SILVACO

GUIDE TO USING
TCAD WITH EXAMPLES

(LINUX AND UNIX VERSION)

Table of Contents

Chapter 1:
Introduction

This manual is intended for the first time user of SILVACO TCAD products. Its intention is to enable a new user to have the software up and running within a few minutes of successful installation.

This tutorial will also demonstrate how to navigate the manuals quickly and efficiently to locate explanations and definitions of all parameters used in the simulators. It will cross reference the user to the relevant physics chapters to understand the equations and underlying physics in which these equations are used.

For further reading and references, the reader will be directed to a wealth of technical material and published paper references located in the technical sections of the SILVACO web pages.

Chapter 2:

Getting Started With Just a Few Clicks of the Mouse

Section 2.1: The DeckBuild Runtime Environment Window

The key to rapid familiarization with SILVACO's TCAD software is the versatile and feature laden runtime environment called "DeckBuild." DeckBuild features include: automatic creation of process input files, editing of existing input files, creation of Design Of Experiments (DOE), powerful parameter extraction routines and the ability to make anything in the input file a variable.

Most importantly for the first time user however, DeckBuild includes literally hundreds of examples for many different types of electrical, optical and magnetic technologies.

Getting Started...

Open a console window and create a directory where you want the temporary files that will be created in this tutorial to reside. For example, to create and relocate to a directory called "tutorial," in the console window type:

> mkdir tutorial <return>
> cd tutorial <return>

Now invoke the DeckBuild runtime environment by typing:

> deckbuild <return>

You should see the DeckBuild runtime environment appear on your monitor and it should look very similar to that shown in Figure 2.1.

The GUI interface is arranged in two parts. The upper window displays the current input file and the lower window displays the output created when running that input file.
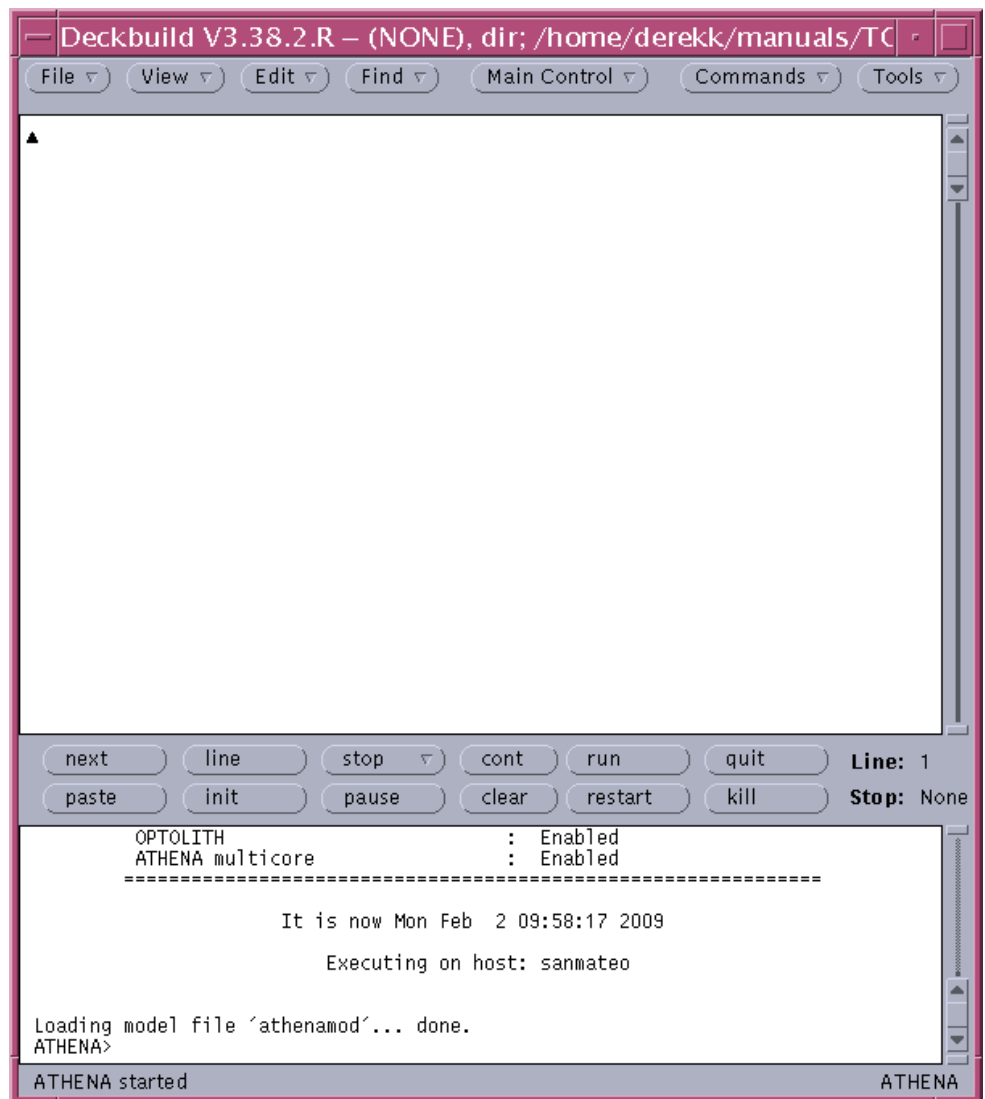

Figure 2.1 The DeckBuild Runtime Interface GUI

Section 2.2: Loading and Running Examples

The input file can be created by the user or can be loaded from the examples library. Familiarization with the software syntax is best executed by loading an example in the first instance. To load an example from the DeckBuild runtime environment GUI, click:

Main Control... Examples...

