# Virtuoso Schematic Composer

Virtuoso Schematic Composer is a schematic design tool from Cadence. In this tutorial you will learn how to put electrical components, make wire connections, insert pins and check for connection error.

Start Cadence Custom IC Design

Before start, you should have necessary files and setup done to be able to run Cadence software. If you have not done this, see tutorial on "How to setup Cadence tools?" (Available on class website)

To start Cadence Custom IC Design software, go to the directory you set up for running Cadence (**~/cadence**), type in **icfb** command as shown in Figure 1.
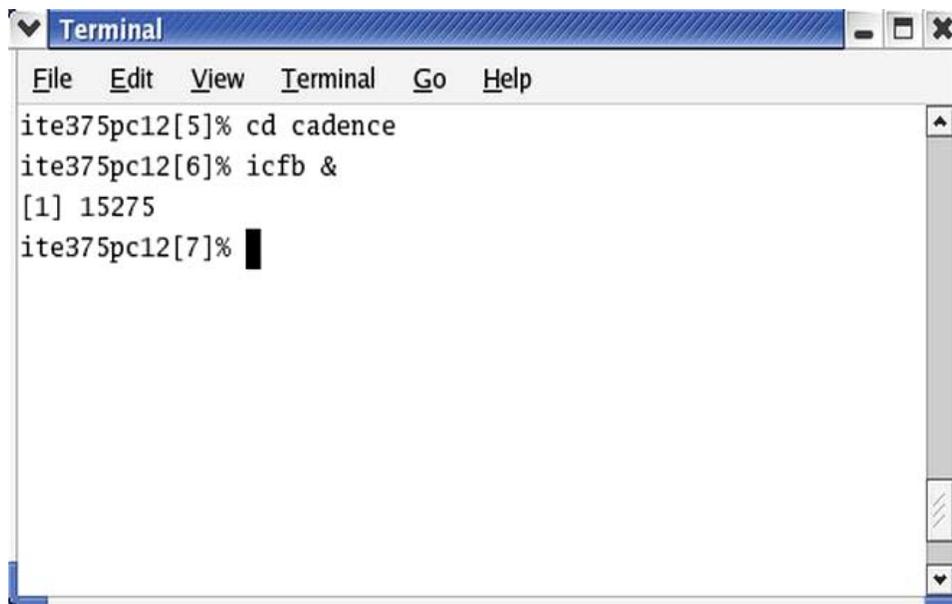
Figure 1: Command for running Cadence tool

The Command Interface Window (CIW) and Library Manager window should appear as shown in Figure 2.
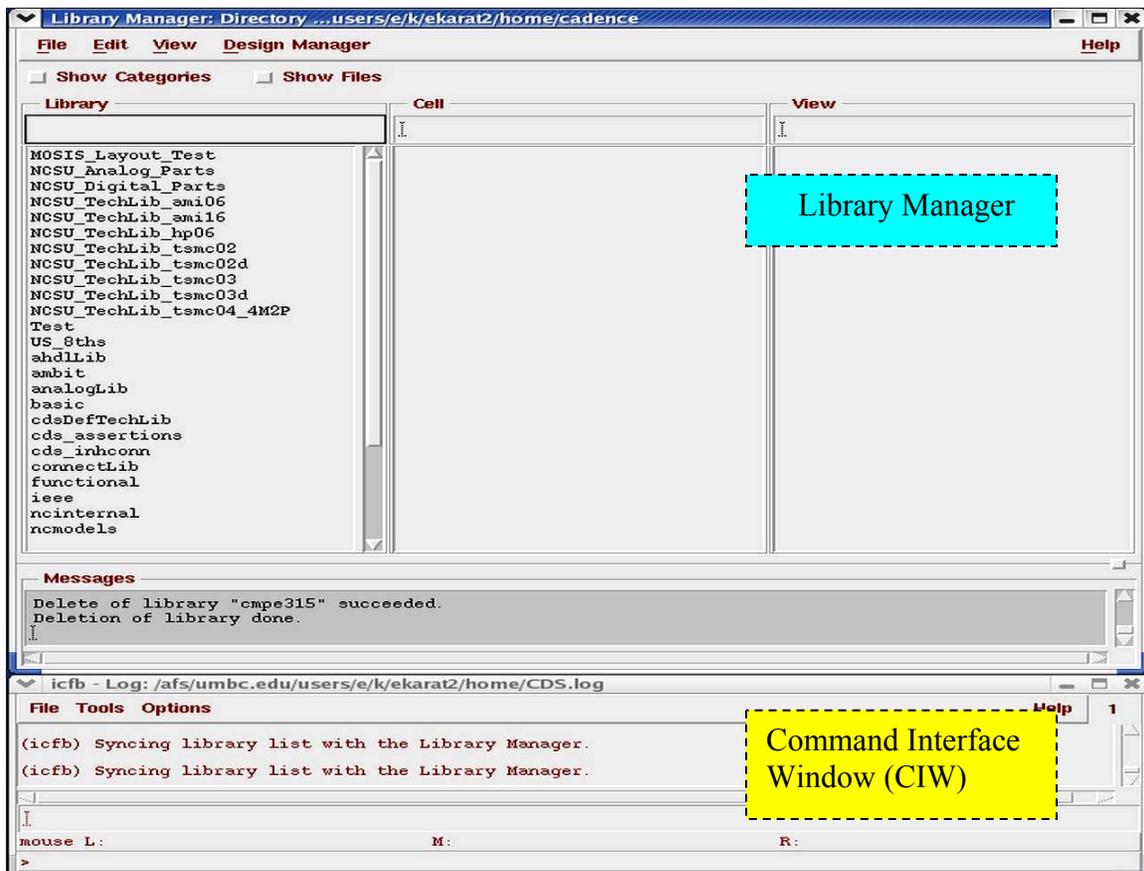
Figure 2: CIW and Library Manager

Create New Library

The design library is a group of cells, you can have multiple views (schematic, layout, symbol, vhdl, etc.) for each cell in the library. The technology (process) defined for the library will be used for all cells in that library.

