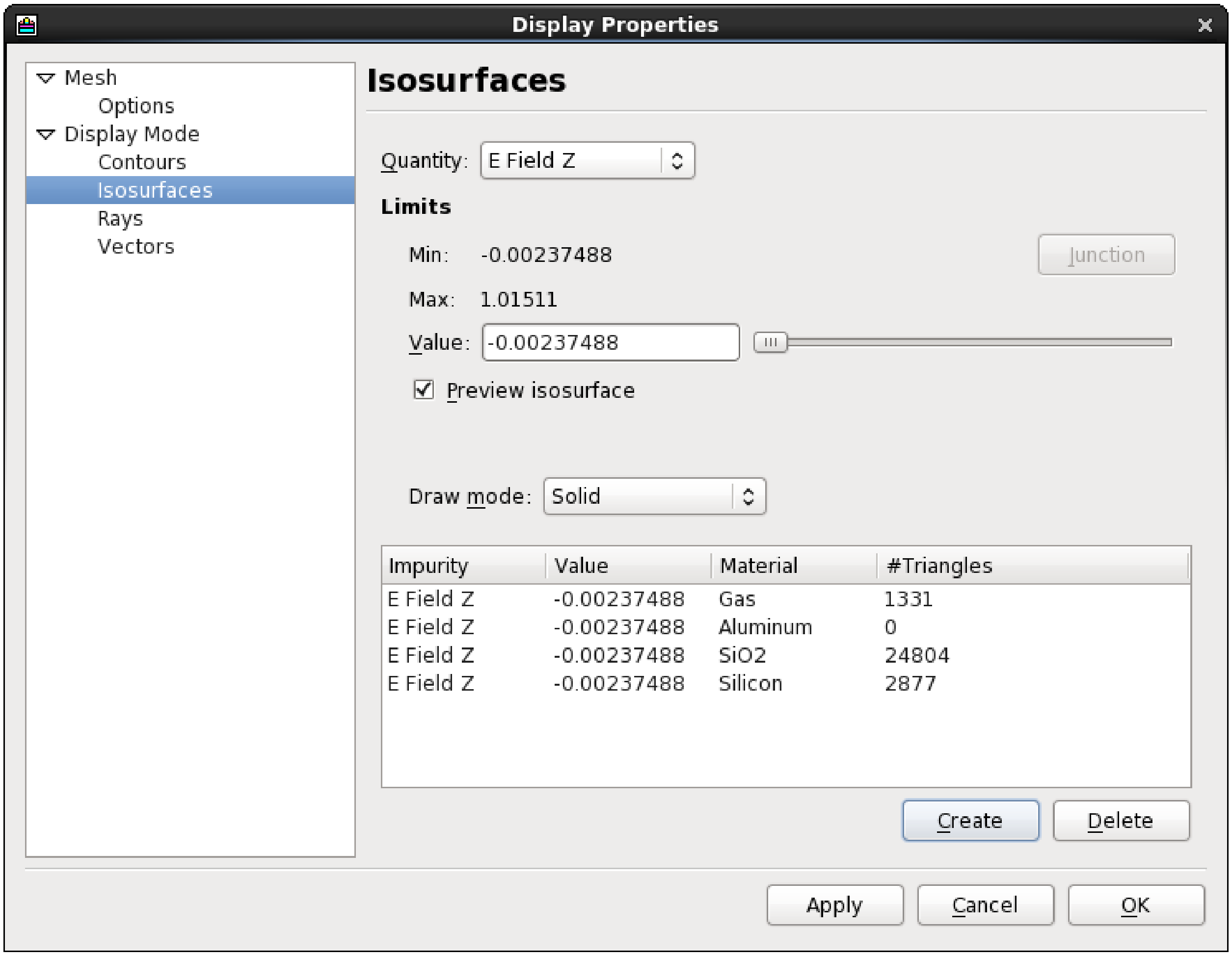


Figure 4-3 Display Mode: Isosurfaces

4.5 Rays

When a 3D Mesh structure is opened that contains ray traces, the rays display mode is automatically selected, and the rays are displayed in the plot. The Display Mode: Rays panel (Figure 4-4) shows a summary of the ray trace data in tabular format.

By default, the ray traces are colored according to the wavelength of each trace. Wavelengths in the infra-red region are displayed as a dark red color, while ultra-violet wavelengths are displayed as dark purple.

The properties of the ray trace display can be examined and edited using the Object Browser (see Section 2.7.1 Object Browser). In this way, rays of specific wavelengths can be displayed or hidden as needed. The ray display can be turned off entirely by unchecking the **View→Rays** menu item.

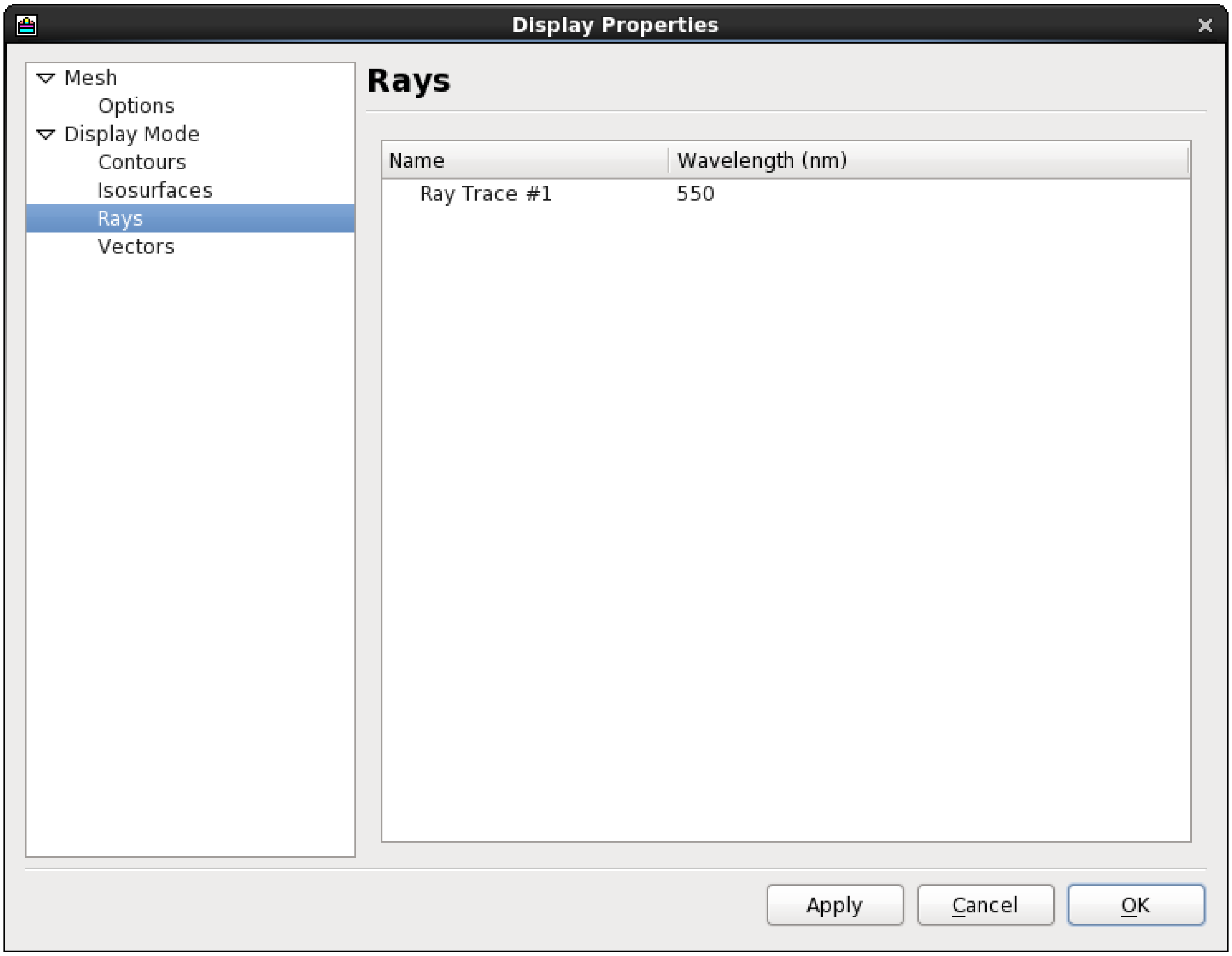


Figure 4-4 Display Mode: Rays

4.6 Vectors

When a 3D structure contains vector data, you can use **Vectors** feature to visualize their directions and magnitudes. The vectors are interpolated and sampled on a uniform axis-aligned grid. Figure 4-5 shows the **Display Mode: Vectors** panel.

To display a set of vectors, use the **Quantity** combo box to select a vector quantity. The number of samples along each axis can be set using the respective boxes for X, Y, and Z. The **Drawing Settings** controls are used to scale the relative length and radius of the displayed vectors. Press the **Create** button to create the vector set. Once created, the vector set will appear in the samples list. To delete one or more vector sets, select them in the list and then click **Delete**.

Note: Creating a vector set may take a few moments if the structure contains many vertices and/or many samples are requested.

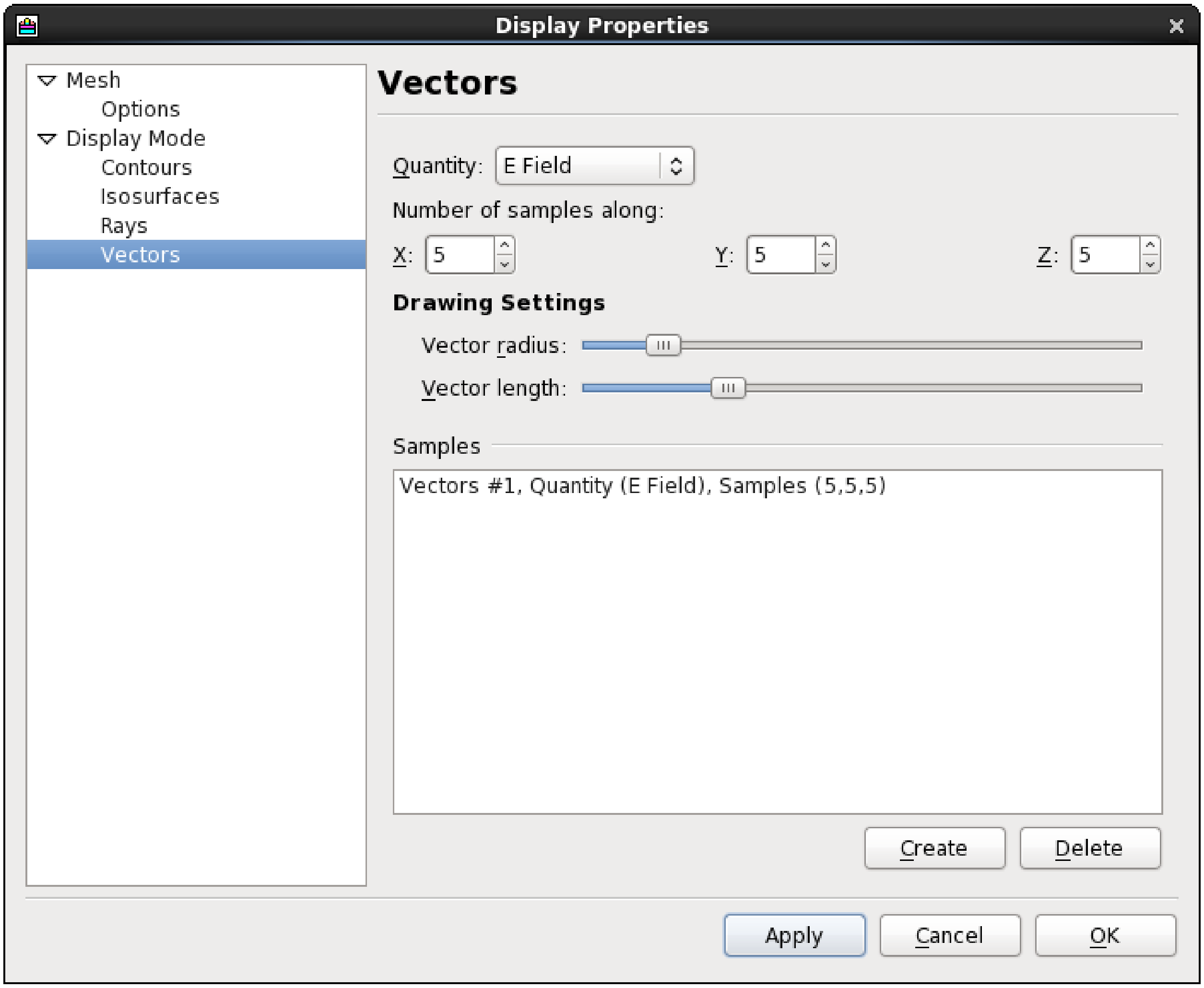


Figure 4-5 Display Mode: Vectors

4.7 Save As Surfaces

3D Meshes can be saved as surfaces-only structures by selecting **File→Save As…**. Surfaces-only structures contain mesh data only at the outer surfaces of the regions. Any data points internal to the regions are discarded. The resulting surfaces-only structure file can have a much smaller file size than the original full structure, and consequently can be much faster to load and display, while still retaining the same outer appearance. However, it is important to note that cut plane slices through a surfaces-only structure will yield only an outline.

5. XY Graph Display

5.1 Overview

The **XY Graph Plot** dialog (see Figure 5-1) shows the current display settings for the first selected graph plot. When the settings on the display dialog are applied, all selected graph plots are affected. This permits much easier application of global changes to similar plots in the view. The **XY Graph Plot** dialog contains all the controls needed for complete control of graph plots.

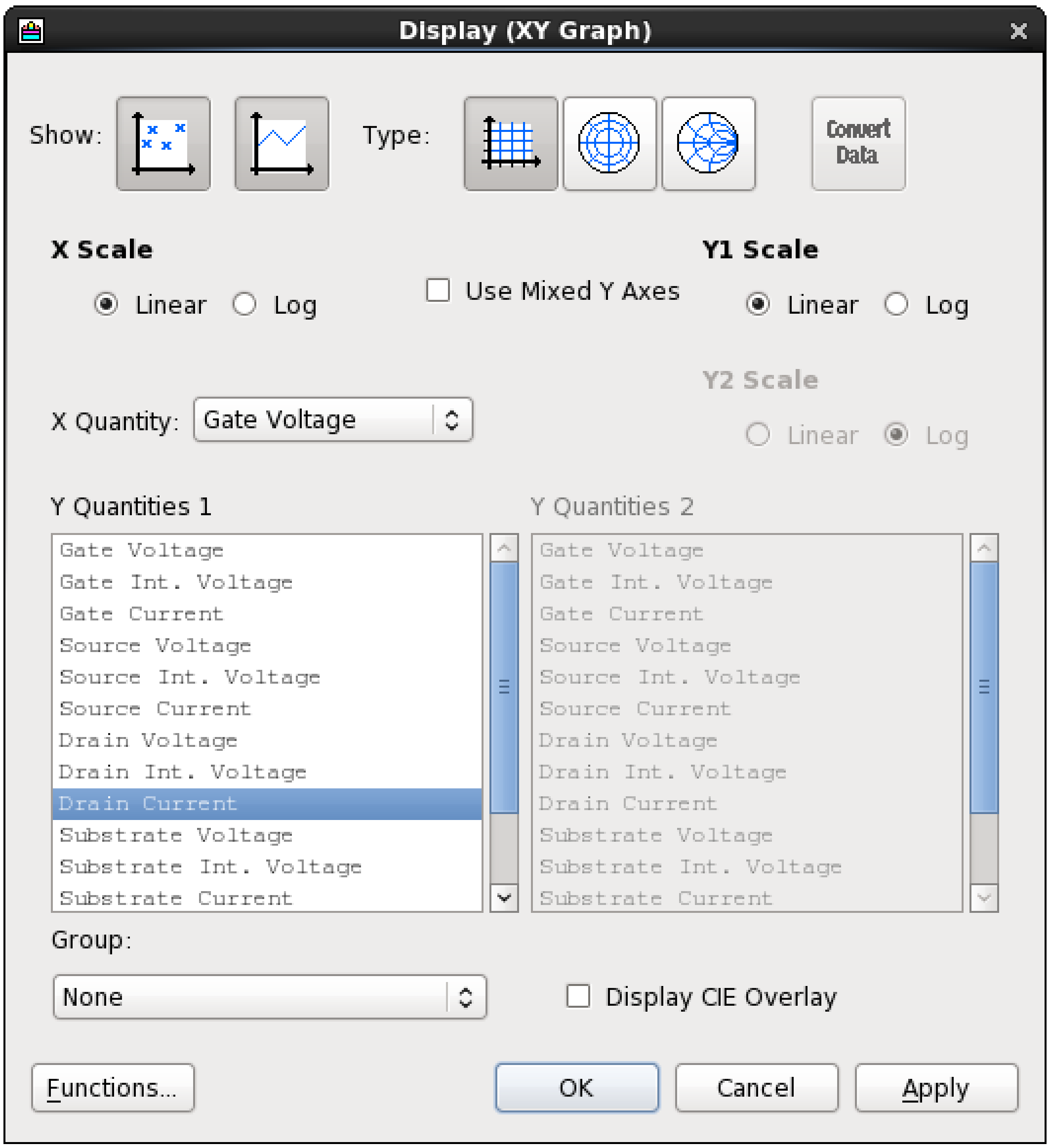


Figure 5-1 Graph Plot Dialog

- **Show**: This allows you to select (**Points**/**Lines**) the way lines are drawn on the graph.
- **Type**: This specifies the type of graph that is to be drawn. The options represent Cartesian graphs, polar plots, and Smith charts. The data that is plotted is mapped onto axes of the chosen type. If the data is in a format that matches the type chosen (e.g. [r, theta] pairs for polar plots), then selecting the **Convert Data** button tells TonyPlot to transform the coordinates before plotting them.
- **Display CIE Overlay**: This displays a diagram of the CIE 1931 color space.
- **Functions**: This displays the **Functions** dialog, which can be used to define the functions that can be selected from the choice of **Quantities**. Functions can be plotted on any axis. The type of graph chosen will affect the controls on the remainder of the **XY Graph Display** dialog.

5.2 Cartesian Graphs

Scales

Since only one quantity can be plotted on the x-axis, there is an item to select a linear or log (base 10) x-axis. Choose the one desired. For the y-axis, more than one quantity can be plotted. If all of them are to be on a linear scale, choose **Linear** for the y1 scale. Choose **Log** if they are all to be log. Choose **Use Mixed Y Axes** if both linear and log quantities are to be plotted on the y-axis.

X Quantity

One quantity can be chosen for plotting on the x-axis. All quantities available in all selected plots appear in this list. If one of the plots does not have data for the chosen quantity, nothing is drawn.

Y Quantities

Any number of quantities can be chosen for plotting on the y-axis. All quantities available in all selected plots appear in two lists. The **Y1 Scale** controls whether the **Y Quantities 1** list is linear or log. The **Y2 Scale** similarly controls the **Y Quantities 2** list. The **Y Quantities 2** list is only available if **Use Mixed Y Axes** is selected.

A context menu attached to each list (accessed by clicking the right mouse button over the list) makes list control a little easier. There are options to move selections from one list to the other, and for rapidly selecting, deselecting and locating choices in the lists.

Group

When Cartesian data is plotted that contains different groups of data sets for the same y quantity, this item can be used to specify which quantity divides the y value into its distinct groups. For example, a structure may contain data to show several plots of drain current against drain voltage for different values of gate voltage. In this case, the x-axis would be set to “drain voltage”, the y-axis to “drain current” and the **Group** item to “gate voltage”. The plot would show one curve of Id vs. Vd for each value of Vg.

5.3 Polar Charts

When the graph type is **Polar**, four subpanels are shown. Each subpanel can be used to display quantities. In other words, up to four polar curves can be plotted.

Two quantities are used to specify each curve. By default, the quantities real and imaginary are used when the data is not converted. If the data is to be converted, the quantities R (radius) and A (angle) are used. The data should only be converted if it appears in (r, theta) form in the structure. When quantities are present that TonyPlot recognizes as being usually displayed on polar charts, TonyPlot tries to automatically select an “i” (or “A”) quantity whenever you choose an “r” (or “R”) quantity.

The real or radius quantity can be logged before plotting, and the angle quantity can be specified in terms of degrees or radians. Choose the setting which corresponds to the data in the structure.

There are some options to control the polar chart drawn. The chart can be drawn proportionally (e.g., concentric circles appear as circles, even if the plot window is not square) and radial labels can be shown in degrees (radians is the default). The radial lines can be drawn at various intervals. Choose the interval desired from the item marked **Radials**.

5.4 Smith Charts

When the graph type is Smith, four subpanels are shown. Each subpanel can be used to display quantities. In other words, up to four Smith curves can be plotted. This is basically the same as Polar charts described previously.

Two quantities are used to specify each curve. By default, the quantities real and imaginary are used when the data is not converted. If the data is to be converted, the quantities R and X are used. The data should only be converted if it appears in R, X form in the structure.

When quantities are present that TonyPlot recognizes displayed on Smith charts, TonyPlot tries to select an “i” (or “X”) quantity whenever you choose an “r” (or “R”) quantity. There are some options to control the Smith chart drawn. The chart can be drawn proportionally (e.g., concentric circles appear as circles, even if the plot window is not square), and axis arms can be drawn in all four Smith quadrants (only the first quadrant is shown by default).

5.5 Transforms

The **Transform XYG** dialog is accessed by selecting **Edit→Transform XYG…**. This dialog allows mathematical operations to be performed between multiple selected XY Graph curves.

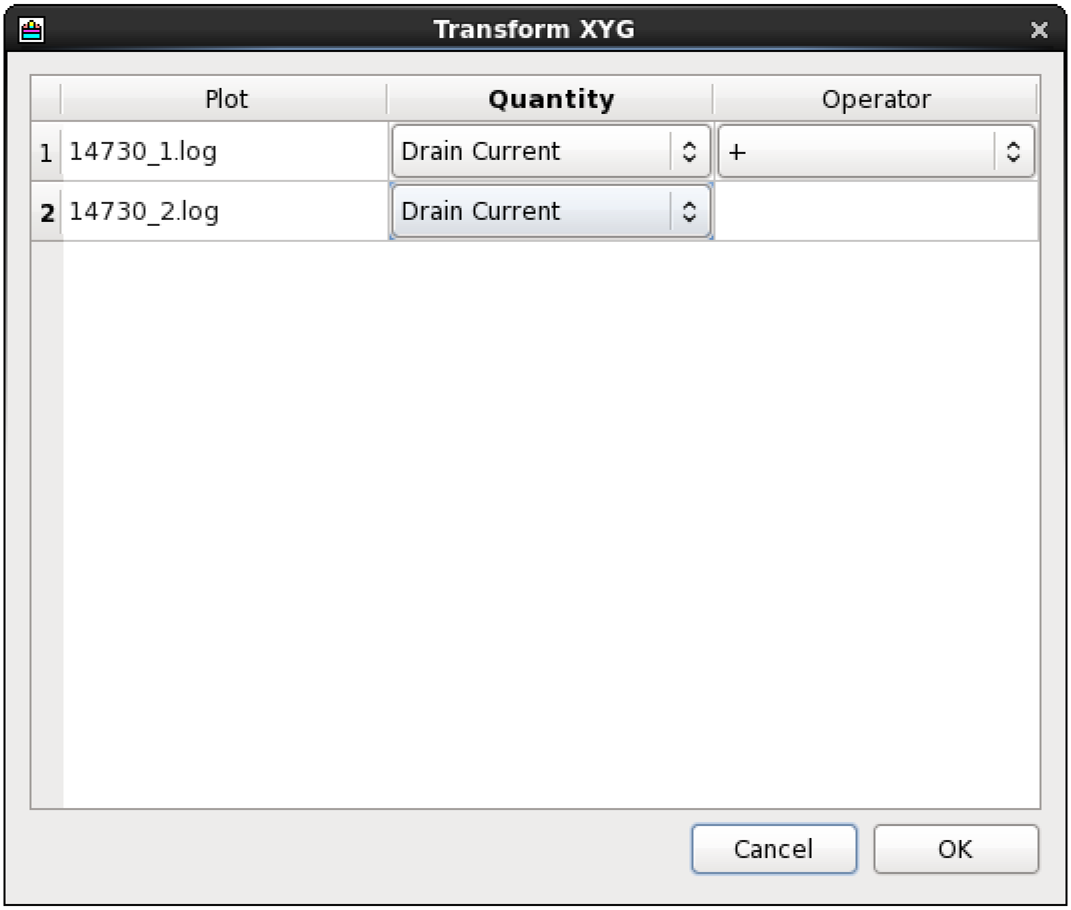


Figure 5-2 Transform XYG Dialog

5.6 Save As

XY Graph plots can be saved to Silvaco Standard Format (`.str`), TonyPlot User Data (`.dat`), Spayn Data File (`.dat`), Neural Net I/O File (`.net`), Spice Raw File (`.raw`) and Comma Separated Values File (`.csv`).

Note: To obtain a .csv file from a .str file, first use an extract command in DeckBuild to obtain a (`.dat`) file, then save this data from TonyPlot as a `.csv`.

To save data, choose one plot that you wish to write to a file and select **File→Save As**. The **Save As** dialog will then appear. Select the option desired. To name the file to be created, enter a base name and a file extension. One file is created for each overlay level in the plot. For example, if you export a three level plot, the basename is `hello`, the extension is `dat`, and the following three files will be created.

```
hello1.dat
hello2.dat
hello3.dat
```

6. Cross Section Display

6.1 Cross Section Display

The **Cross Section** dialog (Figure 6-1) shows the current display settings for the first selected **Cross Section** plot. See Section 2.7.2 Cut Line for details on how to generate **Cross Section** plots. When the settings on the display dialog are applied, all selected **Cross Section** plots are affected. This permits much easier application of global changes to similar plots in the view. The **Cross Section** dialog contains all the controls needed for complete control of **Cross Section** plots.

