Lecture 4: Cadence

CSCE 5730
Digital CMOS VLSI Design

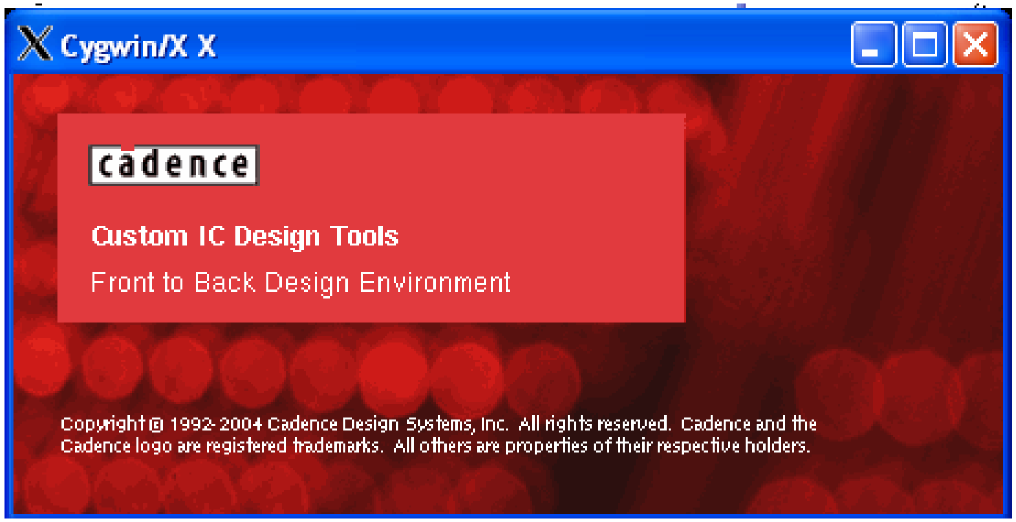
Instructor: Saraju P. Mohanty, Ph. D.

NOTE: The figures, text etc included in slides are borrowed from various books, websites, authors pages, and other sources for academic purpose only. The instructor does not claim any originality.

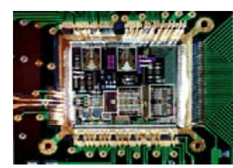
Lecture Outline

- In this tutorial we will learn using the following
  - The Cadence® ICFB Environment and Tools
  - Creating Library, Cellview in Library Manager
  - The Virtuoso Schematic Editor
  - Cadence Analog Design Environment
  - The Calculator Tool
  - Parametric Analysis

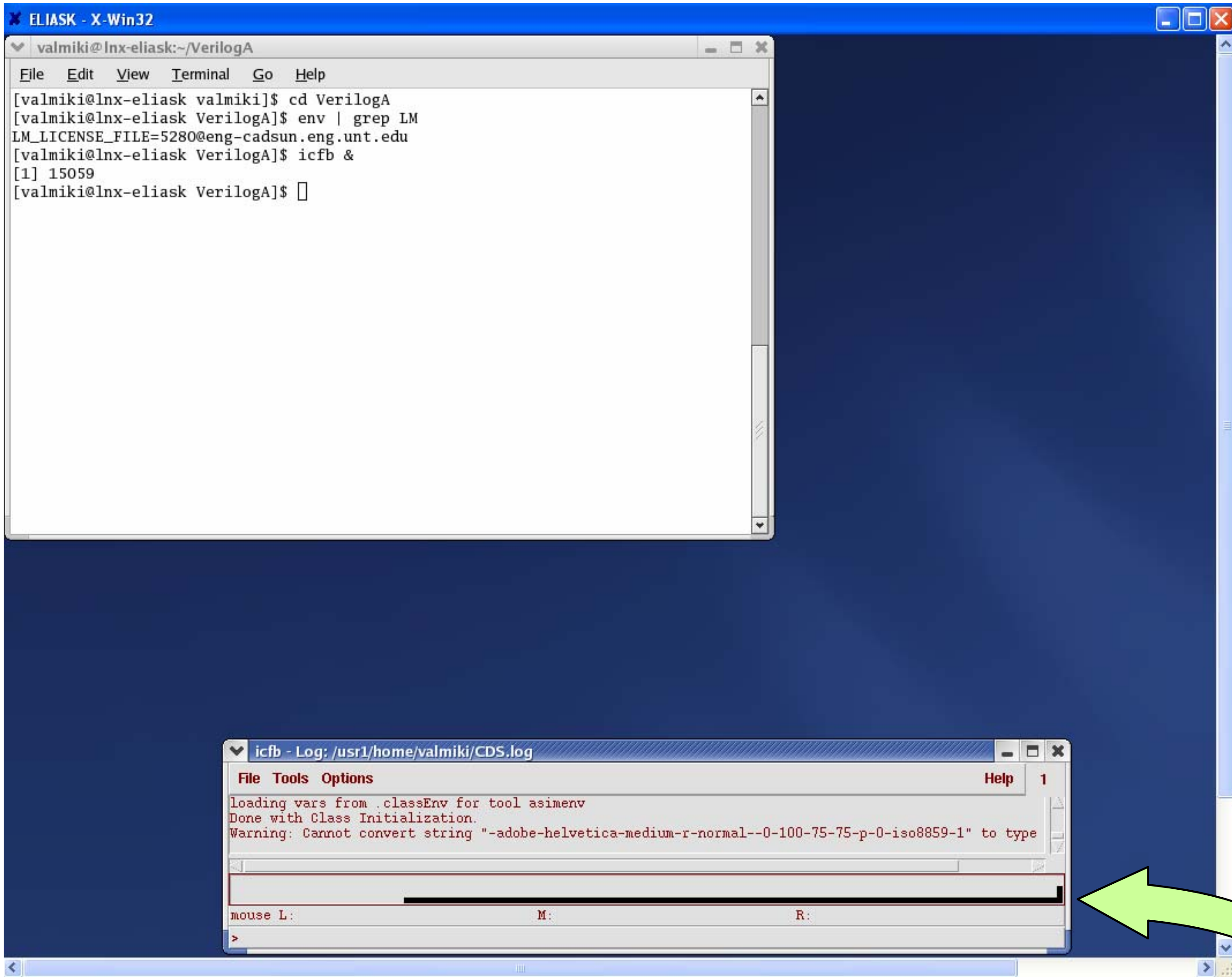
Cadence® ICFB



- The Cadence® ICFB environment is a set of
  - “Custom IC Design Tools”.
- It gives a complete
  - “Front to Back Design Environment”