A pop up window displays a list of 47 different categories of examples. The first 15 categories are displayed as shown in Figure 2.2. To view the remaining categories, simply scroll down using the slider on the right hand side of the window.
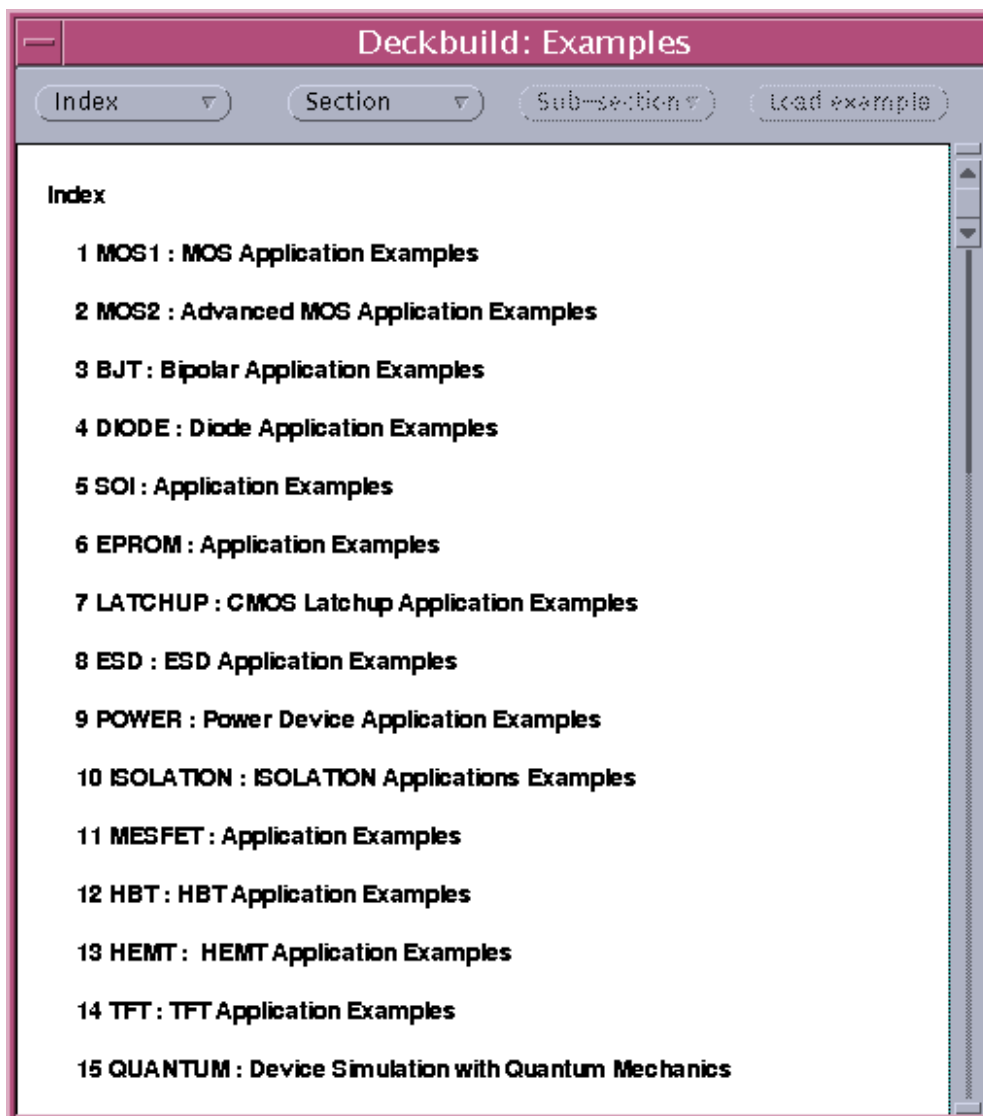

Figure 2.2 The First 15 DeckBuild Example Categories

To view a listing of the examples in each category, double-click one of the categories. If you double-click the first category entitled "MOS1: MOS Application Examples," you will get the listing shown in Figure 2.3. There are 15 examples in this category.

Figure 2.3 MOS Application Example List
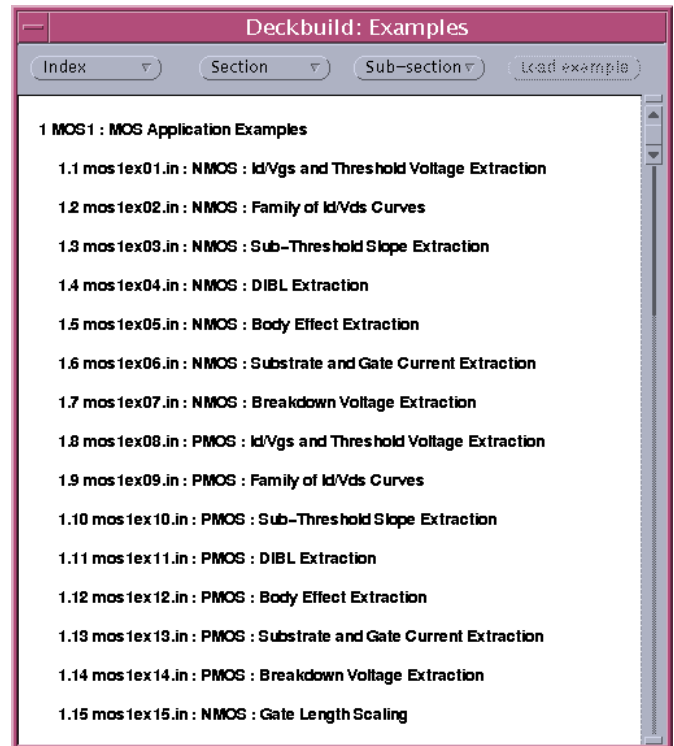
To view an explanation of any particular example simply double click the example in which you are interested. Double click the first example, entitled "NMOS : Id/Vgs and Threshold Voltage Extraction" to view an explanation of that example. The resulting display window is shown in Figure 2.4. Scroll down the window to read the full description.
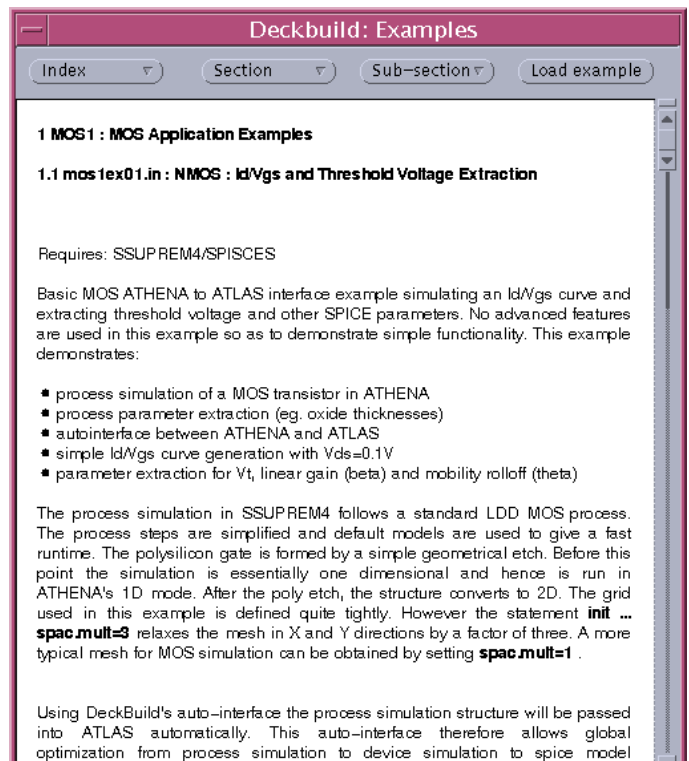