To create new library, go to **File -> New -> Library** from Library Manager menu. The New Library dialog as shown in Figure 3 will appear. Type in the name for this library and then click OK. Another dialog will pop up asking for the technology of this library, choose "**Attach to an existing techfile**" as shown in Figure 4. Next you need to choose the technology, select "**NCSU_TechLib_ami06**" as shown in Figure 5.
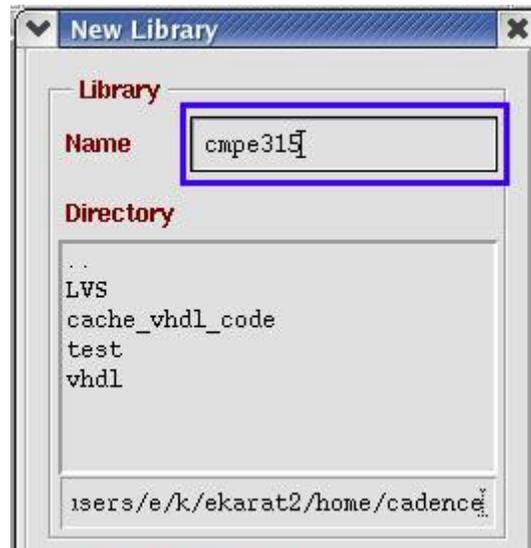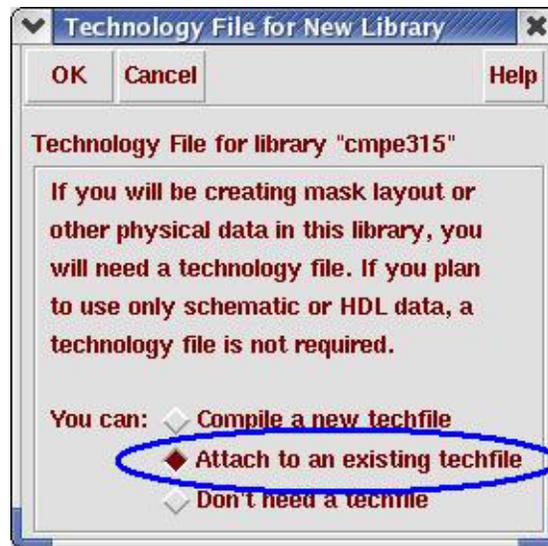
Figure 3: New Library Dialog

Figure 4: Technology File for New Library

Figure5: Attach Design Library

---

i **More info:**
  - Technology Library defines
    - Electrical components available in this technology (MOS transistors, bipolar transistors, analog components, etc.)
    - Layers available for the circuit layouts (e.g., how many Metal layers, oxide layer, etc.)
    - Rules associate with the design (Design rule, extraction rule, etc.)
    - Electrical parameters associate with electrical components and simulation models
    - And many more