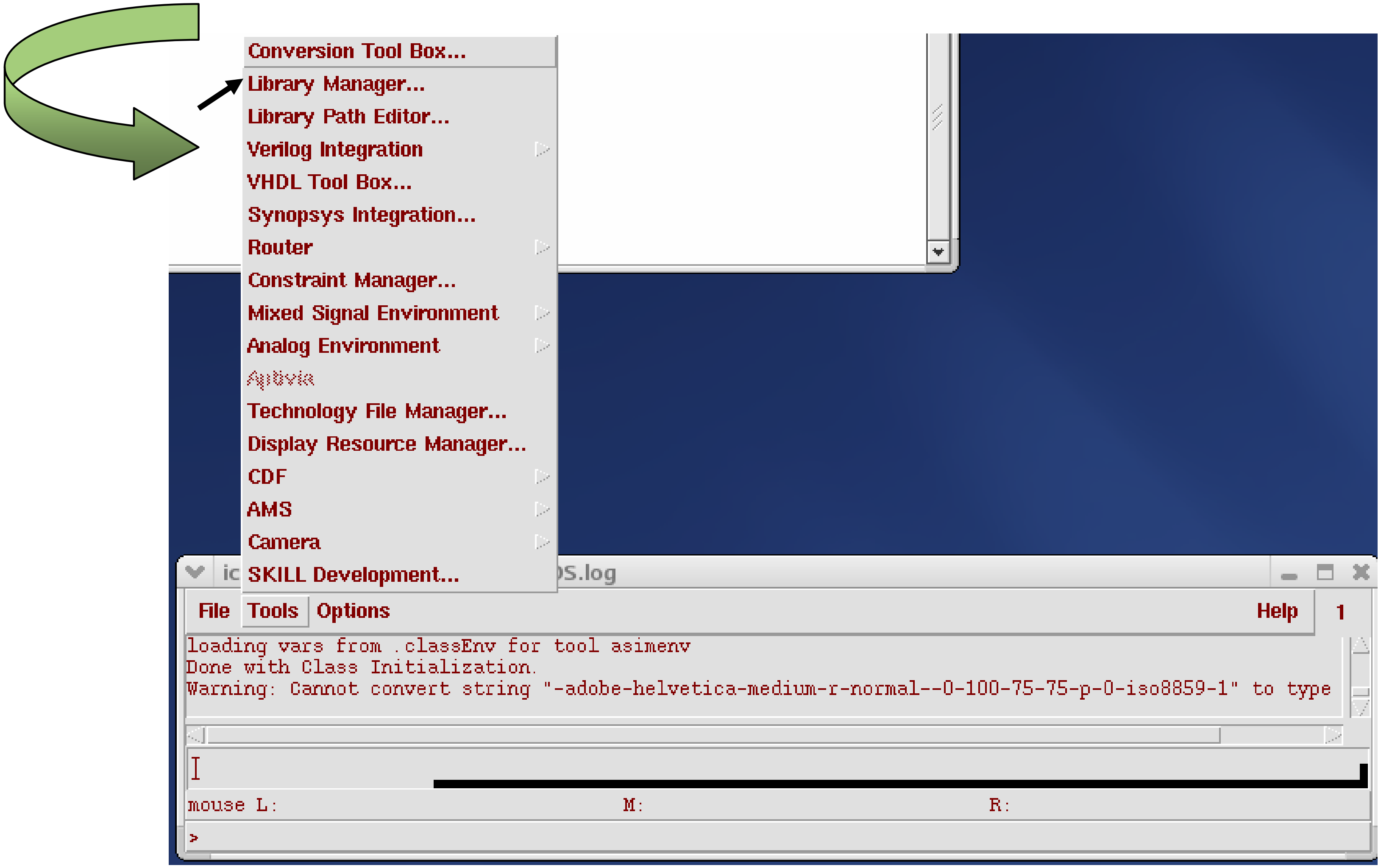
Initiating the ICFB Environment



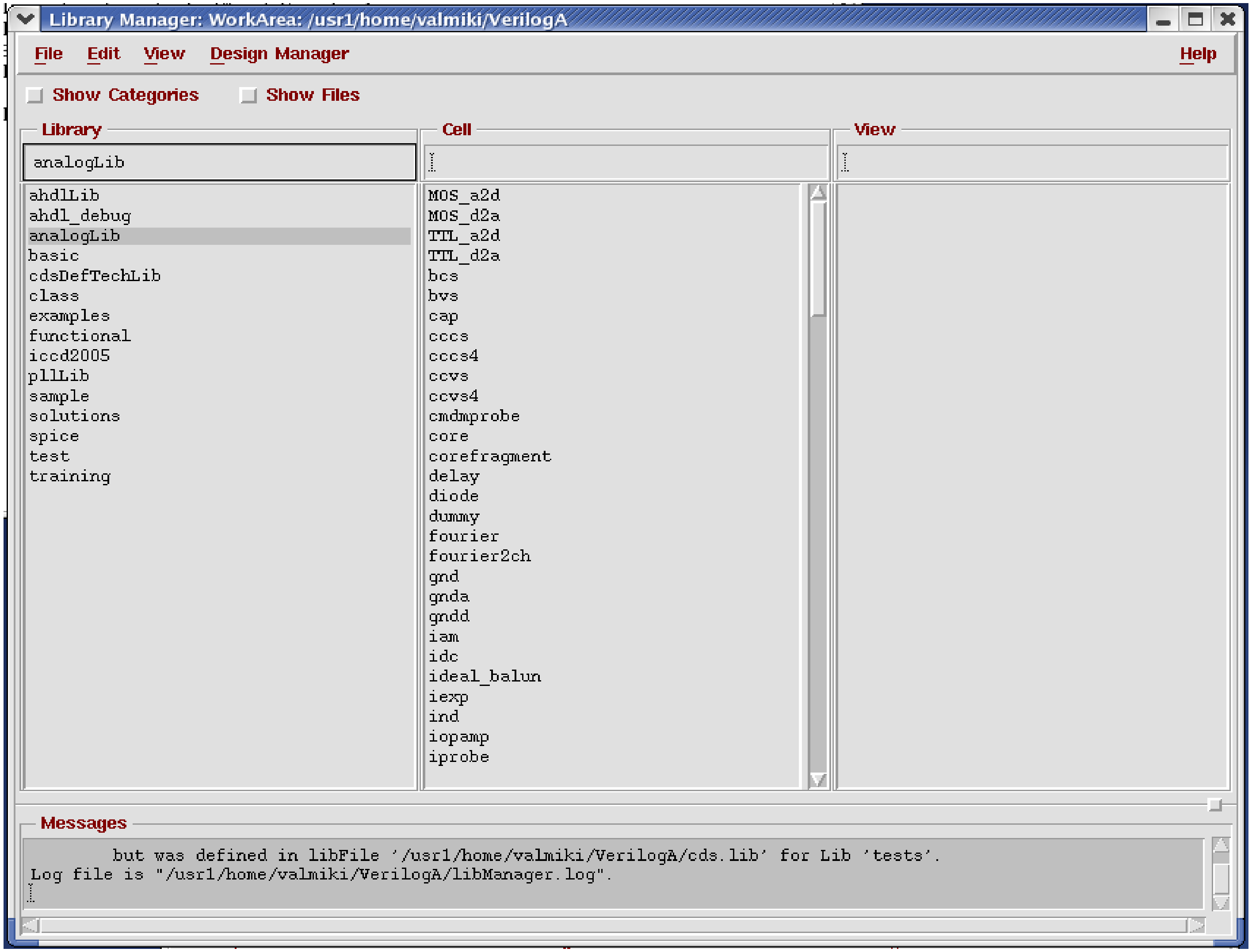
ICFB Command interpreter window

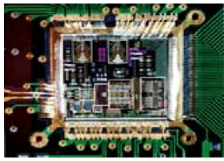
Launching Library Manager

Choose Library Manager from the "Tools" menu

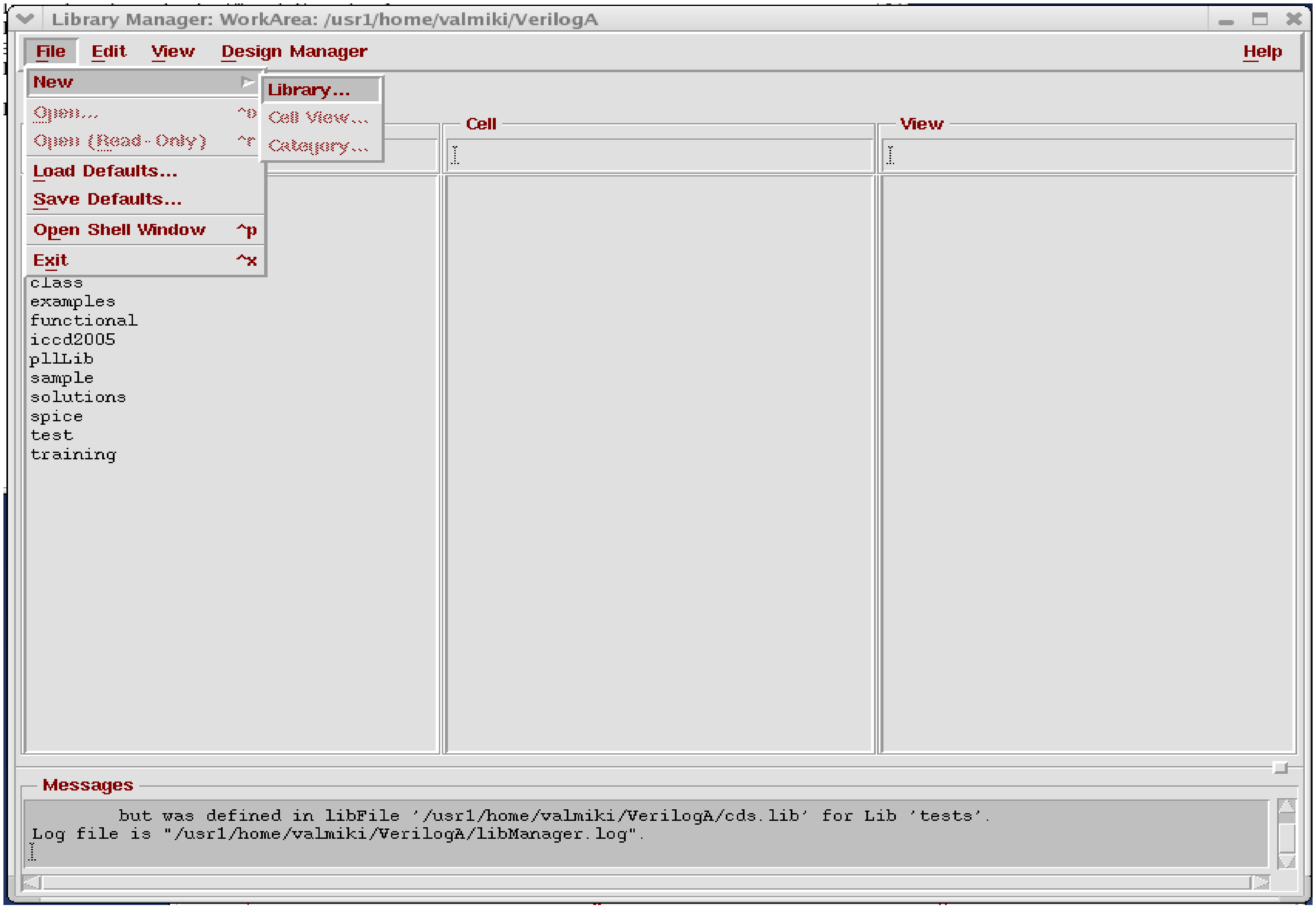


CADENCE – Library manager

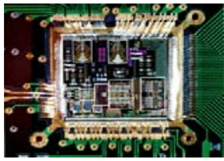




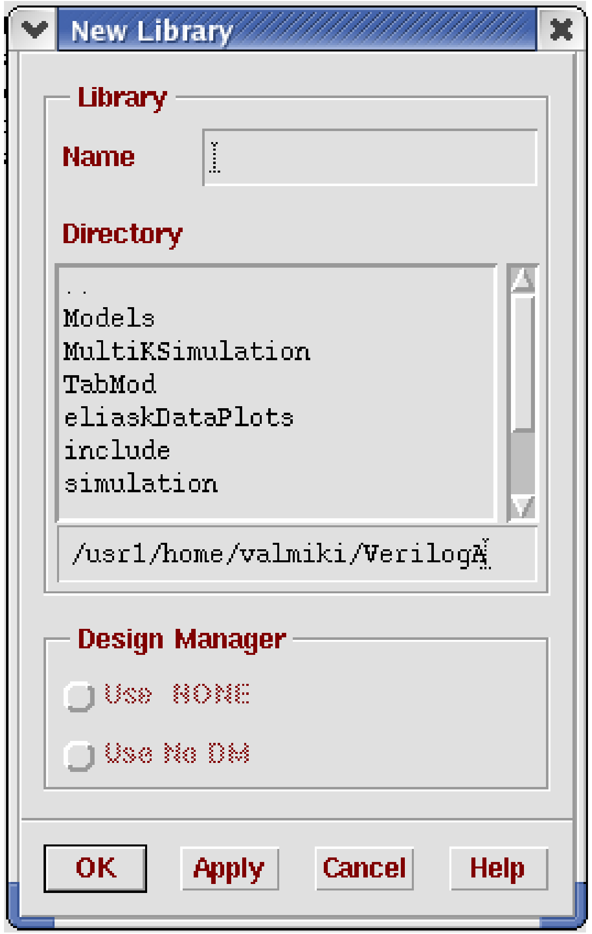
Creating a New Library - 1



File → New → Library

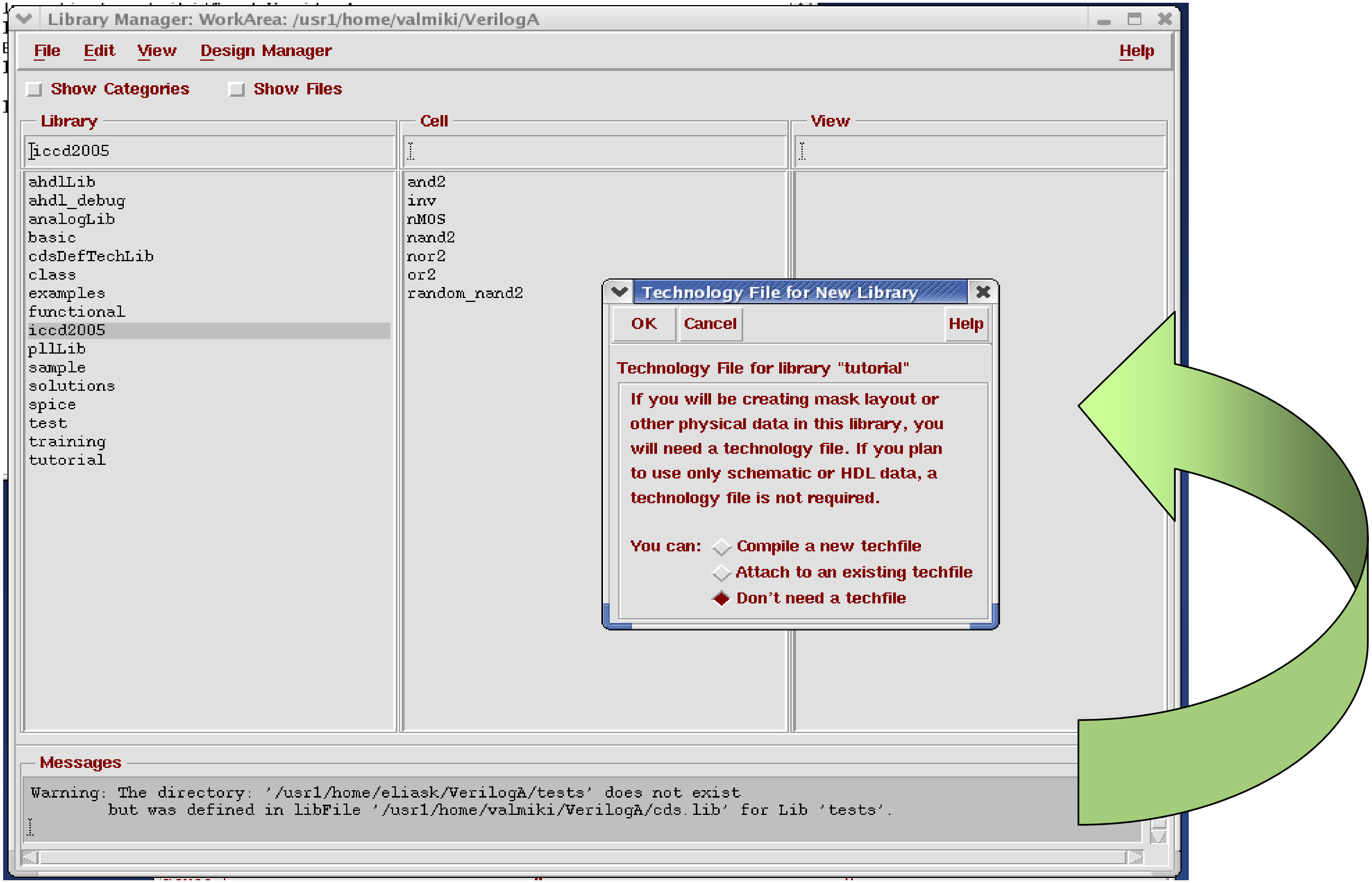


Creating a New Library - 2



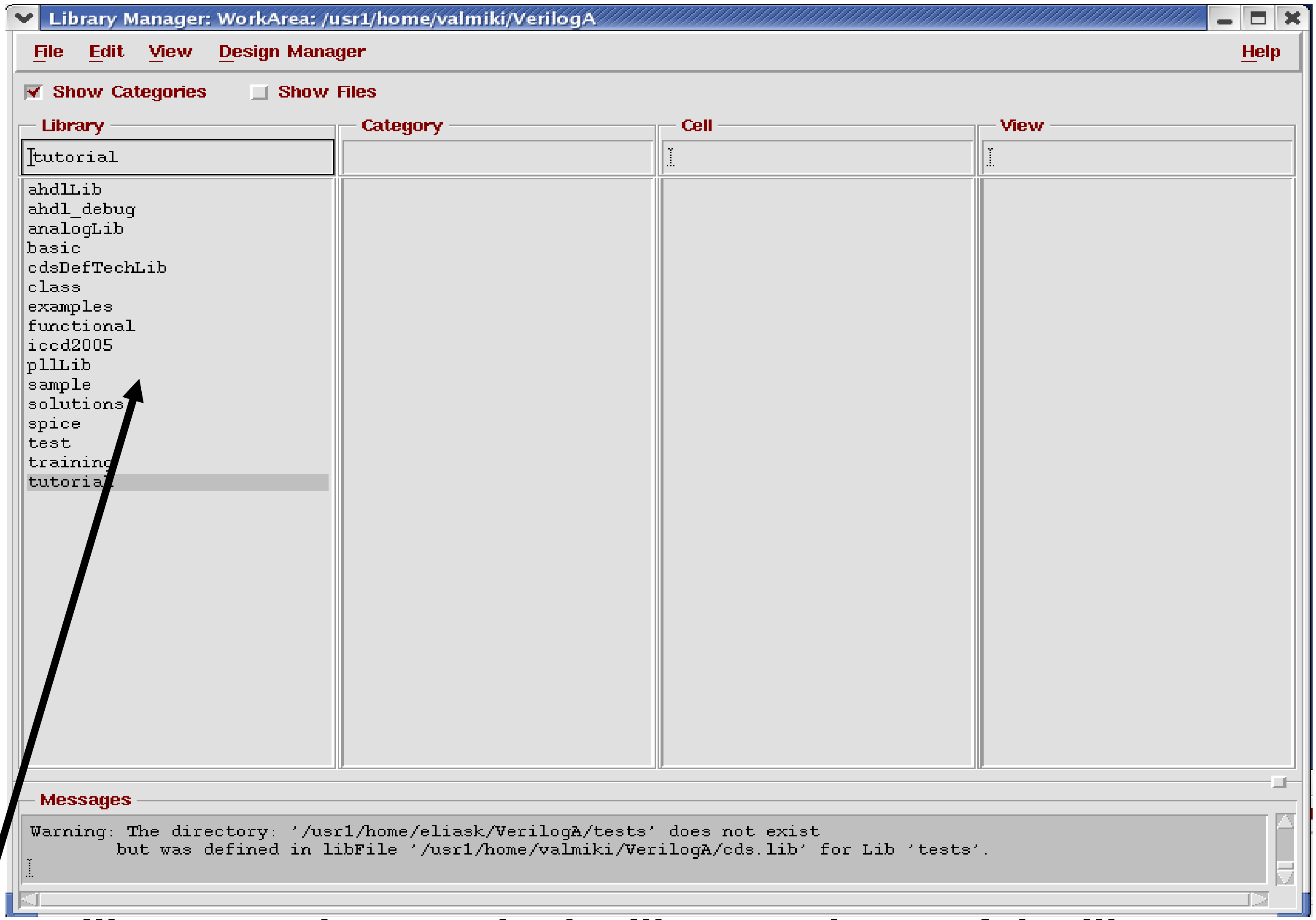
- 1.Choose the directory where you want to place your library/tools
- 2. Name the library
- Hit OK

Creating a New Library - 3



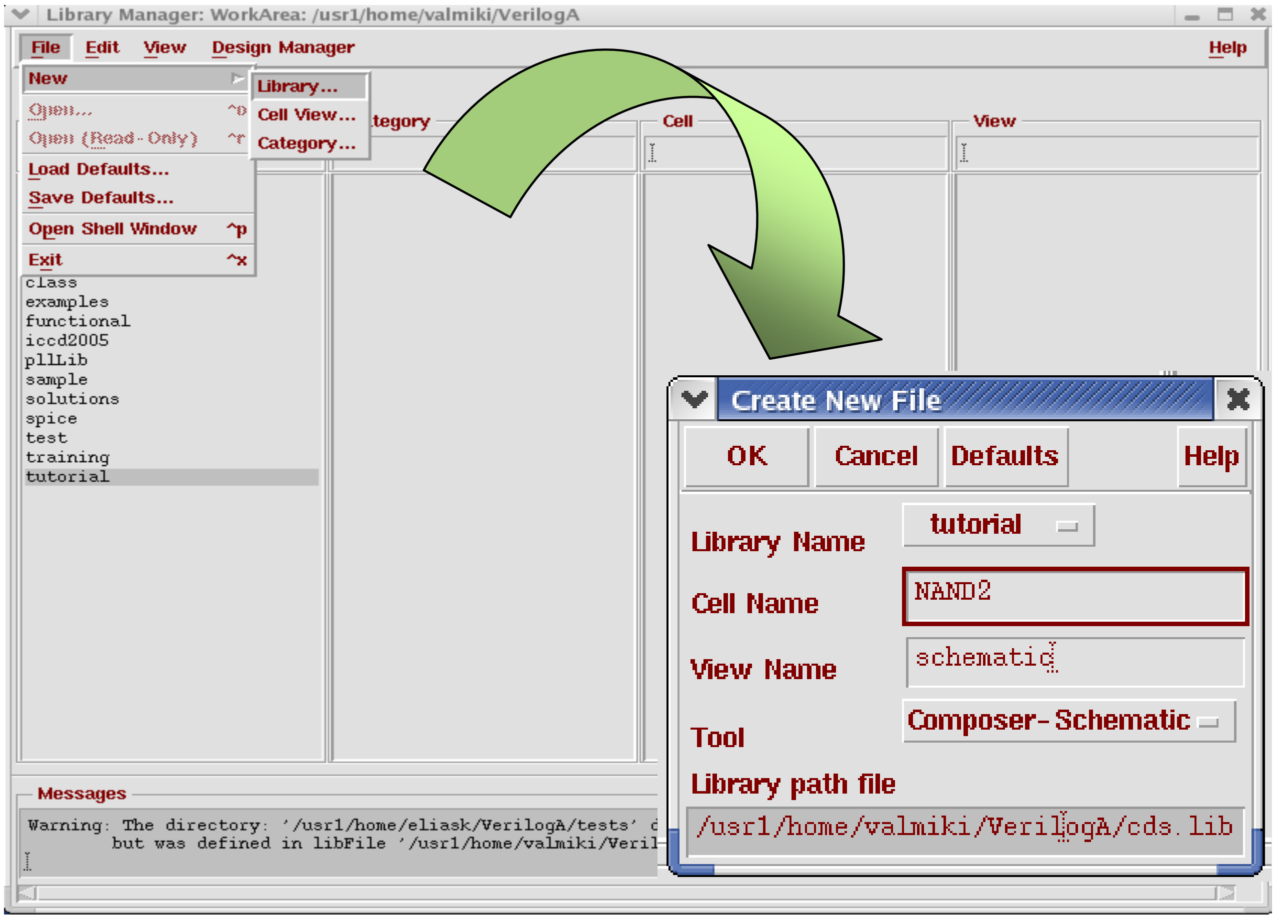
As and when required, associate a "techfile" with your library.

Creating a New Library - 4



The new library can be seen in the library column of the library manager

Creating a Cell View

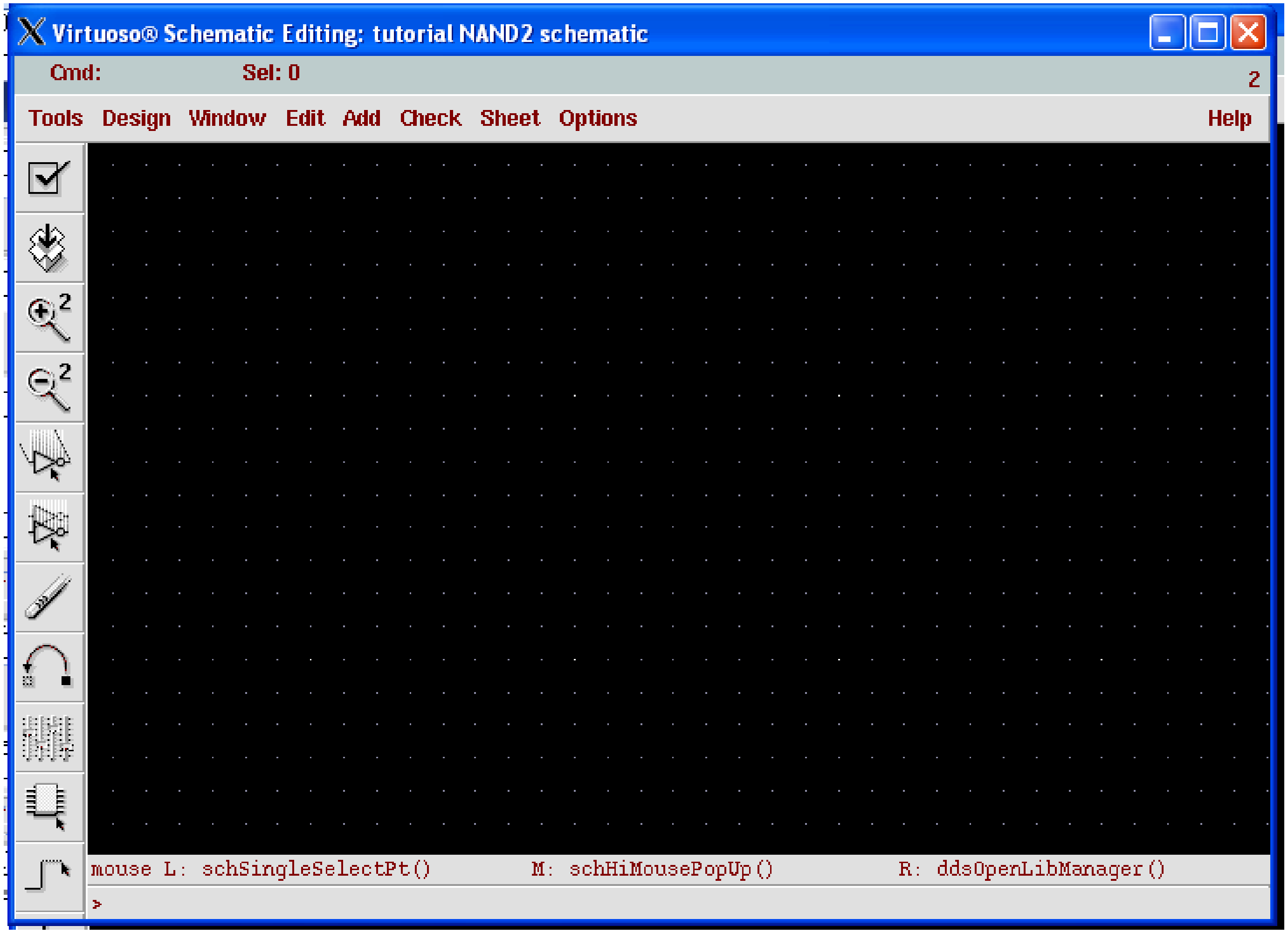


Choose your library, and then create a new Cell View as follows:

File → New → Cell View,

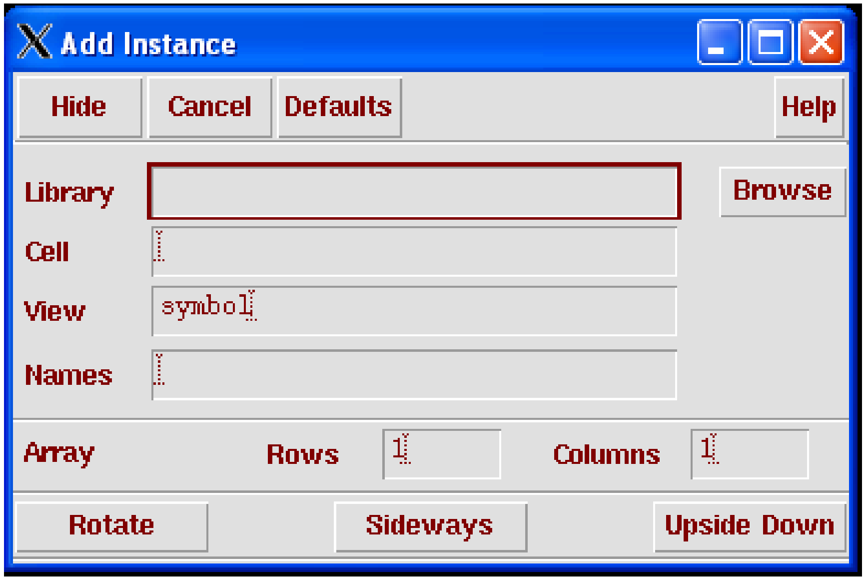
Cell Name → NAND2, View Name → Schematic, Tool → Composer