Figure 2.4 Example Description Window

In order to load this particular example into the DeckBuild runtime environment click the "Load Example" button at the top right of the "DeckBuild: Examples" window. The main DeckBuild window should now appear as in Figure 2.5.
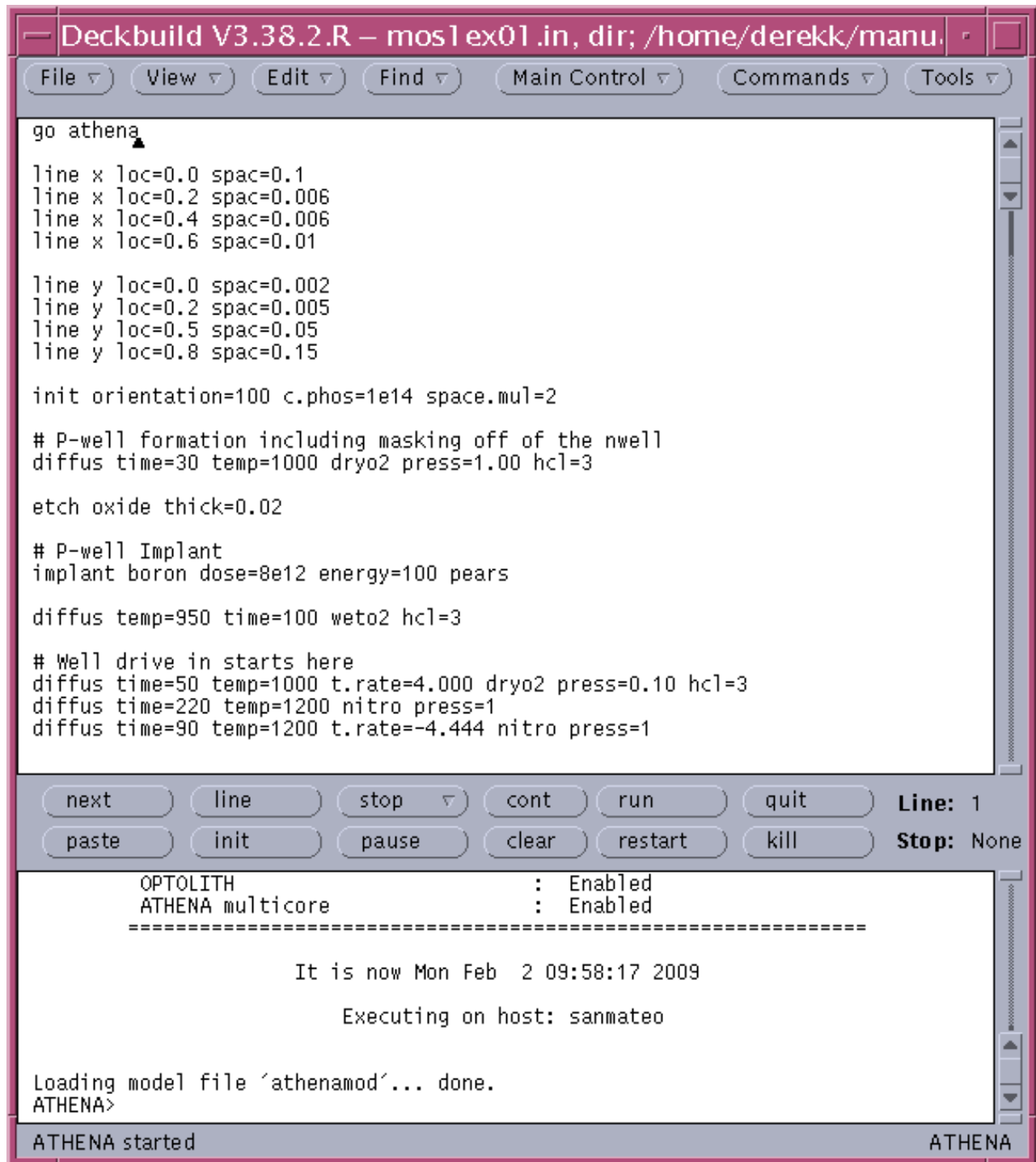
```
Deckbuild V3.38.2.R – mos1ex01.in, dir; /home/derekk/manu.

  File ▽    View ▽    Edit ▽    Find ▽      Main Control ▽      Commands ▽    Tools ▽

go athena

line x loc=0.0 spac=0.1
line x loc=0.2 spac=0.006
line x loc=0.4 spac=0.006
line x loc=0.6 spac=0.01

line y loc=0.0 spac=0.002
line y loc=0.2 spac=0.005
line y loc=0.5 spac=0.05
line y loc=0.8 spac=0.15

init orientation=100 c.phos=1e14 space.mul=2

# P-well formation including masking off of the nwell
diffus time=30 temp=1000 dryo2 press=1.00 hcl=3

etch oxide thick=0.02

# P-well Implant
implant boron dose=8e12 energy=100 pears

diffus temp=950 time=100 weto2 hcl=3

# Well drive in starts here
diffus time=50 temp=1000 t.rate=4.000 dryo2 press=0.10 hcl=3
diffus time=220 temp=1200 nitro press=1
diffus time=90 temp=1200 t.rate=-4.444 nitro press=1


  next        line        stop  ▽    cont      run        quit       Line: 1
  paste       init        pause      clear     restart    kill       Stop: None

       OPTOLITH                        :  Enabled
       ATHENA multicore                :  Enabled
       ============================================================

                  It is now Mon Feb  2 09:58:17 2009

                     Executing on host: sanmateo

Loading model file 'athenamod'... done.
ATHENA>

 ATHENA started                                                     ATHENA
```