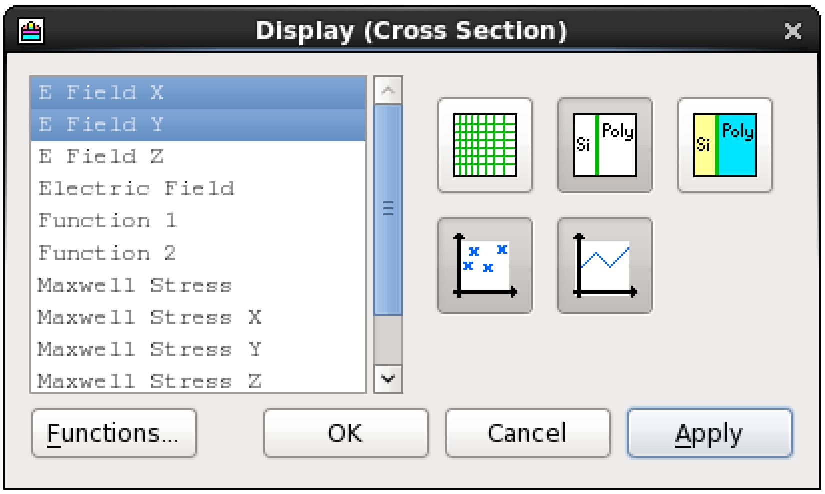


Figure 6-1 Cross Section Dialog

The following items can be specified:

- **Quantity**: This specifies quantities to be plotted. Any number of quantities can be plotted. One of two functions can also be chosen. These functions are defined from the **Functions** dialog. See Section 2.8 Functions for more information.
- **Options**: This allows you to add mesh, interfaces, and materials to the plot. These options are represented by the three icons in the top right of the dialog. To add any of these features to the plot, select the corresponding icon. The icons underneath control the way lines are drawn on the graph.

The icons allow you to draw points or line or both segments.

- **Axis Scales**: Each quantity that you can plot on a **Cross Section** plot has a default flag, which TonyPlot uses to determine whether to use a Linear or Log y-axis. If you plot linear and log quantities simultaneously, two y-axes will be drawn. One to show all log quantities, and the other to show all linear.
- **Functions**: Click on this button to display the **Functions** dialog. This dialog can be used to define the functions that can be selected from the choice of **Quantities**. See Section 2.8 Functions for more information.

The type of y axis drawn depends on the quantities being plotted. If log scale quantities are drawn, such as **Net Doping**, then TonyPlot draws a true log scale axis. If linear quantities are drawn, such as **Potential**, a normal linear axis is drawn. If a mix of the types of quantities are drawn, TonyPlot draws both types of axis: the log axis that appears on the left side of the subwindow, and the linear axis that appears on the right. When reading values from the curve, be sure to use the correct scale.

For dopants, the log axis shows values below 1e12. You can change this value in the **Plot Options** panel of the **Preferences** dialog (see Preferences).

6.2 Save As

Cross Section plots can be saved to Silvaco Standard Format (`.str`) and TonyPlot User Data (`.dat`). To save data, choose one plot that you wish to write to a file and select **File→Save As**. The **Save As** dialog will then appear. To name the file to be created, enter a base name and a file extension. One file is created for each overlay level in the plot. For example, if you export a three level plot, the basename is `hello`, the extension is `dat`, and the following three files will be created.

```
hello1.dat

hello2.dat

hello3.dat
```

7. RSM Display

7.1 Overview

The **RSM Plot** display dialog allows you to control the display of the Response Surface Model (RSM) inputs and outputs. RSM plots can be drawn in one of three modes: **1D graphs**, **2D contours**, or **3D surfaces**.

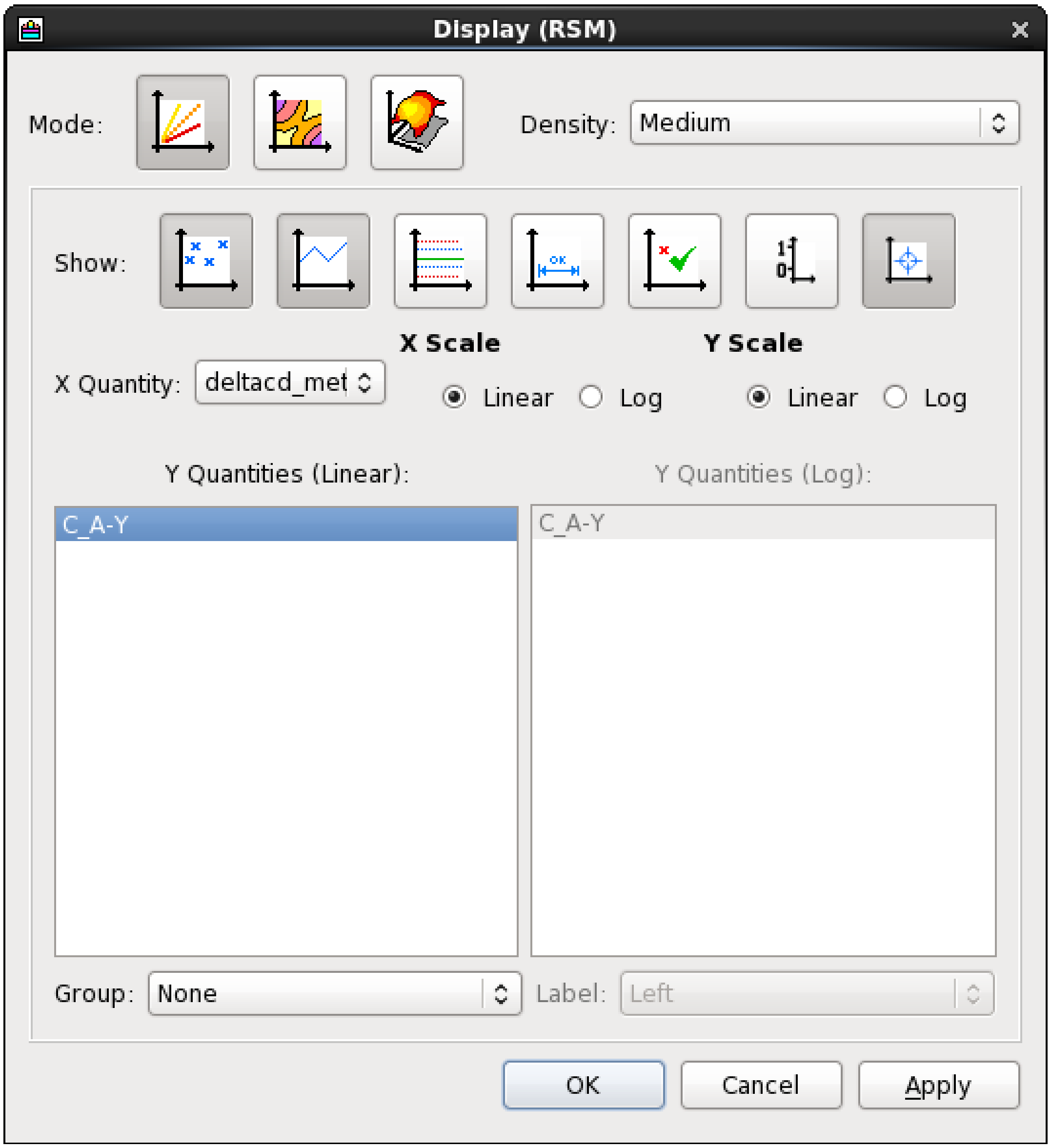


Figure 7-1 RSM Dialog

To draw any RSM plot, TonyPlot "samples" the input(s) a number of times to calculate values for the output. The number of samples taken (which are always regularly spaced) is determined by the setting of the **Density** item. Higher densities generate more points, creating smoother curves or surfaces, but take longer to compute. Low density plots are quick to calculate and draw, but provide only “approximate” plots.

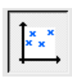
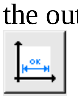
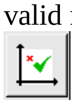
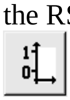
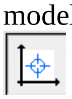
For total control of RSM plots and access to a selection of analysis tools, enable the VWF Production Mode. A description of these features can be found in VWF Production Mode. Without **Production Mode**, the benefit of RSM plots is greatly reduced.

7.2 1D RSM Graphs

When you plot RSMs in the 1D mode, you can select any one input for the x- axis, and you can select any output(s) for the y-axis. All RSMs that contain both the input and output are plotted.

Show

There are several ways to display data on 1D RSM plots. These are selected from the row of icons near the top of the control panel.

-   **Points/Lines**: draw points and line segments at or between sampled points, to draw the curve. **SPC** limits can be added to the plot, if this information is available for the outputs being plotted.
-  activates the **Valid X Range** marker, showing the range of the input that is valid for the model used. The drawn range can be extended in certain cases to values outside the valid range.
-  represents **Measured Points**, which are plotted if measured data was passed with the RSM.
-  normalizes all outputs to a range between 0 and 1, for easier comparison of different models.
-  activates a crosshair, which can be used to track x- and y-coordinates along the curve.
- **X Quantity**: Any input parameter can be chosen for plotting on the x-axis. In addition, values of this input can be plotted on a log scale.
- **Y Quantities(s)**: Any output parameters can be chosen for plotting on the Y axis. There are two lists: one for outputs to be plotted on a linear axis, and one for a log axis. Any combination of the two can be used.

7.3 2D RSM Contours

In the 2D mode, RSM plots show how outputs vary with respect to two independent input parameters. You can specify the way to draw contours the same way contours are drawn in regular 2D Mesh plots from Atlas or Athena.

X and Y Quantities

You must choose two inputs for contour plots: one for the x-axis and one for the y-axis. The inputs chosen must be different. Each of these can be plotted on a log scale if desired.

Z Quantity

The Z Quantity cannot be chosen (it is always the RSM output parameter that is plotted). But you can specify a linear or log scale for the Z axis.

Contour Type

Contours can be drawn as lines or filled areas, and filled areas can be outlined. There are a number of color sets that can be used to create the contours. All these options are controlled with the items in the lower left corner of the control panel.

Mesh

Next to the icons for controlling the contour types is an icon that draws the “sample mesh” on top of the contour plot. This mesh shows the points where outputs were calculated to generate the plot. A triangular mesh is created from these points to draw contours.

Projection

This is not used in the 2D mode. See 3D RSM Surfaces for an explanation.

Output Range

The range over which contours drawn can be selected as one of two options. The first option is to use the highest and lowest output values over the sampled input range. This ensures all contour colors are drawn on the plot. The second option is to use to absolute range of the model output, which can be a greater range than that plotted.

7.4 3D RSM Surfaces

When you choose the 3D mode, TonyPlot draws an RSM as a three-dimensional surface with contours drawn according to surface “height”. Control over the display of these plots is the same as described above for 2D plots, except that projection may also be specified. **Projection** allows you to choose either parallel or perspective projections when the 3D surface is displayed.

Note: These 3D plots can be rotated and scaled (but not zoomed). See Plot Control for details on 3D rotation and scaling.

8. Statistics Display

8.1 Overview

Whenever statistics plots are present, the **Statistics Plot Display** dialog can be used to alter the way the data is displayed. These **Statistics Plots** are often generated from Production Mode tools (see VWF Production Mode).

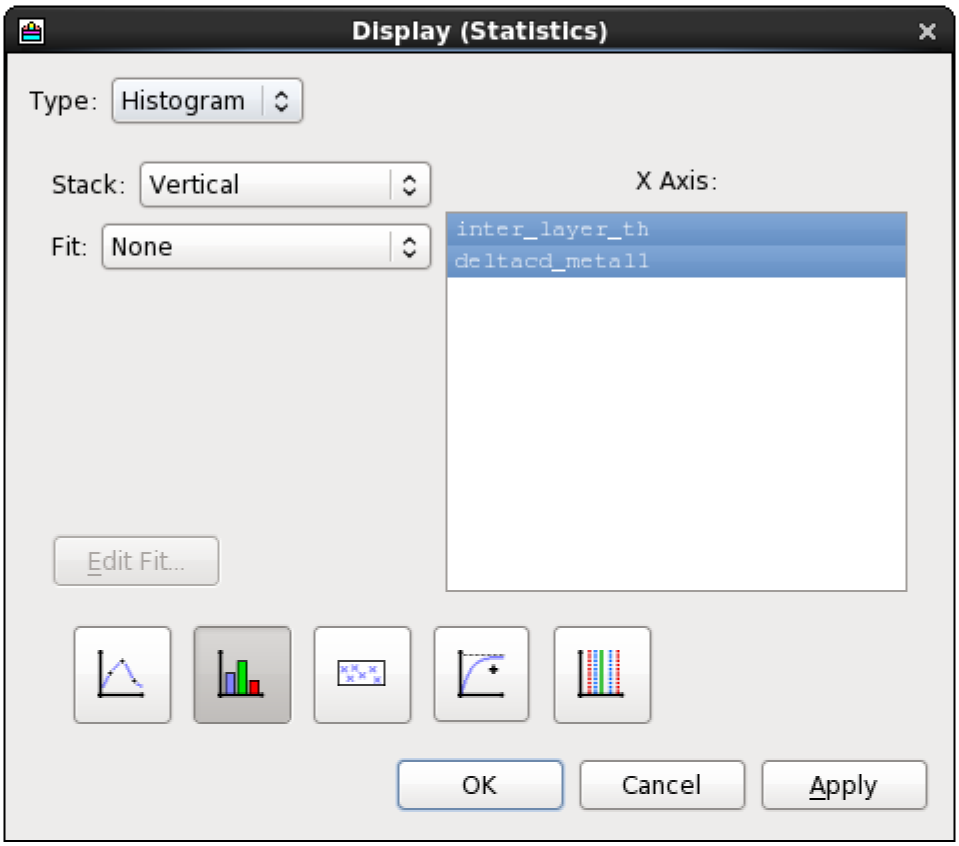


Figure 8-1 Statistics Dialog

Statistics plots can be one of several distinct types. These are **Histogram**, **Pie Chart**, **Scatter Plot**, **Box Plot**, and **Sunray Plot**. The current type is shown at the top of the dialog. To change the type, select it from the **Type** menu. For each type, a different set of control appears beneath, on the lower portion of the dialog.

8.2 Histograms

X Axis

You can plot one or more quantities on a histogram plot. Choose the quantities desired from the scrolling list to the right.

Stack

When you plot more than one quantity, the stack item selects the method used for showing each one on the same axis. You can stack the bars for each quantity vertically on top of each other or horizontally next to each other.

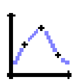
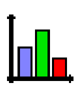
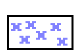
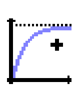
Fit

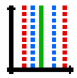
This item allows you to plot a best-guess distribution curve over the data. TonyPlot uses the range, mean and standard deviation of the data to generate a distribution curve of the chosen type.

Show

The icons along the bottom control what items can be drawn on a histogram plot.

Table 8-1 Histogram Plot Controls

 Lines	Draws from one bar to the next.
 Solid Bars	Draws a standard histogram. This is the default option.
 Jitter Plot	Shows the distribution of all the data points.
 Cumulative Curve	Shows the total number of data points over the X axis range.

 SPC Limits	Drawn when RSM output quantities are plotted on the histogram. See SPC Limits for more information.

8.3 Pie Charts

Controlling of pie charts is simple. Just select the quantities to be displayed from the list, and a pie will be drawn for each of them. To remove a slice from each pie, enter the number of the slice into the text field labeled **Remove Slice**. A value of zero means “remove no slice”.

8.4 Scatter Plot

Scatter plots show distributions of data in an x-y graph. By selecting various parameters for the X and Y axes, you can see the correlation of parameters graphically.

X Axis

Choose one quantity to be plotted along the X axis.

Y Axis

Choose one or more quantities to be plotted along the Y axis.

8.5 Box Plot

Box plots are used to examine the overall structure of the data. Use the list to select the quantities to be plotted, and a box will be drawn for each one. You can display the boxes horizontally or vertically. When they displayed horizontally, you can add a jitter plot. A jitter plot shows the distribution of all the data points for each box.

8.6 Sunray Plot

Sunray plots show data values distributed around a central point with the distance of each point from the center proportional to the data value. This yields a star or hedgehog-like plot.

Select the quantities to be plotted from the scrolling list, and a sunray plot will be drawn for each one. The icons along the bottom control how to draw the sunray plots. These are circumference lines, radial lines, and a bounding circle whose radius is the maximum data value.

9. VWF Display

9.1 Overview

The **VWF Plot** dialog (see Figure 9-1) shows the current display settings for the first selected VWF plot. When the settings on the display dialog are applied, all selected VWF plots are affected. This permits much easier application of global changes to similar plots in the view. VWF plots can be drawn in one of three modes: **1D graphs**, **2D contours**, or **3D surfaces**. The **VWF Plot** dialog contains all the controls needed for complete control of VWF plots.

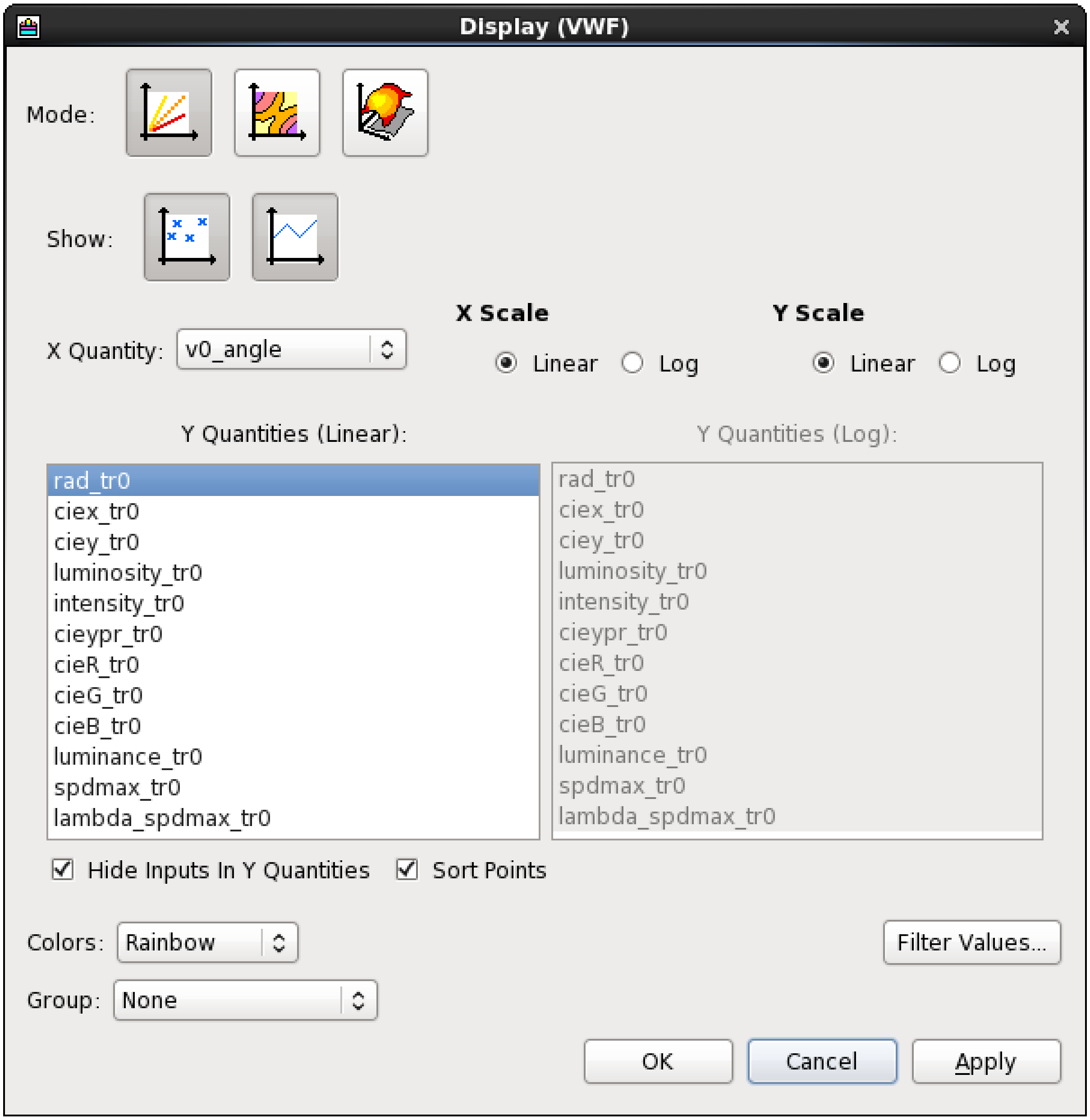
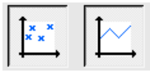


Figure 9-1 VWF Plot Dialog

9.2 1D VWF Graphs

Show

There are different ways to display data on 1D VWF plots. These are selected from the row of icons near the top of the control panel.

- **Points/Lines**: draw points and line segments at or between sampled points, to draw the curve.
- **X Quantity**: Any input parameter can be chosen for plotting on the X axis. In addition, values of this input can be plotted on a log scale.
- **Y Quantities**: Any output parameter can be chosen for plotting on the Y axis. By default, inputs are hidden, but can be enabled as well by unchecking the **Hide Inputs In Y Quantities** check box. There are two lists: one for outputs to be plotted on a linear axis, and one for a log axis.

9.3 2D VWF Contours

In the 2D mode, VWF plots show how outputs vary with respect to two independent input parameters. You can specify the way to draw contours the same way contours are drawn in regular 2D Mesh plots from Atlas or Athena.

X and Y Quantities

You must choose two inputs for contour plots: one for the X axis and one for the Y axis. The inputs chosen must be different. Each of these can be plotted on a log scale if desired.

Z Quantity

You must choose an output for the Z/Contour quantity. You can specify a linear or log scale for the Z quantity.

Contour Type

Contours can be drawn as lines or filled areas, and filled areas can be outlined. There are a number of color sets that can be used to create the contours. All these options are controlled with the items in the lower left corner of the control panel.

Mesh

Next to the icons for controlling the contour types is an icon that draws the “sample mesh” on top of the contour plot. This mesh shows the points where outputs were calculated to generate the plot. A triangular mesh is created from these points to draw contours.

Projection

This is not used in the 2D mode. See 3D RSM Surfaces for an explanation.

9.4 3D VWF Surfaces

When you choose the 3D mode, TonyPlot draws a VWF plot as a three-dimensional surface with contours drawn according to surface “height”. Control over the display of these plots is the same as described above for 2D plots, except that projection may also be specified. **Projection** allows you to choose either parallel or perspective projections when the 3D surface is displayed.

Note: These 3D VWF plots can be rotated and scaled (but not zoomed). See Plot Control for details on 3D rotation and scaling.

10. Overlays

10.1 Overview

One of TonyPlot's most useful features is the ability to directly compare different sets of structure data. This not only means loading several output files at once and looking at each one in the same view, but also overlaying the data sets in the same plot subwindow.

When different files are overlaid, the plot is called an overlay plot, and has some characteristics not shown by normal plots. Each structure is drawn on a separate level in an overlay plot, and the levels are transparently stacked upon one another. Each level uses the same axes and display settings, so that the data can be readily compared. Obviously, each level must be of the same plot type (**2D Mesh**, **Cross Section** or **XY Graph**).

10.2 Making an Overlay

There are two ways to create overlay plots. One way is if the files to be overlaid are already loaded into TonyPlot (each in its own plot subwindow), select the plots to be overlaid (at least two must be selected) and select **Edit→Make Overlay**. A new plot is created containing one level for each plot selected. Another way is if data that is to be overlaid is not loaded, the new files can be overlaid onto an existing plot as they are loaded. To do this, select **File→Overlay…** and use the **Overlay** dialog (Figure 10-1), or the `-overlay` option if loading from the command line.

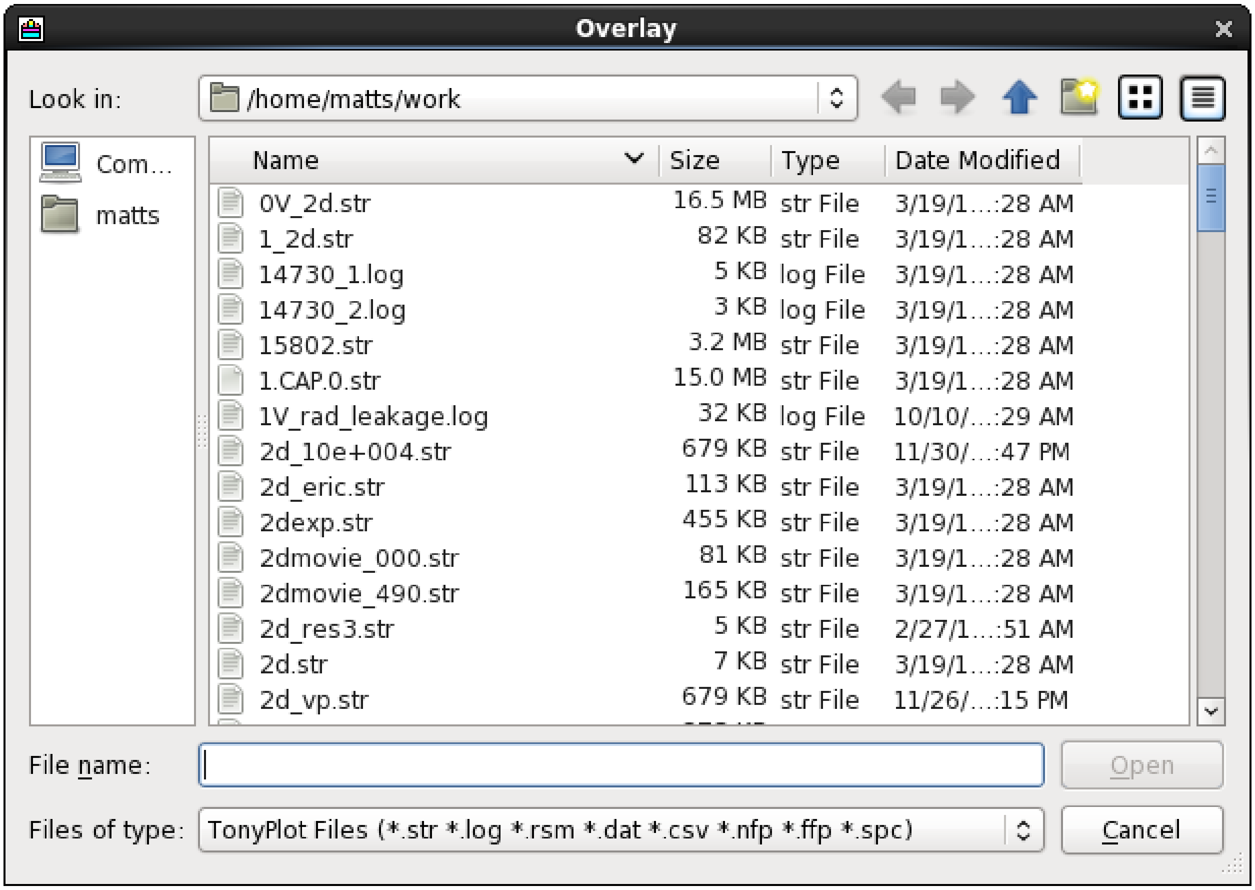


Figure 10-1 Overlay Dialog

10.3 Splitting an Overlay

An overlay plot can be broken down into separate single-level plots by choosing the **Edit→Split Overlay**. The overlay plot to be split must be selected when this option is chosen. One new plot will be created for each level in the overlay.

10.4 Overlay Control

Overlay plots are controlled just like any other plot. It is still a single plot even though several structures are displayed within it. Controls such as zooming, key commands, or labeling are still possible as though only one structure were present.