Virtuoso Schematic Editor

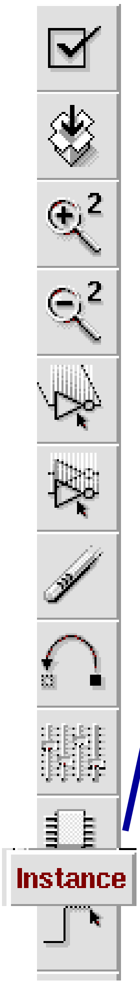


Virtuoso Schematic Editor Window

Adding an Instance of a Cell



Click Browse to get the Library Browser

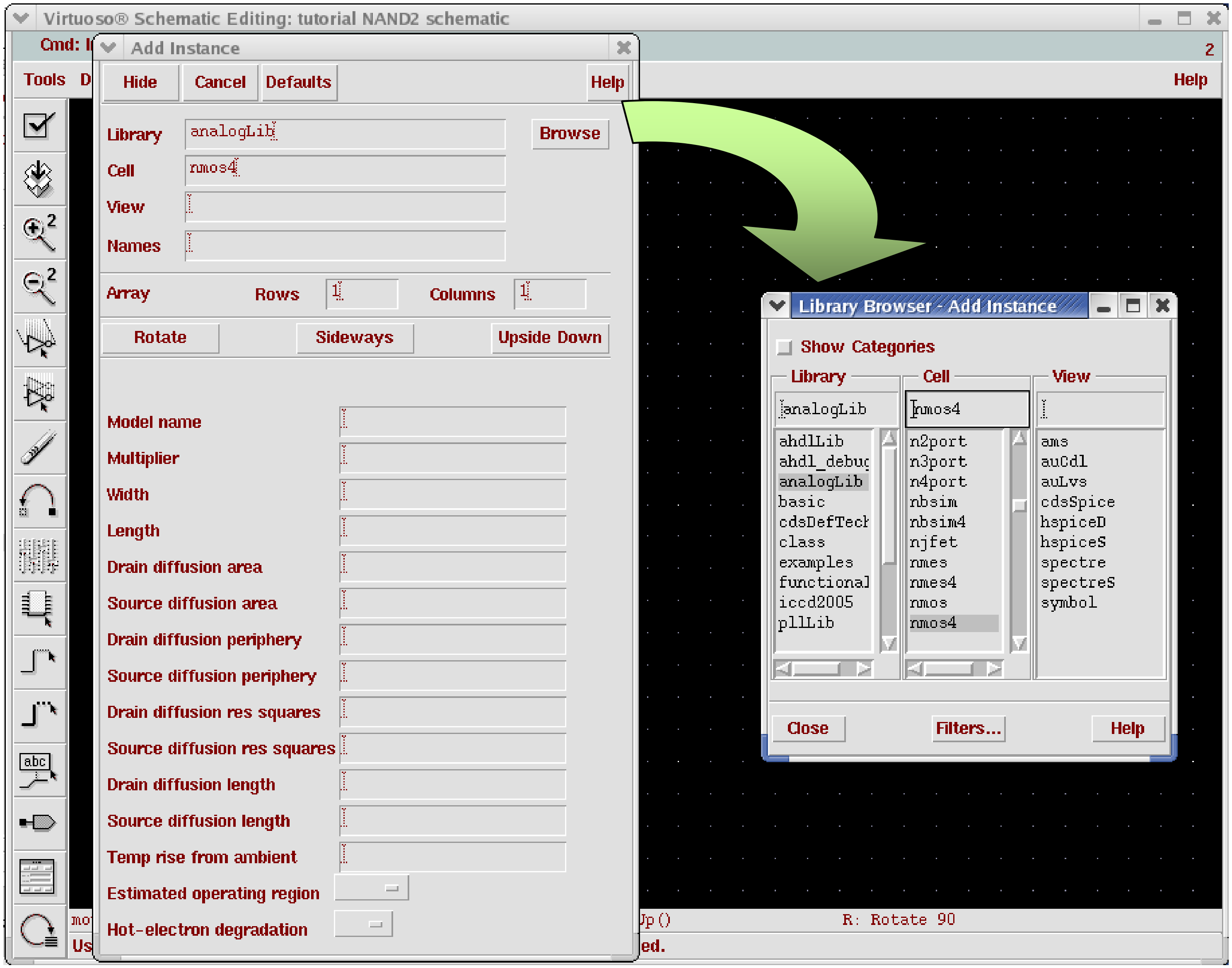


Various shortcuts to common commands such are available in the left toolbox:

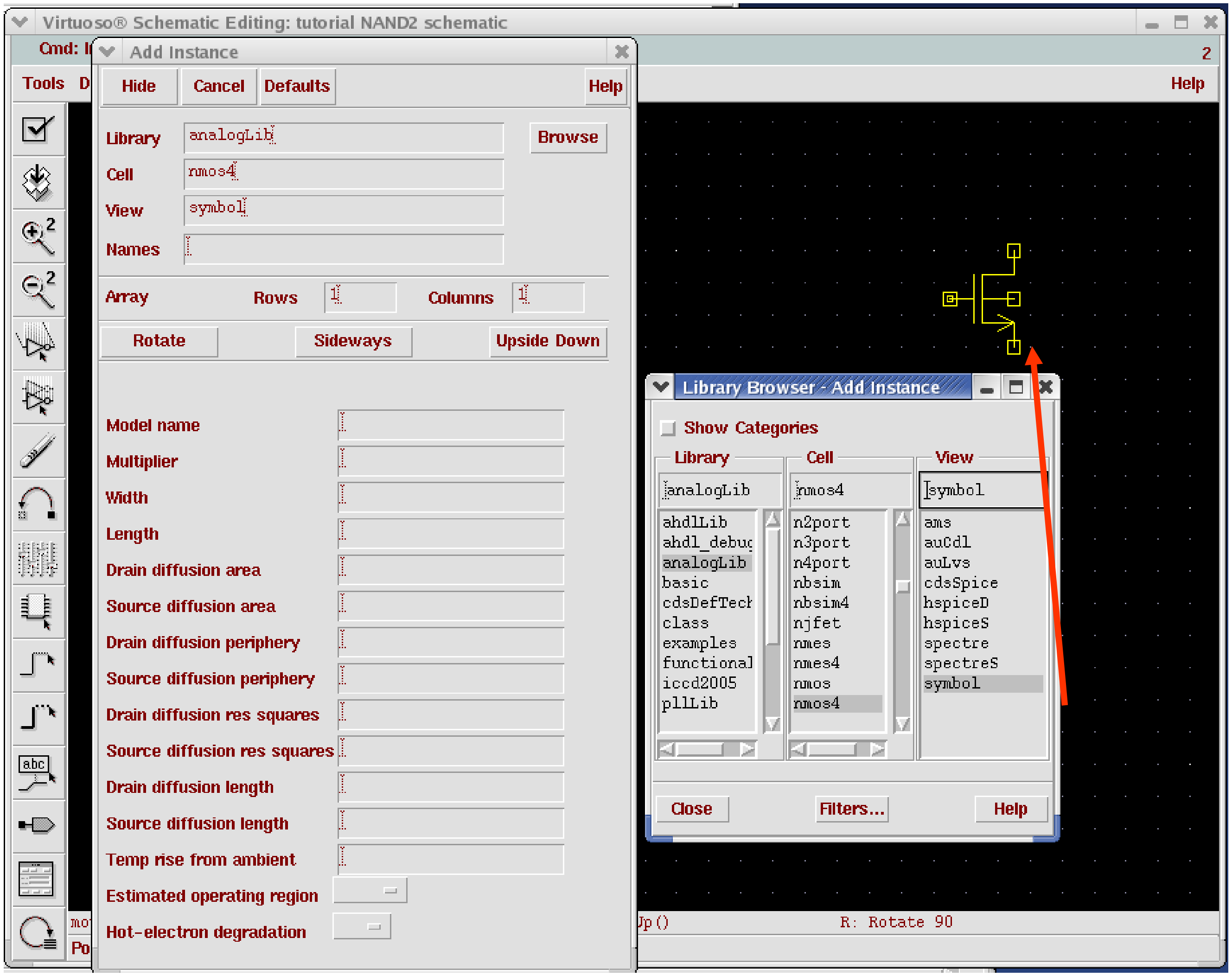
1. placing component instances,
2. drawing wires,
3. placing ports etc.

Short pop-up help messages can be obtained by putting the mouse on top of the icons.

Adding an Instance - 2



Adding an Instance - 3

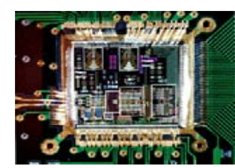


When the view is selected from the library it is automatically placed in the schematic

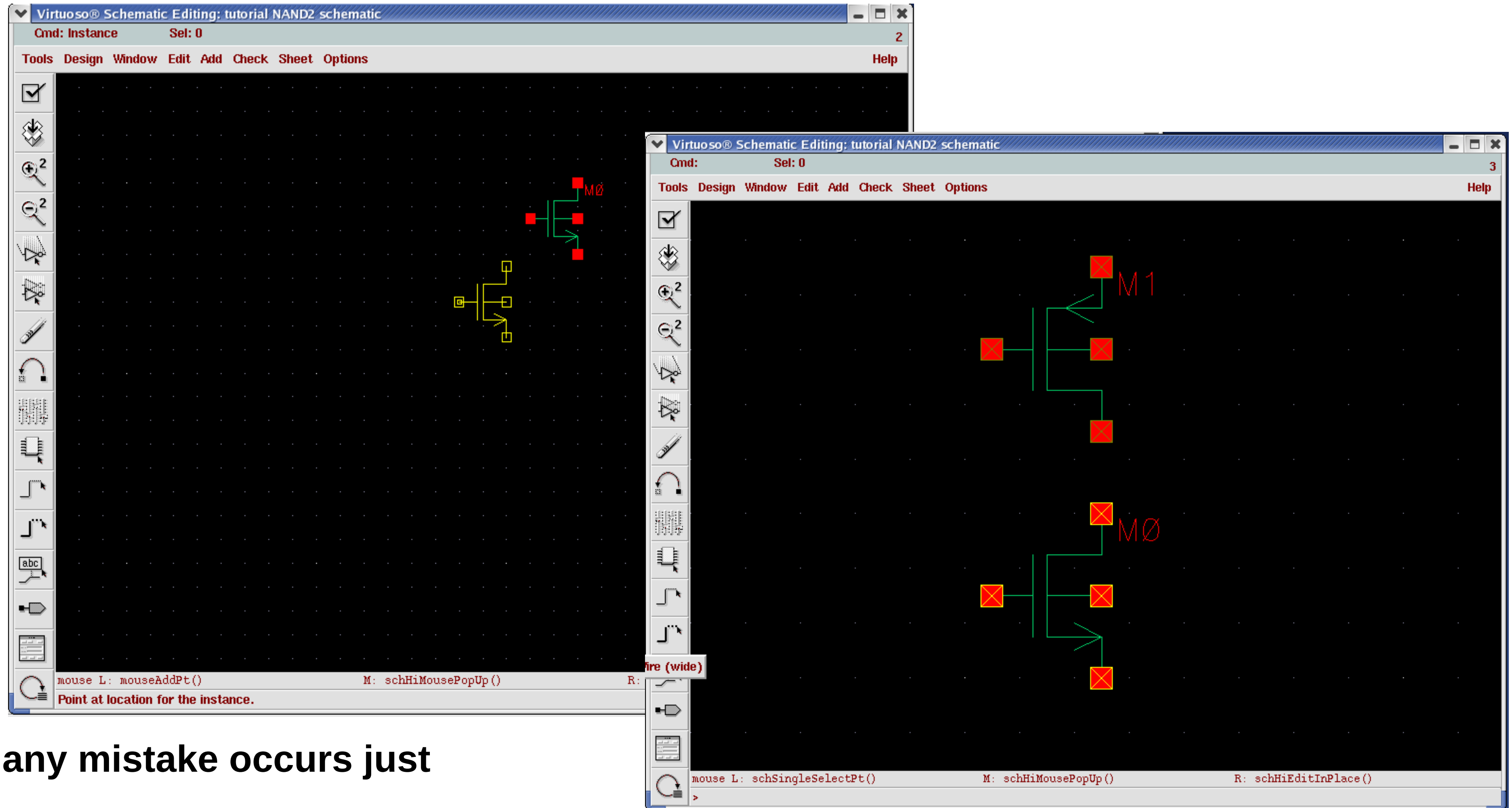
When the mouse is moved on top of the Virtuoso Schematic window an "outline" of the transistor can be seen.

The outline can be moved, rotated, flipped this outline, then by clicking the left-mouse button it can be placed in the schematic.

Multiple ones at a time can be put if required number of transistors are known ahead of time by specifying the number of rows and columns.



Adding an Instance - 4



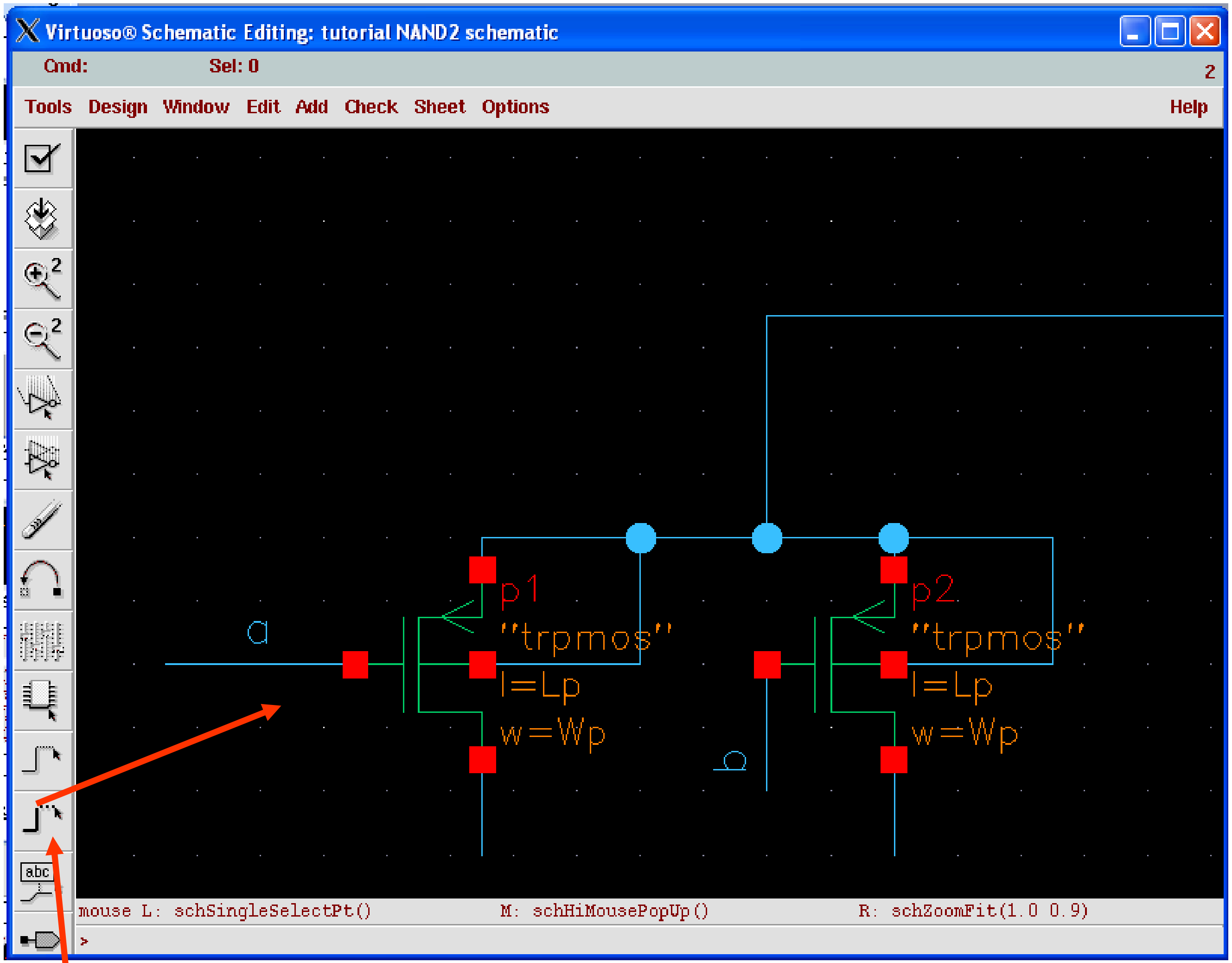
If any mistake occurs just

Edit → Undo and try again.

ESC key on the keyboard gets us out of the place instance mode

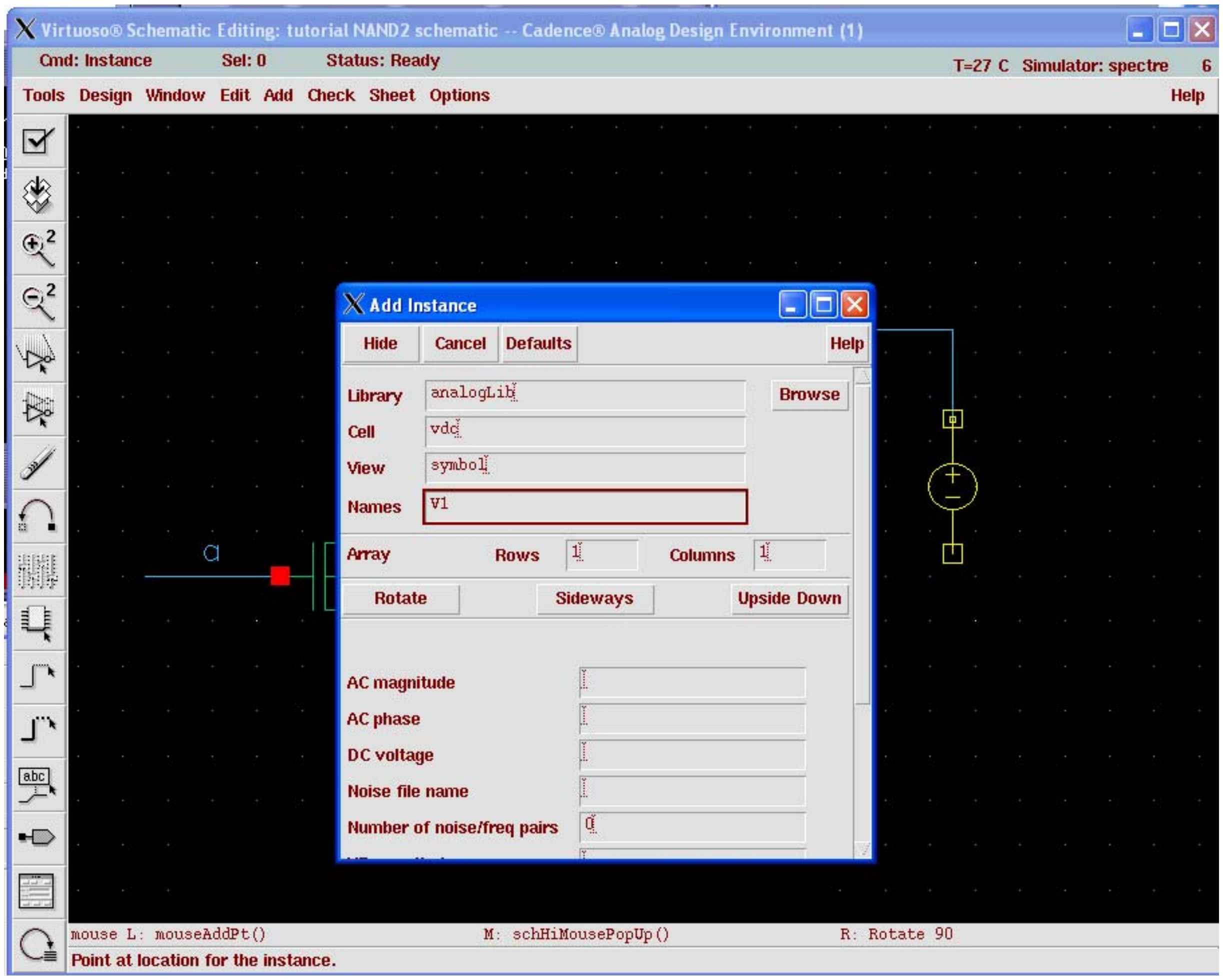
As many instances as possible can be added to the schematic