Figure 2.5 Example Input File Now Loaded Into DeckBuild

Running the example is simply a matter of clicking the "run" button. On completion, two plots will appear on the screen. The first is a color contour plot of the process simulated structure displaying "Net Doping" and the second is a simple threshold voltage, Vg versus Id plot. These plots are shown in Figures 2.6 and 2.7 respectively. It should be noted that it is not necessary to actually run the simulations in order to view the results, since these structure files and line graphs are loaded into the current working directory as soon as the "Load Example" button is selected.
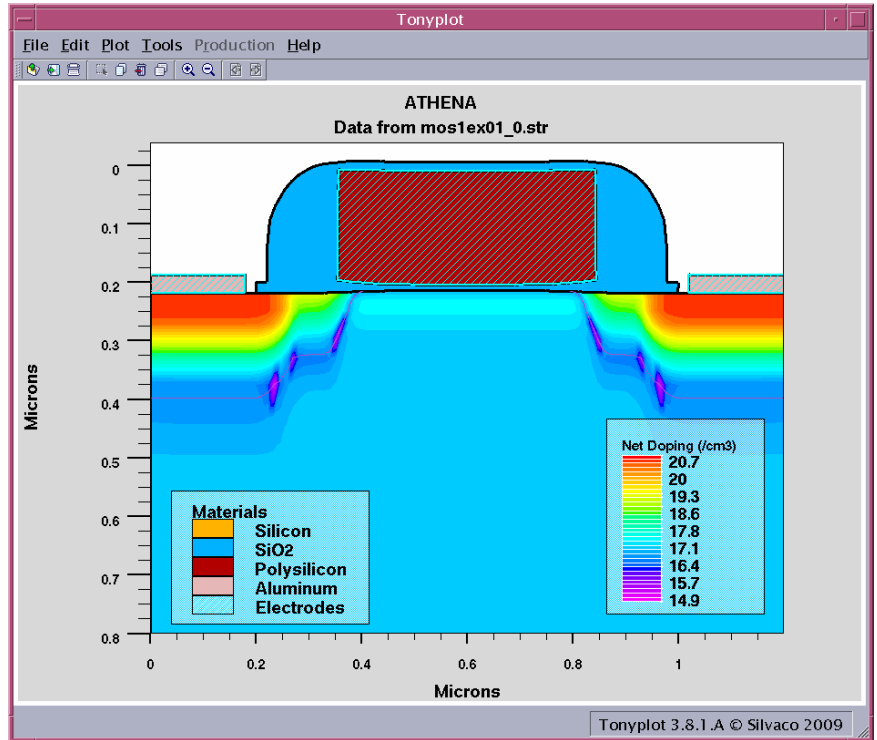

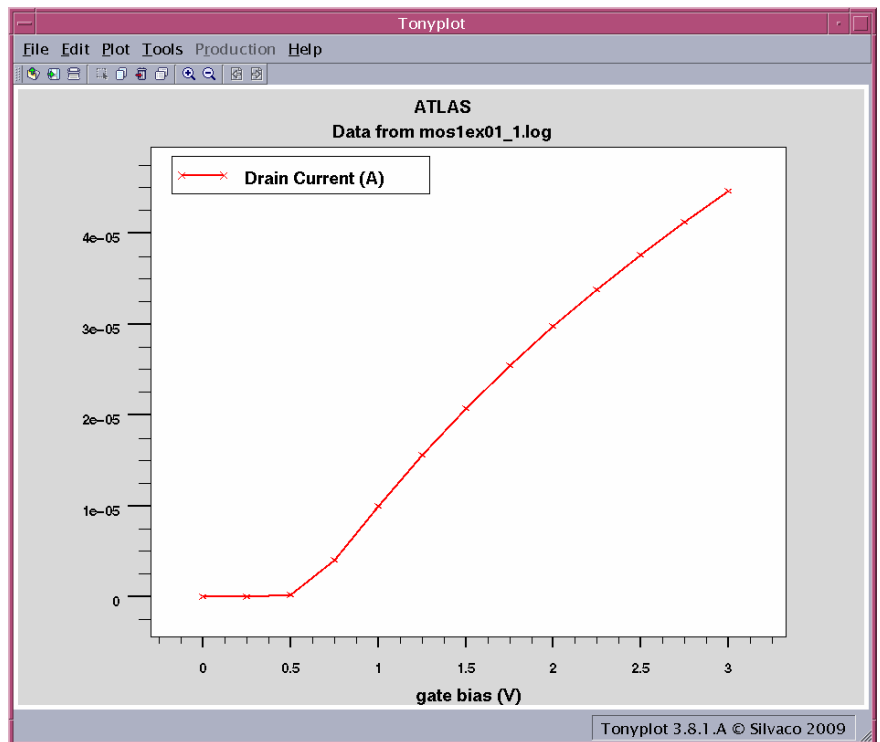**Figure 2.6 Process Simulated Structure Showing Net Doping**

Figure 2.7 Device Simulated Line Graph Showing Vg Versus Id

Section 2.3: Viewing the Results

SILVACO's visualization tool called "TonyPlot" is a feature-packed plotting program. We will only touch on some of its more basic capabilities in this tutorial. When a structure file or line graph is saved in SILVACO's TCAD package, almost every quantity required to create the structure file or line graph, will be automatically included into the file and will be available for plotting.