Create New Schematic

From Library Manager, click to highlight the library (cmpe315 in this tutorial) you want the new cell to be created in. From the menu, go to **File -> New -> Cell View**. The new cell view dialog as shown in Figure 6 should appear. Type in the cell name (we will make an inverter in this tutorial) and make sure you have "**Composer-Schematic**" selected as a tool. Click OK and Virtuoso Schematic window will appear as shown in Figure 7.
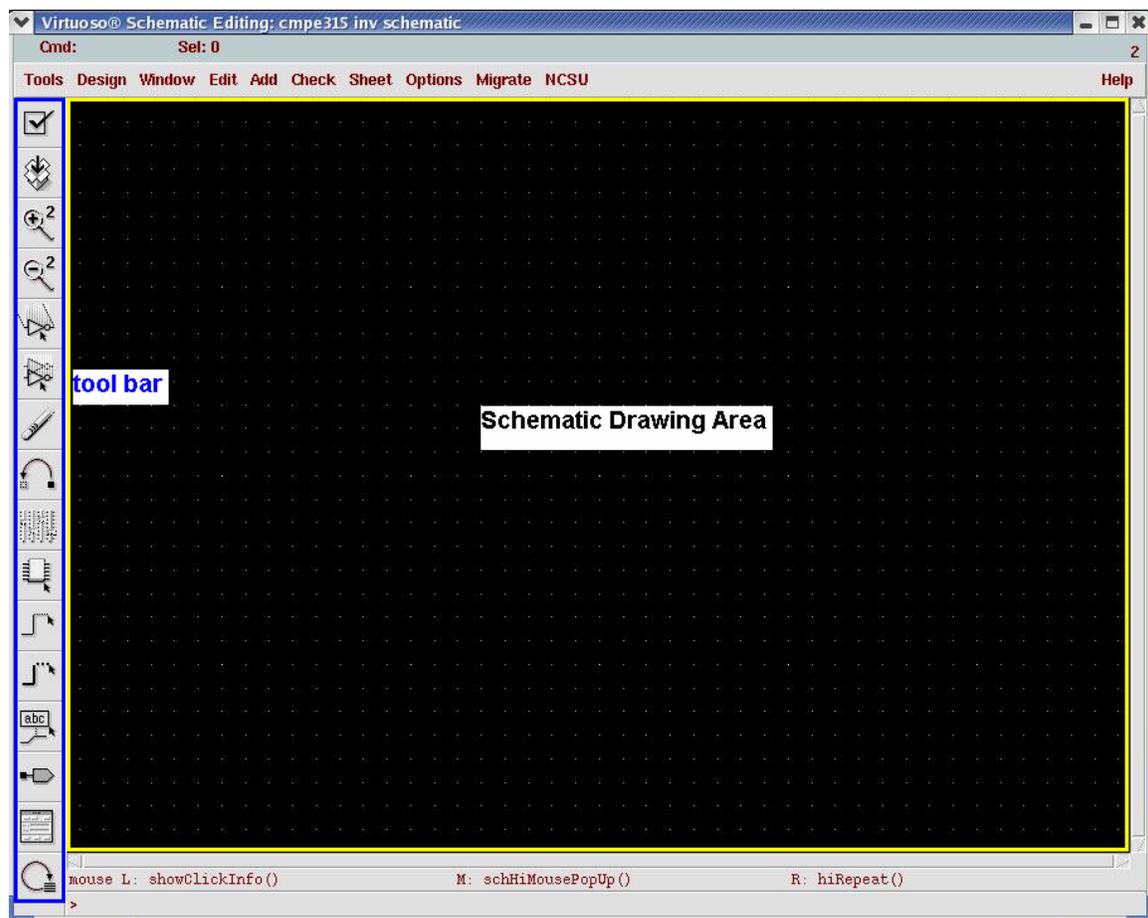

Figure 6: New Cell View

Figure 7: Virtuoso Schematic

Working with Virtuoso Schematic Composer

In this tutorial, we will make a schematic of the inverter. First, we need NMOS on the schematic. To add components to the schematic, from menu go to **Add -> Instance**, the Component Browser and Add Instance dialogs will show up. At the Component Browser, shown in Figure 8(a), change library to **NCSU_Analog_Parts**. You should see N_Transistor in the component list, click on N_Transistors and the component list will change as shown in Figure 8(b). Select nmos4 (4-terminal NMOS), now you can change parameters for the NMOS (e.g., width, length, no. of finger, etc.) at the Add Instance dialog, shown in Figure 9. For this tutorial, we will use transistors with the width of 1.5μm and length of 0.6μm. After changing parameters, now move the mouse over schematic drawing area, left click on the schematic where you want the new component to be placed.

Repeat the steps for placing PMOS, vdd and gnd on the schematic. Using NCSU_Analog_Parts library, you should find PMOS from P_Transistor -> pmos4, vdd from Supply_Nets -> vdd and gnd from Supply_Nets -> gnd. Press **Esc** to end add instance command. (**Esc** can be used to end any commands in Virtuoso Schematic)
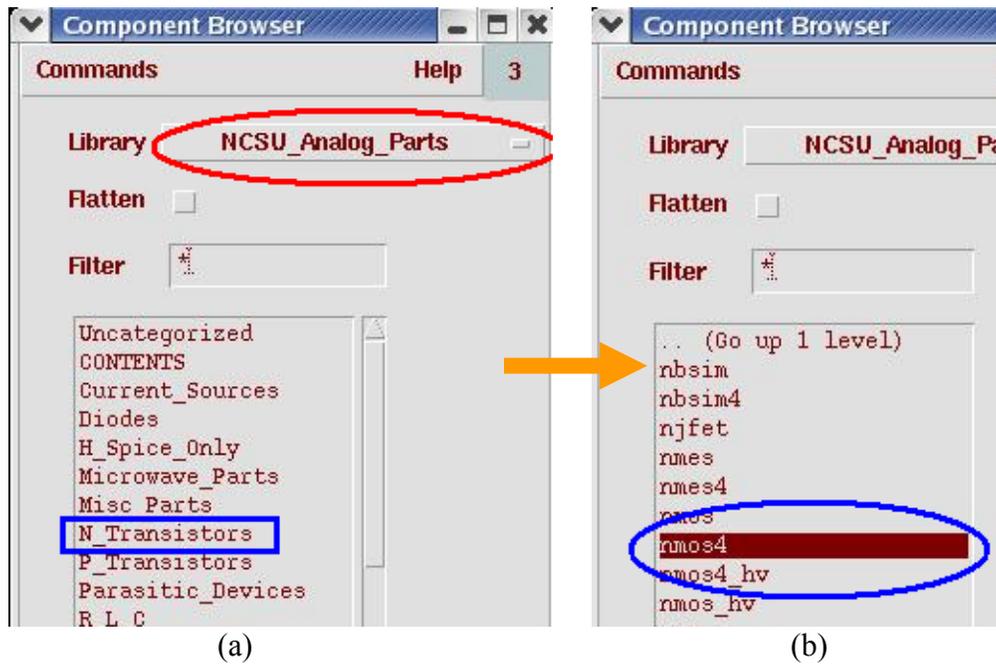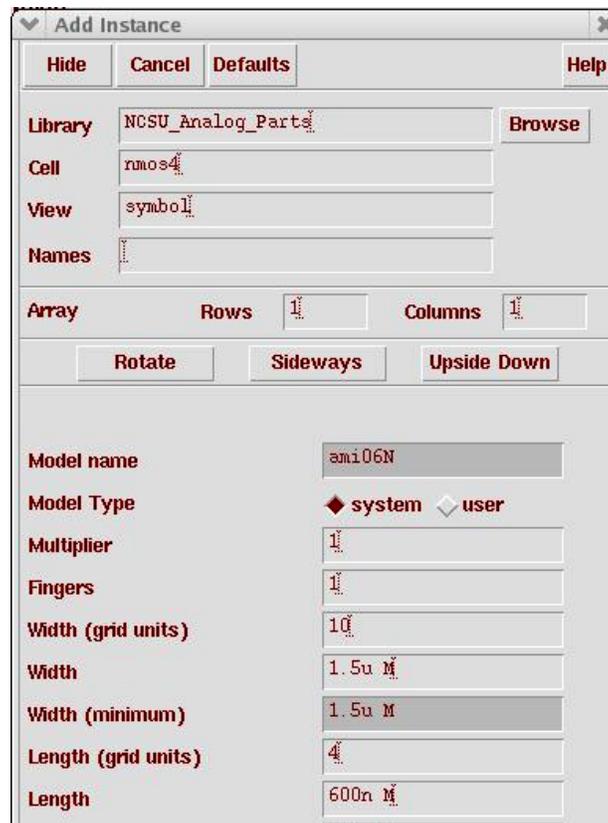
(a) (b)