Connecting Components



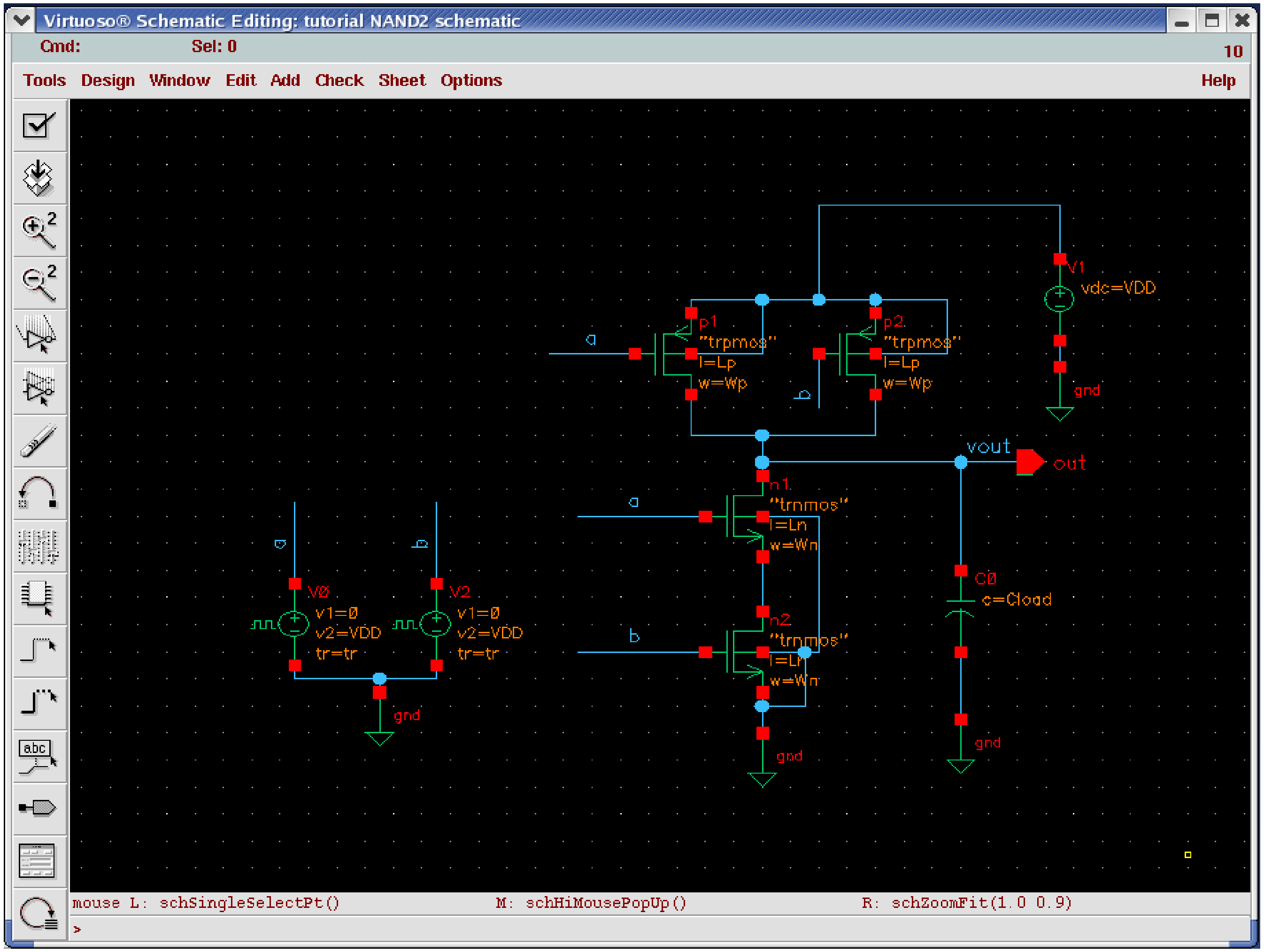
Then add wires (narrow) to connect all the transistors as required.

Adding Voltage Sources



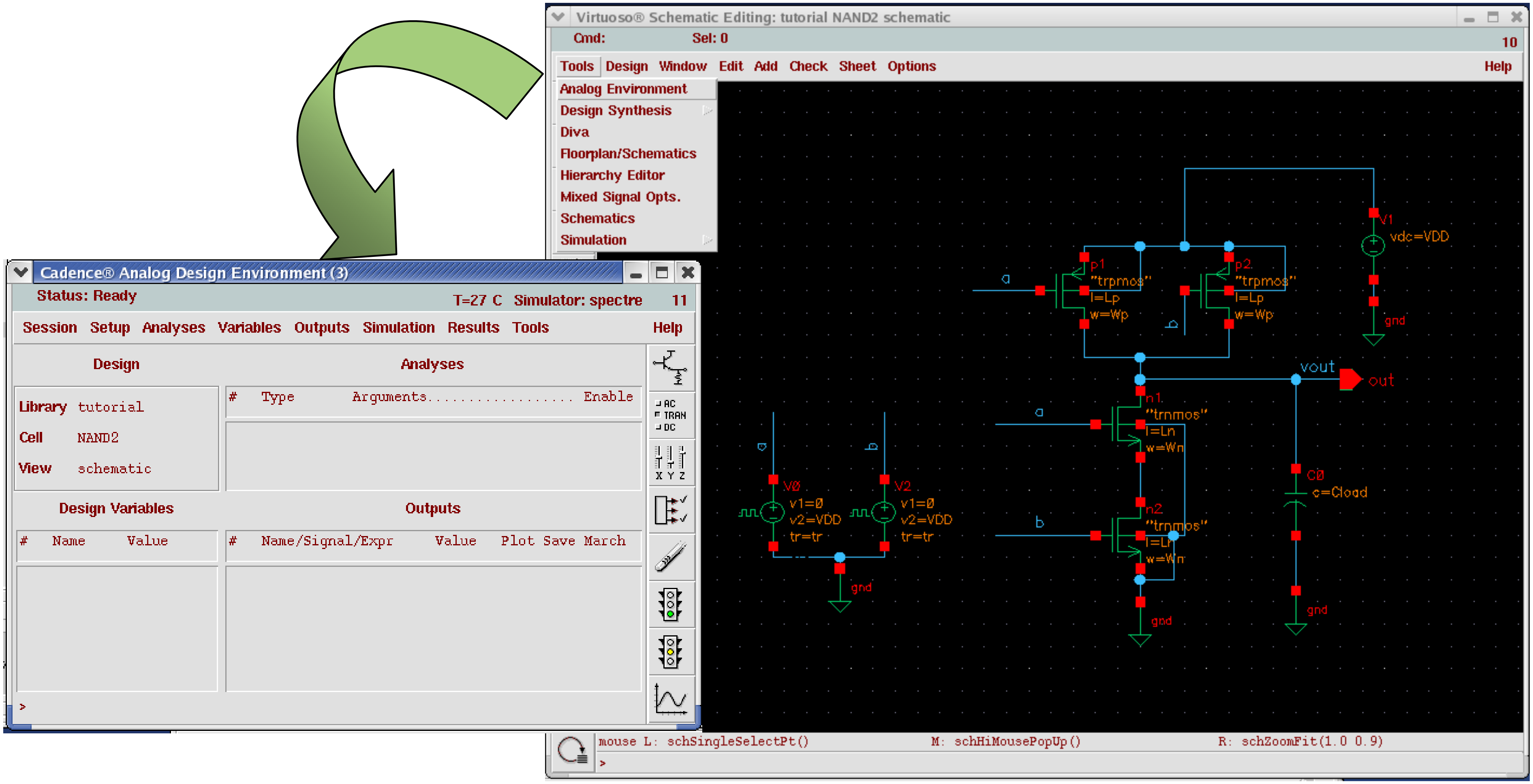
Adding a voltage source is same as adding any other instance. It can be picked up from the “vdc” cell of the “analogLib”

Final Schematic of NAND-2



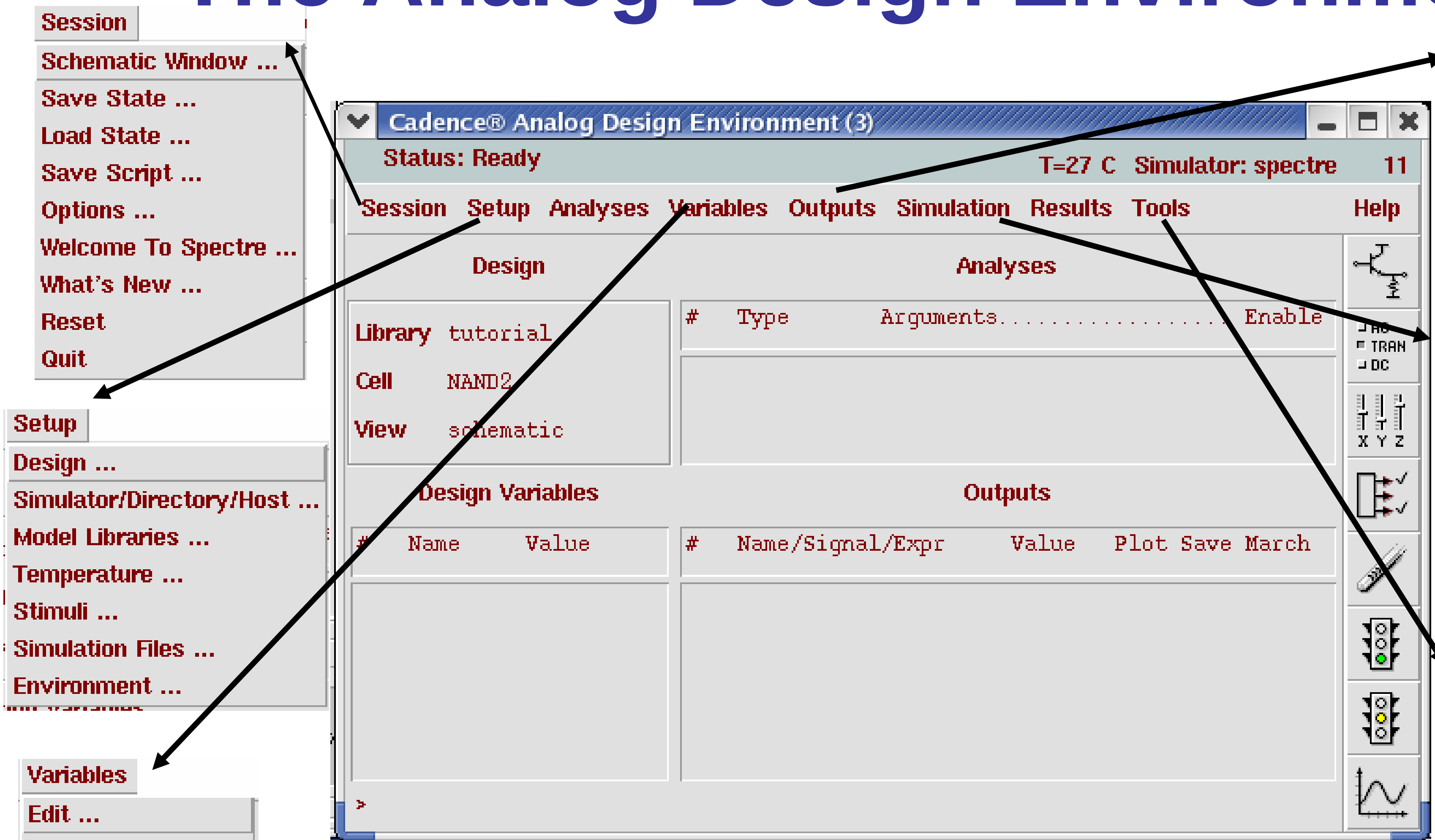
Complete Schematic Diagram and Test Bench of a NAND 2 drawn using the Virtuoso Schematic Editing tool of Cadence

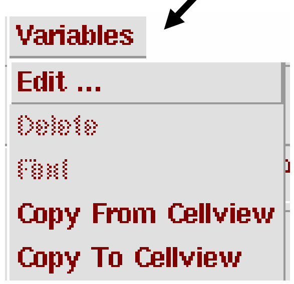
Launching Analog Design Environment

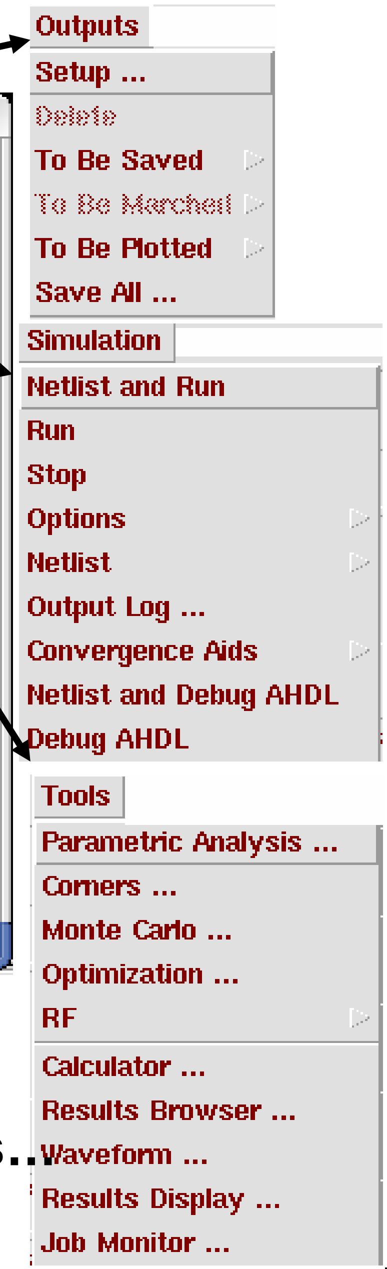


In order to launch the Analog Design Environment, choose Analog Environment from the Tools menu in the Virtuoso.

The Analog Design Environment



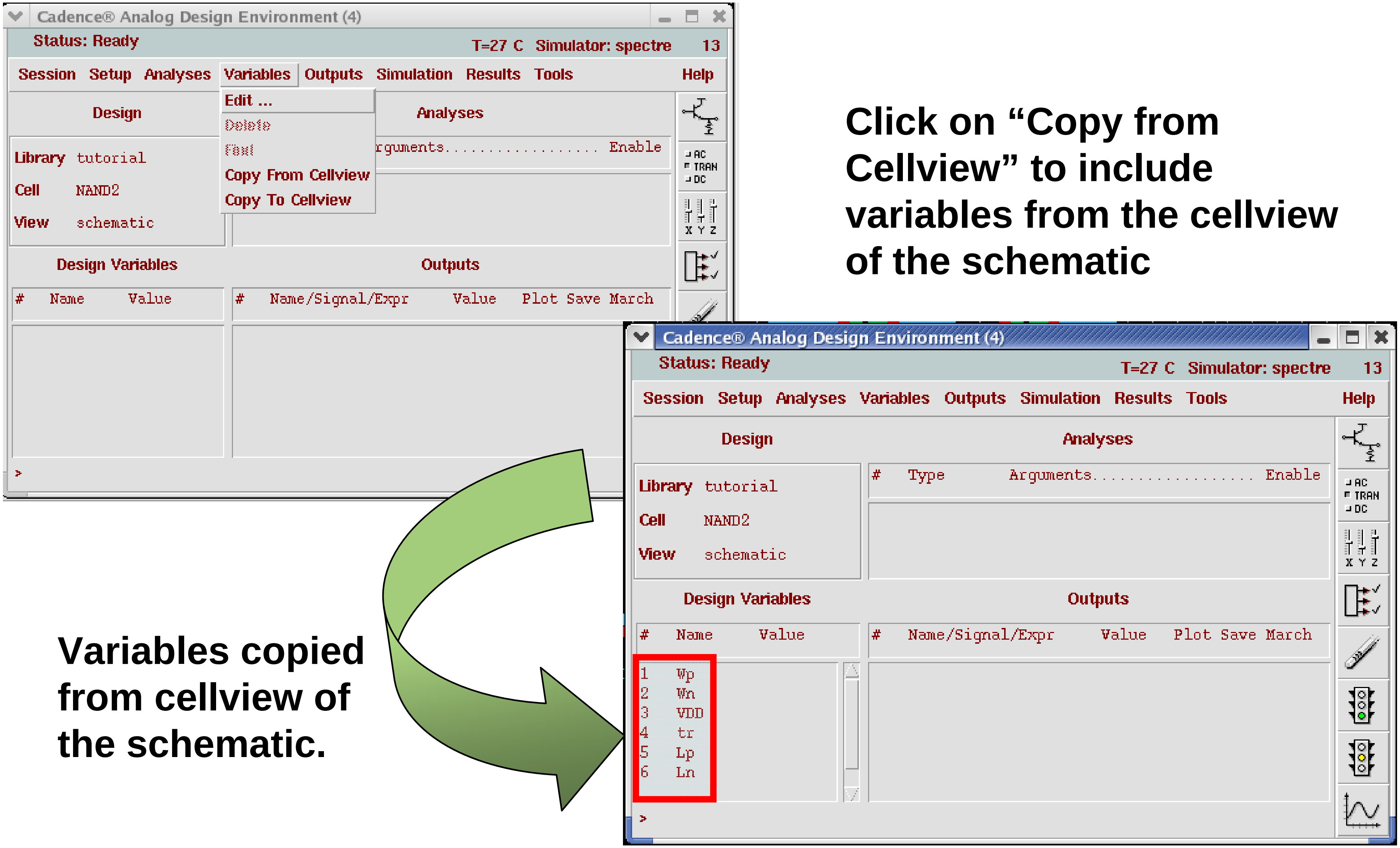




Various important menu options in the ADE include

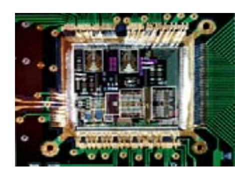
Session, Setup, Variables, Outputs, Simulation, Tools…

Adding Design Variables



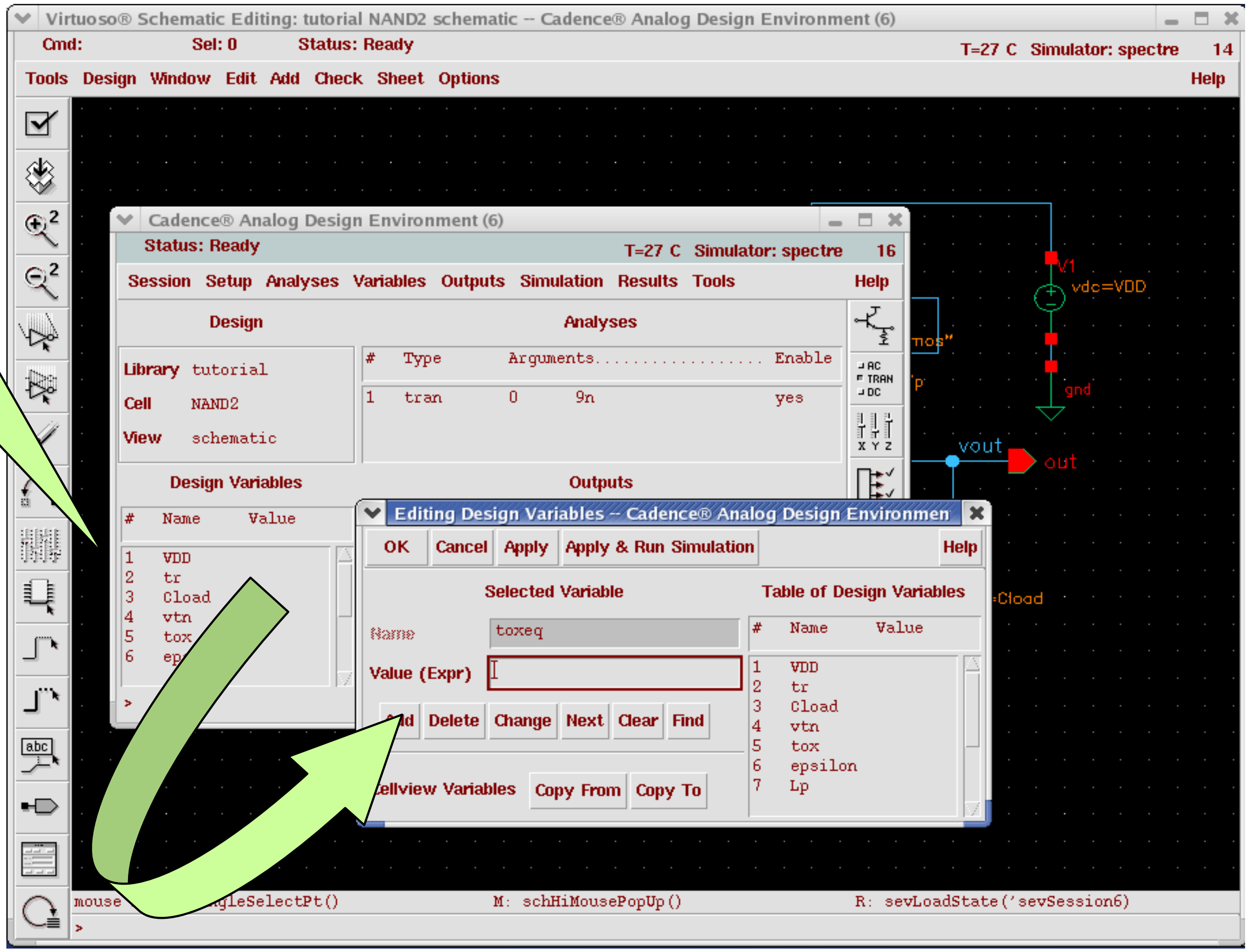
Click on “Copy from Cellview” to include variables from the cellview of the schematic

Variables copied from cellview of the schematic.