For example, in the structure file plot, if you wish to plot other calculated quantities such as "Donor Concentration," click the following sequence:

Plot... Display... Define... Contours...

then pull down and scroll the "Quantity:" box to obtain a list of plottable quantities. If you followed the sequence, two additional GUI displays should appear as in Figure 2.8. Select any new quantity and click "Apply."
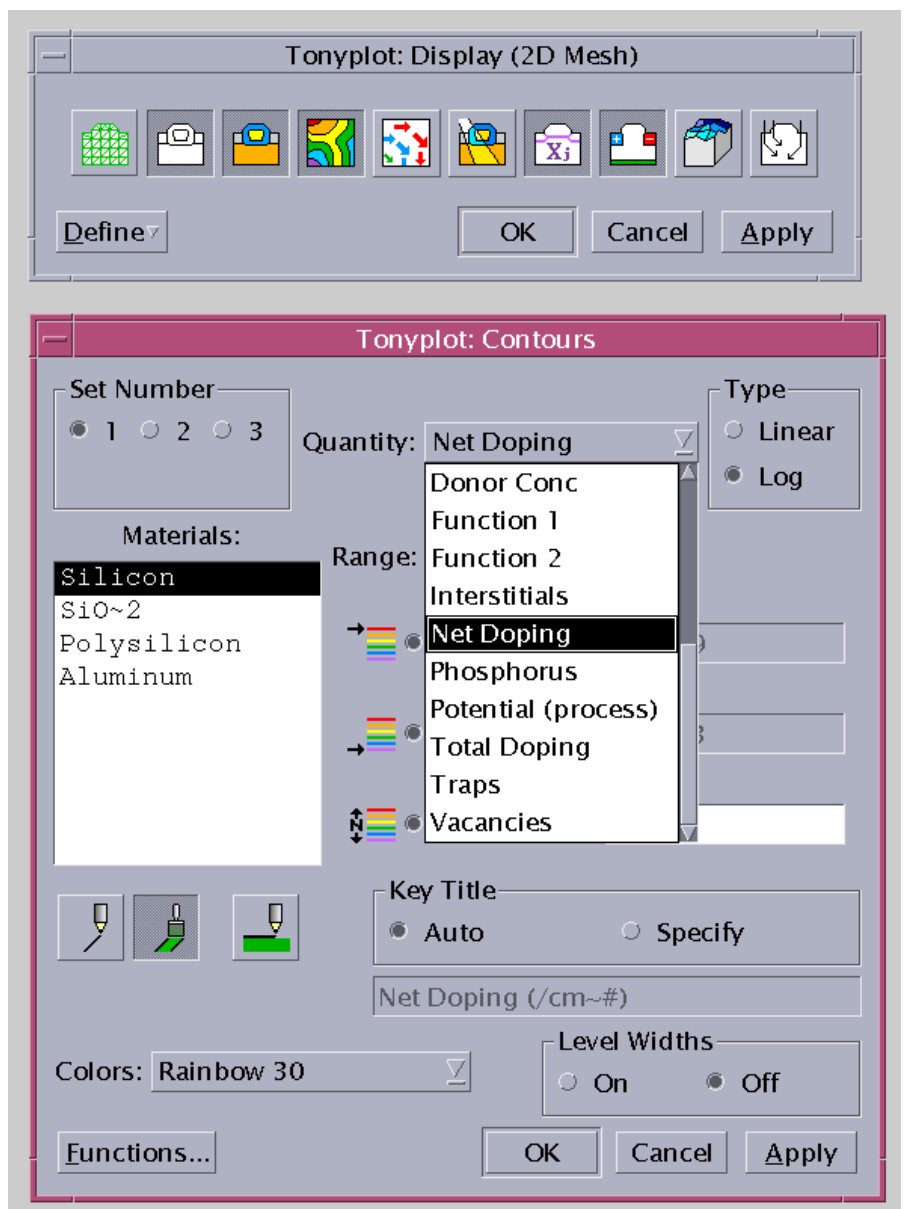

Figure 2.8 The GUIs Related to Selecting Plottable Quantities

It will often be necessary to view a "cutline" slice through a plot. To do this, close other plotter related windows by clicking "OK," then return to the main plot and select the following sequence:

Tools... Cutline...

For now we will use the default "vertical cutline" box, but the horizontal or diagonal cutline box could also be selected in the "cutline" GUI. Return to the main plot, click and hold down the left mouse button where you want to start the cutline. Drag the mouse vertically and let go of the left mouse button where you wish to finish the cutline. When you let go of the left mouse button, the cutline plot will automatically appear showing a cutline of the currently displayed quantity. Click "OK" on the cutline box if you are now done with making cutlines.

To overlay other quantities on the new cutline graph, left click the line graph to make it the current active plot (a white border around the plot indicates which plot is currently active). Now click:

Plot... Display...

and select other quantities to overlay on the line graph. For example, if you add Arsenic, Boron and Phosphorus to the line graph, it should now appear as in Figure 2.9. A similar sequence on the Vg versus Id curve will also allow other electrical quantities to be overlayed on that line graph.