Figure 8: Component Browser
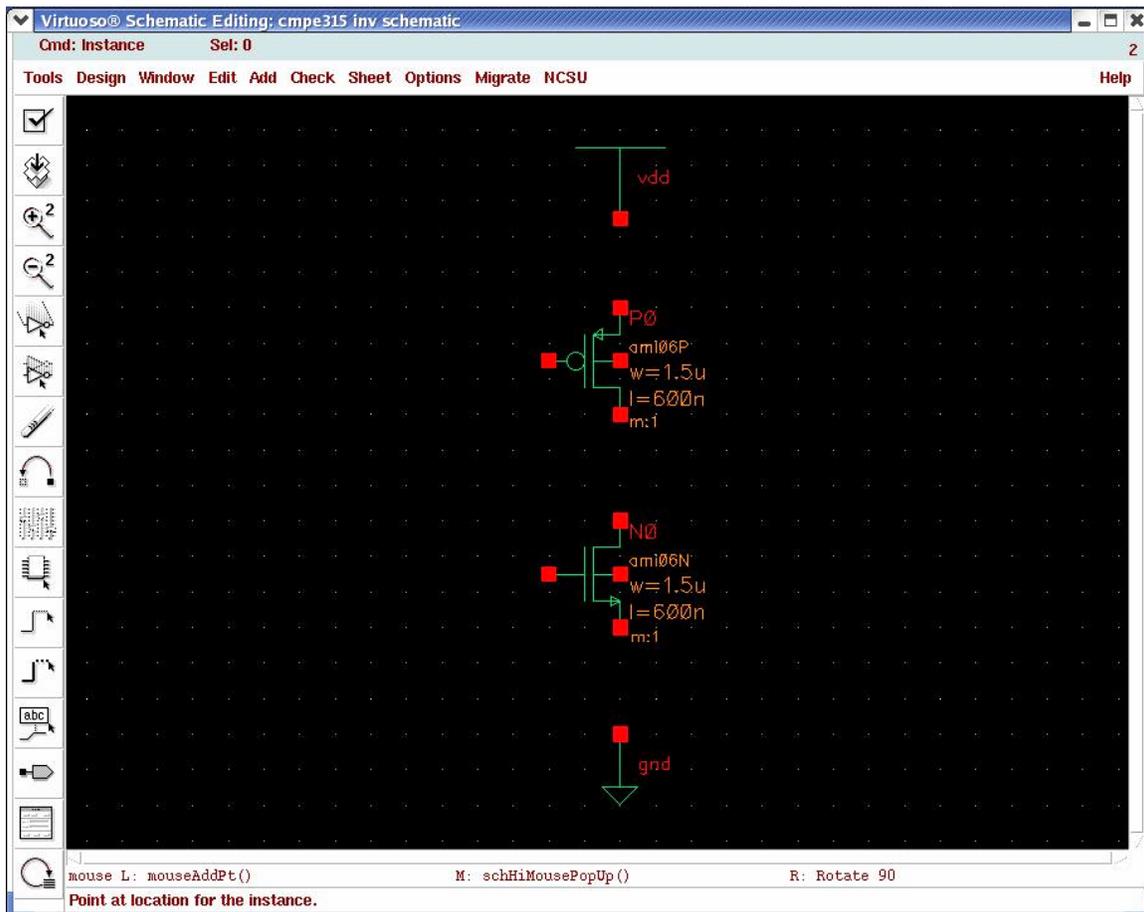

Figure 9: Add Instance

Figure 10: Schematic after placing components

Figure 10 shows the schematic after placing components. You can move components around by go to **Edit -> Move**, left click at the component you want to move, move the mouse to new location and left click to place(do not drag the component, click once on the component and click again at the new location).