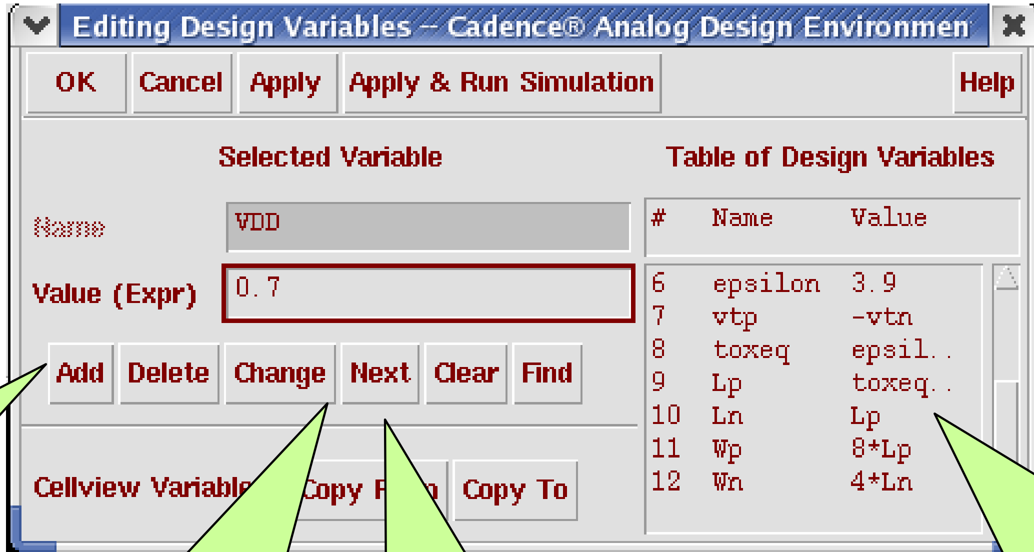
Adding and Editing Design Variables

Double click on any of the variables to launch the Design Variable Window



The Design variables can be added/edited by launching the "Edit Design Variables" window in the ADE and then giving values corresponding to the names of the variables.

Adding and Editing Design Variables



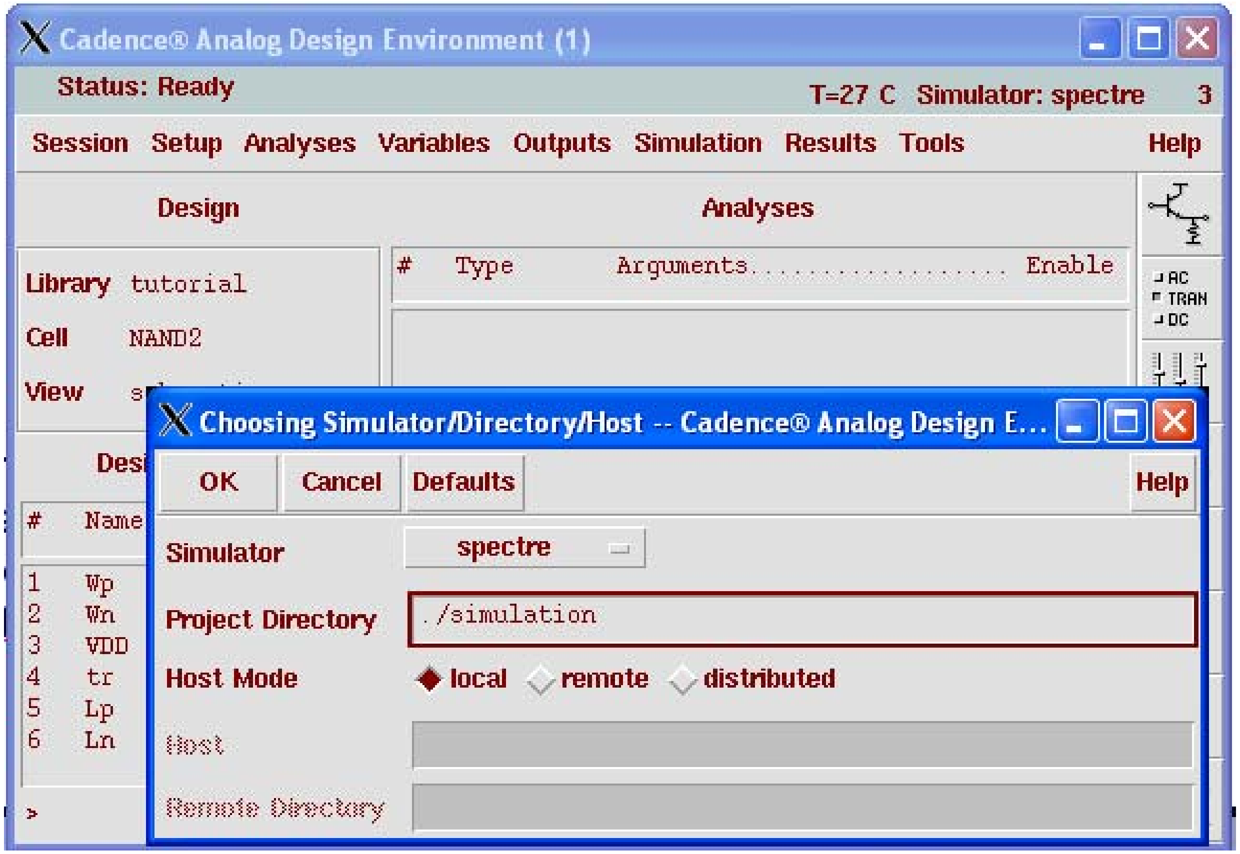
Add the variable

Don’t forget to click “Change” if you change a value.

When the value is added/changed, it can be seen in the table

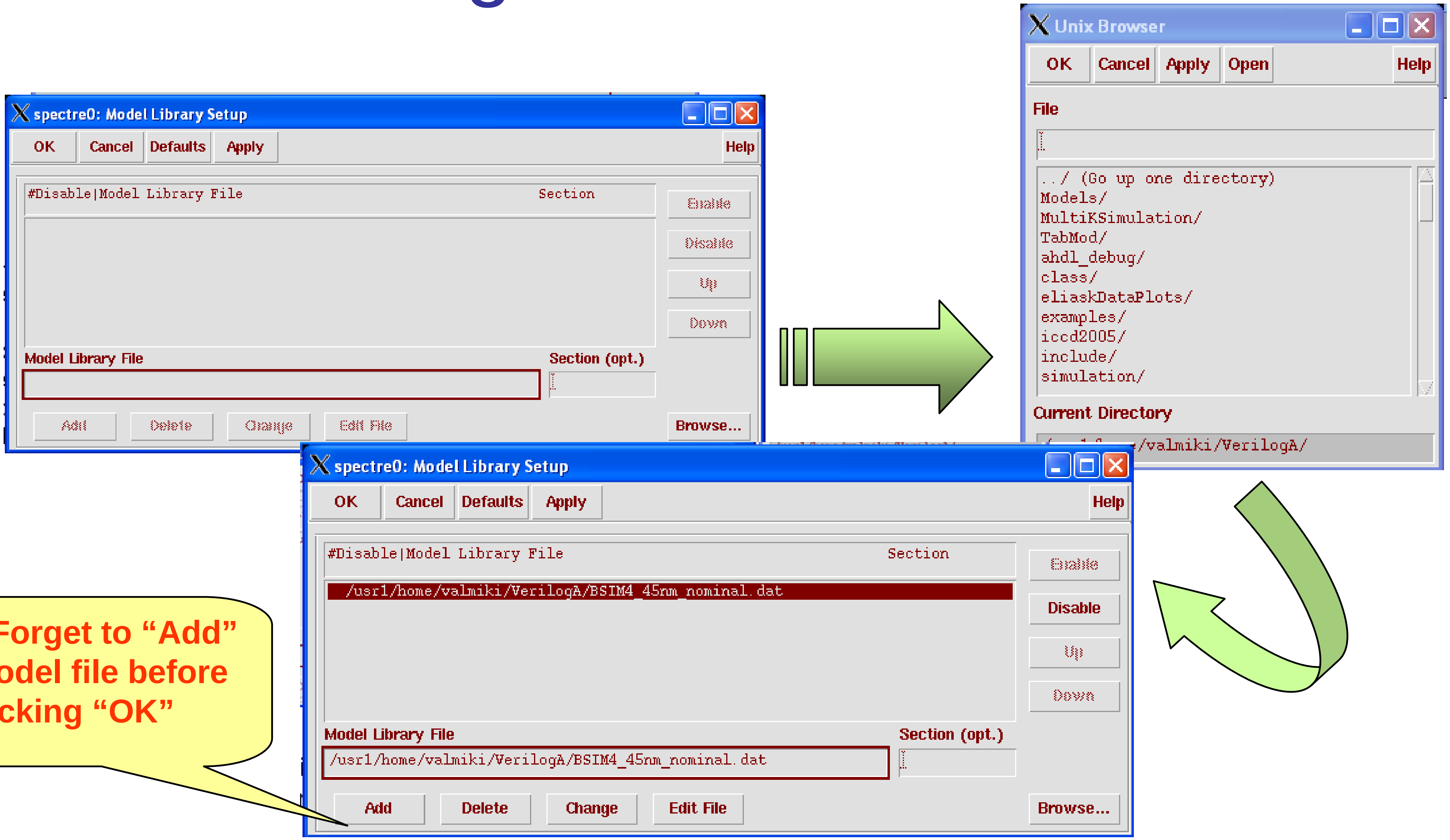
Also you can Delete,
Go to the “Next” Variable, “Clear” a value and “Find” a variable

Choosing a Simulator



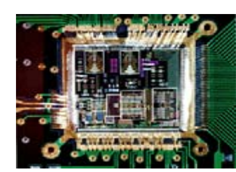
After the schematic design phase is over, we chose a simulator to simulate the design. In our case we chose Spectre.
Go to Setup -> Simulator/Directory/Host, and choose Spectre in the pop-up window, then click OK

Adding Model Libraries

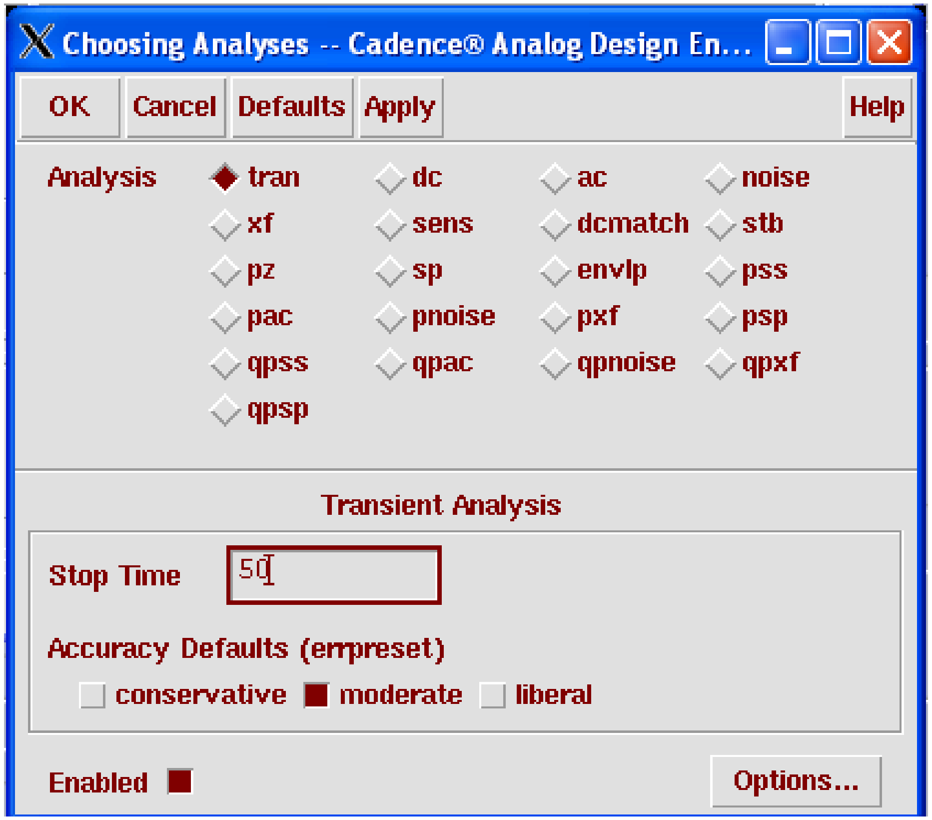


Don't Forget to "Add" the model file before clicking "OK"

Go to Setup → Model Libraries and choose (you can type directly or use Browse) an appropriate library then click Add (this is important, don't forget to do it), which adds the model for the chosen model to the simulation environment.

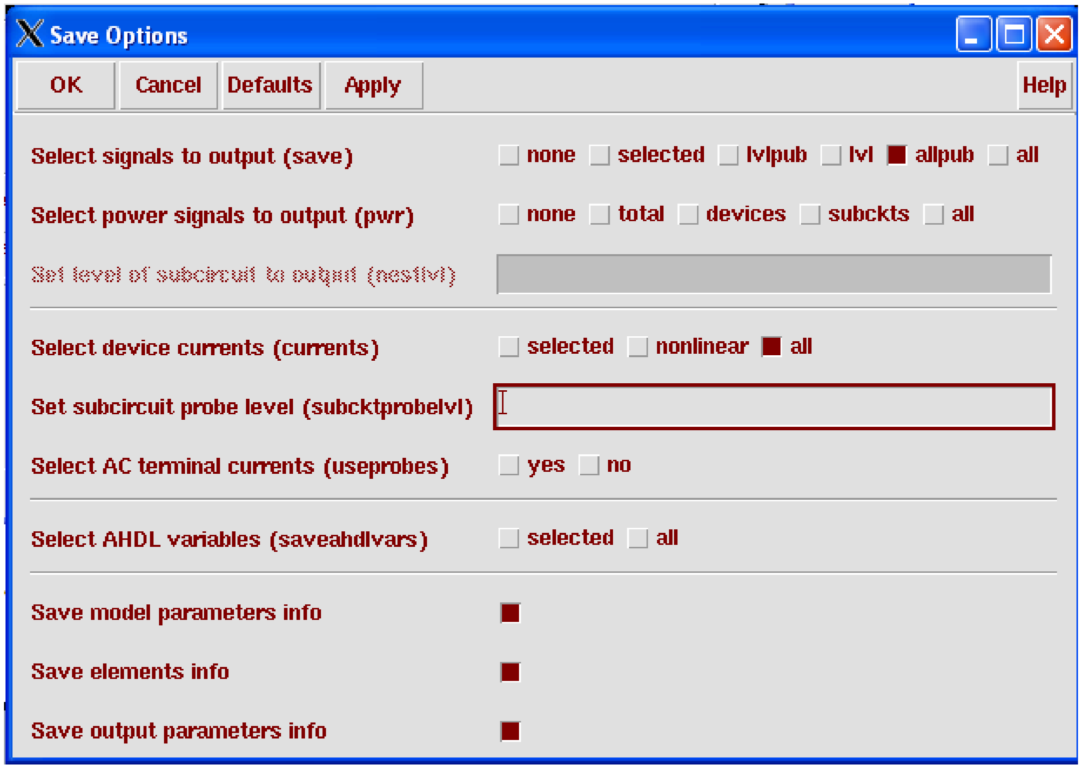


Choosing Type of Analysis



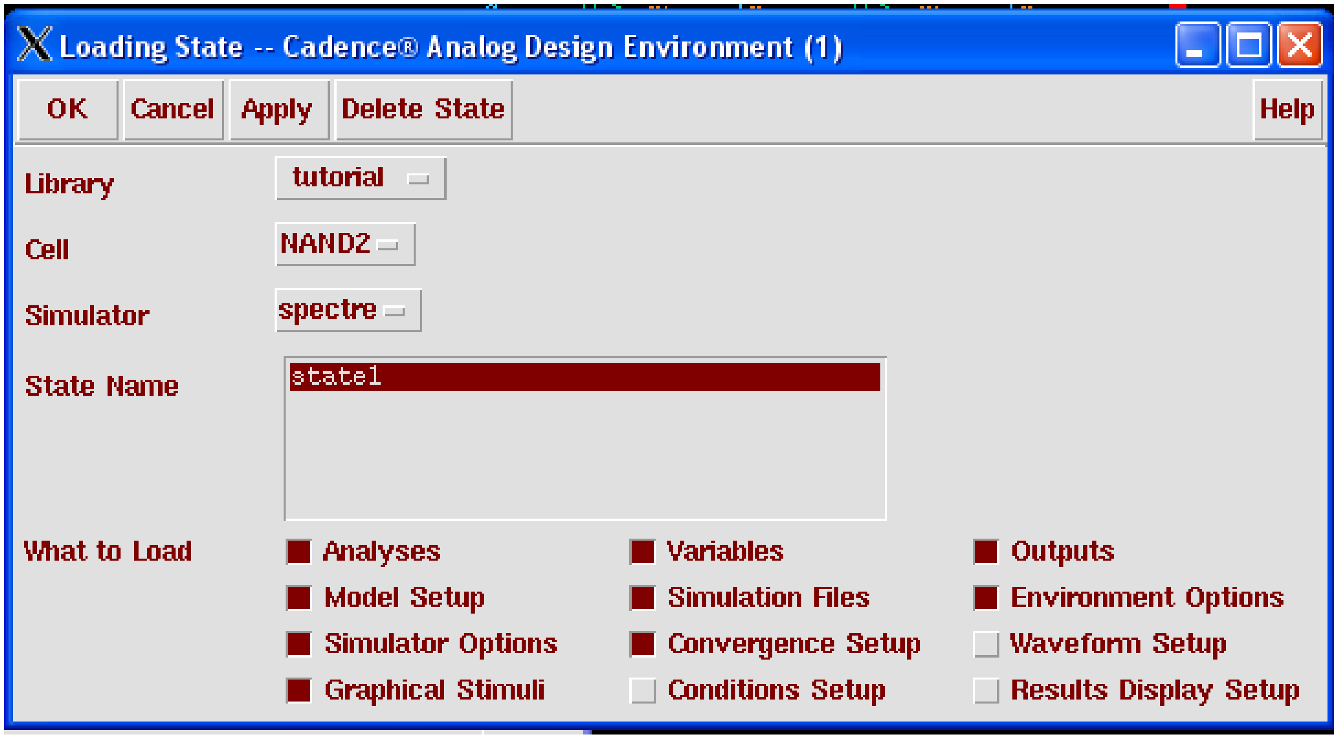
Choose the type of simulation, go to Analyses -> Choose...
We choose a "Transient" by clicking on the "tran" radio button.
We also assign a "Stop Time" here it's 50 which signifies 50 cycles.
Finally click OK.