On a final note, you can plot a file by typing "tonyplot <filename>" into a terminal or console window or you can plot structures directly in DeckBuild by highlighting a structure file name and clicking the following sequence:

Tools... Plot... Plot Structure

To plot a structure file created during a simulation automatically, simply add the lines:

        structure outfile=<filename>
        tonyplot <filename>

into your input file and the simulator will save the structure file created up until that point in the input file.It will then plot it automatically once the simulator reaches that point. You can save and plot as many structure files as you wish at any point during the simulation.
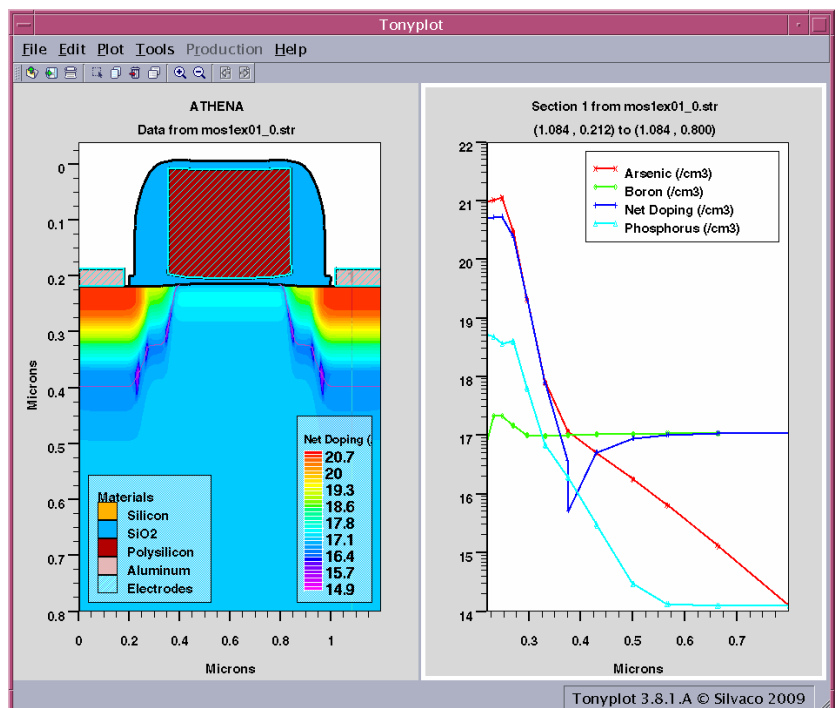

Figure 2.9 Cutline Graphs

Section 2.4: Automated Input File Creation and Editing

By now you should have a pretty reasonable idea how to load, run and view the results of TCAD examples. We will now demonstrate just how simple it is to create your own process simulation input files or to modify existing ones.

The input files are all simple ASCII text format so you can create an input file with any ASCII editor. However, DeckBuild includes a feature that creates process simulation input files automatically via a simple GUI.

For simplicity, rather than take the user through the repetition of creating a process simulation input file from scratch, we will demonstrate how to add lines of code to our existing input file. The same technique can be used to create an entire input file by simply repeating this procedure for each process step.

Once the simulation space and initial substrate have been defined using the "line" and "init" statements (see the first few lines of the loaded example), a process simulation simply consists of numerous consecutive diffusions, implants, etches and deposits. Every one of these steps can be written automatically using DeckBuild.

Return to our example input file. Suppose we wanted to add a second, low energy 10keV "P-Well Implant" after the first one. Left click in the top DeckBuild window, one line after the first P-well Implant. Now click the following sequence of buttons:

Commands... Process... Implant

The "DeckBuild: ATHENA Implant" window should appear. Creating the additional implant line is now just a matter of filling in the boxes, using the sliders or filling in the values followed by depressing the return key. For a 10keV boron implant, with a dose of 1e11/cm2 at an angle 7 degrees to the perpendicular and a rotation of 27 degrees to the major flat, the implant GUI should appear as shown in Figure 2.10. Once you are done inputting the information, click "Write" and the new line will appear in the input file.

All deposits, etches, implants, diffusions, electrodes, mesh creation, wafer initialization and structure file saves, etc., can be created in exactly the same way. Since the input files are simple ASCII text, editing parameters in the intuitive syntax input files is simply a matter of using the usual copy, paste and delete functionality included in DeckBuild.
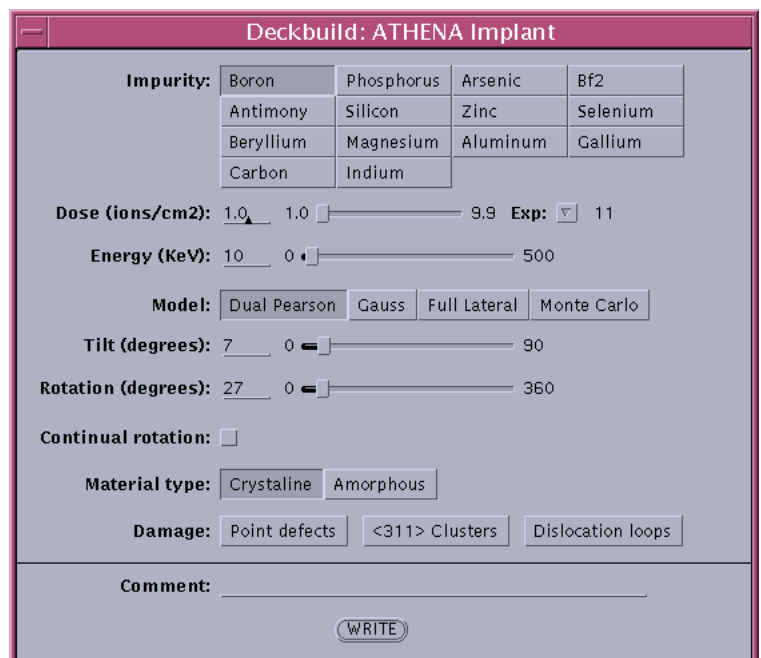

Figure 2.10 The Completed Implant GUI

Chapter 3:
Navigating the Manual and Self Training the Easy Way

At first sight, the size of the manuals may seem a little daunting, but this section will show the new user just how simple it is to find the specific information required by using the examples to navigate the manual.

Using the input file we loaded in Section 2.2 as an example, let us say we wanted to understand what the "SRH" parameter does in the device simulation section of the input file, which appears on the line beginning with "models."

In both the process and device simulator, the first word on any line is known as the "statement." In this example, since the first word on the line is "models" we now know that we need more information on the "models statement" if we are to find out what the parameter "SRH" means.