Now, we need to make connections between components. From the menu, go to **Add -> Wire (narrow)**, the add wire dialog as shown in Figure 11 will show up, you can change wire color and style from this dialog. Click on the pin of the component you want to make a connection (when you move mouse over the pin, yellow circle around pin will appear), move mouse to another pin you want to make connection, click again to end the wire. Note that you can make a 90 degree turn on the wire by left click at the turn. You can also end the wire without connecting to the pin by double click.

Figure 12 shows the schematic after add wires. You can check the schematic connection and save the schematic by go to **Design -> Check and Save**. Since we did not make the complete connections, the error/warning dialog will appear as shown in Figure

13. You can see the explanation of errors/warnings on ICW window as highlighted in red in the figure.
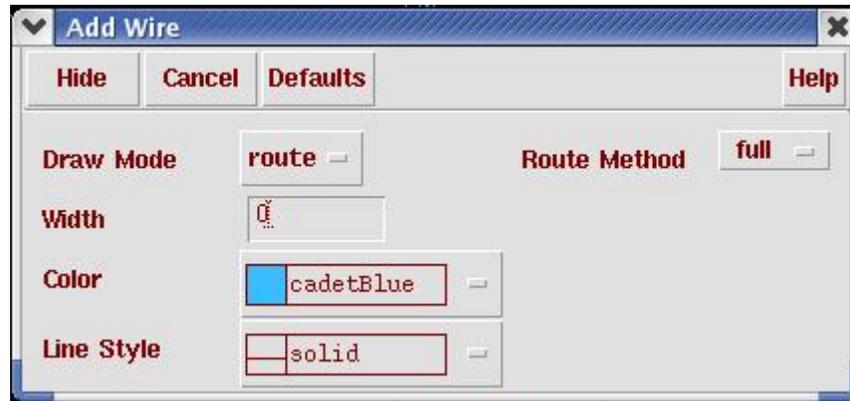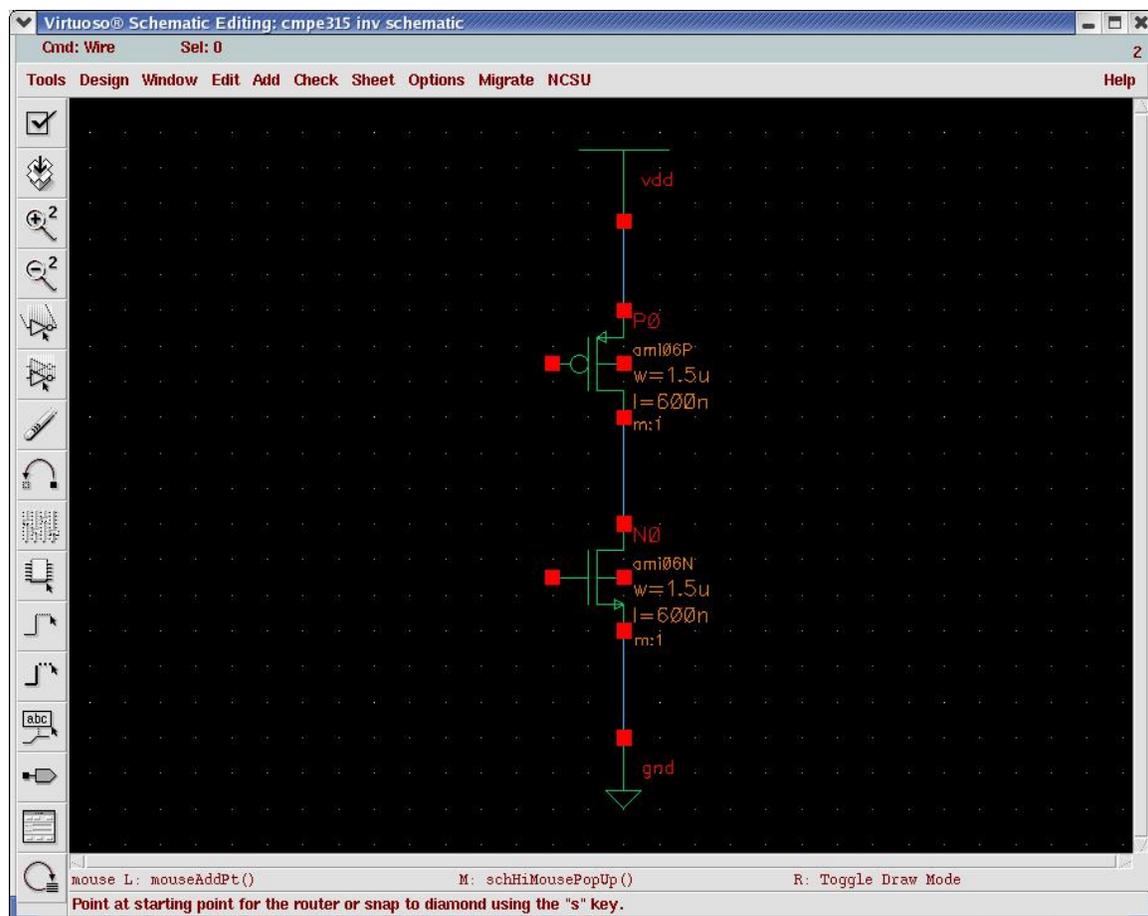