10.5 Overlay Display

Plot display for an overlay plot is the same as for a normal plot, except that multiple data sets are affected. All the data sets (i.e., every level) is displayed according to one common display setting. It is impossible, for example, to show contours in one level and vectors in another. Since each level must be of the same plot type, the dialog used to change the display settings are the same as described for normal plots.

Some levels may not be plotted if the display settings include quantities or options that are not present in the data for that level. The display dialogs, however, shows all quantities from all levels. For example, one level may contain `boron` and another level just `arsenic`. The display dialog shows both `boron` and `arsenic`. If you only choose `boron`, only the first level will be drawn.

10.6 Identifying Data

Data from separate levels can be identified by the overlay legend. This legend indicates which profile or graph line corresponds to a certain data file. The quantities plotted are identified with the same legend used in single-level plots. Each level is shown in a different color, (line type for monochrome screens) and each quantity with a different mark type. The colors used are determined by the current set of **Sequence colors**. See Preferences to determine how to set these colors.

Mesh plots, when overlaid, use the same color for each level. Because of the large amounts of information that can be portrayed in a 2D Mesh plot, the number of levels is limited to three. For **XY Graph** and **Cross Section** plots, there is no practical limit.

10.7 Level Names

Each level in an overlay is named from the file from which its data was taken. If needed, these names can be changed with the **Level names** dialog. Choose **View→Level Names...** to display this dialog.

To change the name of a level, first select the old name from the scrolling list. Then, enter a new name into the text field labeled **Name** and click on the **Apply** button. The plot updates to show the new names.

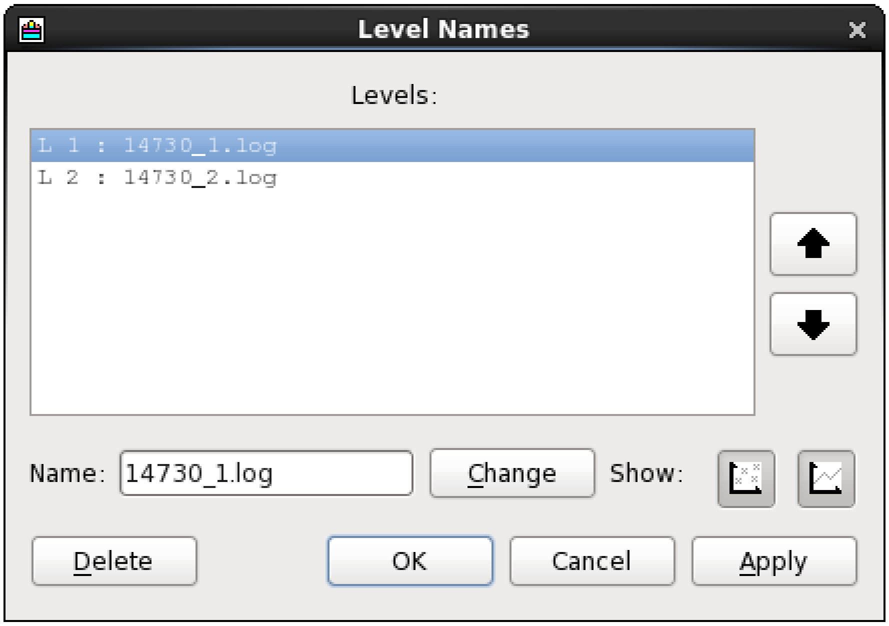


Figure 10-2 Level Names Dialog

10.8 Cutlines

When the **Cutline** tool is used on an overlay 2D Mesh plot, a section is taken from each level. TonyPlot automatically overlays each of these when it creates the new cross section plot.

10.9 Preferences

There are some TonyPlot preferences that apply to overlays. These are accessed by choosing **1D/2D Plot** from the Preferences dialog. Although explained in the Preferences, they are briefly described here.

- **Creating**: When creating overlays from existing plots, TonyPlot just creates a new plot. The old plots remain in the view. If you choose the **Deletes originals**, however, the plots that made up the overlay will be deleted when you create the overlay.

11. Legacy VWF Production Mode

11.1 Overview

TonyPlot provides the graphics to examine and interact with response surface models (RSMs) in one, two or three dimensions. These RSMs can be examined with a selection of **Production Mode** features, such as **Failure Analysis**, **Disposition**, and **Synthesis**. This section discusses the advanced **Production Mode** features available.

An RSM consists of a "response parameter", sometimes called an "output" or simply "model", and a number of "input parameters". The output is calculated form the inputs according to the RSM definition, which is passed from VWF to TonyPlot.

RSM plots are displayed as either simple XY Graph plots, where the model is plotted against the variation of one input, or as 2D or 3D contour plots, where the model is plotted against two inputs. In all cases, inputs not plotted are held at fixed values, although you can set these fixed values.

Controlling RSM plots is described RSM Display. See those pages for an explanation of how to draw RSM graphs, contours and surface plots.

11.2 Enabling Production Mode

There are three ways to enable the RSM graphics in TonyPlot. The simplest way is to let TonyPlot do it automatically: whenever an RSM plot is loaded, the **Production Mode** is started. The second way is to choose **File→Production Mode**. The third way is to do it via Spayn/VWF. In VWF, select **Tools→Spayn** to load a worksheet into Spayn. From Spayn, use **Tools Regression** to create the RSM and then start TonyPlot (see the VWF and Spayn User Manuals for further details).

When **Production Mode** is enabled, the **Production** menu (between the **Tools** and **Help** menus) becomes active.

This menu allows access to each of these **Production Mode** features (**Interactive control**, **Failure Analysis**, **Calibration**, **Synthesis** and **Yield Analysis**) as well as some parameter editing dialogs (e.g., Input range, SPC limits). Choosing any of the main features displays the Production dialog in the appropriate mode. Selecting one of the editing dialogs displays that dialog.

11.3 The Production Mode Dialog

By selecting an option from the main **Production** menu, the **Production Mode** dialog appears. This dialog shows all the RSM input and output parameters currently in use, and allows the selection of some advanced features. The dialog can be set to one of five "modes" (**Interactive control**, **Failure Analysis**, **Calibration**, **Synthesis** and **Yield Analysis**) by either choosing the mode from the **Production** menu, or from the choice item at the top of the dialog.

Below the mode selector is a panel whose contents depend on the mode currently selected. In the Interactive mode, for example, it shows the current **Process Name** (if any) and a row of four buttons for control of the input sliders.

The input sliders appear on the next panel down. Each input parameter in use is represented as a slider/toggle combination. Inputs can be “selected” by pressing the toggle — it is grayed out when not activated.

The dialog can show up to twenty-four inputs. You can expand or reduce size of the dialog to show more or less sliders. The bottom of the dialog has the usual **Cancel** and **OK** buttons.

11.3.1 Interactive RSM Control

The Interactive mode is the basic use for the **Production** dialog. When set to this mode, the mode control panel shows the current **Process Name**. This name can be changed if needed. Under the process name are three buttons that control the input parameter sliders:

- **Reset to nominal**: Clicking on this button sets all slider positions to the `nominal value` for each input. The nominal value is defined by the RSMs, but can be changed to any other value with the **Input Parameter Ranges** dialog (see Input Parameter Ranges) or with the next button.
- **Store as nominals**: Clicking on this button uses the current position of all selected input sliders as a new nominal value for those inputs. You can also set nominal values from the **Input Parameter Ranges** dialog.
- **Fix Y Axis**: When you toggle this option, the Y-axis ranges of the selected RSM plots remain when you move the input sliders.

This allows interactive user control over regression model parameters to study the effects on the response surface.

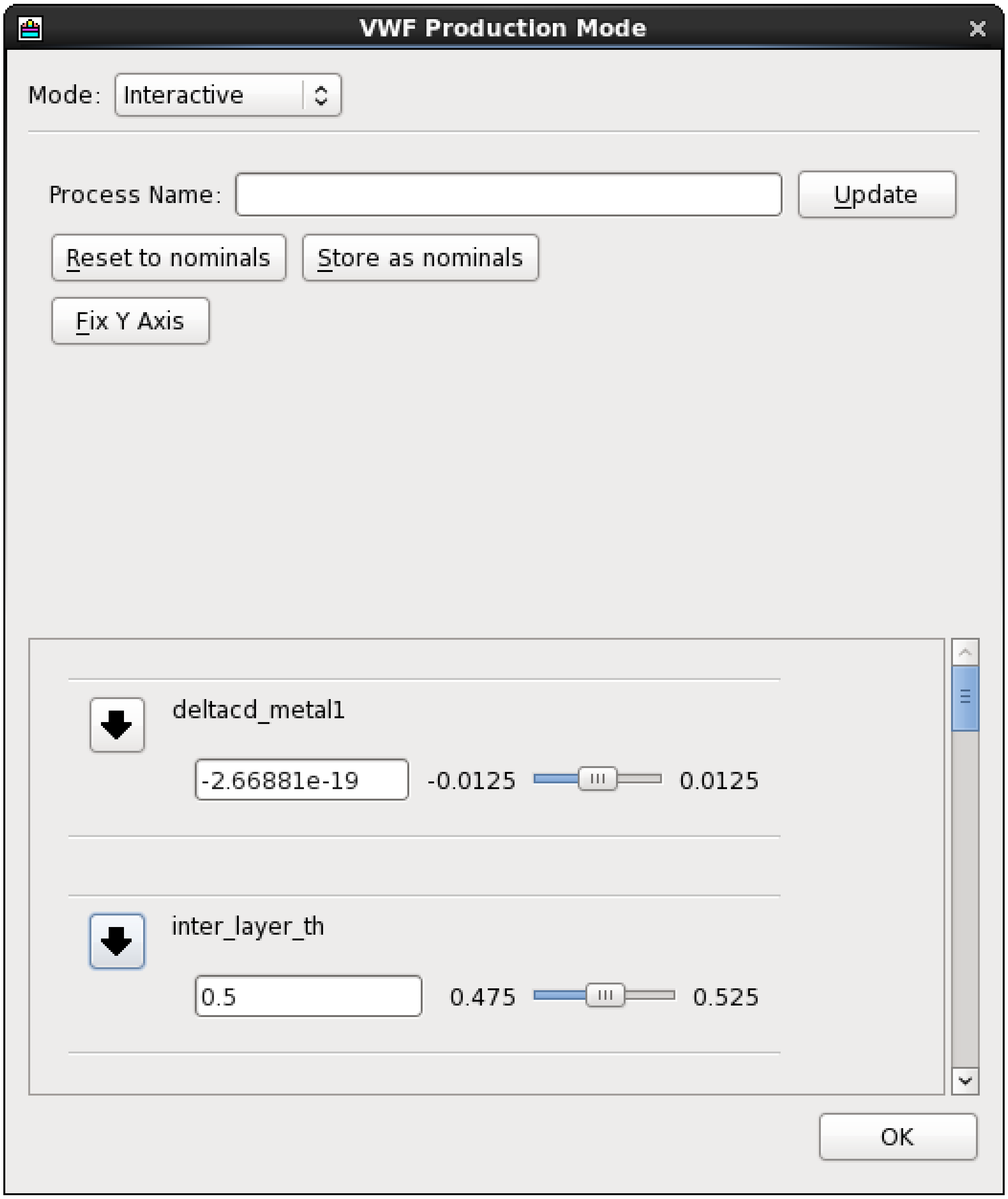


Figure 11-1 VWF Production Mode Dialog showing Interactive

11.3.1.1 The Input Sliders

Any input slider that is selected can be dragged left and right to interactively change the current value of that input. As the value changes, any selected RSM plots in the current view updates to reflect these new values. Any selected RSM plot is updated as follows:

- Any 1D RSM plot that has the input plotted on the X-axis moves the crosshair along the curve to match the changing X-axis value. If the input is not the X (or Y) quantity, the

whole curve will be recomputed for the new value.
- Any 2D plot that doesn't have the input as either the X or Y axis quantity is redrawn with the contours recalculated for the new input value.
- Any 3D plot that doesn't have the input as either the X or Y axis quantity is redrawn with the contours and surface recalculated for the new input value.

Note: Since plots are updated interactively, it is usually preferable to set the RSM plot density to "low". This speeds up the calculations and greatly improves the response time.

This interactive control is available regardless of the current dialog mode. That is, sliders can be dragged in any of the production modes, and plots are still updated interactively as described in Interactive RSM Control.

11.3.2 Failure Analysis

The **Failure Analysis** feature can be used to predict which input parameter is the most likely cause of an error in production. In other words, an output parameter or measurement that exceeds the specification or design limits.

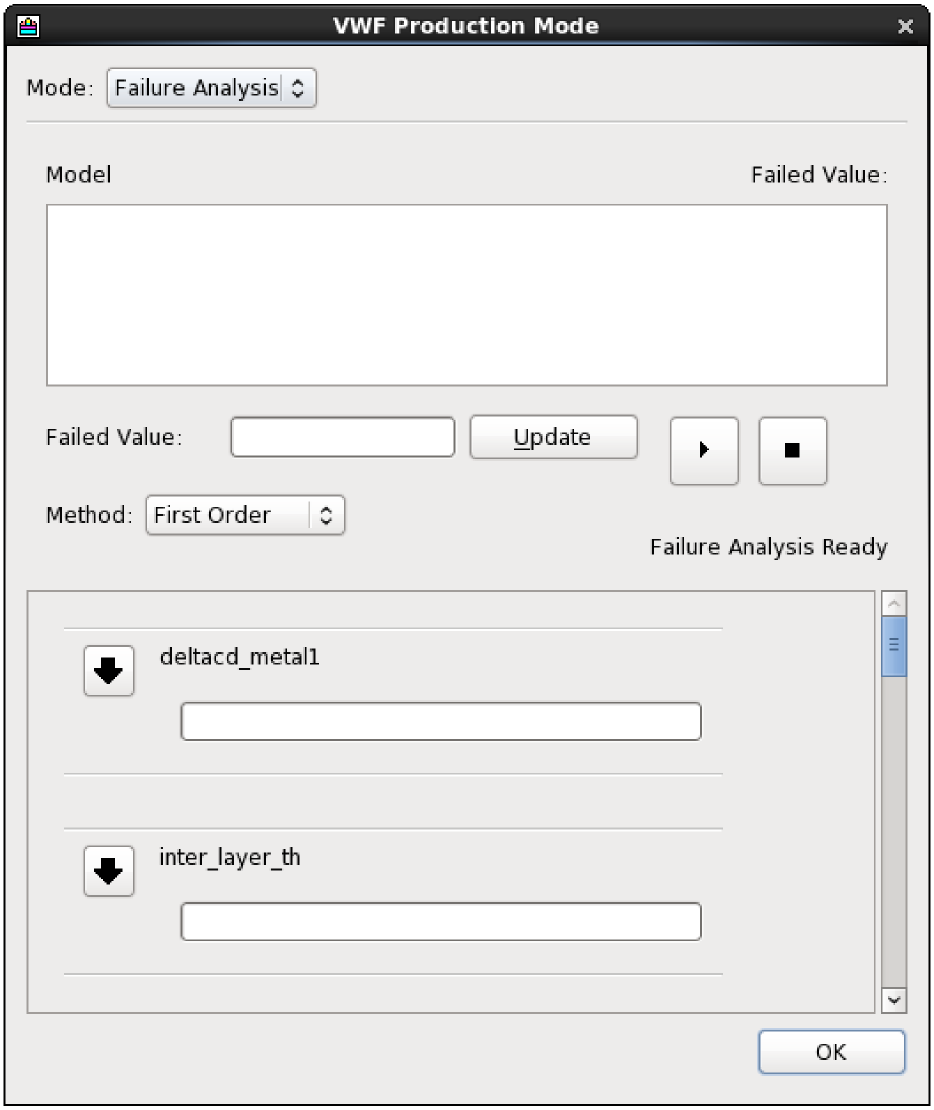


Figure 11-2 VWF Production Mode Dialog with Failure Analysis selected

11.3.2.1 Setup

When the **Production** dialog is set to **Failure Analysis**, there is a list of all output parameters, a text field for entry of "failed values" and **START** and **STOP** buttons. There is also a **Method** item.

- **Failed Values**: For each output parameter, enter the failed value that was measured at the production site. Press the Return key to submit this value to the list.
- **Method**: This controls the error tolerance method used in the failure analysis. Higher orders are less error tolerant, lower orders are more tolerant.
- **Select Inputs**: From the set of input sliders, choose the ones to be used in the analysis. TonyPlot tries to find which of these inputs is the most likely cause of failure in all the

outputs. You must at least select two inputs because the algorithm compares relative probabilities of failure cause.

- **START**: Click on this button to start the failure analysis. As each of the selected input parameters are tested, its slider moves from the minimum to the maximum value. Once all inputs have been tested, a plot of the results appears.
- **STOP**: Click on this button to stop the analysis at any time. All calculations are aborted, and no result plot appears.

11.3.2.2 Result Plot

When you run the failure analysis, a histogram appears (which is a form of the general **TonyPlot Stat** plot). The histogram shows the relative probabilities of each input being the cause of the failed values.

Note: Each input is tested individually. The analysis tries to find the one single input that could cause the specified failure in all outputs. The highest bars show inputs that are most likely to have caused the failure(s).

TonyPlot also adds a set of labels to the plot. These labels indicate the values of each input which would have generated the failed value, if that input had been the cause of the failure.

11.3.3 Calibration

The initial condition of the calibration function requires an overlaid display of the measured and RSM data to be made. At this time, choose which parameters are to be varied to do the calibration. Your choice is to set the process parameters to be fixed while the model parameters are allowed to vary.

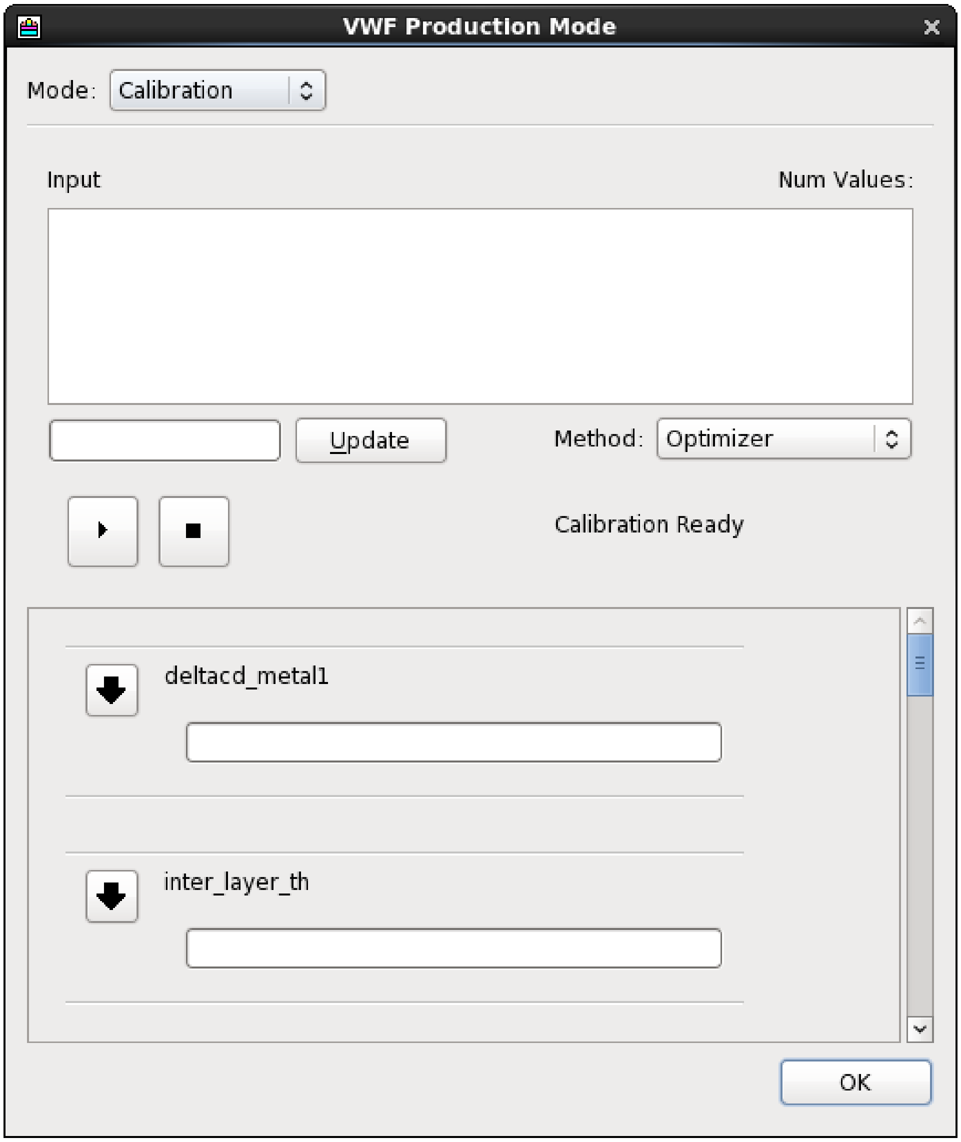


Figure 11-3 VWF Production Mode Dialog with Calibration selected

11.3.3.1 Setup

To use the calibration tool for VWF, do the following:

1. Generate a Design of Experiments (DOE) employing the "Most Practical and the Most Important" Process Parameters and the "Most Important" Calibration Parameters. Split lots can be Orthogonal DOEs or Random DOEs.
2. Run the Simulation Split Lot in Parallel on MP machine.
3. Generate RSM for each Process Parameter.
4. Load the (Calibration Parameters and Processing Parameters) RSMs into TonyPlot.
5. Load in the measured data into the Production Tools. Production Tools will find values of

Calibration Parameters that will best fit the measured data.
6. Save these RSMs for later use with Production Tools Yield Improvement Failure analysis.
7. Update the Calibration Coefficients for future use in future baseline input decks.

11.3.4 Synthesis

In **Synthesis** mode, TonyPlot tries to find a value for each selected input that produces a desired set of outputs. Two algorithms are provided to perform this reverse calculation — a “Levenberg- Marquardt” optimizer and a method known as Adaptive Simulated Annealing.

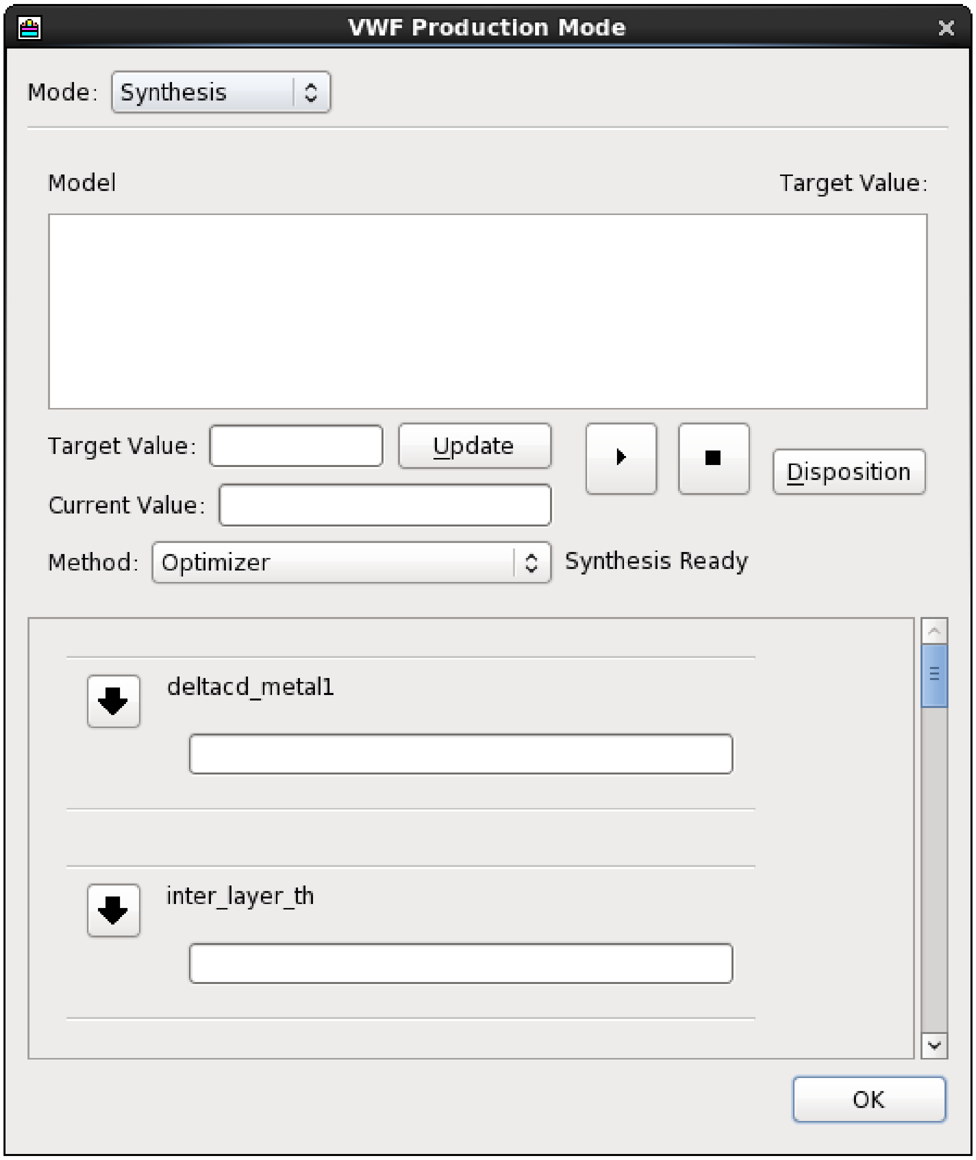


Figure 11-4 VWF Production Mode Dialog with Synthesis selected

11.3.4.1 Setup

Before starting the synthesis calculations, you must set the desired output values. Use the text field labeled **Target Value** to enter a desired value for the output currently selected in the list on the left. Press the Return key to submit this value.

When all target values are entered correctly, select the desired method. There is a choice between the **Levenberg-Marquardt** optimizer and **Adaptive Simulated Annealing**. Select the method desired.

Select the input sliders that are to be used in the synthesis: not all of them need to be used.

Unselected inputs will be fixed at their current values when the RSM computations use them. Certain operational parameters specific to each synthesis method can be customized to help obtain the required results. See Optimizer Setup or ASA Setup for further details.

Click on the **START** button to start the synthesis procedure. As the calculations progress, the latest results achieved will be displayed in the **Current value** text field. A status message describes the current state.

To abort the synthesis at any point, click on the **STOP** button. The procedure is cancelled and reset.

11.3.4.2 Results

The inputs sliders changes as the synthesis calculations progress. When the procedure is successful and complete, the input slider positions shows the values of the inputs needed to produce the output value(s) that were specified in the list of targets.