Save Options for a State



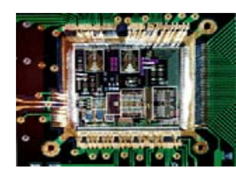
Now go to Outputs -> Save All and click on allpub for signals to save (default). In general, for large schematics, you want to save only a subset of signals so that you save computing resources, but this schematic is small enough that it is OK to just save all. Click OK.

Loading Saved States

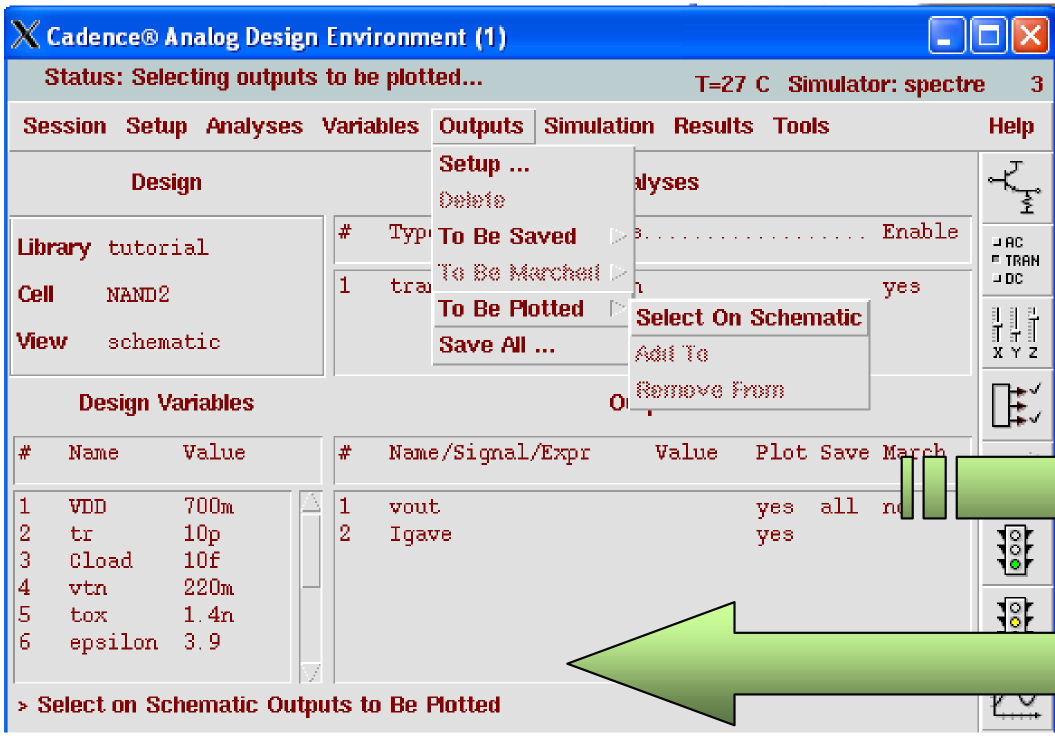


A saved state for a schematic contains all the information that has been saved regarding the schematic, including the Library, Cell Name, the Simulator and the information to be loaded.

Just choose the state to be loaded and click "OK"

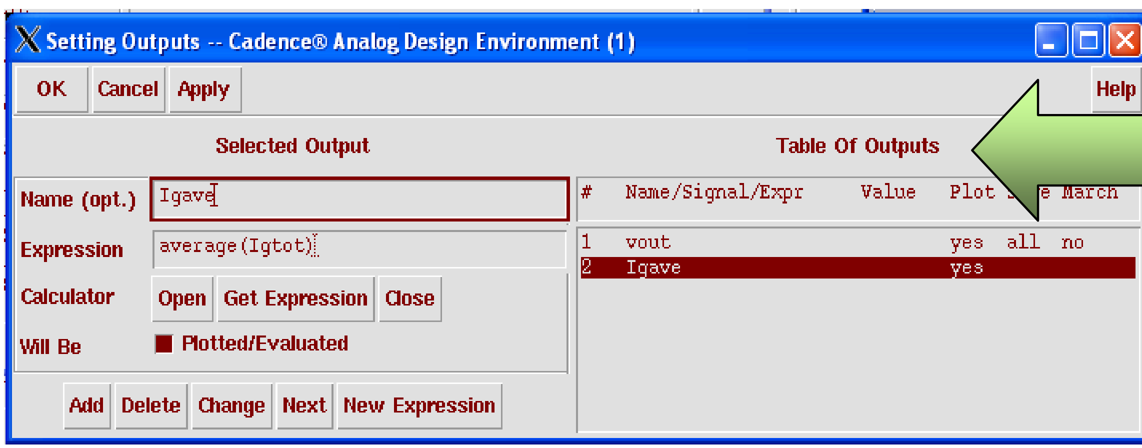


Setting up Outputs

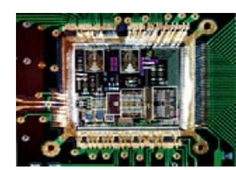


You can specify the Outputs to be plotted or saved from the Outputs menu option.

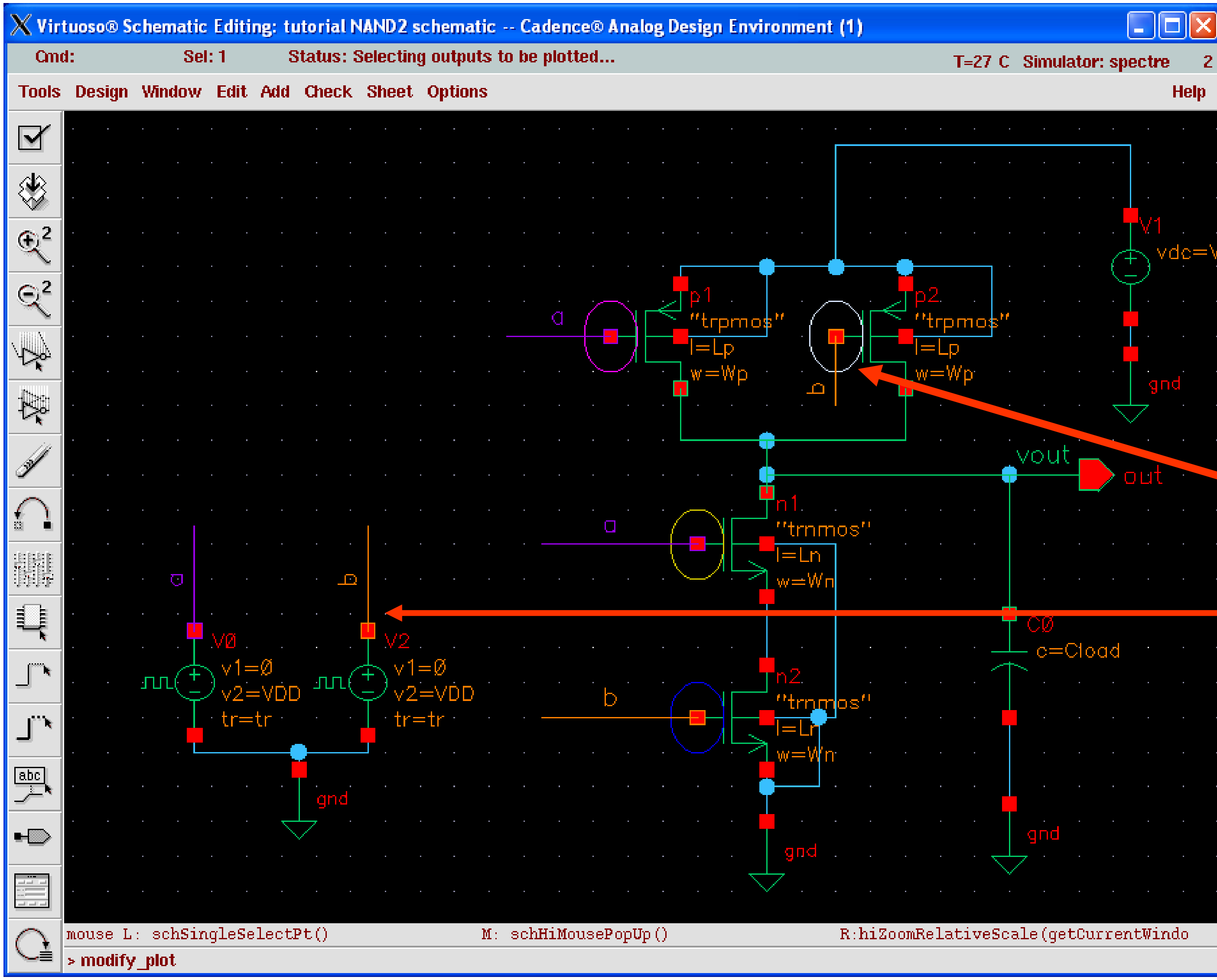
Initially you can choose them from the schematic



Then you can use the Setting Outputs window to add/edit them.



Choosing Outputs from Schematic



For choosing voltage click on the wire.

For choosing current click on the point/node.

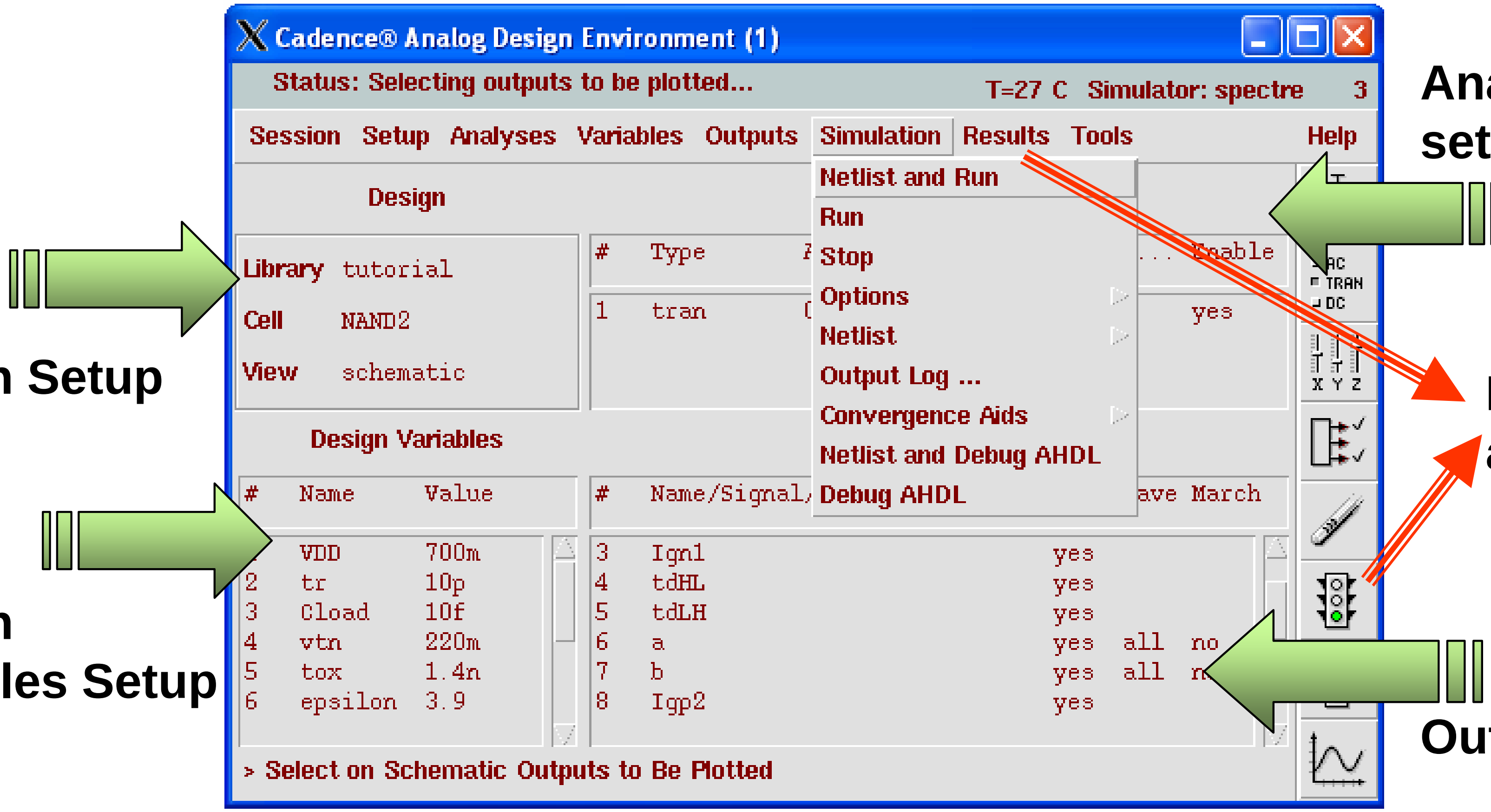
A choice for a current node is shown by a elliptical ring and choosing a voltage changes the color of the wire. These colors correspond to the color of corresponding line plotted in the simulation.

Starting Simulation

Analysis setup

Design Setup

Design Variables Setup



Netlist and Run

Outputs Set

When all the setup is complete, you are ready for the Simulation.

Simulation can be started either from the menu as shown or by clicking the “Green Traffic Light” button from the right hand toolbox

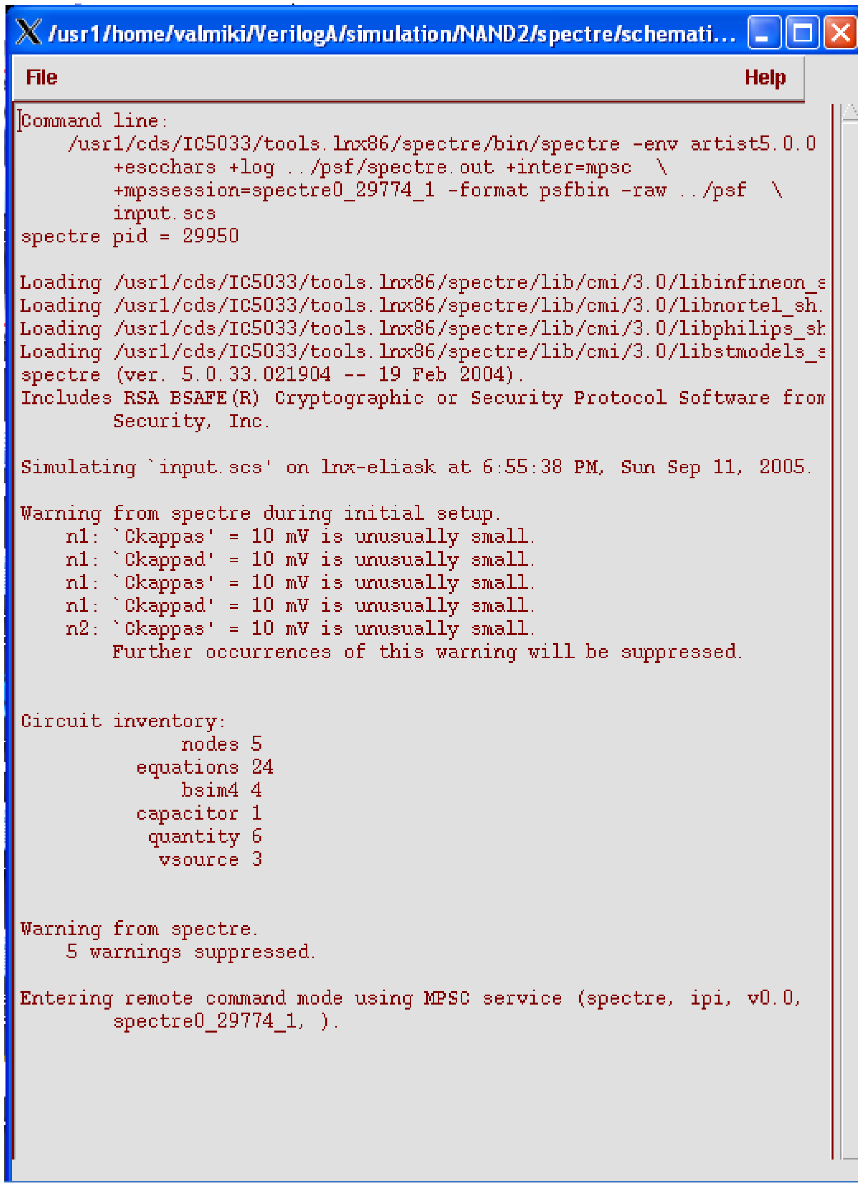
Simulation Results

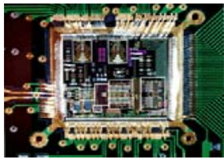
When the Simulation is done and there are no errors in the Simulation. You can see a window with the results of the simulation as shown here.

With an error free simulation, next you can see the “Waveform Window”

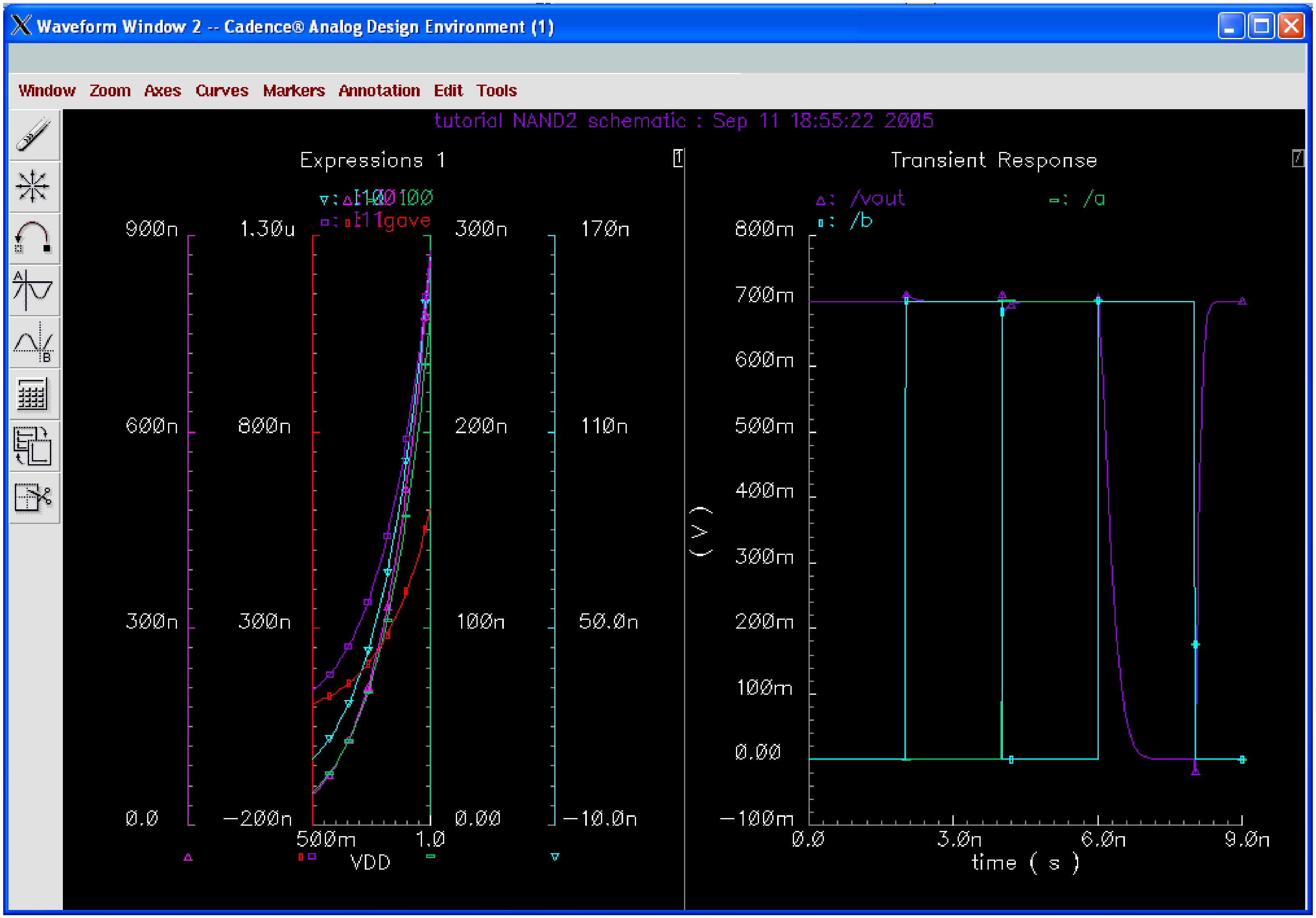
If you have any errors, you can always go back to the “Virtuoso Schematic Editor” window and correct them

Always remember to do a “Check and Save” when you are done with any correction.