Figure 12: Add Wire

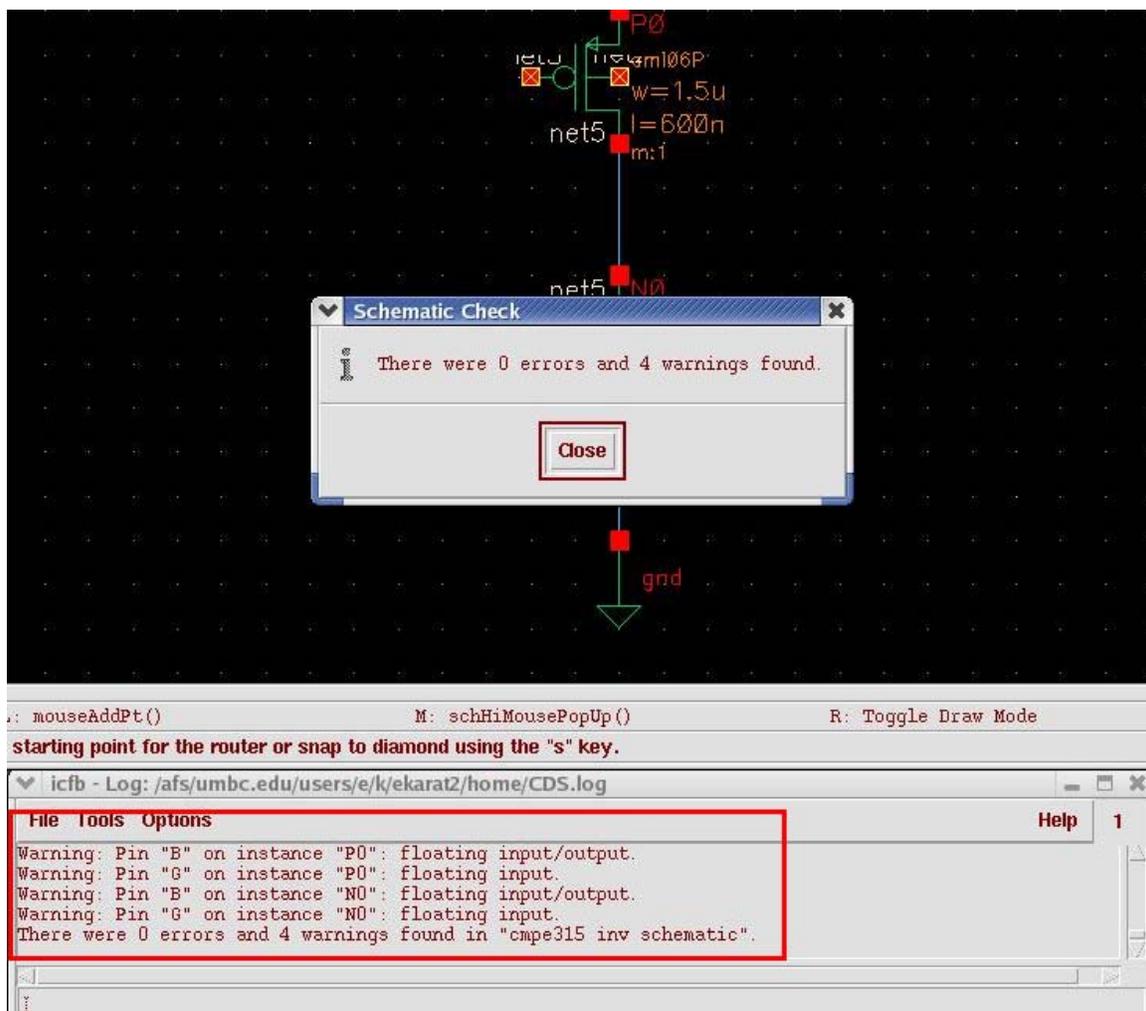*Figure 13: Schematic after add wires*

Figure 14: Schematic errors & warnings

Now we need to add input and output pins to the schematic. To add pin, go to **Add -> Pin**, the dialog as shown in Figure 15 should show up. Type in pin name (you can type multiple pin names using space to separate each pin name), and change the direction to appropriate direction for the pin. If you do not have appropriate directions set, you will find problems when trying to simulate the schematic. Figure 16 shows the schematic after 2 pins are placed (INPUT and OUTPUT).

To complete the connections, we need to tie bulk terminal of PMOS to vdd and bulk terminal of NMOS to gnd. We also need to connect gate of NMOS and PMOS to INPUT pin and make connection from drain of both transistors to OUTPUT pin. Figure 17 shows the complete schematic. Run Check & Save again, you should have no errors/warnings as shown in Figure 18.
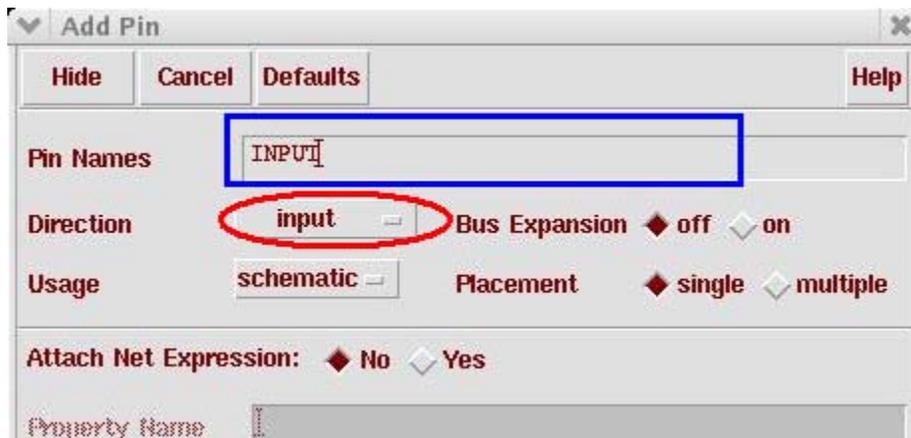
CMPE 315/CMPE640   **Virtuoso Schematic Composer**