11.3.5 Yield Analysis

Yield Analysis can predict the characteristics of output yield from a production situation from known experimental data. TonyPlot generates large numbers of output measurements from a statistical sample of many inputs. By specifying the probability distributions of each input parameter, a total distribution of all outputs can be obtained.

The statistical distributions of input parameters are obtained in several ways. First, they can be passed to TonyPlot along with the actual RSM data that uses the inputs. If none pass, TonyPlot generates a default distribution. Finally, all input distributions can be specified using the **Input Distributions** dialog (see Input Distributions).

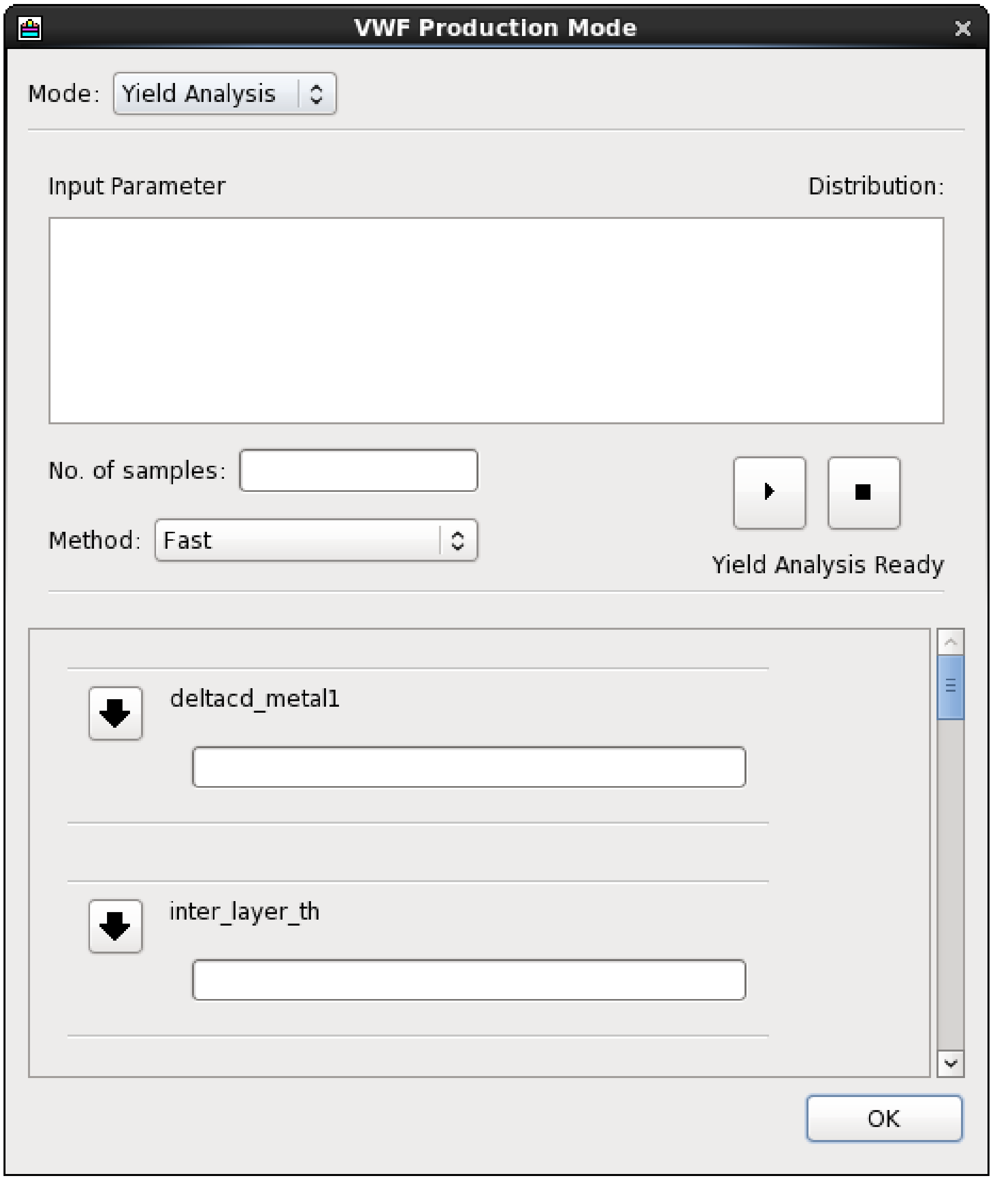


Figure 11-5 VWF Production Mode Dialog with Yield Analysis selected

11.3.5.1 Setup

The first thing to set up before running a **Yield Analysis** is the input distributions. The scrolling list on the **Production** dialog shows the distribution types for each input.

To examine the distributions in detail, and possibly alter them, use the **Input Distributions** dialog.

The number of samples taken by TonyPlot of each input parameter can be set to any value. The default is 5000 samples. The more samples that are taken, the more accurate the results, but it takes longer to perform the analysis.

Click on the green **START** button to start the analysis. The number of samples taken is continuously displayed next to the start and stop buttons. To abort the analysis at any time, click on the **STOP** button.

When all samples have been taken, TonyPlot takes a few moments to collate the data it has obtained, and then a histogram is plotted.

11.3.5.2 Results

The default result plot that Yield Analysis creates is a histogram showing the distribution of the output parameter(s). This plot is an instance of a standard **Statistics** plot, and can be manipulated further with the normal dialogs for **Statistics** plots.

The plot also contains data for all the input values that were used: these can also be plotted. Use the Statistics plot **Display** dialog to select sets of data to be plotted, and the type of plot. It is possible to draw pie charts, scattergrams, box plots, and other figures, to examine the data. See Statistics Display for more information.

11.3.6 Input Parameter Ranges

As RSM data is loaded into TonyPlot, a record is kept of the greatest range of each input parameter. Along with these minimum and maximum values, a “nominal” value is stored, which represents the normal, or default, value of that input.

The range and nominal value of any input can be changed by using the **Input Ranges** dialog (Figure 11-6).

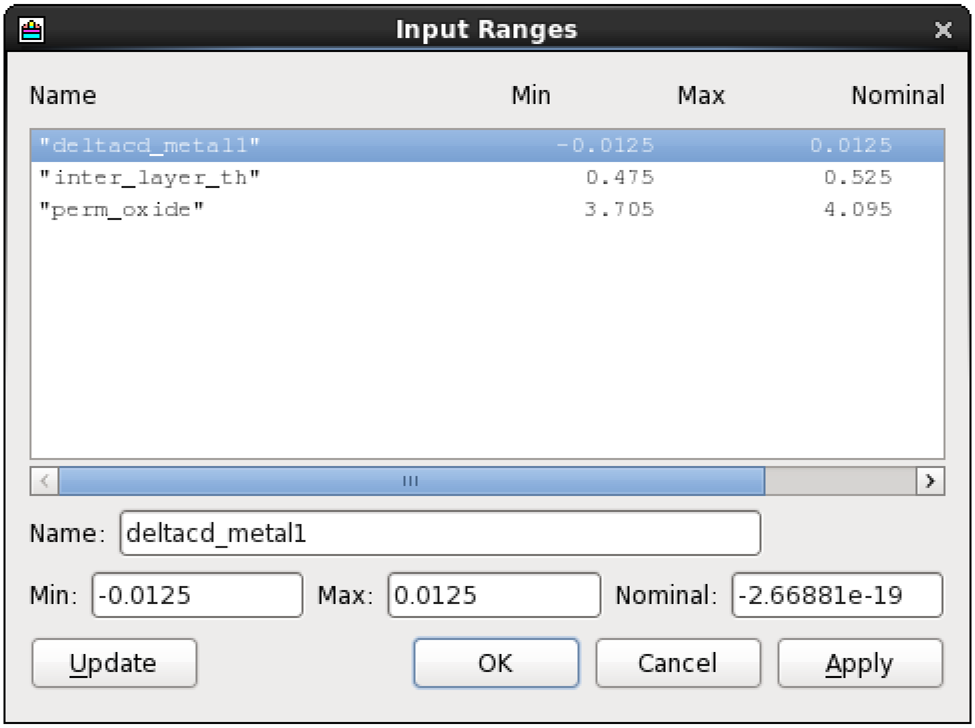


Figure 11-6 VWF Production Mode showing the Input Ranges Dialog

11.3.6.1 Control Items

The dialog shows a list of all inputs from all loaded RSMs, and shows the minimum, maximum, and nominal values for each one. Underneath the list, the name and values for the input currently selected are displayed: the values can be changed but the name cannot. Press the **Update** button on each text field to submit new values to the list. TonyPlot makes sure that the values are logical. In other words, the minimum is less than the maximum and the nominal lies between them.

Once you made all the desired changes to the list, click on the **Apply** button. This will submit these new values to TonyPlot. Any RSM plots are updated to use these new ranges and nominal values.

11.3.6.2 Uses

The **Input Parameter Ranges** dialog affects the range over which values are taken for each input when RSM outputs are computed. The sliders on the **Production** dialog reflect the current range for each input.

Some models may only be valid for specific ranges of their inputs. Since it is possible to exceed these ranges, there is a feature that allows the valid range to be identified on a 1D plot (see RSM Display for more information).

11.3.7 Input Distributions

As RSMs are loaded into TonyPlot, a default distribution is assigned to each one. This distribution represents the statistical “spread” of values that would be obtained for this input parameter in an experimental situation. Using this data, TonyPlot can simulate real-life input values by sampling data with the given distribution parameters.

The default distribution given to each input is Gaussian with a mean value halfway between the minimum and maximum value of that input. The standard deviation will be 10% of the mean.

To alter the distributions for any input, use the **Input Distributions** dialog (Figure 11-7).

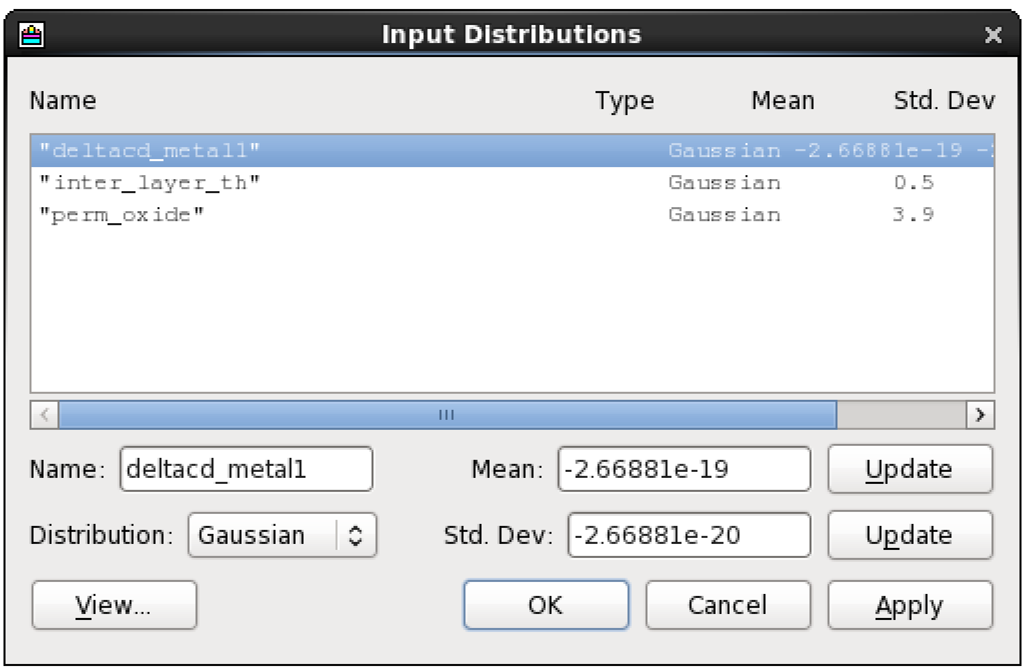


Figure 11-7 VWF Production Mode showing the Input Distributions Dialog

11.3.7.1 Control Items

The **Input Distribution** dialog shows a list of all inputs currently loaded and the type of distribution for each one, along with its mean and standard deviation.

Underneath the list, the values for the currently selected input are shown. The name of the input is fixed and cannot be changed but the other items can. Press the **Update** button on the **Mean** and **Std. Dev** text fields to submit changes to the list. If you select a new distribution type, the list will update automatically.

When you made all the desired changes, click on **Apply** to commit the new values. If the **Production** dialog is set to **Yield Analysis** mode, the new distribution types are shown in the scrolling list.

To examine and alter the distributions graphically, press the **View...** button. This summons a small window showing a histogram distribution of the selected parameter. Select other parameters to view their distributions also. The sliders under the histogram can be used to modify the mean and standard deviation of the parameter. Click on **Apply** to save the values back to the original scrolling list.

11.3.7.2 Uses

Input distributions are used in **Yield Analysis**. When TonyPlot takes input samples, it uses these distribution parameters to generate realistic values that may be found in a real-life scenario. For accurate **Yield Analysis** results, each input distribution should be set up so that it matches a closely as possible that found by experimentation.

11.3.8 SPC Limits

Each output parameter used in **Production** mode has a set of Statistical Process Control (SPC) parameters, which can be used to monitor the value of some measured value. There are five SPC limits:

- Upper spec limits (maximum values permitted)
- Lower spec limits (minimum values permitted)
- Upper control limits (ideal maximum)
- Lower control limits (ideal minimum)
- Center limit (ideal value)

These limits are abbreviated to **USL**, **LSL**, **UCL**, **LCL**, and **CL** respectively. Values for each of these are sometimes passed to TonyPlot through the RSM data. If not, or if they need to be modified, the **SPC Limits** dialog can be used to add or modify **SPC Limit** values for any output. To display these values, select **Production→SPC Limits** and the **SPC Limits** dialog will appear (Figure 11-8).

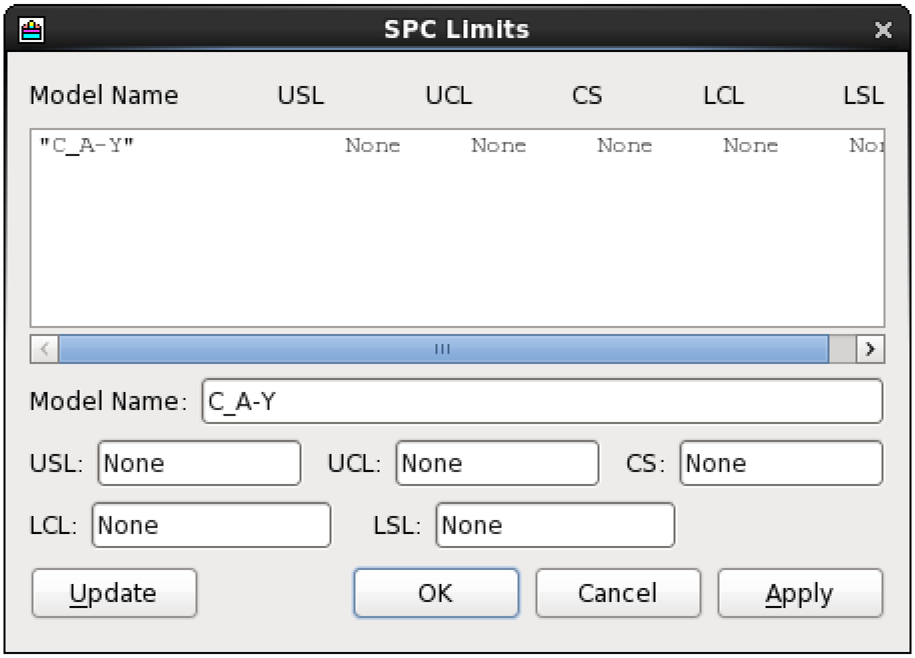


Figure 11-8 VWF Production Mode showing the SPC Limits Dialog

11.3.8.1 Control Items

The scrolling list on the **SPC Limits** dialog shows all output parameters and the USL, UCL, CL, LCL, and LSL values for each one. Underneath the list are five text fields where these values can be changed. Use the scrolling list to select the output that is to be modified, and enter new SPC values into text fields provided. Press the **Update** button in each text field to submit changes to the list.

If an output parameter has no defined SPC limits, use the word **None** to indicate missing values. To remove defined values, enter the word **None** into the text field and click on the **Return** key. When you made all desired changes, click on the **Apply** button to store the new values. Values in the list are not stored until the **Apply** button is clicked.

11.3.8.2 Uses

SPC Limits are used to monitor measured output values to ensure that these values stay within predefined boundaries and generate some warning when the boundaries are crossed.

Outputs generated by RSMs in TonyPlot can be compared to SPC Limits in a similar way. If these values are defined, the control lines can be added to any 1D RSM plot. See RSM Display for details on how to add these SPC limits to a plot.

11.3.9 Experimental Results

Each output parameter modeled by an RSM has an associated experimental value that was measured when the process input parameters used in the model were set to their nominal values. If no experimental value is given for an output, or one needs to be changed, select **Production→Experimental Results** and the Experimental Results will appear (Figure 11-9).

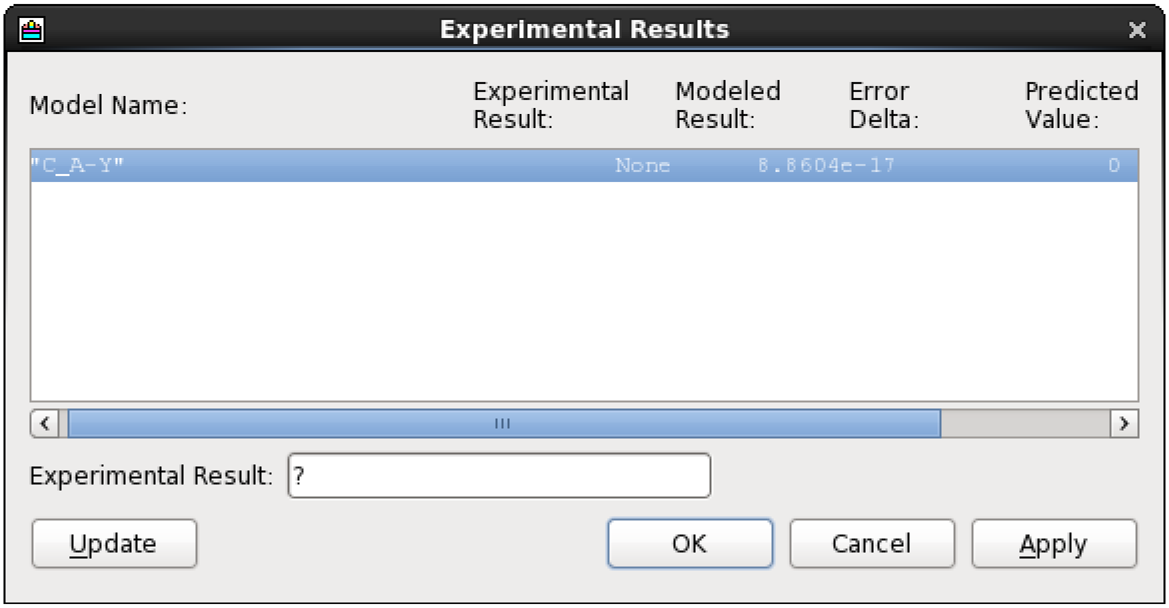


Figure 11-9 VWF Production Mode showing the Experimental Results Dialog

11.3.9.1 Control Items

The dialog shows a scrolling list with a line for each output. Next to the name of each output are four values: **Experimental Result**, **Modeled Result**, **Error Delta**, and **Predicted Value**.

- **Experimental Result**: This is the measured value. It can be set from information provided by the RSM as it is passed from the VWF, or it can be added or modified later using the text field.
- **Modeled Result**: As previously mentioned, the experimental result is measured when all inputs are set to their nominal values. The modeled result shows the value that is calculated by the RSM and is a measure of the accuracy of the model. It also used to make a first-order error correction of the model.
- **Error Delta**: The error delta is the difference between the experimental measured value and the modeled value when all inputs are nominal. The smaller this error, the more accurate the model. This delta is used to error-correct the model.
- **Predicted Value**: This shows the modeled result after error-correction has been applied. It is the same as the experimental results, to show that after error-correction, the model is

more accurate (when the inputs are nominal).

To change the experimental result for any output, select the appropriate row from the list, enter the new value in to the text field provided, and press the **Update** button. Once all new values have been made, click on the **Apply** button to commit the changes.

11.3.9.2 Uses

A measured result for an output parameter allows TonyPlot to make a simple error correction to a model of that parameter. The value shown as the “error delta” is added to values obtained from a model to “shift” the response curve to a more accurate position. The nominal case is used for this calibration (i.e. when all inputs are at their nominal vales), and the result applied for all modeled values.

11.3.10 Optimizer Setup

The **Synthesis** mode of the **Production** dialog provides two synthesis methods. One method is the Levenberg-Marquardt optimizer. The parameters for the optimizer can be changed in the **Optimizer Setup** dialog (Figure 11-10). The available parameters and their default values are:

- Max. no of iterations (100)
- Norm of gradient bound (1e-6)
- Sum of squares difference (1e-11)
- Marquardt initial value (0.1)
- Marquardt scale factor (2)
- Marquardt upper bound (1000)
- Switch (0.1)
- Maximum Jacobian (100)
- Maximum RMS error (1)
- Maximum average error (1)
- Maximum maximum error (1)
- Maximum iterations (4)
- Noise level (1e-18)
- Error ceiling (100)

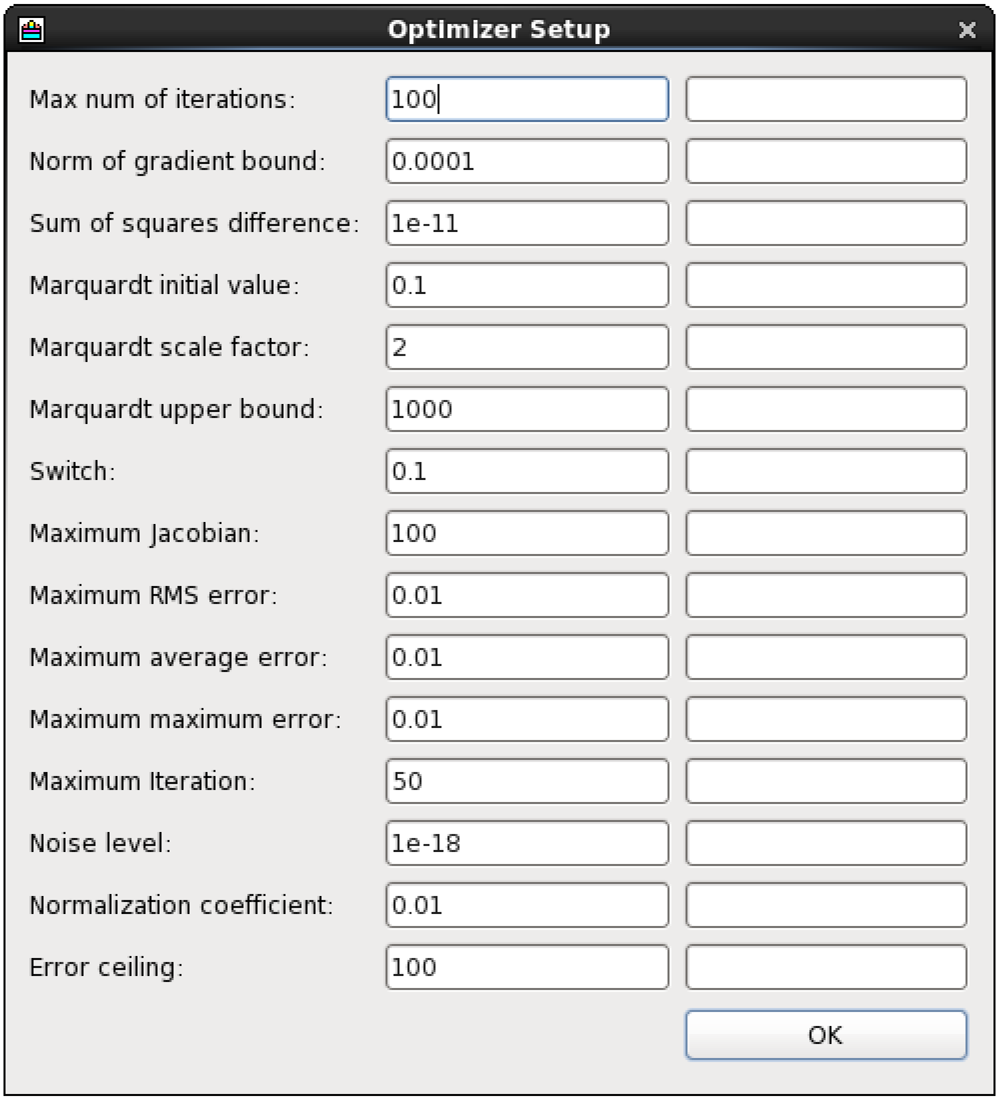


Figure 11-10 VWF Production Mode with Optimizer Setup selected

11.3.11 ASA Setup

The **Synthesis** mode of the **Production** dialog provides two synthesis methods. One method is Adaptive Simulated Annealing (ASA). The parameters for ASA can be changed in the ASA Setup dialog (Figure 11-11). The available parameters and their default values are:

- Limit acceptances (10000)
- Limit generated (99999)
- Cost precision (5%)
- Maximum cost repeat (1)
- Number cost samples (5)
- Cost parameter scale (1)

- Temperature anneal scale (100)
- Testing frequency modulus (5)

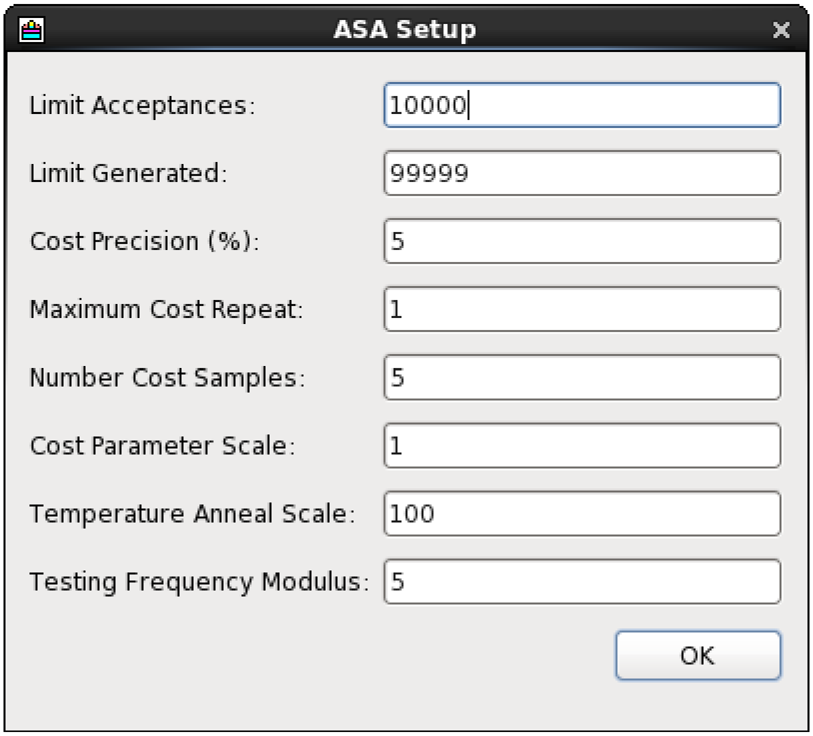


Figure 11-11 VWF Production Mode with ASA Setup selected

ASA code is made available under terms of the GNU general public license for libraries. See the file `$SILVACO/etc/gnu_license` for details about this license.