All SILVACO products include manuals in pdf format. We are looking for information in the device simulator, so we need to invoke the ATLAS manual. The manuals can be accessed via the "sman" facility which can be invoked by typing "sman" in a console or terminal window. Figure 3.1 shows what the manual should look like.
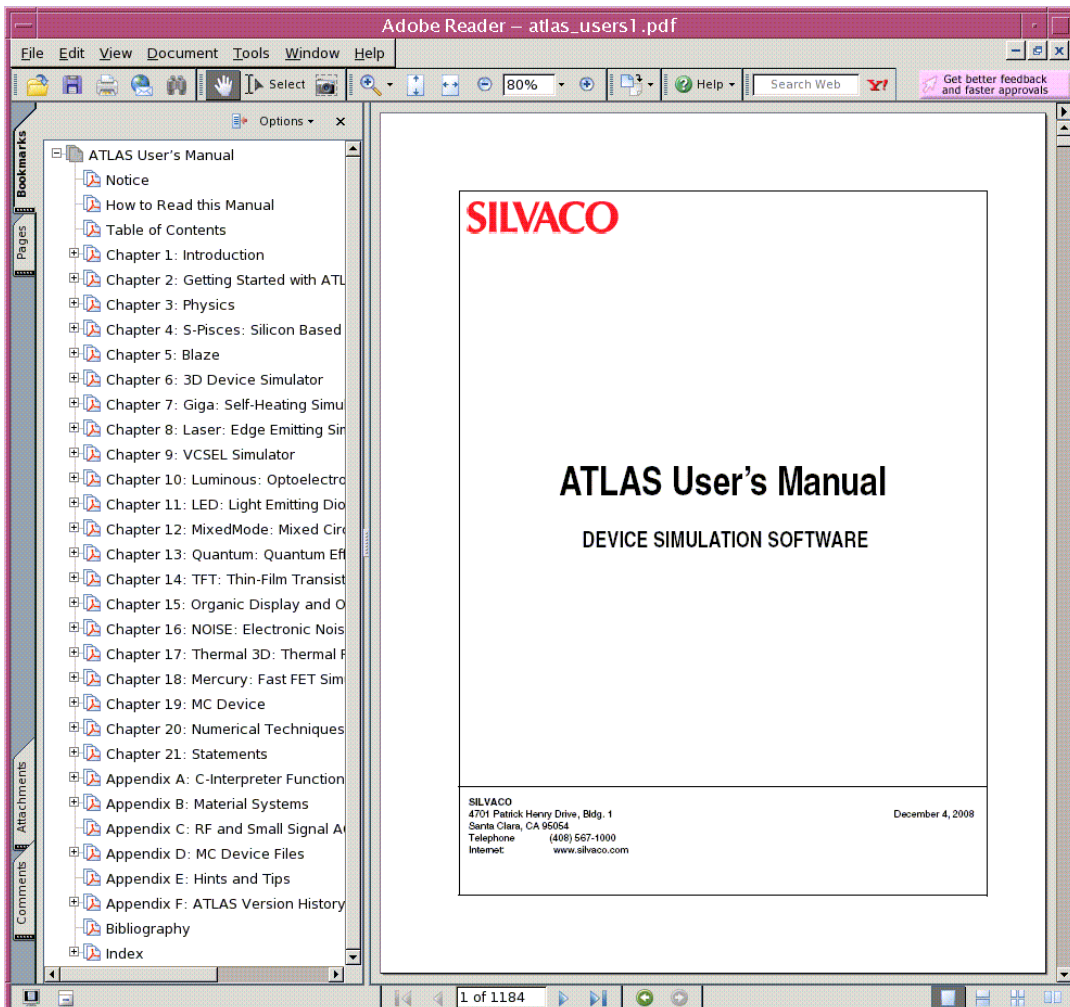

Figure 3.1 The ATLAS Manual PDF Interface

One of the most powerful features of the manual is the "Statements" chapter. This chapter contains an alphabetical listing of all the statements in the device simulator. Since we are looking for information on the "models" statement, click on the expansion "+" symbol next to the Statements chapter in the Index located on the left side of the page. Scroll down the alphabetical listings until you reach the "Models" statement. Click the expansion "+" sign next to the Models statement to see its contents.