UMBC   **Tutorial**   **Ekarat Laohavaleeson**
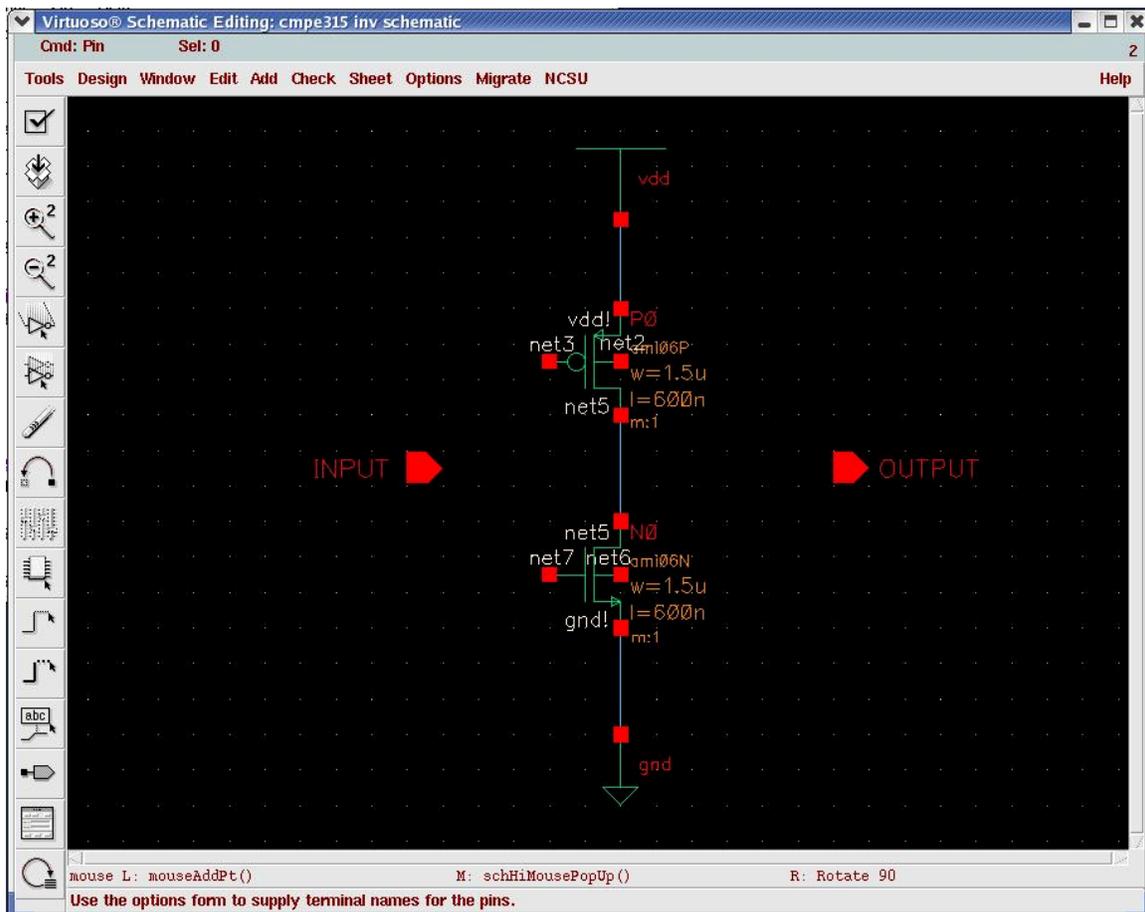
Figure 15: Add Pin
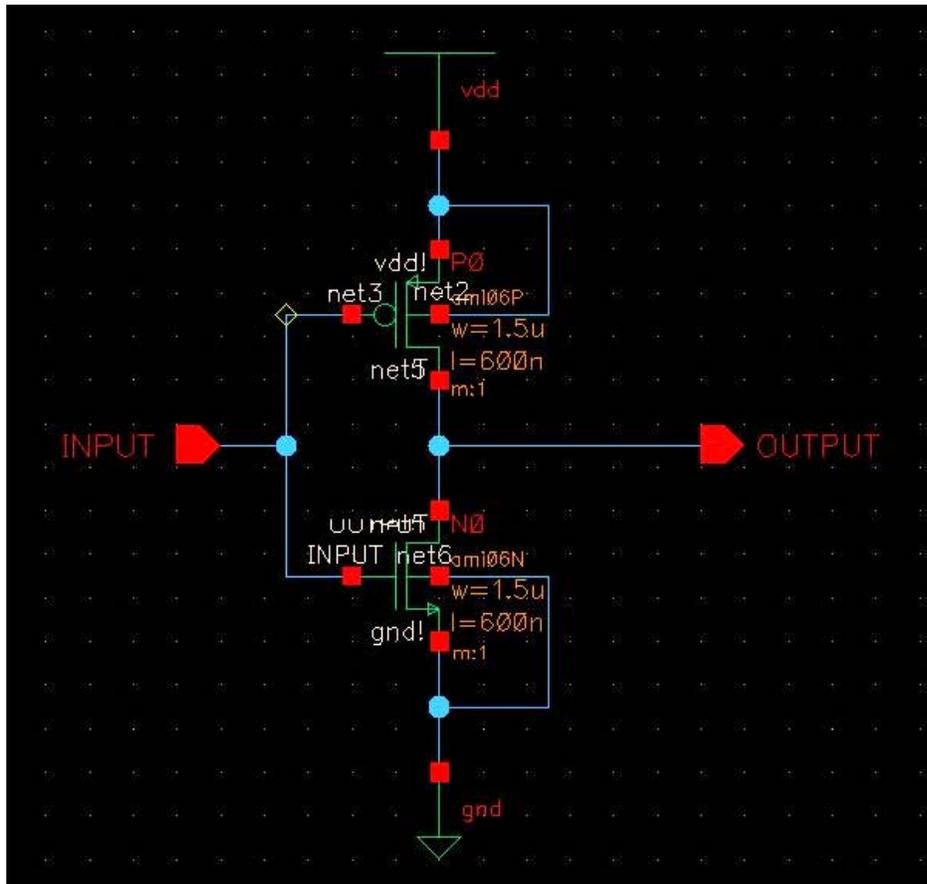