12. Preferences

12.1 Overview

TonyPlot has many preferences that you can set. You can alter these preferences to suit your needs. All preferences can be viewed and modified using the **Preferences** dialog. To open the preferences dialog, select **Edit→Preferences…**.

12.2 Manage Preferences

The Preferences dialog can import and export settings. Figure 12-1 shows the Manage Preferences settings in the Preferences dialog.

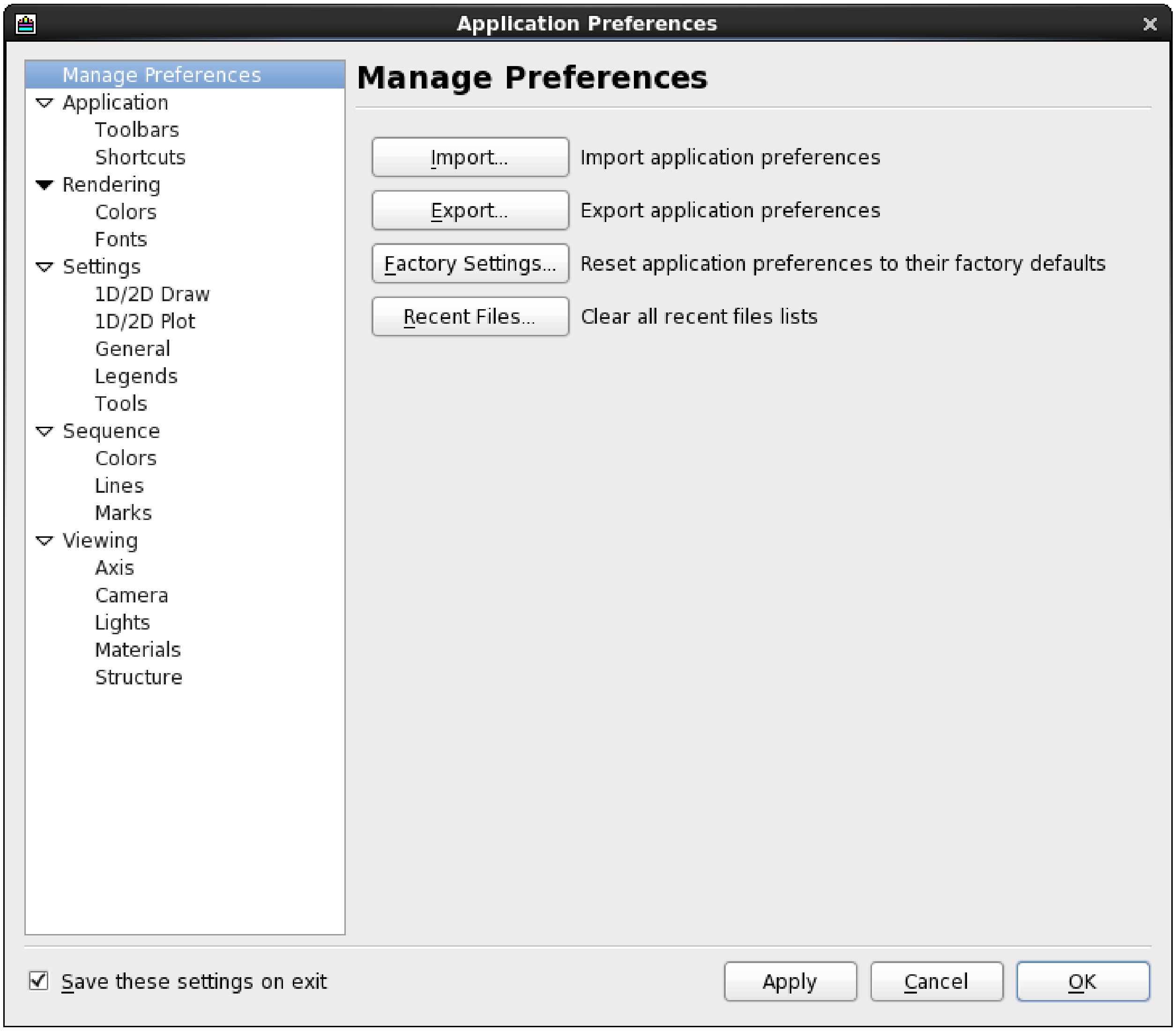


Figure 12-1 Manage Preferences

12.2.1 Exporting Preferences

To export settings to a file (*.spf), click the Export button. The Export Preferences dialog will then appear (Figure 12-2). This dialog has two parts: the name of the file to be exported and the desired preferences to export. First, browse to the desired path and type a filename in the dialog. Then, use the tree to expand and collapse the preferences to export. Checking the box next to All checks all preferences boxes in the tree. Unchecking the box next to All unchecks all preferences boxes. If the checkbox next to All has a gray background, some box or boxes below were unchecked. Finally, click the Export button. The *.spf file is written with all desired preferences. This file can be distributed to other users and then imported into TonyPlot to acquire the preferences.

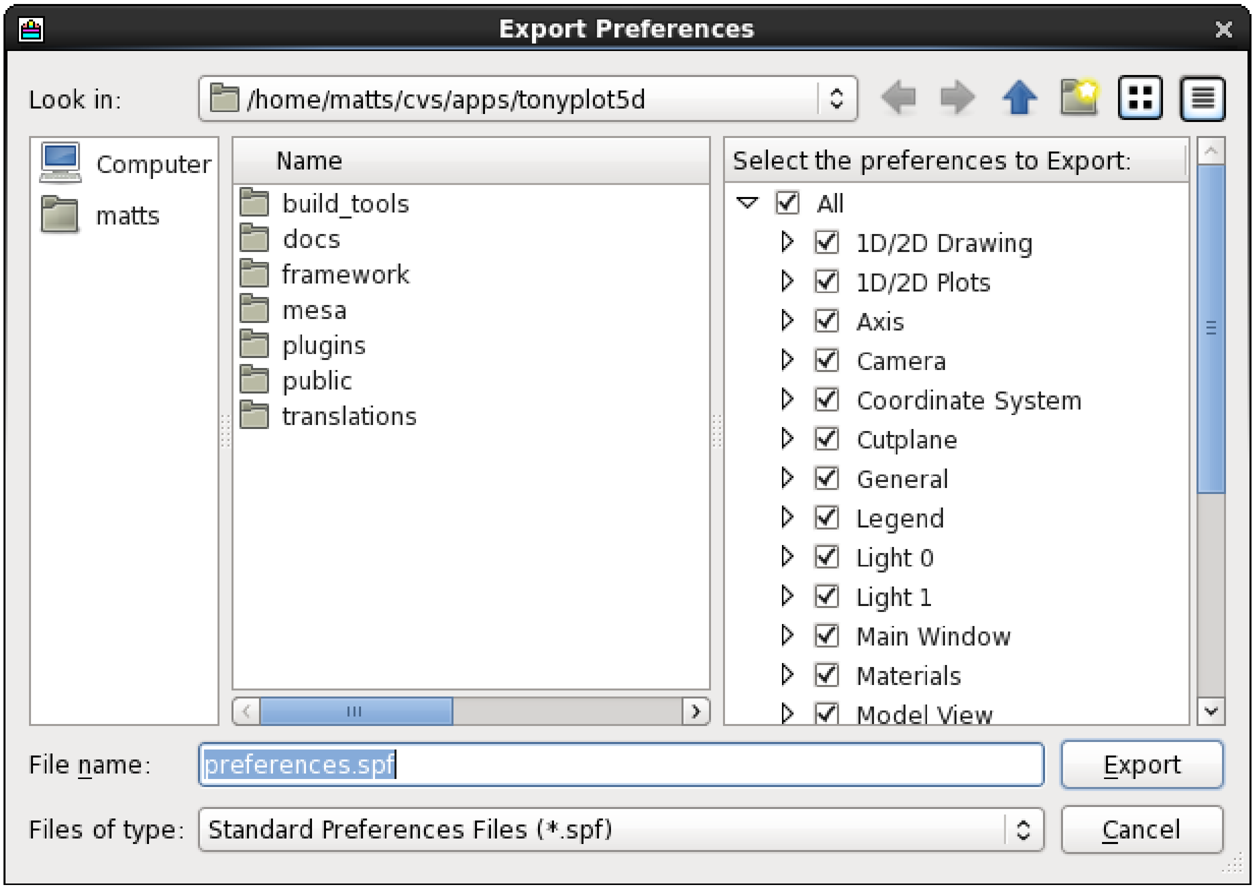


Figure 12-2 Export Preferences Dialog

12.2.2 Importing Preferences

After settings are exported to a *.spf file, they may be imported into TonyPlot on any computer. To import a preferences file, click the Import button on the Manage Settings window of the Preferences dialog. The Import Preferences dialog will appear (Figure 12-3). In this example in Figure 12-3, only the Materials from the preferences.spf preferences file will import into the existing TonyPlot preferences. This makes it easy to pick and choose which settings you may want to import from different preferences files. To import preferences from multiple files, repeat the steps to import, checking only the preferences you want from each specified file.

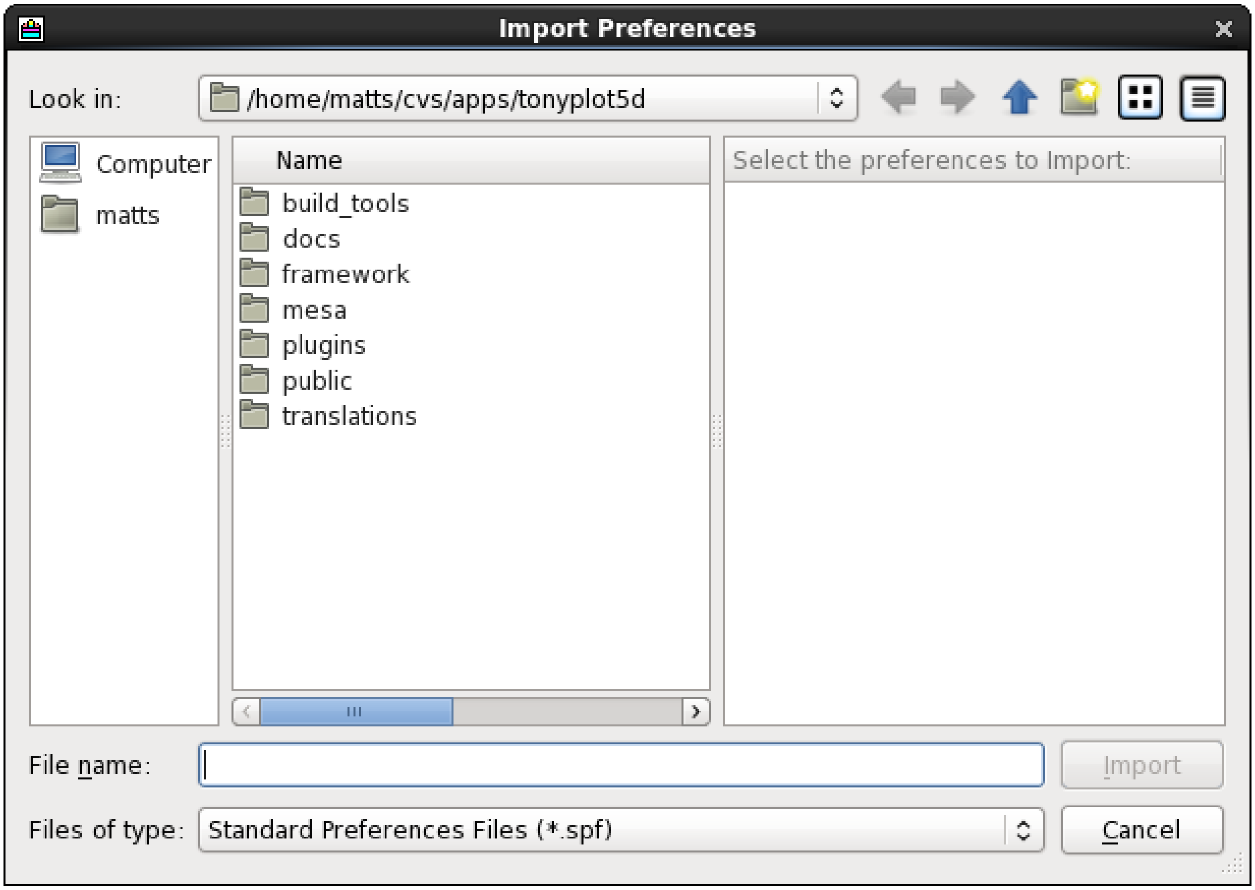


Figure 12-3 Import Preferences Dialog

12.2.3 Factory Settings

To reset the preferences back to the original defaults, click the Factory Settings button. The dialog in Figure 12-4 appears to confirm whether you want to reset to the default settings.

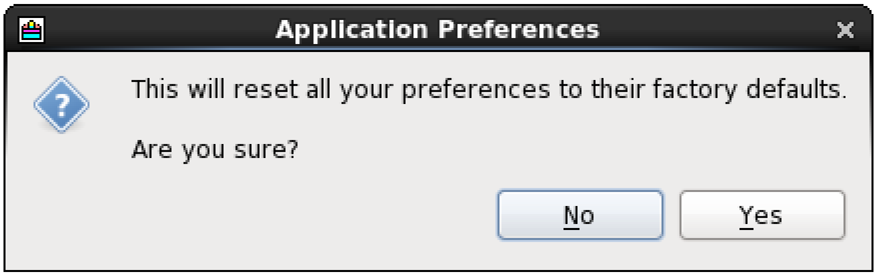


Figure 12-4 Confirm Factory Settings Dialog

12.2.4 Recent Files

To clear the recent file menus from memory, press the Recent Files button, then click Yes on the confirmation dialog.

12.3 Application

12.3.1 Toolbars

- **Display large icons**: This controls the size of the icons on the toolbar.
- **Display text labels**: This controls whether descriptive text is shown beneath the icons on the toolbar.
- **Display hints**: This controls whether tool tips are displayed when you hover the mouse cursor over an icon on the toolbar.
- **Customize**: Clicking this button displays a dialog that allows you to customize which buttons are currently shown on the toolbar.

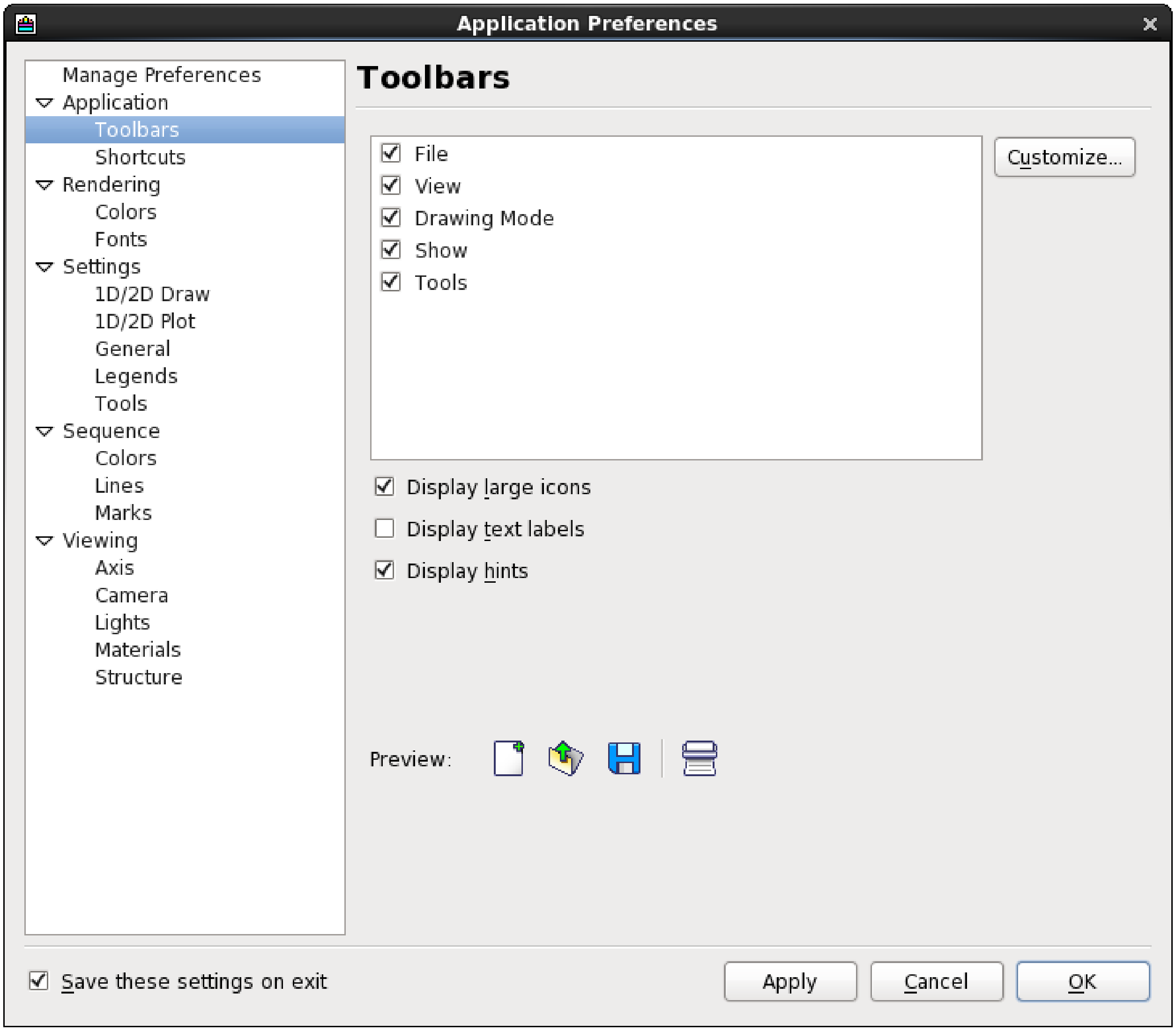


Figure 12-5 Toolbars Settings

12.3.2 Shortcuts

- **Assign**: This customizes which keyboard shortcuts you use for any menu item. Select the menu item you wish to customize. If there is a shortcut currently assigned to it, it will appear in the **Current Keys** box. Press your new desired shortcut (e.g., Ctrl+A) and it will appear in the **Press new shortcut key** box. It will also tell you if the shortcut is currently being used for another menu item. Press the **Assign** button to assign the new shortcut to the menu item.
- **Remove**: This removes the currently assigned keyboard shortcut from the current menu item.
- **Details**: This gives an overview of all the currently assigned shortcuts.

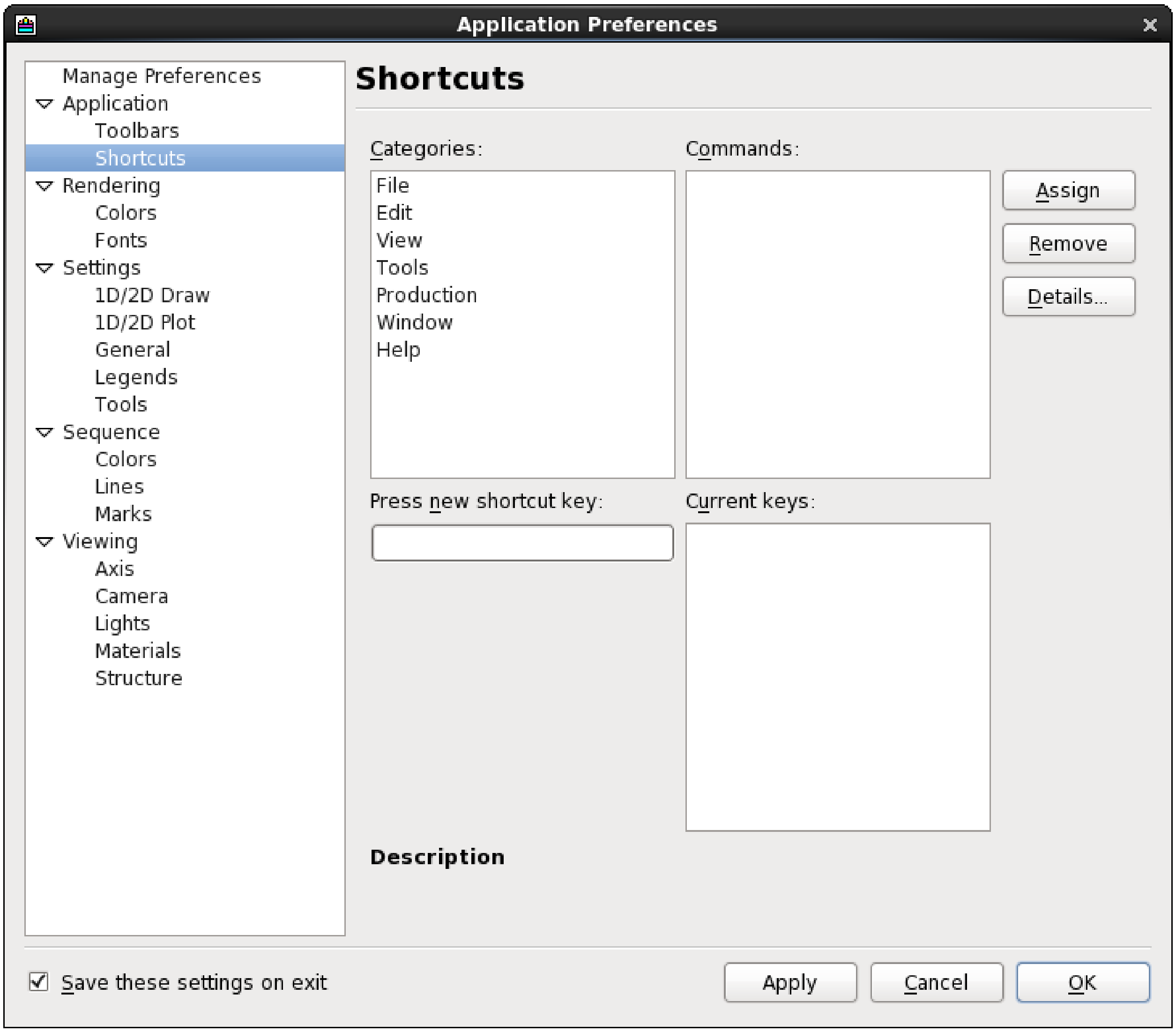


Figure 12-6 Shortcuts Settings

12.4 Rendering

12.4.1 Colors

This allows you to set the default colors for the foreground and background, as well as the various parts of the structure being displayed.

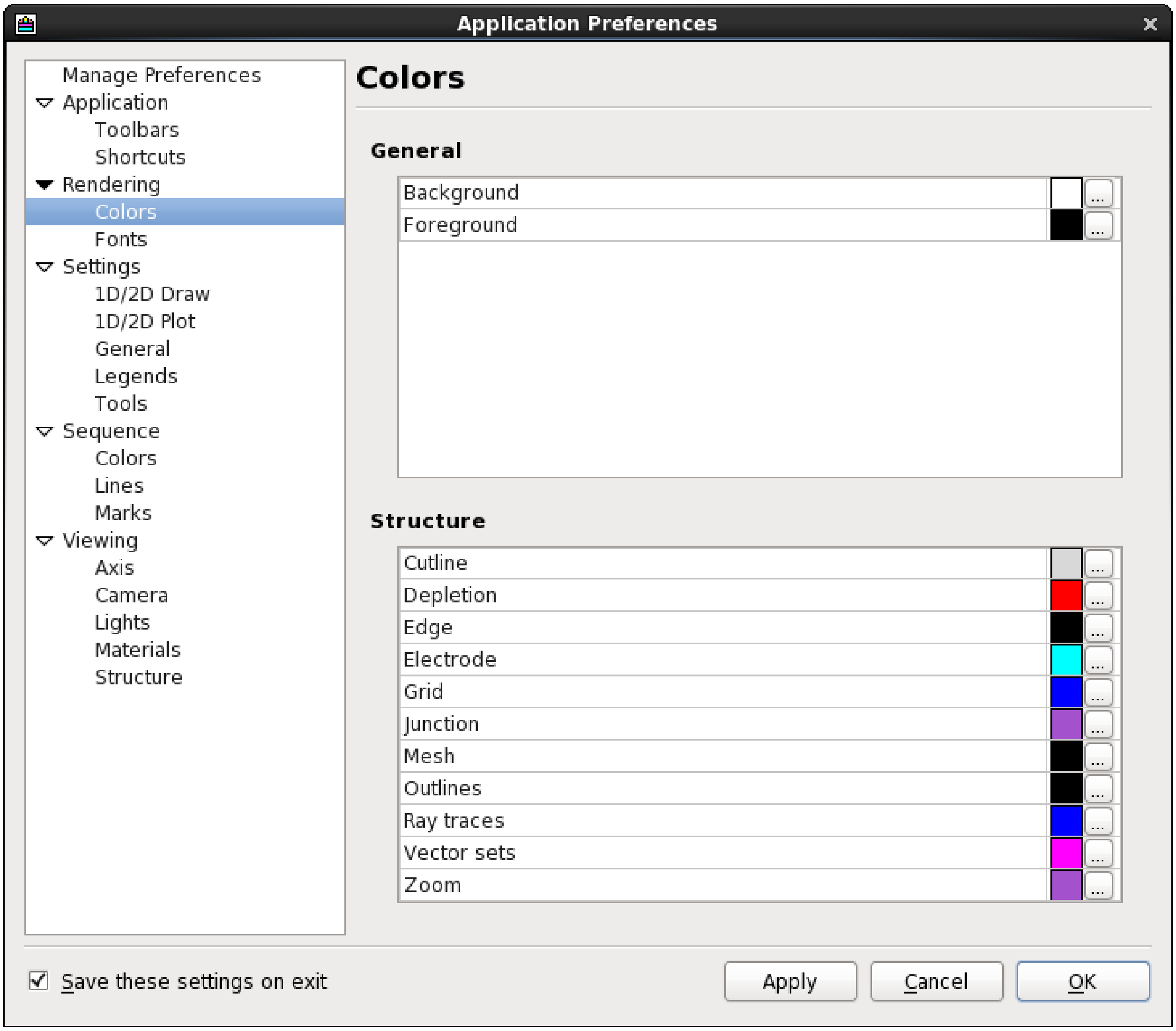


Figure 12-7 Rendering – Colors Settings

12.4.2 Fonts

This allows you to set the default font family used in TonyPlot.