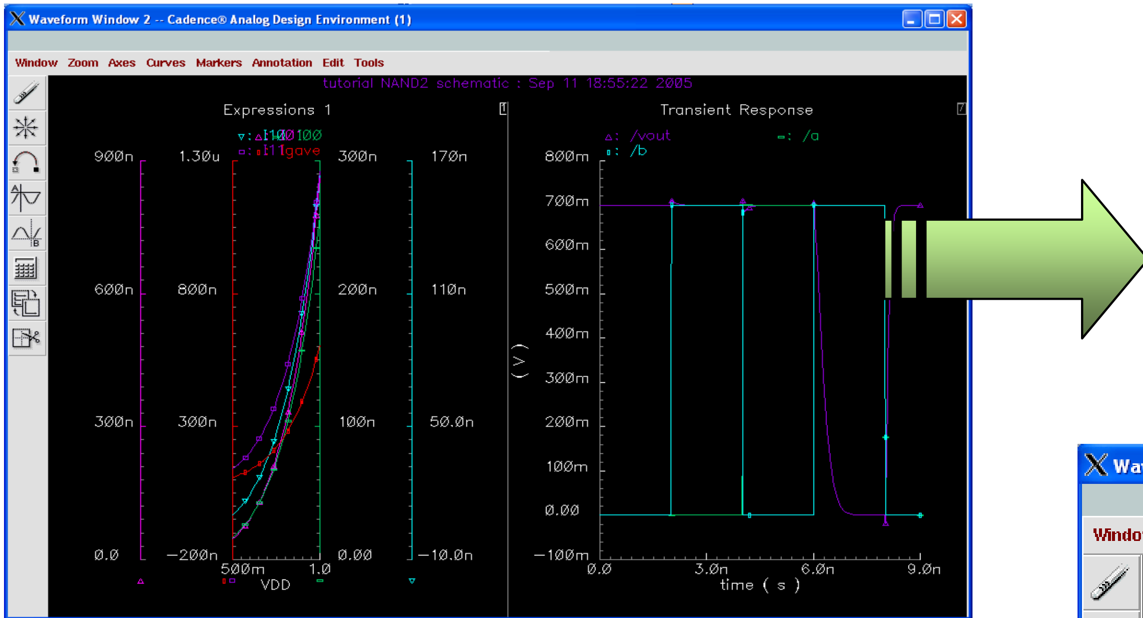

/usr1/home/valmiki/VerilogA/simulation/NAND2/spectre/schemati...
File
Help
```
Command line:
    /usr1/cds/IC5033/tools.lnx86/spectre/bin/spectre -env artist5.0.0
        +escchars +log ../psf/spectre.out +inter=mpsc  \
        +mpssession=spectre0_29774_1 -format psfbin -raw ../psf  \
        input.scs
spectre pid = 29950

Loading /usr1/cds/IC5033/tools.lnx86/spectre/lib/cmi/3.0/libinfineon_s
Loading /usr1/cds/IC5033/tools.lnx86/spectre/lib/cmi/3.0/libnortel_sh.
Loading /usr1/cds/IC5033/tools.lnx86/spectre/lib/cmi/3.0/libphilips_sh
Loading /usr1/cds/IC5033/tools.lnx86/spectre/lib/cmi/3.0/libstmodels_s
spectre (ver. 5.0.33.021904 -- 19 Feb 2004).
Includes RSA BSAFE(R) Cryptographic or Security Protocol Software from
        Security, Inc.

Simulating `input.scs' on lnx-eliask at 6:55:38 PM, Sun Sep 11, 2005.

Warning from spectre during initial setup.
    n1: `Ckappas' = 10 mV is unusually small.
    n1: `Ckappad' = 10 mV is unusually small.
    n1: `Ckappas' = 10 mV is unusually small.
    n1: `Ckappad' = 10 mV is unusually small.
    n2: `Ckappas' = 10 mV is unusually small.
        Further occurrences of this warning will be suppressed.


Circuit inventory:
              nodes 5
          equations 24
              bsim4 4
          capacitor 1
           quantity 6
            vsource 3


Warning from spectre.
    5 warnings suppressed.

Entering remote command mode using MPSC service (spectre, ipi, v0.0,
        spectre0_29774_1, ).
```

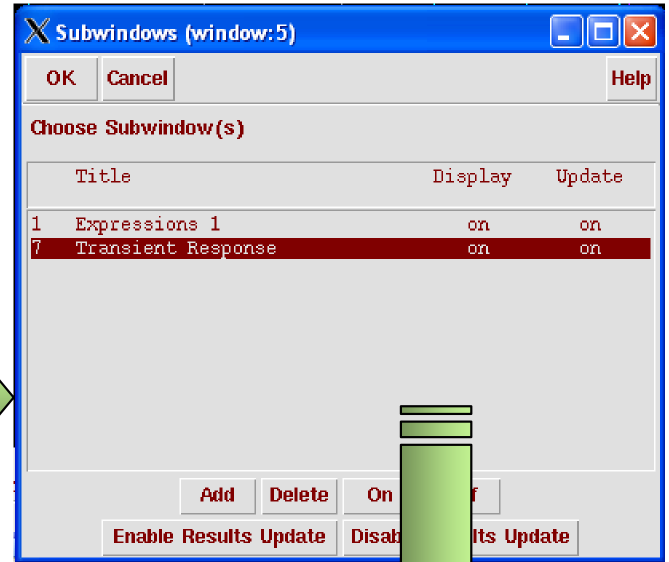



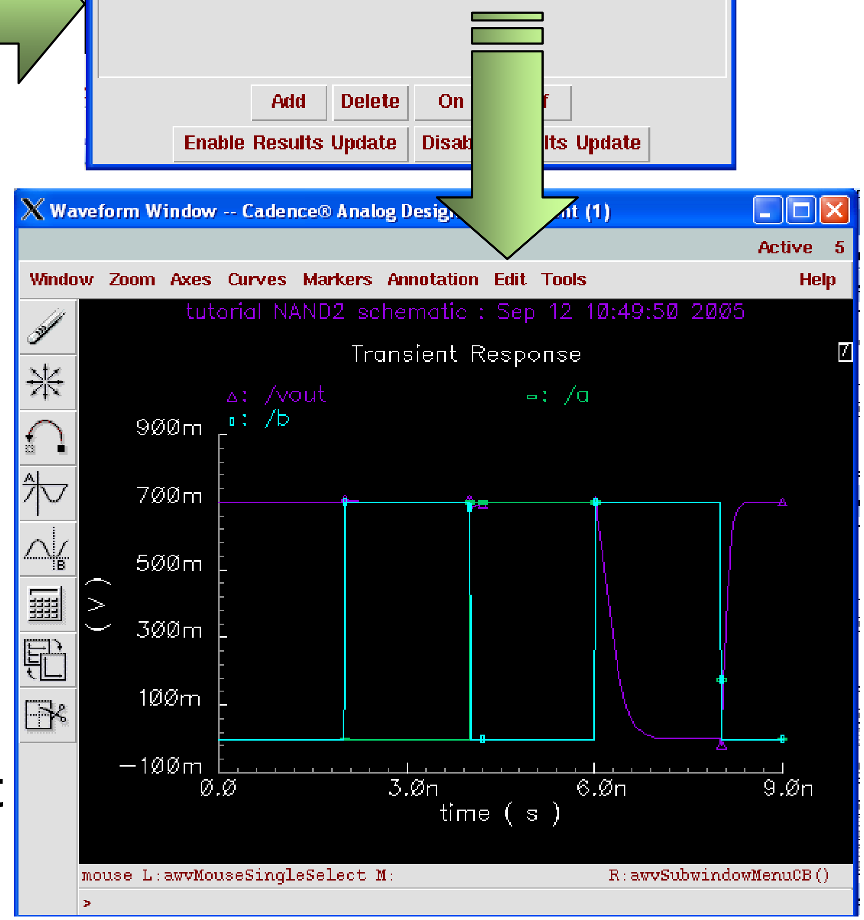
Waveform Window



Waveform Sub-windows

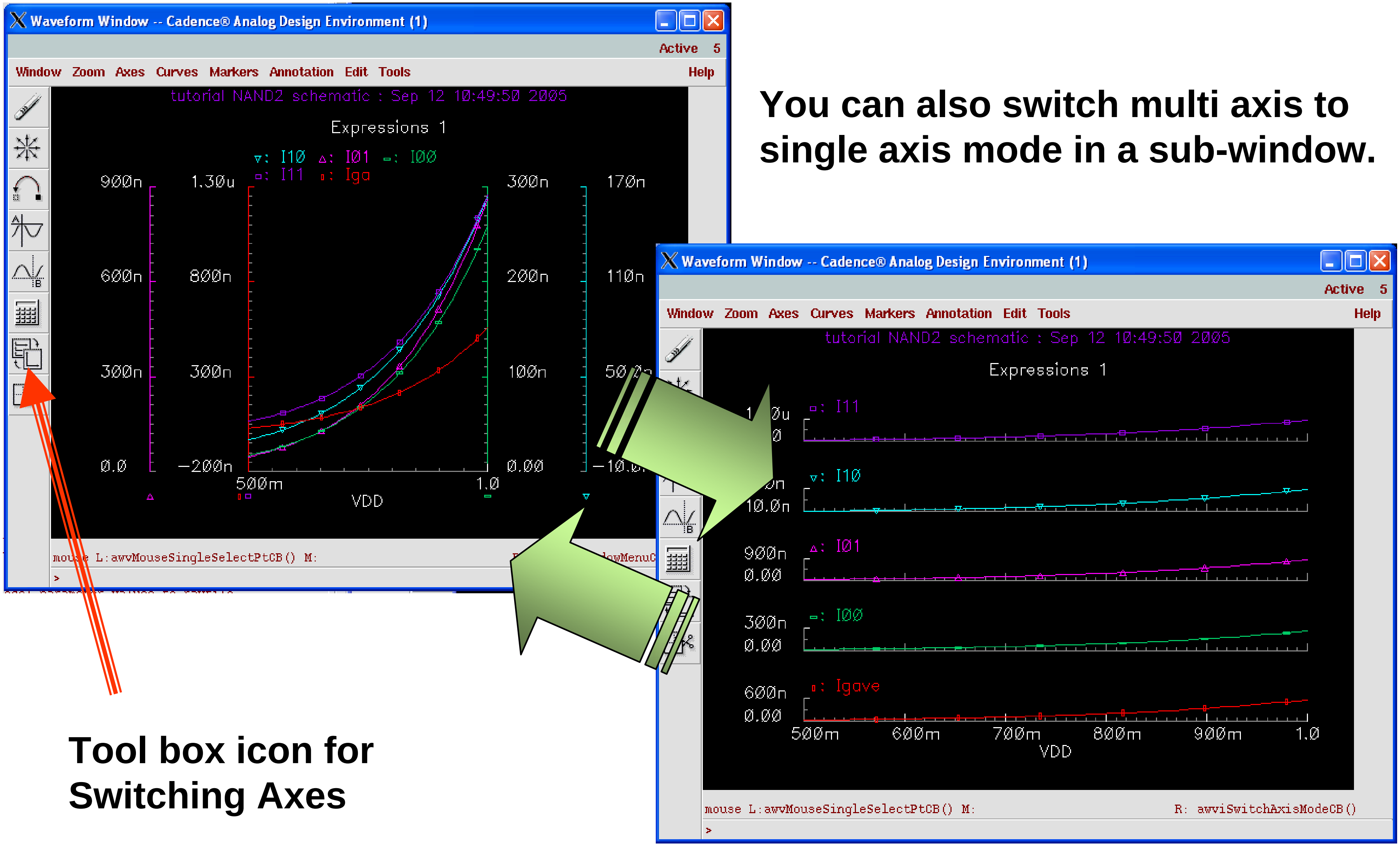






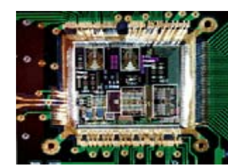
In case of multiple expressions generating numerous sub-windows, you can get the sub-window you want by choosing Windows→Subwindows
Which launches the sub-window management box. Just turn “ON” or “OFF” the display for the particular window or just “Add” or “Delete” them. Then press “OK”

Switching Axis Modes

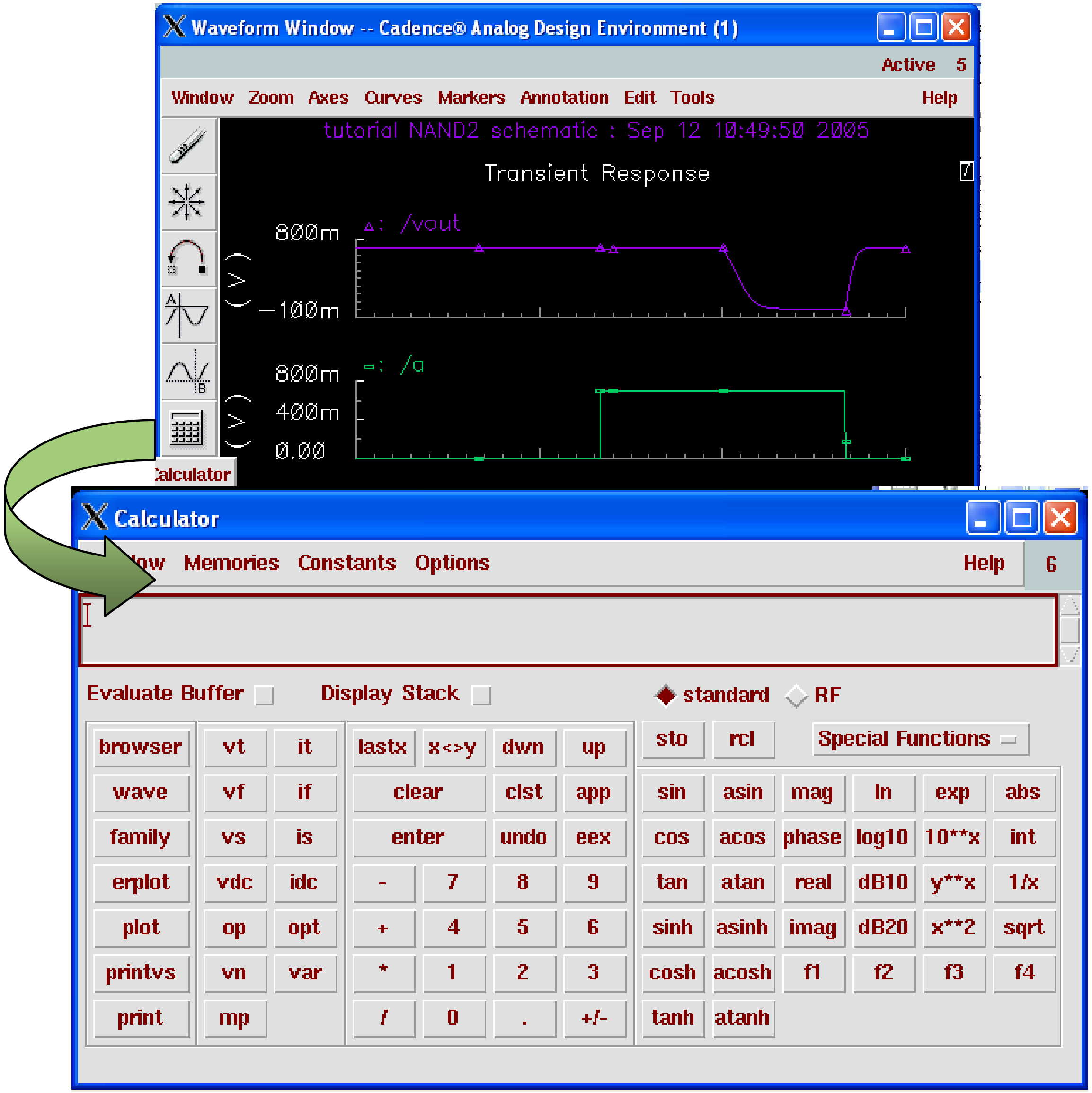


You can also switch multi axis to single axis mode in a sub-window.

Tool box icon for Switching Axes



The Calculator

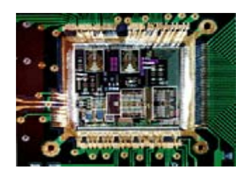


The calculator is an extremely efficient tool that can be used to perform various operations.