At this point, we now have a choice. If you have already guessed that "SRH" is an acronym for "Schottky Read Hall" recombination, then click the section titled "Recombination Models Flags." If you had no idea what the parameter "SRH" was, click the "MODELS" heading to start at the beginning of this section. At the beginning of this section is another alphabetical listing of all parameters associated with the models statement. Scroll down until you reach "SRH" and the table will inform you: what type of parameter it is, any default value applied to it and what the units of the parameter are. Once you know what kind of parameter it is, scroll down the rest of the section until you find it in the description section below. You could also use the "Edit... Find..." feature to search for "SRH" in the description section. Whichever method used, you should now be located at the section where a brief description of the SRH parameter is given, as shown in Figure 3.2.
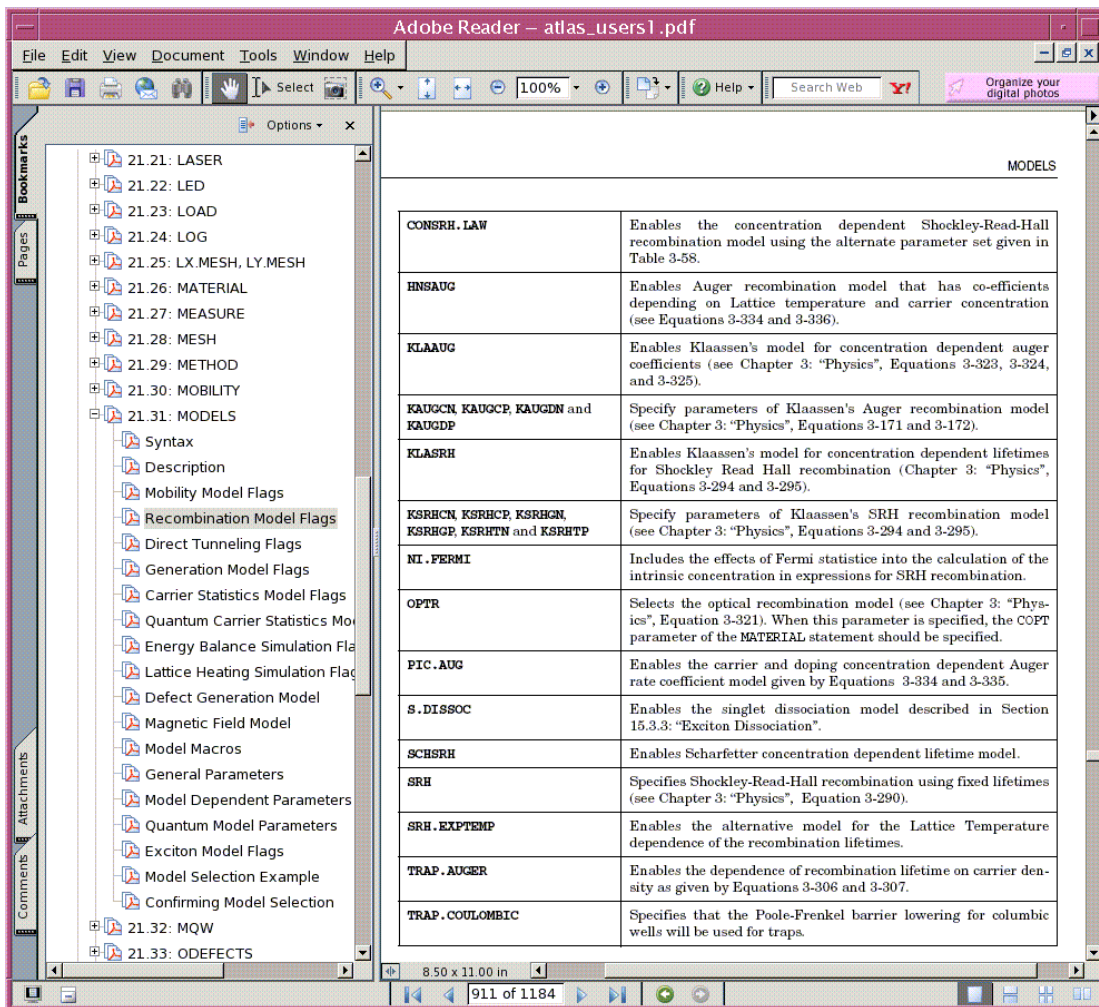

Figure 3.2 Locating the Description of the "SRH" Parameter

This information may be sufficient for the user to know what the parameter is and its default values. If however, you now wish to understand the physics and formulas in which this parameter is used, the description for 'SRH" in the Statements Chapter, cross references the reader to the Physics Chapter 3, Equation 3-290 to find out more information. If you cross references to Equation 3-290, we find a full description or the SRH model, as shown in Figure 3.3.

Shockley-Read-Hall (SRH) Recombination

Phonon transitions occur in the presence of a trap (or defect) within the forbidden gap of the semiconductor. This is essentially a two step process, the theory of which was first derived by Shockley and Read [188] and then by Hall [77]. The Shockley-Read-Hall recombination is modeled as follows:

$$R_{SRH} = \frac{pn - n_{ie}^2}{\text{TAUP0}\left[n + n_{ie}\,exp\left(\frac{\text{ETRAP}}{kT_L}\right)\right] + \text{TAUN0}\left[p + n_{ie}\,exp\left(\frac{-\text{ETRAP}}{kT_L}\right)\right]} \qquad 3\text{-}290$$

where ETRAP is the difference between the trap energy level and the intrinsic Fermi level, T_L is the lattice temperature in degrees Kelvin and TAUN0 and TAUP0 are the electron and hole lifetimes. This model is activated by using the SRH parameter of the MODELS statement. The electron and hole lifetime parameters, TAUN0 and TAUP0, are user-definable in the MATERIAL statement. The default values for carrier lifetimes are shown in Table 3-56. Materials other than silicon will have different defaults. A full description of these parameters are given in Appendix B: "Material Systems".

Table 3-56. User-Specifiable Parameters for Equation 3-290			
Statement	Parameter	Default	Units
MATERIAL	ETRAP	0	eV
MATERIAL	TAUN0	1×10^{-7}	s
MATERIAL	TAUP0	1×10^{-7}	s

Note: This model only presumes one trap level which, by default, is ETRAP=0 and it corresponds to the most efficient recombination centre. If the TRAP statement is used to define specific trap physics then separate SRH statistics are implemented as described earlier in "Trap Implementation into Recombination Models" section on page 3-16.

Figure 3.3 The Full Description of the SRH Model Cross Referenced

With a few clicks of the mouse, we have found what a parameter is, together with a full physical description of how this parameter works in the simulator and what other user definable parameters are in the equation that are under the users control. Now you understand how the manuals work in conjunction with the examples.

Chapter 4:
Other Web Based information

The SILVACO website offers a wealth of technical information in our library section currently located at:

http://www.silvaco.com/tech_lib/index.html

Here information is sorted into four categories, namely:

      1/ Simulation Standard (a SILVACO publication)
      2/ Presentation Materials
      3/ Books published referencing SILVACO Software
      4/ Published Papers (Peer Reviewed Technical Publications)

SILVACO's Simulation Standard is a quarterly TCAD publication that includes detailed application notes and results of simulations covering a wide range of technologies. There is also a Hints and Tips section that answers some frequently asked questions. All Simulation Standard publications, dating back for many years, are available on the SILVACO web site. Use the search facility to find an article on the subject you are interested in.

The documents under the heading "Presentation Materials" contain several types of presentations. There are PowerPoint presentations showing application examples and explanations of syntax etc. There are also academic course materials showing how to incorporate SILVACO TCAD software into coursework. Other sections list patents aided by SILVACO software plus a listing of text books referencing SILVACO software.

The final two sections in the Technical Library section list text books and published papers under numerous technology headings that have all used SILVACO software in their creation.

If this all seems like information over-load, SILVACO has once again made searching for information as simple as possible by including a search box that quickly and efficiently takes the user to the information required.

In conclusion we hope we have demonstrated just how simple it is to use SILVACO software and to find the information you need to teach yourself how to succeed in your TCAD simulations.