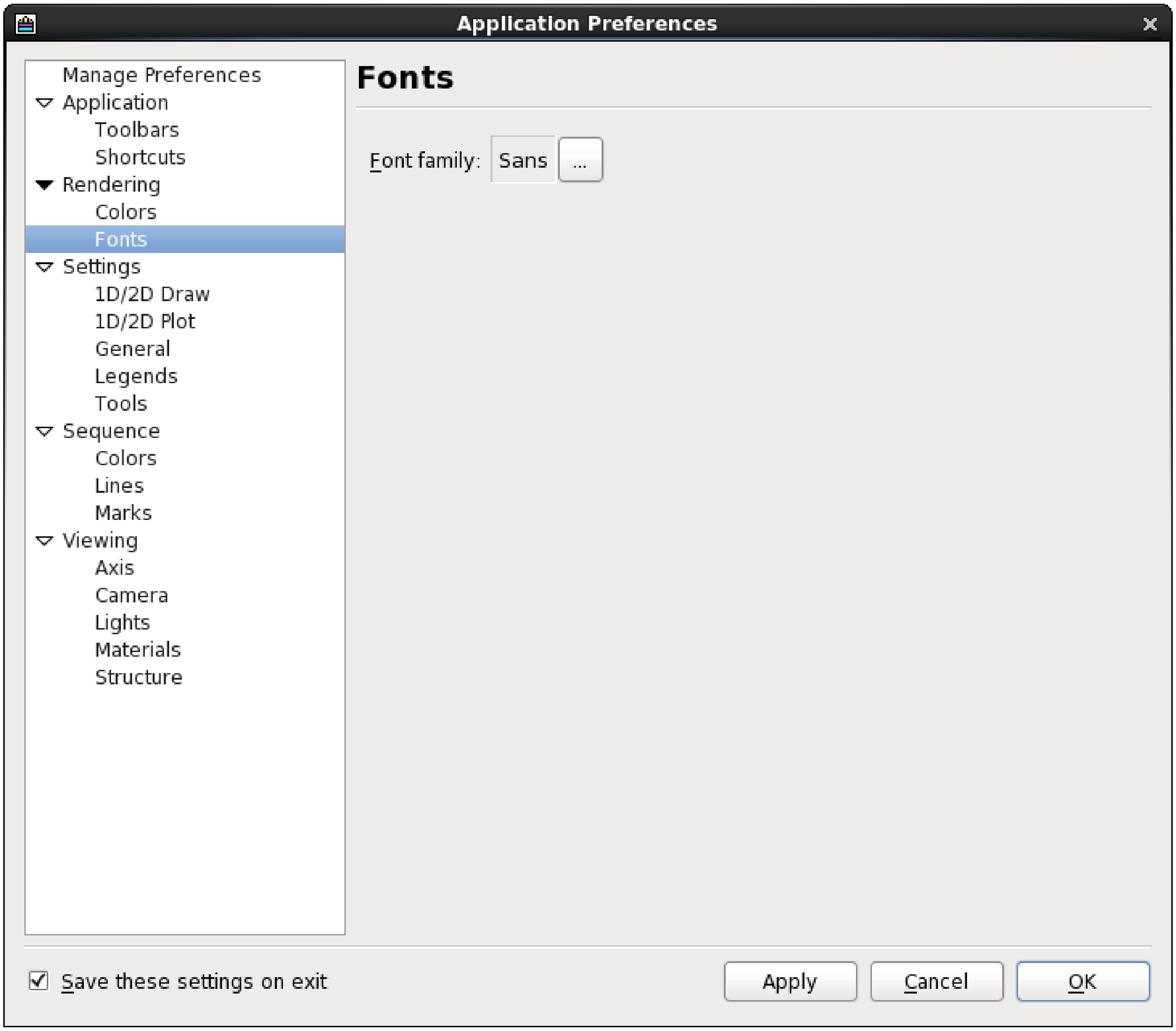


Figure 12-8 Rendering – Fonts Settings

12.5 Settings

12.5.1 1D/2D Draw

- **Grid level**: This controls whether the axis grid is drawn on top of the displayed structures or underneath them. The axis grid is controlled from the **Annotation** dialog.
- **Mesh level**: This controls whether the simulation mesh is drawn on top of the displayed structures or underneath them. The simulation mesh is controlled from the **Display** dialog for either **2D Mesh** plots or **Cross Section** plots.
- **Mark frequency**: For line plots, this controls the number of marks or points that are drawn along the curve. 1 draws a mark at every data point, while any other value draws marks at the specified frequency.
- **Log axis numbers**: When log axes are displayed on either **Cross Section** or **XY Graph** plots, the way numbers are drawn depending on this item. The number 3 is used as an example.
- **Linear axis numbers**: When linear axes are displayed, the way the numbers are drawn can be

controlled with this preference.

- **Line widths**: This group of controls sets the thickness of the lines used to draw **Graph Lines**, **Edges** (2D Mesh, Cross Section), **Vectors**, **Cutlines**, **Junctions**, and **Mask** lines. 1 represents normal thickness, with 2 to 4 representing increasingly thicker lines.
- **Electrodes Drawn**: When electrodes are drawn in mesh plots, TonyPlot indicates them by using a hashing in the defined color for electrodes. If this property is set to **With Names**, then the names of the electrodes (where defined) are also drawn at a point near to the actual electrode region.
- **Min Contour**: This determines whether the minimum contour is displayed as transparent.

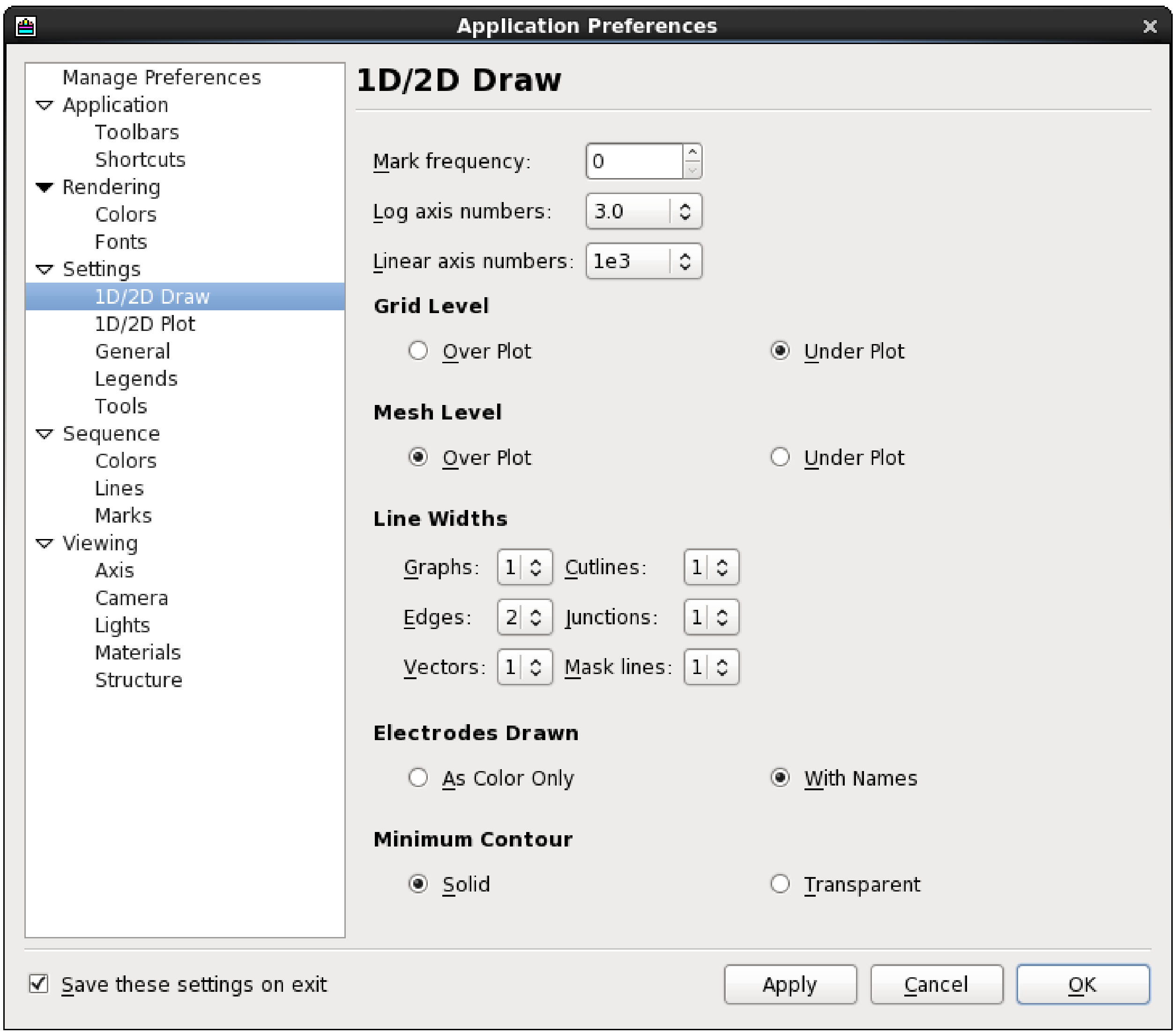


Figure 12-9 1D/2D Draw Settings

12.5.2 1D/2D Plot

- **XY plot ratio**: This controls the relative scaling in the X and Y directions of the data. If **To Window** is chosen the axes scale independently so that the whole window is occupied. If **To Data** is chosen, the axes scale together so that the aspect ratio of the data is maintained. In this mode, at least one of the axes span the subwindow.
- **Snap Mode**: This controls the snap mode when drag-zooming or using the ruler tool in TonyPlot. It is turned off by default. You can choose to have TonyPlot snap to points on a 2D Mesh, or to the edges of 2D structures.
- **Creating Overlays**: When creating overlays, this item either allows you to leave the original plots in place (choose **Leaves Originals**) or delete them (choose **Deletes Originals**). If you delete the originals, you can retrieve them by splitting the overlay. If they remain, splitting the overlay duplicates the originals.
- **Monitor Plots**: This allows you to have TonyPlot monitor loaded files for any changes. If the file changes, TonyPlot will reload it. You can change how often TonyPlot scans files for changes.
- **Anti-Alias Graphics**: This controls whether TonyPlot uses anti-aliasing when drawing graphics primitives.
- **Duplicate**: When a plot is duplicated, the display settings can be applied to the new plot. This causes the new plot to look identical to the original. If you select **Structure Only**, it will use a default display setting for the new plot.
- **Contour Color Sets:** Sets the default color palette used when contour plots are drawn.
- **Default Number Of Contour Colors:** Sets the default number of steps used when drawing contour type plots.

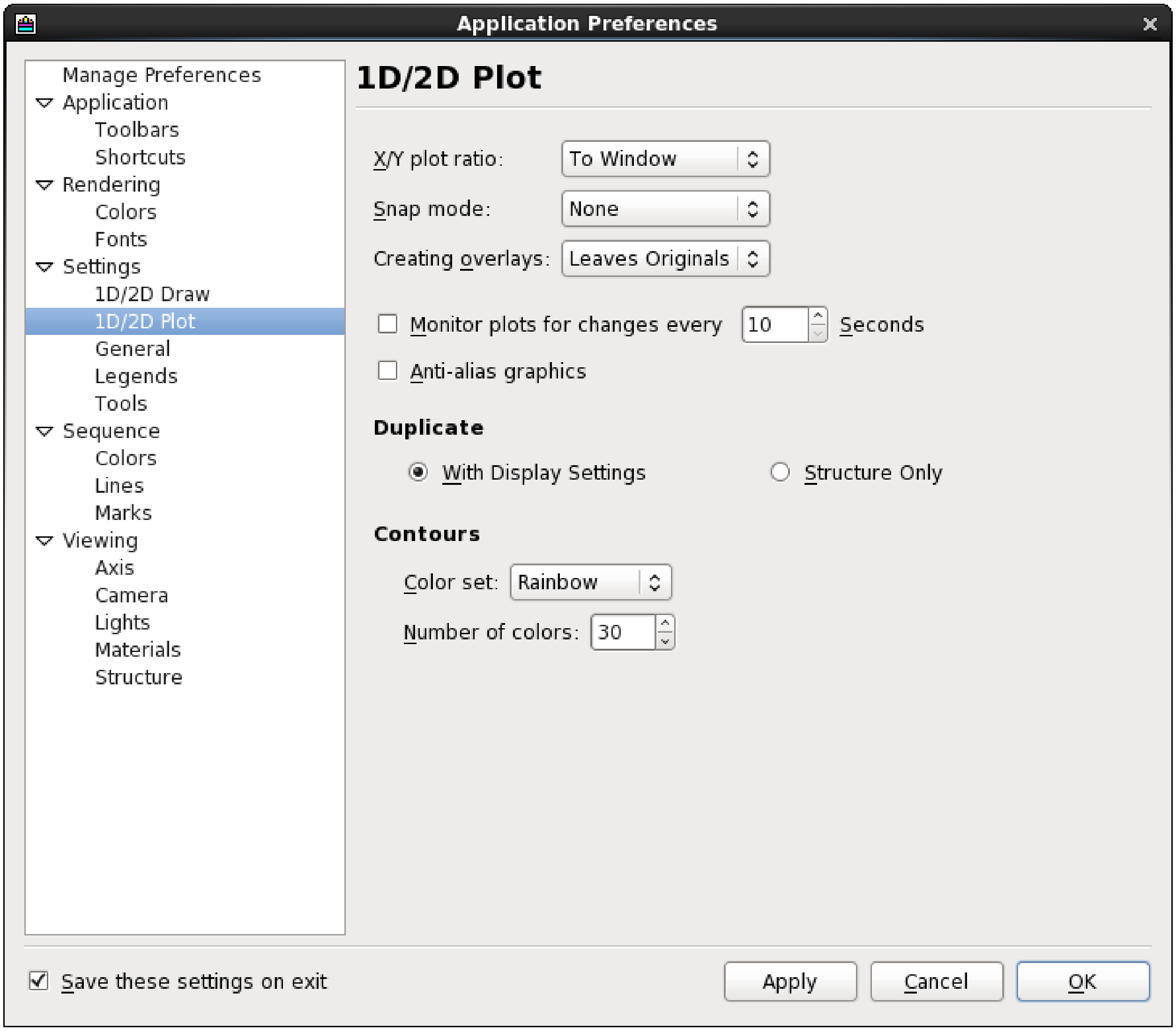


Figure 12-10 1D/2D Plot Settings

12.5.3 General

- **Load/Save Dialog**: If checked, TonyPlot will remember the last working directory used in load and save dialogs.
- **Quantity/Impurity Lists**: This allows you to display quantities in dialogs such as the Contours Dialog in alphabetical order or unsorted.
- **Minimum doping**: This either displays doping concentrations to their actual minimum values (choose **Actual**) or to a specified value (choose **Clip to**) with the value in the text field indicating the minimum value at which doping is clipped.
- **Log of zero**: When plotting results that involve the logarithm of zero (in any base), TonyPlot uses this predefined number as the result. **Zero** is used by default.

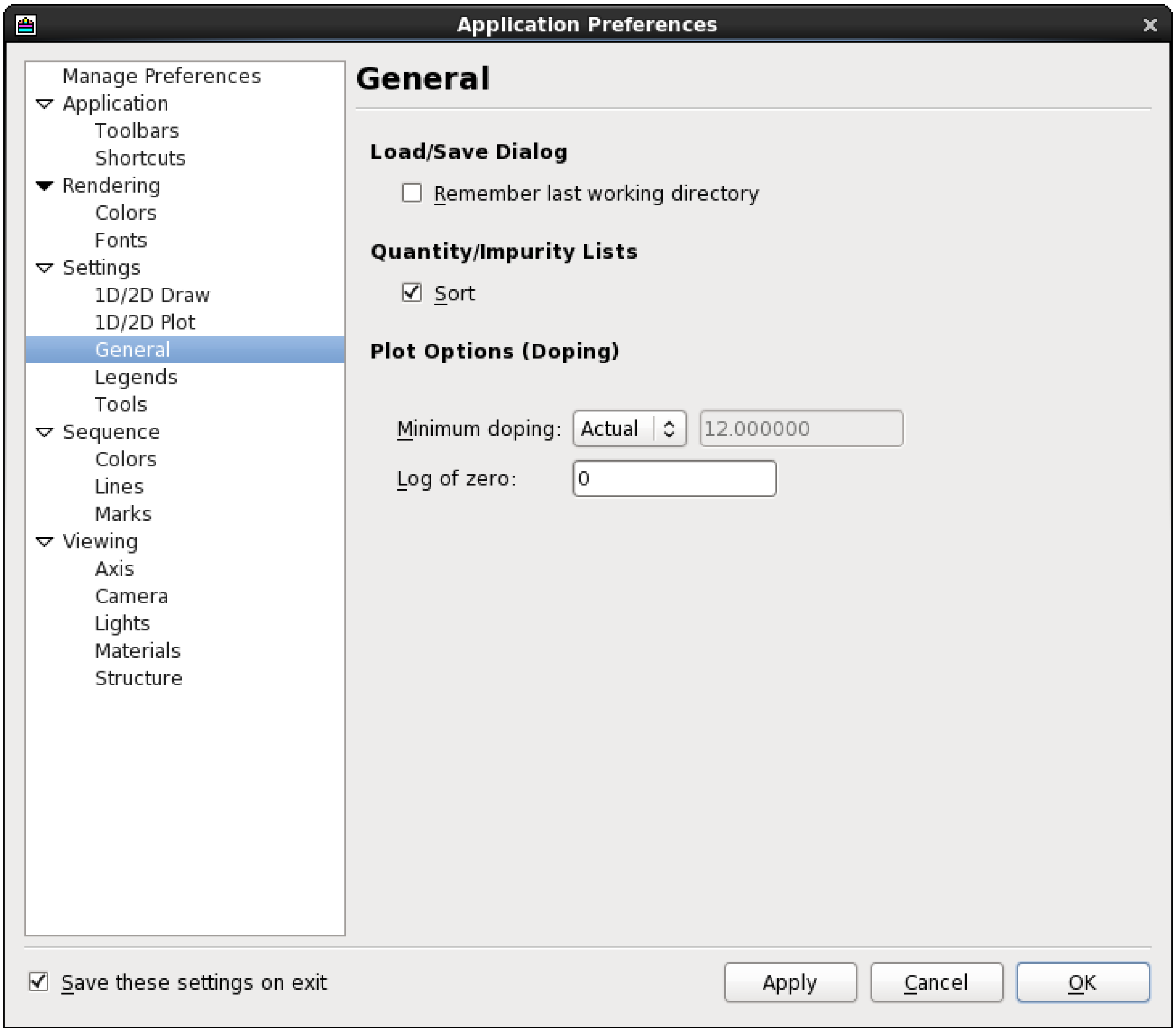


Figure 12-11 General Settings

12.5.4 Legends

Each legend position item provides eight options. If you select **Off**, then that legend will not be drawn. If you select any of the six specific positions, the legend will be drawn in that position in the plot. If the icon with the arrow is displayed, then the legend has been positioned by hand and is in none of the six standard locations. For more information, see Legends.

- **Contours**: There are three contour legends: one for each set. There are three items to control each one separately.
- **Regions**: This indicates the color used to distinguish each material region or region

parameter in 1D and 2D Mesh plots.

- **Graphs**: The Line legend for any graph plot, showing line colors and mark types.
- **Vectors**: The vector legend for 2D Mesh plots.
- **Levels**: The level legend for overlay plots.
- **Legend Type**: This sets the ways the legends are drawn in plots. **Transparent** (the default) allows the legend box to show the plot underneath. **Opaque** covers over any part of the plot under the legend.
- **Function Label**: This determines whether functions appear on legends as names (e.g., Function 1) or as their definitions.

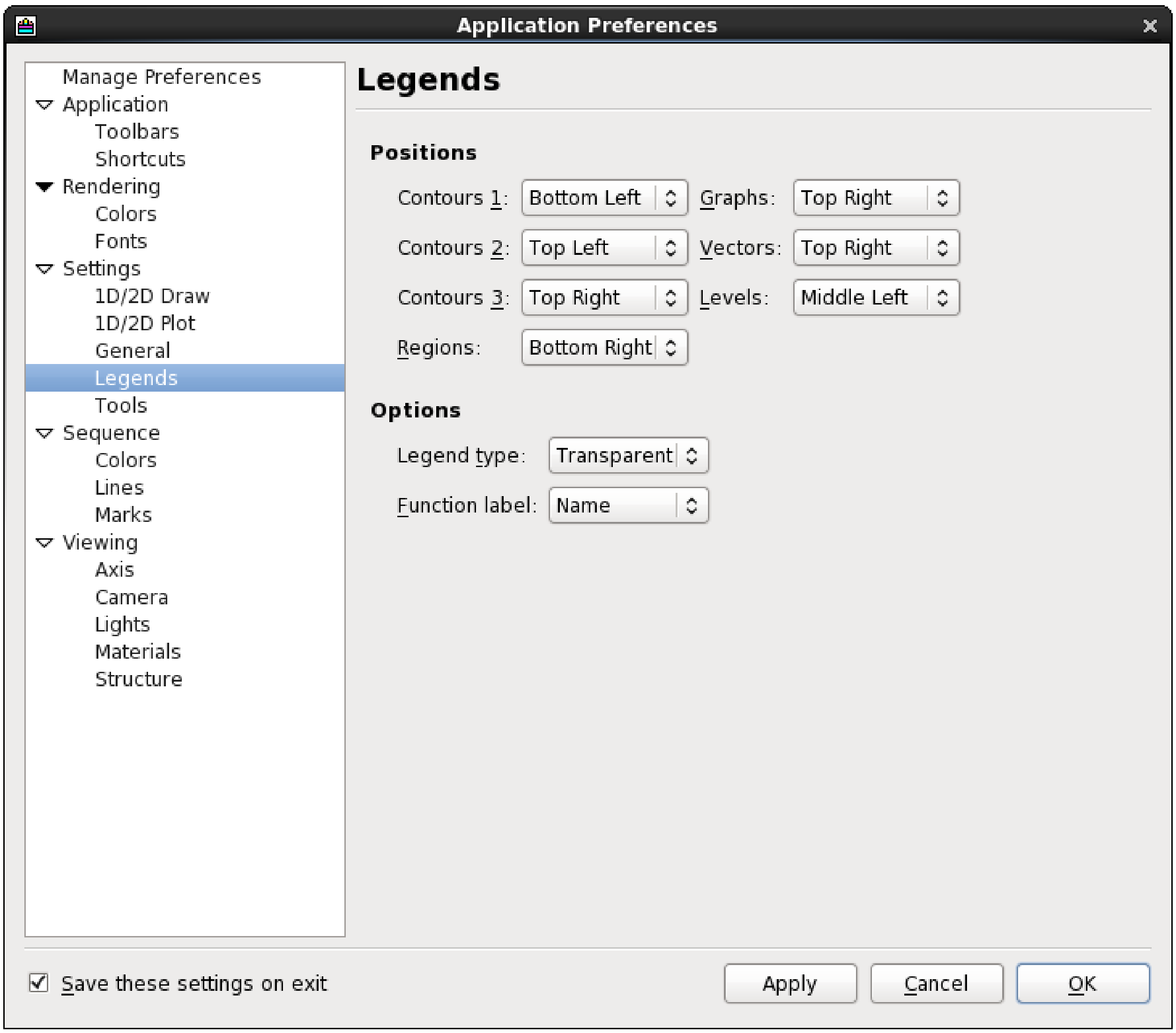


Figure 12-12 Legends Settings

12.5.5 Tools

- **Cutline X Axis**: There are three methods for marking the X axis on cutline profiles. By default, the X coordinate is relative to the start of the line, and so measures distance along the line. If you select **Absolute**, the axis shows the X coordinate of the original mesh (if the cutline is horizontal) or the Y coordinate (if the cutline is vertical). Cutlines that are neither horizontal nor vertical show a **Relative** X axis. If you select **Interface**, the X axis centers around the first material interface in the cross section. For example, the first interface is at X=0. If no interface exists, a **Relative** X axis will be drawn.
- **Probe Impurities**: This controls how impurity values reported by the **Probe** tool are shown. By default, true linear scale values are shown. Select **Log of value** to see the values on a log scale.
- **Panner Jump**: When panning by using the arrow keys, this controls the amount of the jump, given as a percentage of the screen width.
- **Pointer Readout Precision**: This controls the precision of the pointer readout in the status bar.

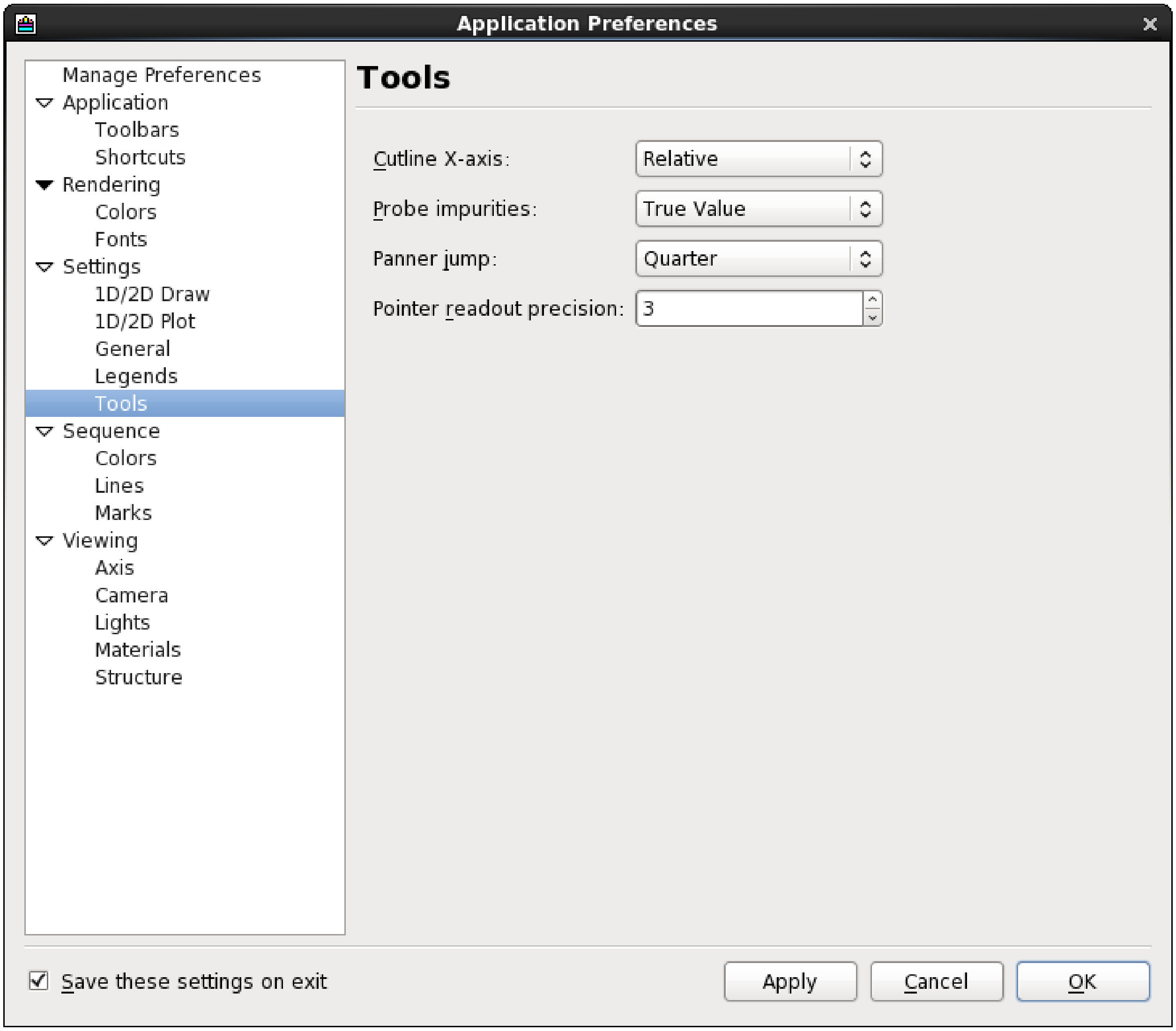


Figure 12-13 Tools Settings

12.6 Sequence

12.6.1 Colors

TonyPlot uses sequence colors whenever a group of items are plotted, and each needs its own color. Examples are lines on items, such as Graph and Cross Section plots, regions, and levels. The colors used are defined by the list of sequence colors indicated on this dialog. For example, if the first color is red, the level one is Red, the first cross section profile is Red, region 1 is Red, and so on. If you need more than twelve colors, the color sequence repeats at one.