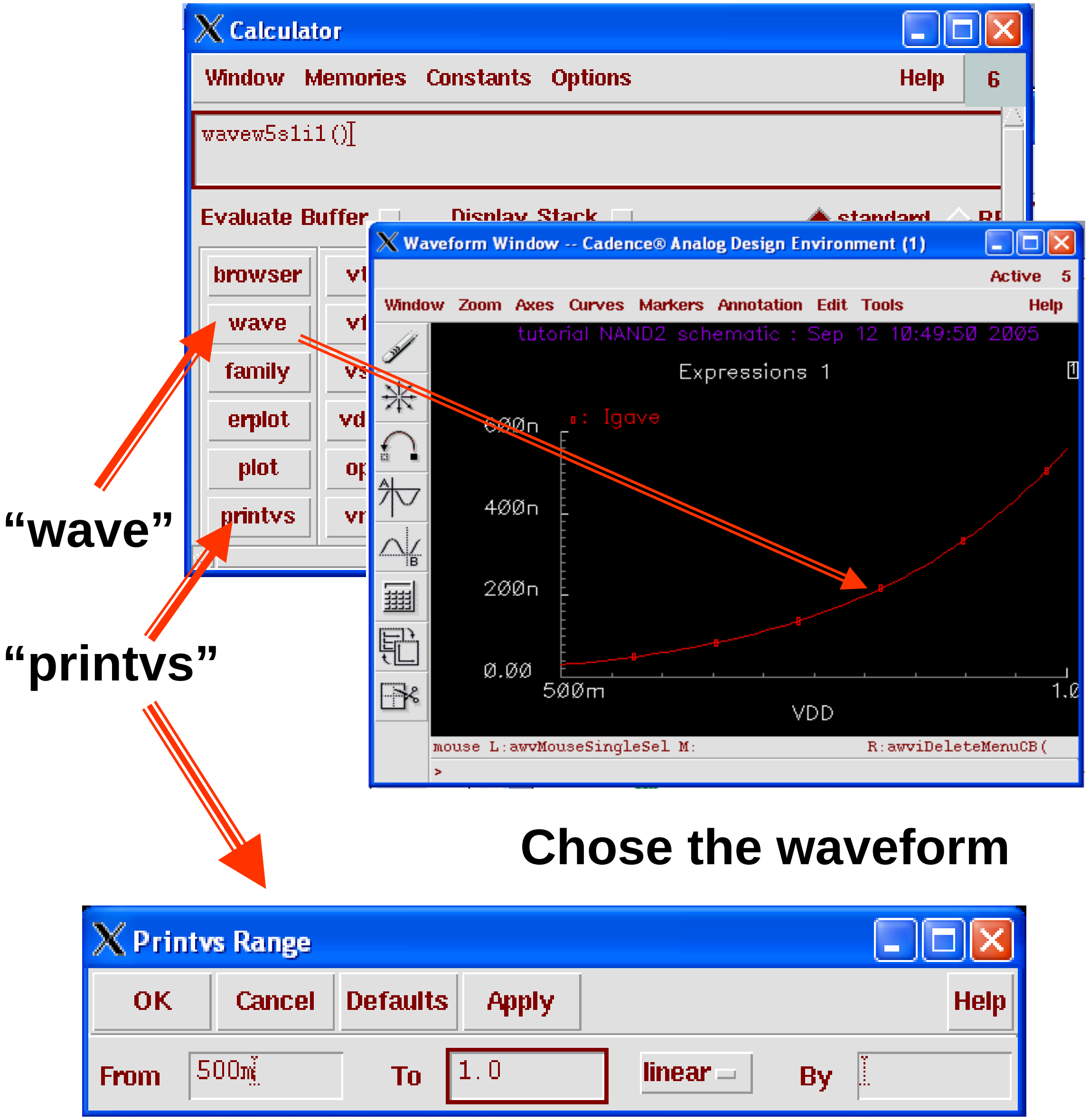
You can launch the calculator by clicking on the calculator icon on the tool box on the left of the "Waveform Window"

It can be used to draw and plot the desired waveforms and evaluate and set up variables and expressions.

An efficient use of the calculator can make a lot of difference to the way you design and analyze the circuits.



Using the Calculator

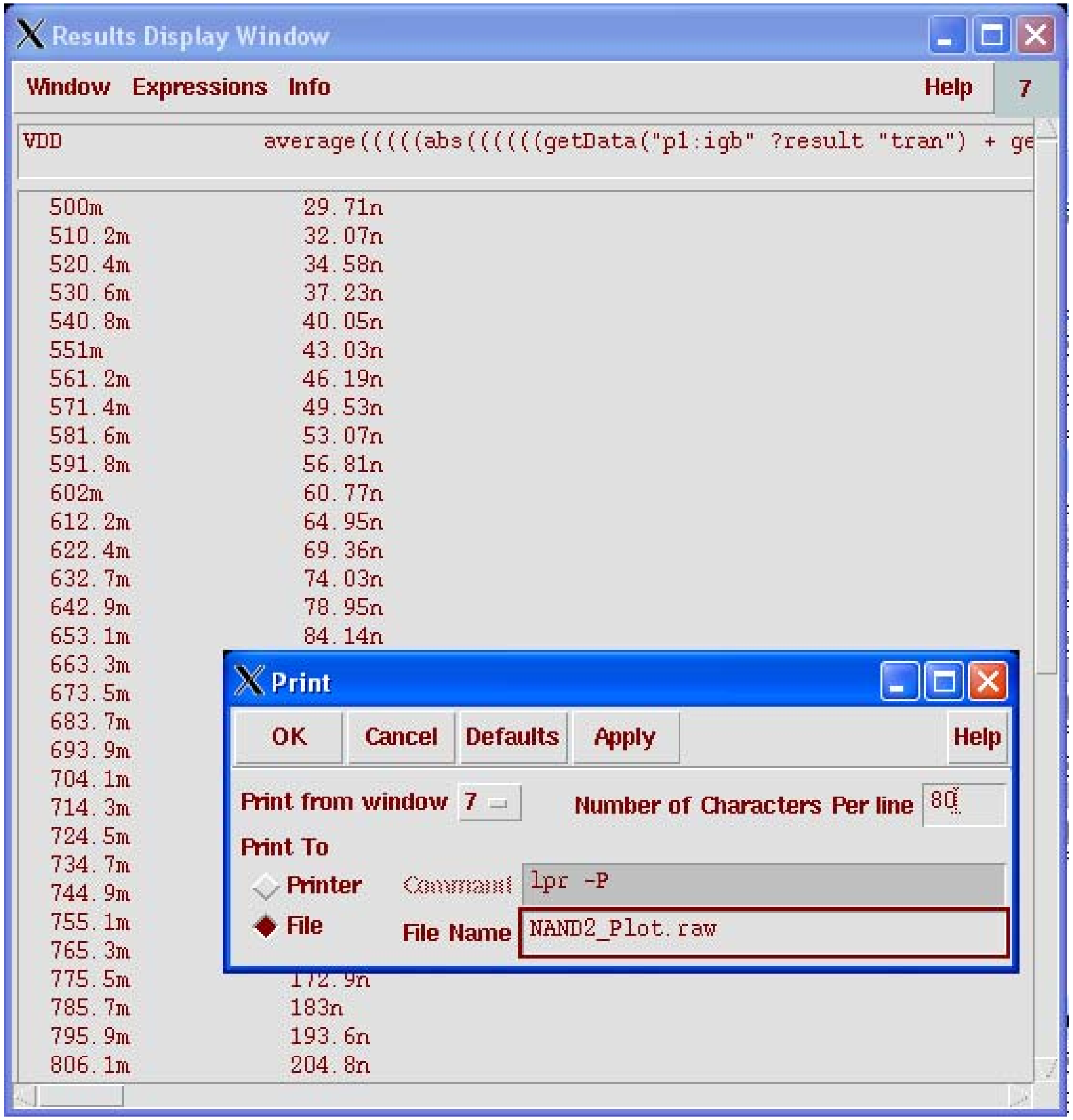


Chose the waveform

Using the calculator we can save the data from a waveform and use it later.

1. Click on “wave” in “Calculator”
2. Click on the desired wave on the “Waveform Window”
3. Click on “printvs”
4. In the “Printvs Range” window give the range.
5. Click OK
6. You will get the “Results Display” window

Saving Waveform Data



After giving the range in the "Printvs Range" window, you click "OK" you get the adjoining "Results Display Window"

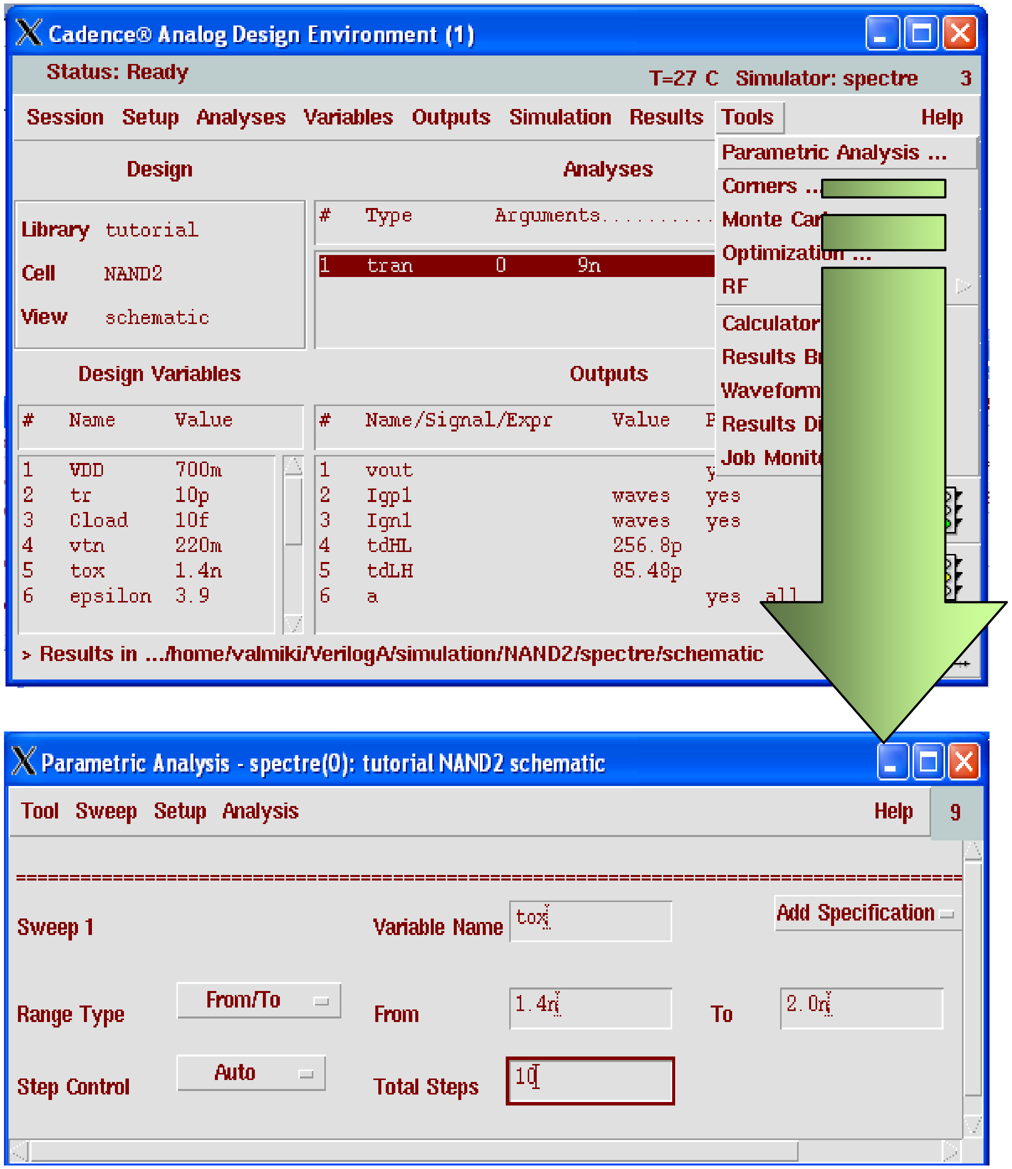
The data from this window can be either printed directly to a printer or it can be printed to a file.

Choose,

Print To → File option and give the file name.

Your simulation data is now saved!

Using Parametric Analysis



Parametric Analysis allows you to run automatic simulations for over a range of values of a specified parameter.

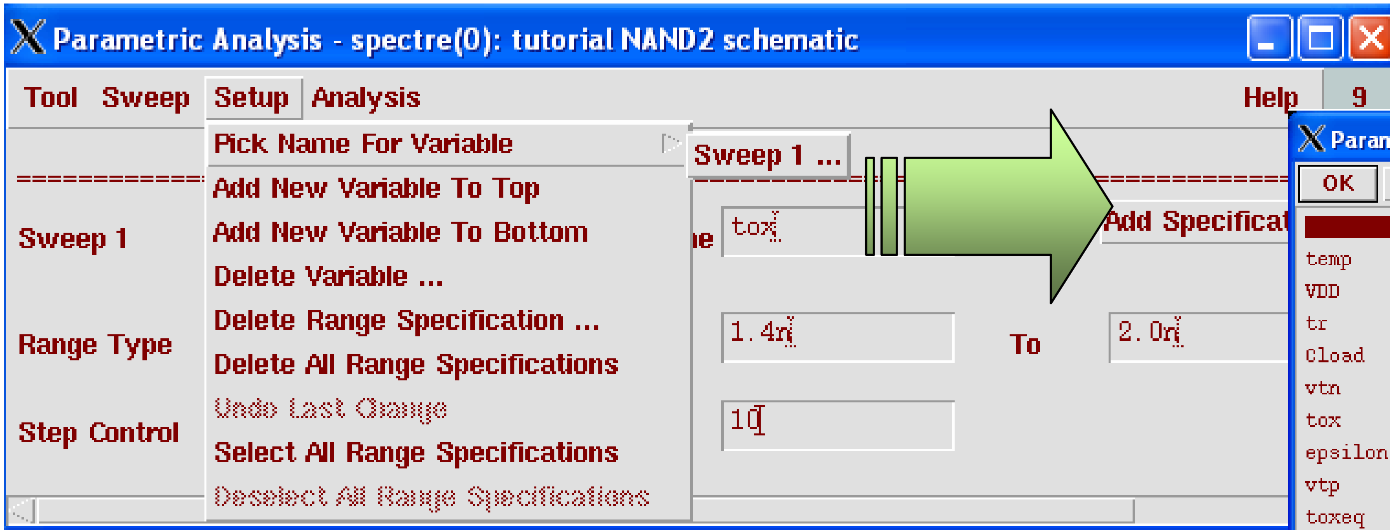
You can launch the parametric analysis tool from the Cadence® Analog Design Environment Tools menu.

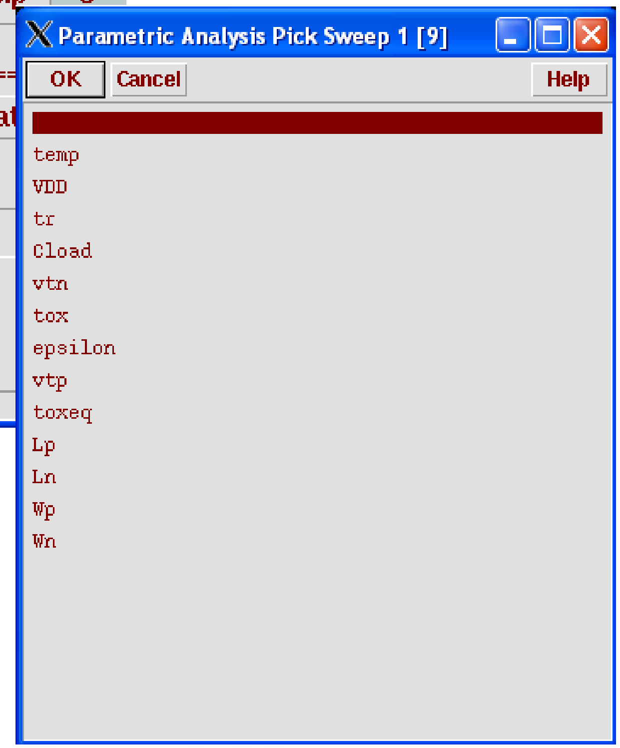
Choose:

Tools → Parametric Analysis

Specify the "Variable Name" on which you want to run the analysis. Specify it's range and Total Steps and click "OK"

Setting up Parameter





In order to run Parametric Analysis chose the variables from

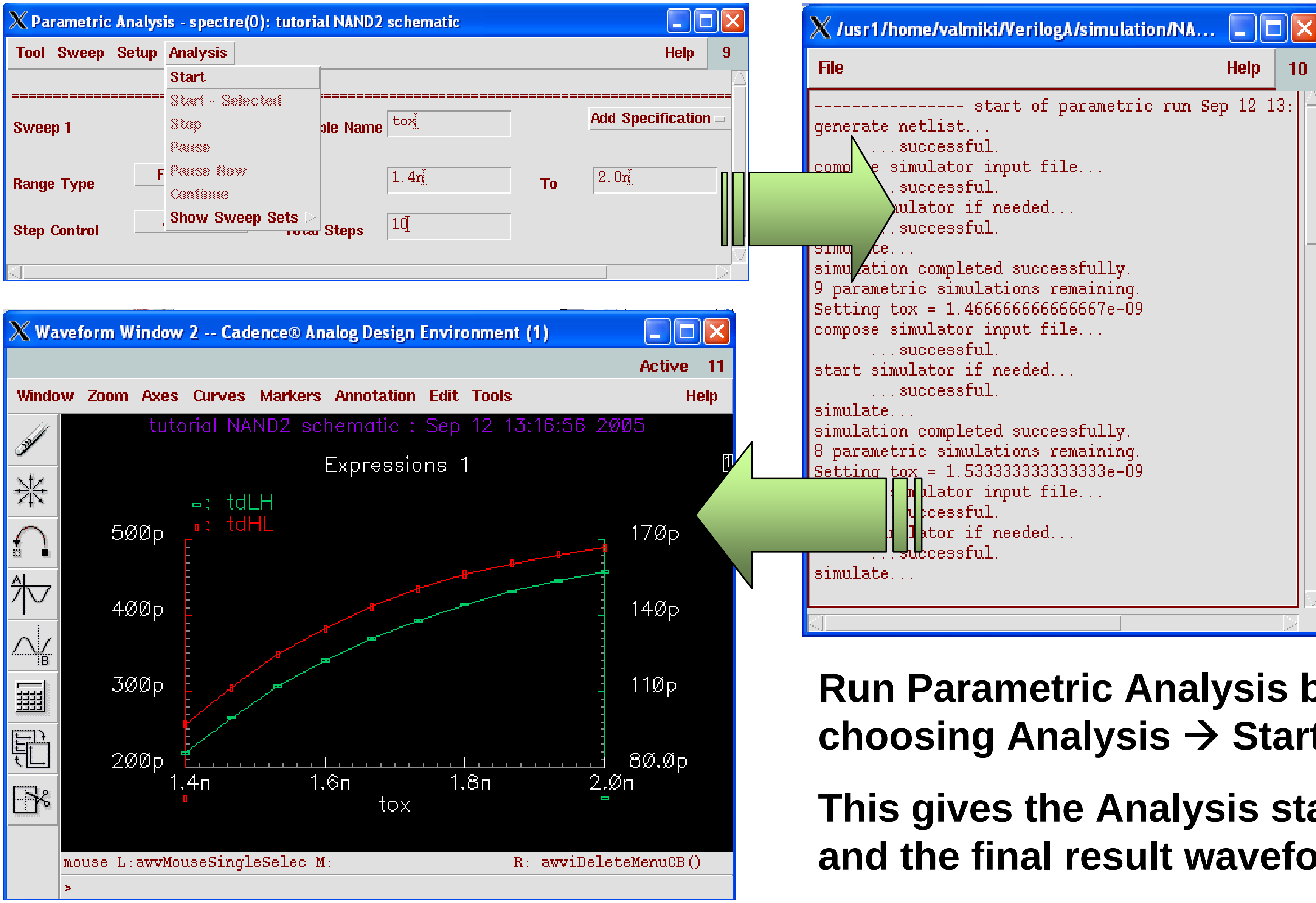
Setup → Pick Name For Variables → Sweep1

And click "OK"

This will fill the variable field.

You can choose multiple parameters for multiple sweeps.

Performing Parametric Analysis



Run Parametric Analysis by choosing Analysis → Start.

This gives the Analysis status and the final result waveform.

Summary

- In this presentation we learnt how to use the various components of the ICFB design environment of Cadence®
- We also discussed in detail the features of
  - The Virtuoso® Schematic Editor
  - The Calulator tool,
  - The Waveform Window and
  - The Parametric Analysis tool.

References

- Cadence Design Systems Manual
- Cadence Tutorials from various sources:
  - University of Virginia
  - Virginia Tech
  - Portland University
  - Worcester Polytechnic Institute
  - And more…!