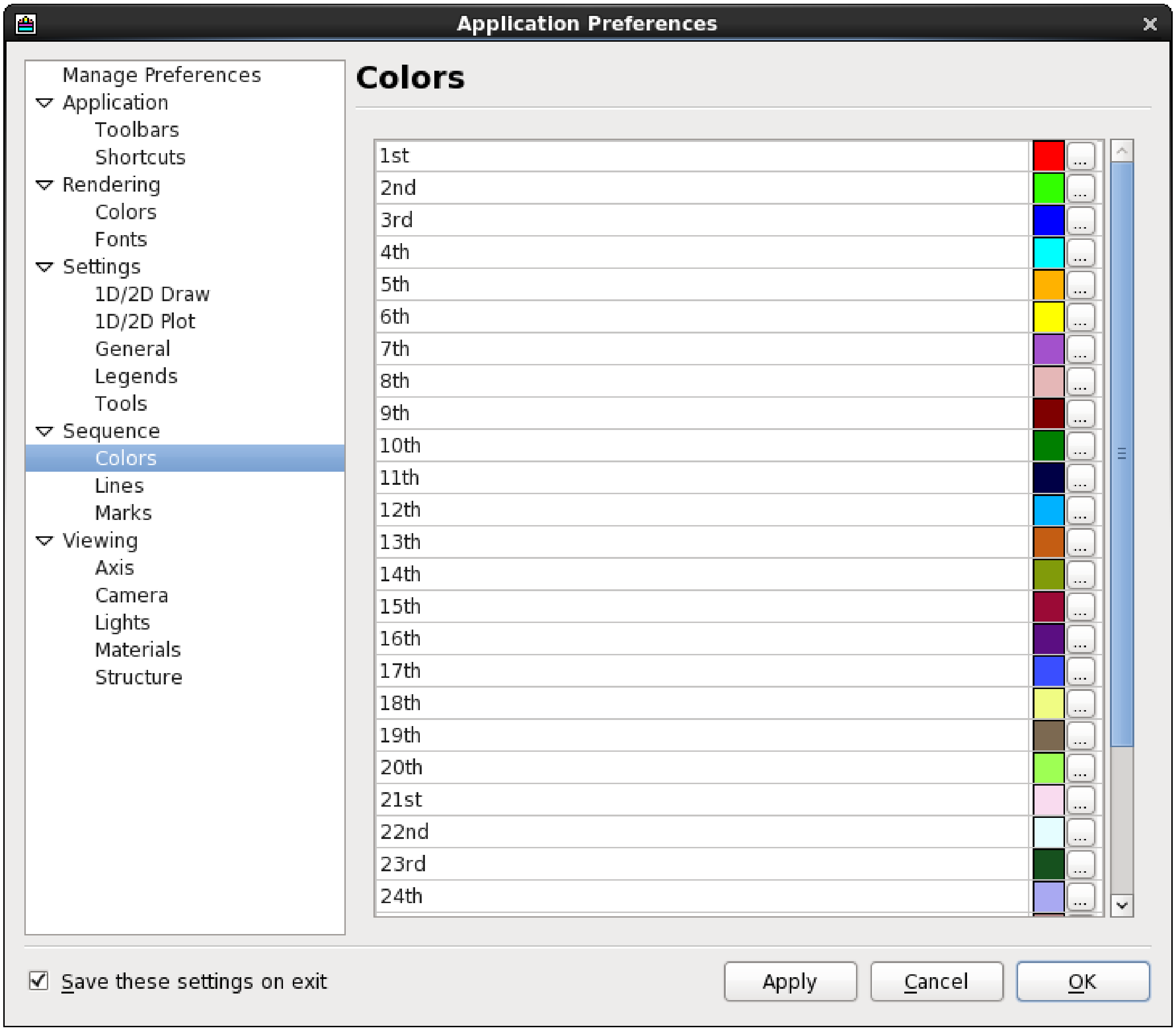


Figure 12-14 Sequence – Colors Settings

12.6.2 Lines

Graph lines are represented by **Lines** and **Marks**. TonyPlot allows you to change the line type of these graph lines. You can choose among **Solid**, **Dash**, **Dot**, **Dash Dot**, and **Dash Dot Dot**.

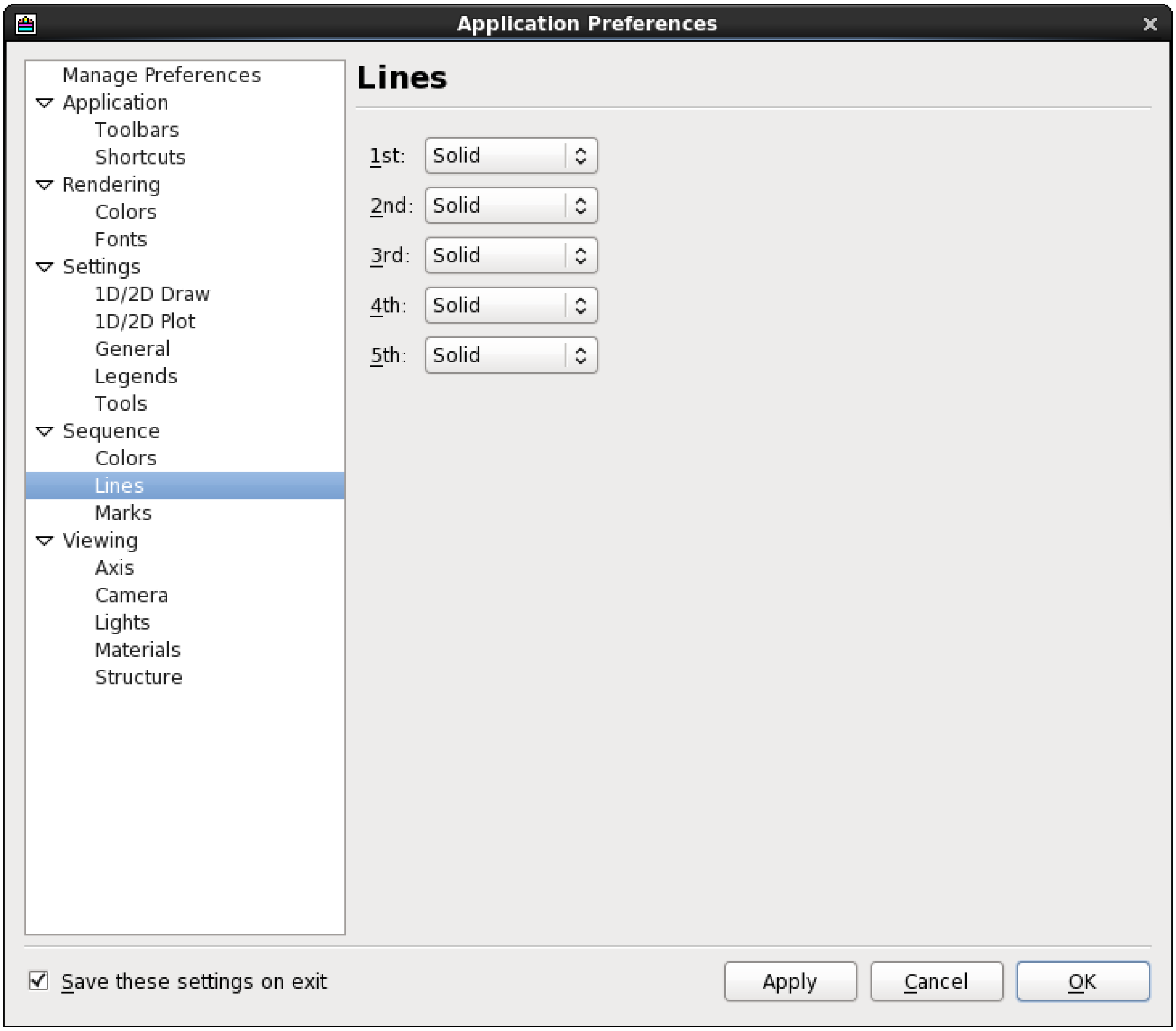


Figure 12-15 Sequence – Lines Settings

12.6.3 Marks

Graph lines are represented by Lines and Marks. TonyPlot uses different mark types to represent either quantities or levels. You can define these mark types in this dialog. The types are: **Cross**, **Circle**, **Plus**, **Triangle**, **Square**, and **Star**. This dialog also allows you to change the size and width of the marks.

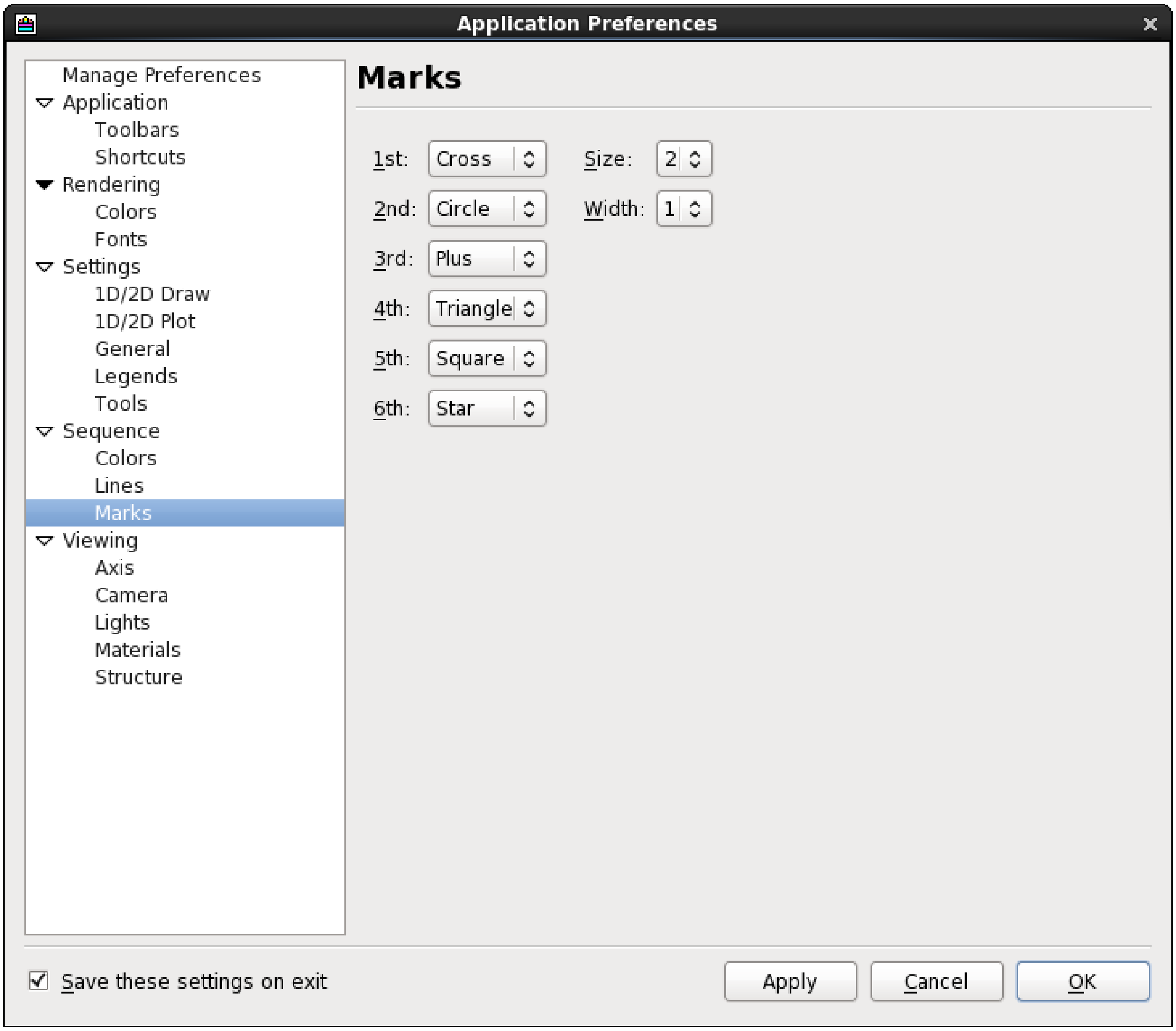


Figure 12-16 Sequence – Marks Settings

12.7 Viewing

12.7.1 Axis

- **Appearance**: Determines the initial visibility and 3D drawing mode of the axes. When the 3D Draw mode is set to Lines, the axes are initially drawn as thin lines. When it is set to Cylinders, the axes are drawn using three dimensional cylinders and cones. The initial appearance can be overridden for each plot by modifying the axis properties in the Object Browser. For more information, see Section 2.7.1 Object Browser.

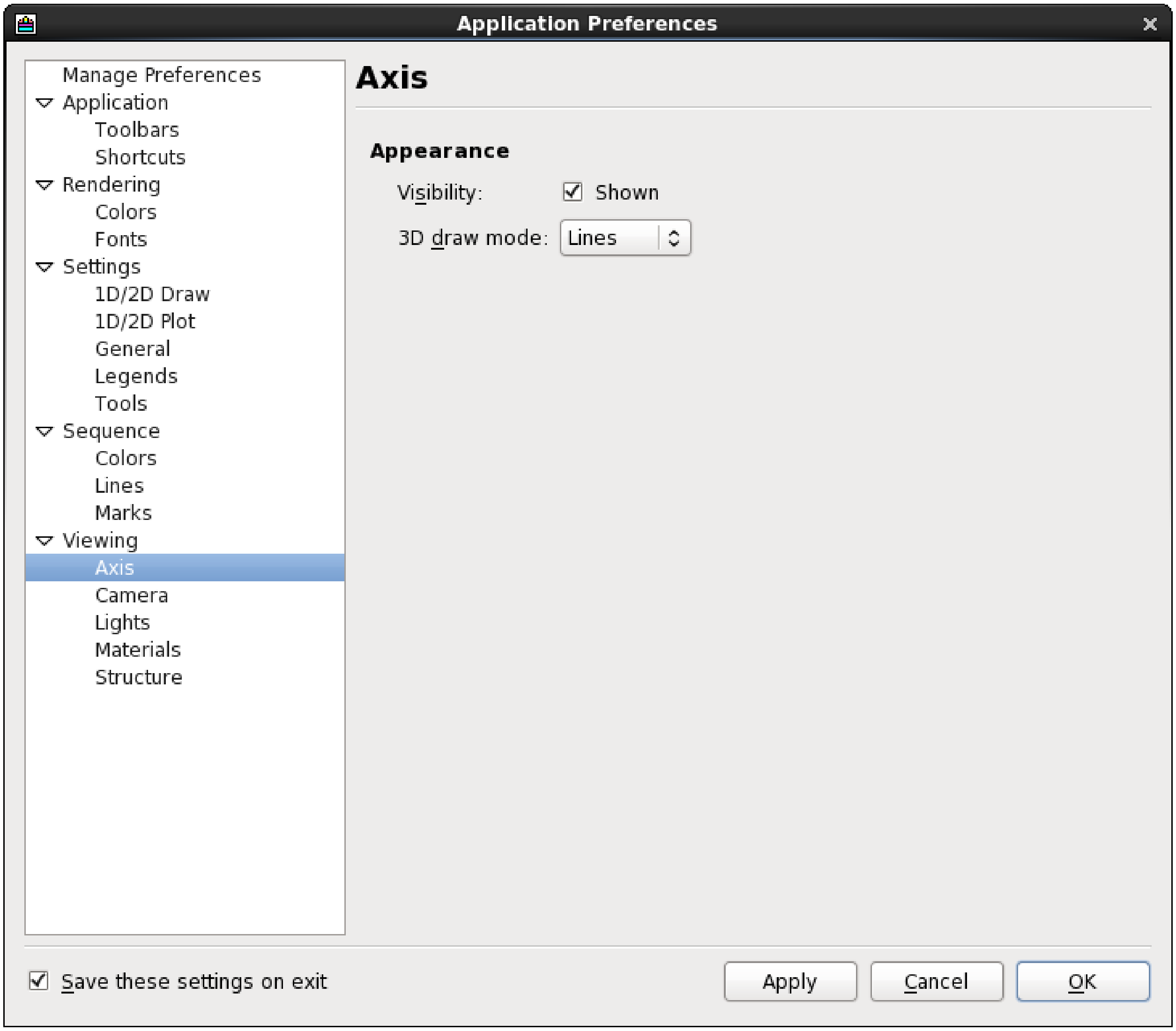


Figure 12-17 Axis Settings

12.7.2 Camera

- **Projection**: This option controls whether 3D structures are drawn using a perspective or orthographic view. The angle control sets the vertical field of view, measured in degrees.
- **Scaling**: Restricts scaling (zooming) to the selected objects axes.
- **Coordinate System**: When the coordinate system Selection is set to Auto, TonyPlot will determine the orientation of the coordinate system by examining the source of the structure data. This can be overridden by setting the Selection to User-Defined and picking either a Right-handed or Left-handed convention.
- **Depth Cue**: This creates the illusion of fog within a 3D scene. Distant objects blend into the fog more than near objects.

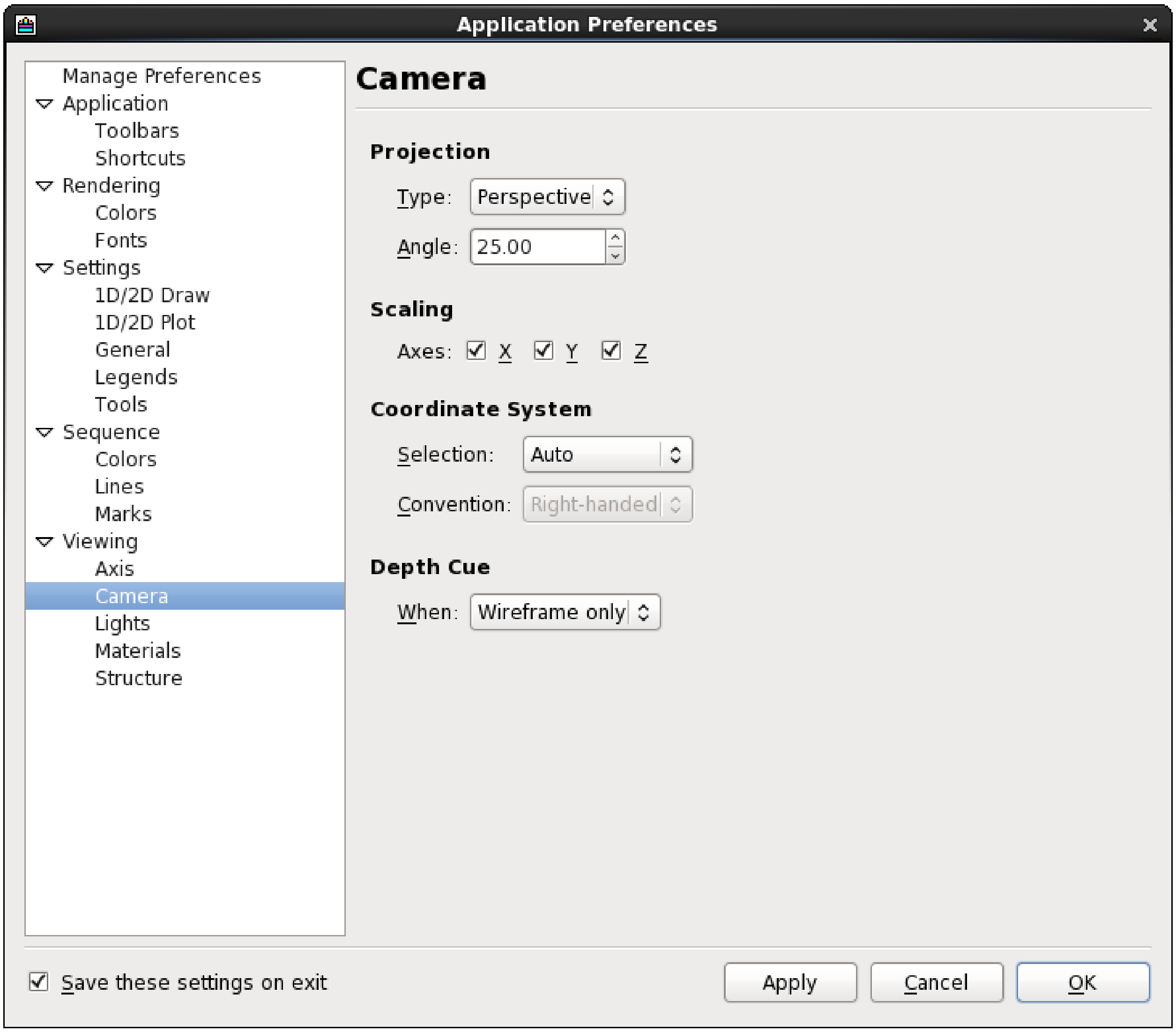


Figure 12-18 Camera Settings

12.7.3 Lights

TonyPlot uses two directional light sources to illuminate 3D scenes. Each light source has the following properties:

- **State:** This controls whether the light source is switched on or off.
- **Color:** Sets the color of the light source.
- **Intensity:** Sets the light source’s intensity (brightness)
- **Latitude:** Sets the latitudinal position of the light source in degrees. A latitude of 0 degrees represents a position on the equator, and 90 degrees represents the north pole.

- **Longitude:** Sets the longitudinal position of the light source in degrees. A longitude of 0 degrees represents a position closest to the observer, and 90 degrees represents the right-hand side of the display.

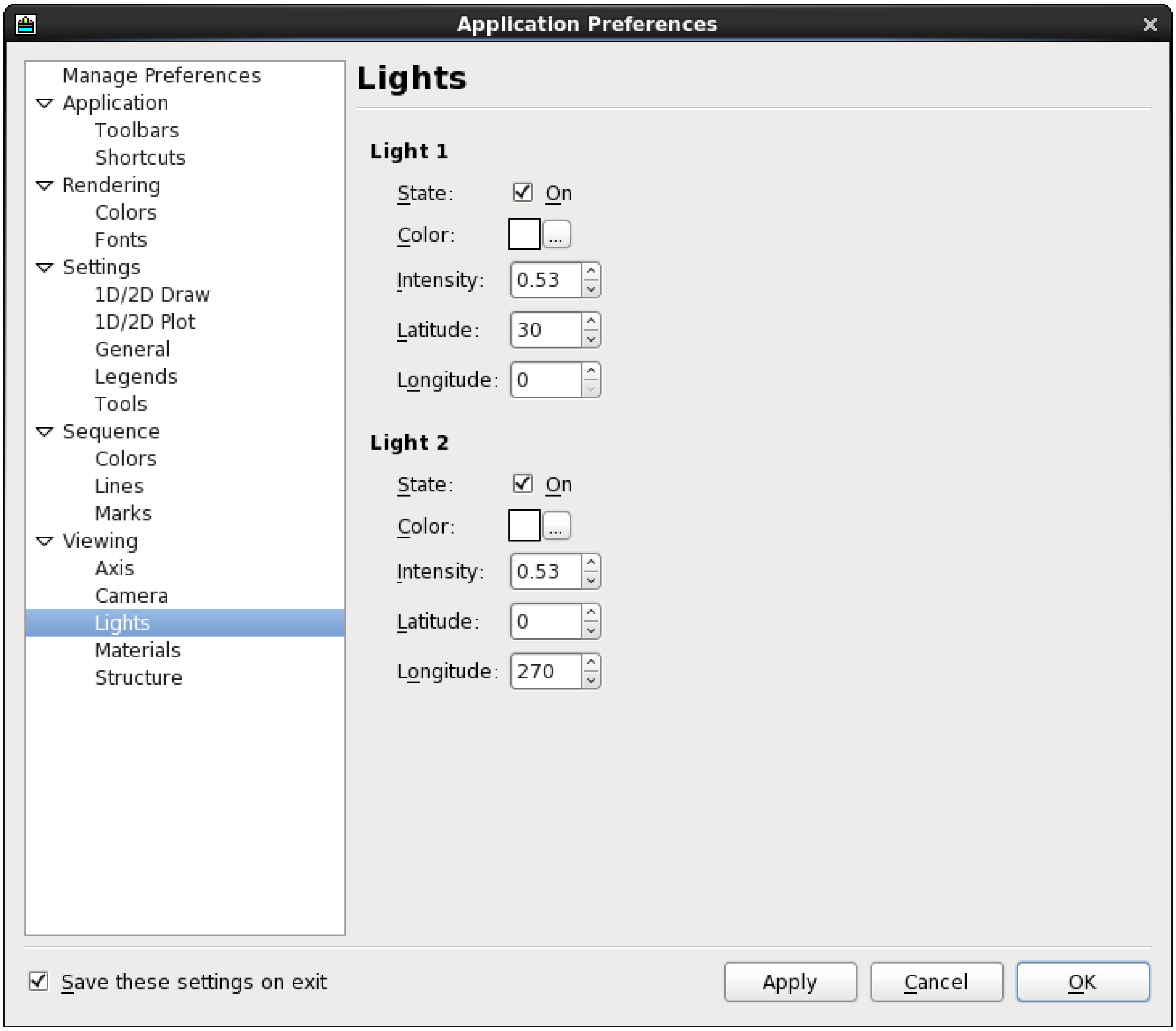


Figure 12-19 Lights Settings

12.7.4 Materials

This allows you to customize the colors of materials. You can also choose if the material is displayed on the materials legend or not.

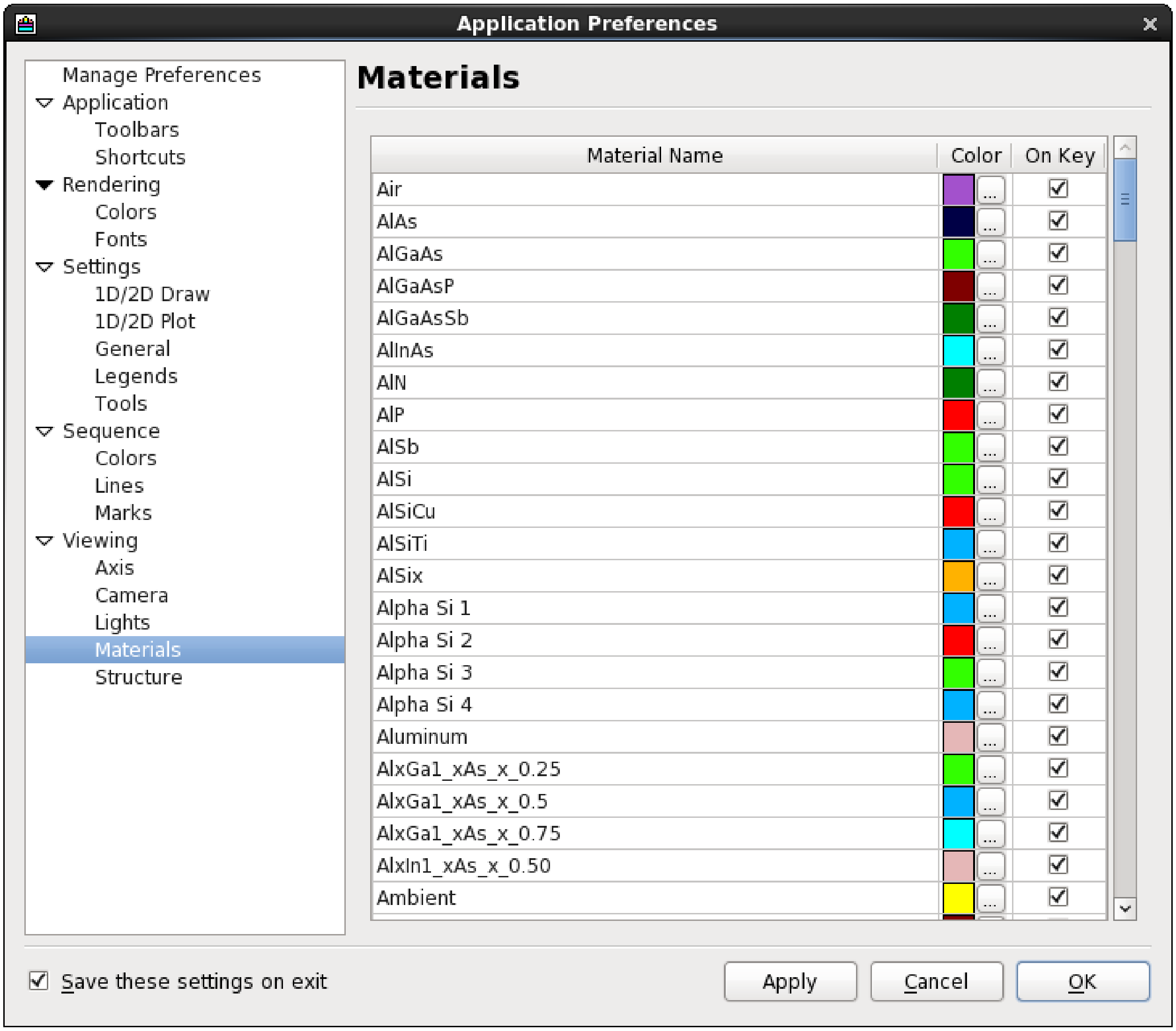


Figure 12-20 Materials Settings

12.7.5 Structure

- **Contour Outlines**: This sets whether or not stepped contours are drawn with an outline. If **Enabled** is checked, the stepped contours are drawn an outline thickness set by **Width**.
- **Materials**: **Show air** determines whether or not the material “air” is drawn on the plot. **Show gas** does the same for materials “gas” and “gas backside”.
- **Other Settings**: **Mesh width** sets the width (thickness) of lines drawn in Mesh views.

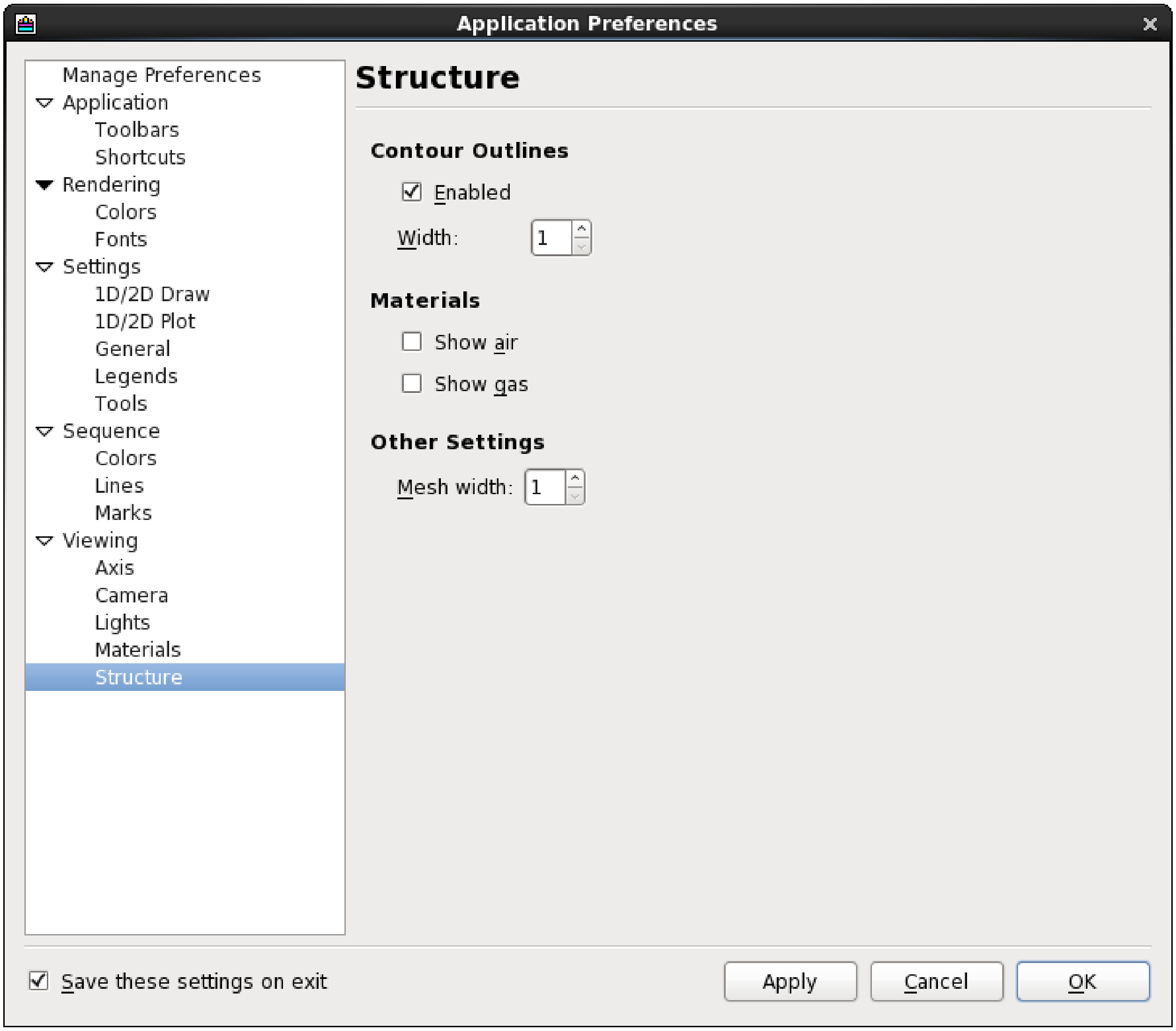


Figure 12-21 Structure Settings

13. Appendix A Data Files

13.1 User Data Files

User data files are ASCII text files that can be loaded into TonyPlot and have an easy-to-read format. This means that data from external sources can easily be read into TonyPlot. Those who have their own sets of data already, maybe from other software packages or from experimental results, find this format useful, since it allows you to apply the features of TonyPlot to that data. You can also compare data files to other files types. This makes it simple to compare simulation results from Silvaco simulators with data obtained from experiments.

13.1.1 Data Format

Data files should be constructed in the following manner.

```
<title line>
<r> <c> <t>
<title 1>
<title 2>
...
<title t>
<x11> <x12> ..... <x1c>
<x21> <x12> ..... <x2c>
<x31> <x12> ..... <x3c>
... ... ..... ...
<xr1> <xr2> ..... <xrc>
```

title line is a one line sequence of ASCII characters. No quotation marks should be used, but spaces are fine. This is used as the main plot title, and must be present.

r is the number of rows in the file. This must be present, but can be set to zero, in which case TonyPlot works out how many rows there are.

c is the number of columns. This must be present, but can be set to zero, in which case TonyPlot works out how many columns there are.

t is the number of column titles. If not present, it defaults to zero (i.e., no titles).

title X is the title that is assigned to the data in column number X. These titles are optional, but if given, the number of titles must match the `t` parameter in the second line.

xij is a data value, in row `i`, column `j`.

Each column in the file represents data values for a certain quantity. If no titles are given, these quantities assume default names `User data 01` for the first column, `User data 02` for the second, and so on. The titles allow you to add more meaningful names.

13.1.2 Examples

The following are examples of some user data files and descriptions of each. These can be entered into a text file and tested with TonyPlot, if needed.

13.1.2.1 Equation

This simple example plots a section of a curve of the equation `y=sin(x)`. This type of file is the simplest.

```
y = sin(x)
10 2
0.1 0.09983
0.2 0.19867
0.3 0.29552
0.4 0.38942
0.5 0.47943
0.6 0.56464
0.7 0.64422
0.8 0.71736
0.9 0.78333
1.0 0.84147
```

13.1.2.2 Transistor

This example contains information about an NMOS transistor. This example shows how columns are named.

```
1.0 um NMOS Id/Vg
10 2 2
gate bias (V)
drain current (A)
0.0 1.0E-14
0.1 1.0E-13
```

```
0.2 1.0E-12
0.3 1.0E-11
0.4 1.0E-10
0.5 1.0E-09
0.6 1.0E-08
0.7 1.0E-07
0.8 5.0E-07
0.9 8.0E-07
1.0 8.9E-07
```

13.1.3 Display

User data files are treated in the same way as normal **XY Graph** plots in TonyPlot. This display dialog for these plots is exactly the same as the Graph display dialog. In fact, once loaded into TonyPlot, there is no difference between these two types of data at all.

13.2 Set Files

When a plot has been set up to display the desired set of information (e.g., using the display dialogs, labels, annotation), it is possible to save this information into a .setx file so that the display can be recreated automatically by TonyPlot. A .setx file contains XML-based instructions that tell TonyPlot the steps needed to recreate the same display that was visible at the time the set file was created. Therefore, when you load the same data file into TonyPlot later, loading the .setx file avoids having to go through all the dialogs again.

Note: TonyPlot version 5 and later uses the newer .setx XML file format. You are still able to read the older, legacy .set files, but you can only save to .setx. In addition, you now specify one .setx file per plot.

13.2.1 Creating

To create a .setx file, choose **File→Save Set File…**. Move to the directory where the .setx file is to be created, and enter the name of the desired file into the field marked **File name**. If the file already exists, select it from the scrolling list (use the **Filter** to screen out undesired files from the list). When you click on the **Save** button, the .setx file will be created. Confirmation is required if the file is overwritten.

13.2.2 Loading

To load a previously created .set or .setx file, locate the file by select the **File→Load Set File…**. Then, click on the **Open** button and the view will update. You can also load .set or .setx files from the command line with the `-set` option. If an error occurs when loading the file, a warning notice appears.

Set files cannot record any actions you performed to create overlay plots, although they can store setup information about the overlay itself.

Cross section plots that are generated using the **Cutline** tool can be created from the original 2D Mesh plot using .setx files. Therefore, you can save a view with a mesh and a cross section and recreate it from the mesh alone. For example, suppose you load a mesh file called `pmos.str`, creates a cutline cross section, and then save a set file called `cut.setx`. You can then recreate the view with:

```
% tonyplot pmos.str -set cut.setx
```

The set file automatically takes a cutline from the PMOS mesh and display the cross section plot as it appeared when the set file was created.

If you have two separate plots, each with its own saved .setx file, you would load it like this:

```
% tonyplot struct1.str -set struct1.setx struct2.str -set
struct2.setx
```

13.3 TonyPlot Spreadsheet Files

TonyPlot spreadsheet files are ASCII text files that can be loaded into TonyPlot and have an easy-to-read, comma-separated format. This means that data from external sources can easily be read into TonyPlot. You can also compare data files to other files types. This makes it simple to compare simulation results from Silvaco simulators with data obtained from external sources such as a spreadsheet.

13.3.1 Loading TonyPlot spreadsheet Files

TonyPlot automatically recognizes data files in this format. Therefore, it is unnecessary to use any special command line options to load them.

13.3.2 Data Format

Data files should be constructed in the following manner.

13.3.2.1 2D File

```
#spreadsheet
#columns,<second axis>
<columns num>,<chv1>,...<chvc>
<first axis>,<output>
<x11>,<x12>, ..... <x1c>
<x21>,<x22>, ..... <x2c>
<x31>,<x32>, ..... <x3c>
... ... ..... ...
<xr1>,<xr2>, ..... <xrc>
```

#spreadsheet is a required line, and tells tonyplot the type of information that follows.

#columns is required on the second line for 2D file types. The remainder of this line denotes the second axis name.

columns num is the number of columns. This must be present. The values that follow on this line are the column header values.

xij is a data value, in row `i`, column `j`.

13.3.2.2 1D File

```
#spreadsheet
<first axis>,<output>
<x11>,<x12>
<x21>,<x22>
<x31>,<x32>
... ...
<xr1>,<xr2>
```

#spreadsheet is a required line, and tells tonyplot the type of information that follows.

first axis is required on the second line for 1D file types, and denotes the first axis name.

output denotes the output name.

xij is a data value, in row `i`, column `j`.

13.3.3 Examples

The following are examples of some TonyPlot spreadsheet data files. These can be entered into a text file and tested with TonyPlot, if needed.

13.3.3.1 1D Example

```
#spreadsheet
#time, temperature
0,            1000
0.5,          1100
1.0,          1200
```

13.3.3.2 2D Example

```
#spreadsheet
#columns, depth
3,    0,       0.1,     0.2
time, temperature
0,    1000,    1100,    900
0.5,  1100,    1200,    1000
1.0,  1200,    1300,    1100
```

14. Appendix B Models and Algorithms

14.1 Introduction

Models and Algorithms used by one dimensional (1D) electrical solvers in DeckBuild and TonyPlot.

Note: This appendix is intended to serve as a quick reference only. A detailed description of the semiconductor device physical models is provided in the Atlas User's Manual.

1D electrical solvers, available by using the `extract` command in DeckBuild or in TonyPlot, are based on the iterative solution of the Poisson equation:

$$div(\varepsilon\nabla\Psi) = q(p - n + N_D^+ - N_A^-) - \rho_F \quad \textbf{14-1}$$

where ψ is the potential, ε is the dielectrical permittivity, n and p are the electron and hole concentrations, and ρ_F is the fixed charge.

QUICKBIP uses the continuity equations to calculate n and p:

$$\frac{1}{q} div\,\overrightarrow{J_n} - U_n = 0 \quad \textbf{14-2}$$

$$\frac{1}{q} div\overrightarrow{J_p} - U_p = 0 \quad \textbf{14-3}$$

Where:

$$\overrightarrow{J_n} = q\mu_n\overrightarrow{E_n} \cdot n + qD_n\nabla n \quad \textbf{14-4}$$

$$\overrightarrow{J_p} = q\mu_p\overrightarrow{E_p} \cdot p + qD_p\nabla p \quad \textbf{14-5}$$

$$D_n = \frac{kT}{q}\mu_n\,, D_p = \frac{kT}{q}\mu_p \quad \textbf{14-6}$$

14.1.1 Physical Models

All electrical solvers take into account the following models and effects:

- Temperature dependence, such as kT/q or Eg
- Concentration-dependent mobility (with built-in temperature dependence)
- Field-dependent mobility (perpendicular field with built-in temperature dependence)
- Material work function (for MOS structures)
- Fixed interface charge

14.2 Concentration Dependent Mobility

The concentration dependent mobilities for *n* and *p* respectively are:

$$\mu_n^D = \mu_{nmin} + \frac{\Delta\mu_n}{1 + N_{total}/N_{nref}} \quad \textbf{14-7}$$

$$\mu_p^D = \mu_{pmin} + \frac{\Delta\mu_p}{1 + N_{total}/N_{pref}} \quad \textbf{14-8}$$

Where:

$$\mu_{nmin} = 88 \cdot \left(\frac{Y}{300}\right)^{-0.57} \quad \textbf{14-9}$$

$$\mu_{pmin} = 54.3 \cdot \left(\frac{Y}{300}\right)^{-0.57} \quad \textbf{14-10}$$

$$\Delta\mu_n = 1252 \cdot \left(\frac{Y}{300}\right)^{-2.33} \quad \textbf{14-11}$$

$$\Delta\mu_p = 407 \cdot \left(\frac{Y}{300}\right)^{-2.33} \quad \textbf{14-12}$$

$$N_{nref} = 1.432 \cdot 10^{17} \left(\frac{Y}{300}\right)^{2.456} \quad \textbf{14-13}$$

$$N_{pref} = 2.67 \cdot 10^{17} \left(\frac{Y}{300}\right)^{2.456} \quad \textbf{14-14}$$

14.3 Field Dependent Mobility Model

The field dependent mobilities for *n* and *p* respectively are:

$$\mu_n = \frac{\mu_n^D}{\sqrt{1 + 1.54 \cdot 10^{-5} \cdot E}} \tag{14-15}$$

$$\mu_p = \frac{\mu_p^D}{\sqrt{1 + 5.35 \cdot 10^{-5} \cdot E}} \tag{14-16}$$

14.4 Sheet Resistance Calculation

After solving the Poisson equation, the sheet resistance for each semiconductor layer is estimated using:

$$R_{sh} = \int_{xleft}^{xright} \frac{dx}{q\mu_n \cdot n + q\mu_p \cdot p} \qquad \textbf{14-17}$$

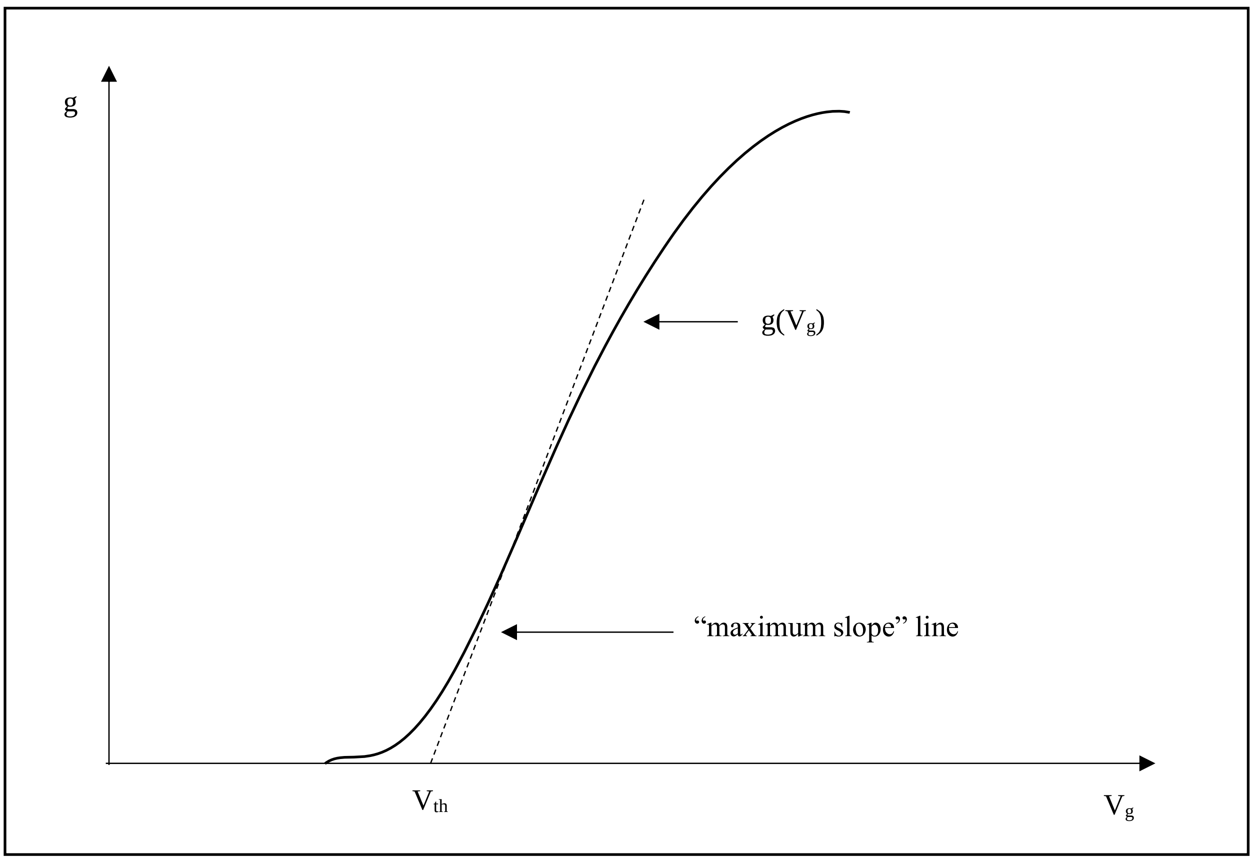
xleft and *xright* are determined by the *p*-*n* junction locations and the semiconductor material boundaries.

14.5 Threshold Voltage Calculation

Threshold voltage calculation is based on the calculated sheet resistance. In MOS mode (1D vt extraction), the solver will calculate threshold voltage automatically. First, the conductance of the channel region will be calculated for each gate voltage applied. If an NMOSFET structure is assumed, then:

$$g(V_g) = \int_{O}^{xinv} {(q\mu_n)}/{ndx} \qquad \textbf{14-18}$$

O corresponds to the oxide-silicon interface and *xinv* is the boundary of the inversion layer. Threshold voltage will be determined using the $g(V_g)$ curve as an intersection with the V_g axis of the straight line drawn through two points on the $g(V_g)$ curve, corresponding to the maximum slope region shown below.



14.6 Breakdown Voltage Calculation

Breakdown voltage calculation is based on estimation of ionization integrals for electrons and holes. Breakdown is determined by the condition that one of the integrals is greater than 1. The ionization rates are calculated using the following equations (See the Selberherr model in the Atlas User's Manual):

$$\alpha_n = AN \cdot exp\left[-\left(\frac{BN}{E}\right)^{BETAN}\right] \quad \textbf{14-19}$$

$$\alpha_p = AP \cdot exp\left[-\left(\frac{BP}{E}\right)^{BETAP}\right] \quad \textbf{14-20}$$

where:

```
AN = AN1 if E < EGRAN        AN = AN2 if E > EGRAN
AP = AP1 if E < EGRAN        AP = AP2 if E > EGRAN
BN = BN1 if E < EGRAN        BN = BN2 if E > EGRAN
BP = BP1 if E < EGRAN        BP = BP2 if E > EGRAN
```

The values of the parameters `AN1`, `AN2`, `AP1`, `AP2`, `BN1`, `BN2`, `BP1`, `BP2`, `BETAN`, `BETAP`, `EGRAN` are user-definable (through the `extract` command or pop-up menu). Their default values are:

AN1=7.03e5 cm^{-1}

AN2=7.03e5 cm^{-1}

BN1=1.231e6 V/cm

BN2=1.231e6 V/cm

AP1=6.71e5 cm^{-1}

AP2=1.582e6 cm^{-1}

BP1=1.693e6 V/cm

BP2=2.036e6 V/cm

BETAN=1.0 (unitless)

BETAP=1.0 (unitless)

EGRAN=4e5 V/cm

15. Appendix C Greek and Math Characters

Table 15-1 Greek and Math Characters

Character	Tag	Character	Tag	Character	Tag
α	<alpha>	Δ	<Delta>	∃	<exist>
β	<beta>	Ε	<Epsilon>	∅	<empty>
γ	<gamma>	Ζ	<Zeta>	∇	<nabla>
δ	<delta>	Η	<Eta>	∈	<isin>
ε	<epsilon>	Θ	<Theta>	∉	<notin>
ζ	<zeta>	Ι	<Iota>	∏	<prod>
η	<eta>	Κ	<Kappa>	∑	<sum>
θ	<theta>	Λ	<Lambda>	√	<radic>
ι	<iota>	Μ	<Mu>	∞	<infin>
κ	<kappa>	Ν	<Nu>	∧	<and>
λ	<lambda>	Ξ	<Xi>	∨	<or>
μ	<mu>	Ο	<Omicron>	∩	<cap>
ν	<nu>	Π	<Pi>	∪	<cup>
ξ	<xi>	Ρ	<Rho>	∫	<int>
ο	<omicron>	Σ	<Sigma>	≈	<asymp>
π	<pi>	Τ	<Tau>	≠	<ne>
ρ	<rho>	Υ	<Upsilon>	≡	<equiv>
σ	<sigma>	Φ	<Phi>	≤	<le>
τ	<tau>	Χ	<Chi>	≥	<ge>
υ	<upsilon>	Ψ	<Psi>	⊂	<subset>
φ	<phi>	Ω	<Omega>	⊃	<supset>
χ	<chi>	Å	<Angstrom>	°	<deg>
	<psi>	¬	<not>	×	<times>

Character	Tag	Character	Tag	Character	Tag
ω	<omega>	±	<plusmn>	⌊	<lfloor>
å	<angstrom>	·	<middot>	⌋	<rfloor>
Α	<Alpha>	→	<rarr>	⌈	<lceil>
Β	<Beta>	∀	<forall>	⌉	<rceil>
Γ	<Gamma>	∂	<part>		

16. Index