Figure 16: Schematic after adding pins
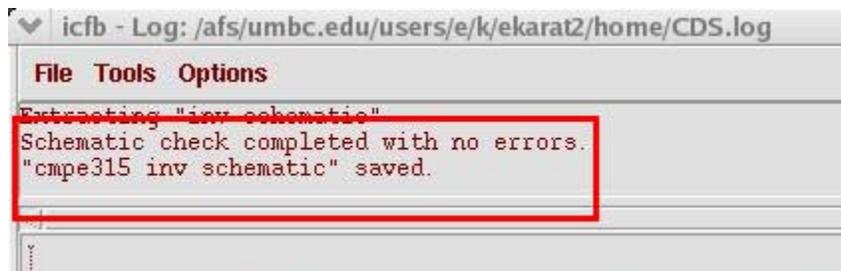
Figure 17: Complete Schematic

Figure 18: CIW showing no errors/warnings after Check & Save

> *Tips:*
> - Using following shortcut keys help working with schematic faster
>   - **i** to Add Instance
>   - **m** to Move
>   - **w** to Add Wire (narrow)
>   - **p** to Add Pin
>   - **u** to Undo
>   - **c** to Copy
> - To zoom in on the schematic, hold **right-botton** mouse to create zooming area and release to zoom in.
> - To zoom out, press **Shift+z**
> - To fit whole schematic on screen, press **f**

Create Symbol View

The symbol views are used in hierarchical design. You can create a symbol view of a cell from its schematic and then use the symbol of that cell in a higher level cell. For example, a buffer cell can be created using 2 inverter cells.

To create symbol view of the schematic, go to Design -> Create Cellview -> From Cellview, the create cellview dialog as shown in Figure 19 should appear. Make sure that you have To View Name set to "symbol" and Tool/Data Type set to "Composer-Symbol". Click OK and new symbol view will pop up as shown in Figure 21. Close the new symbol view.
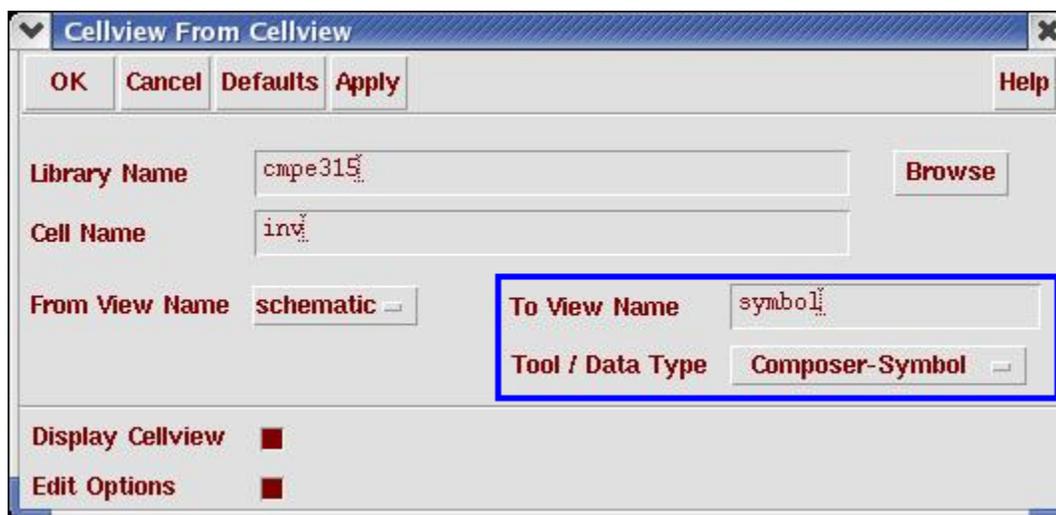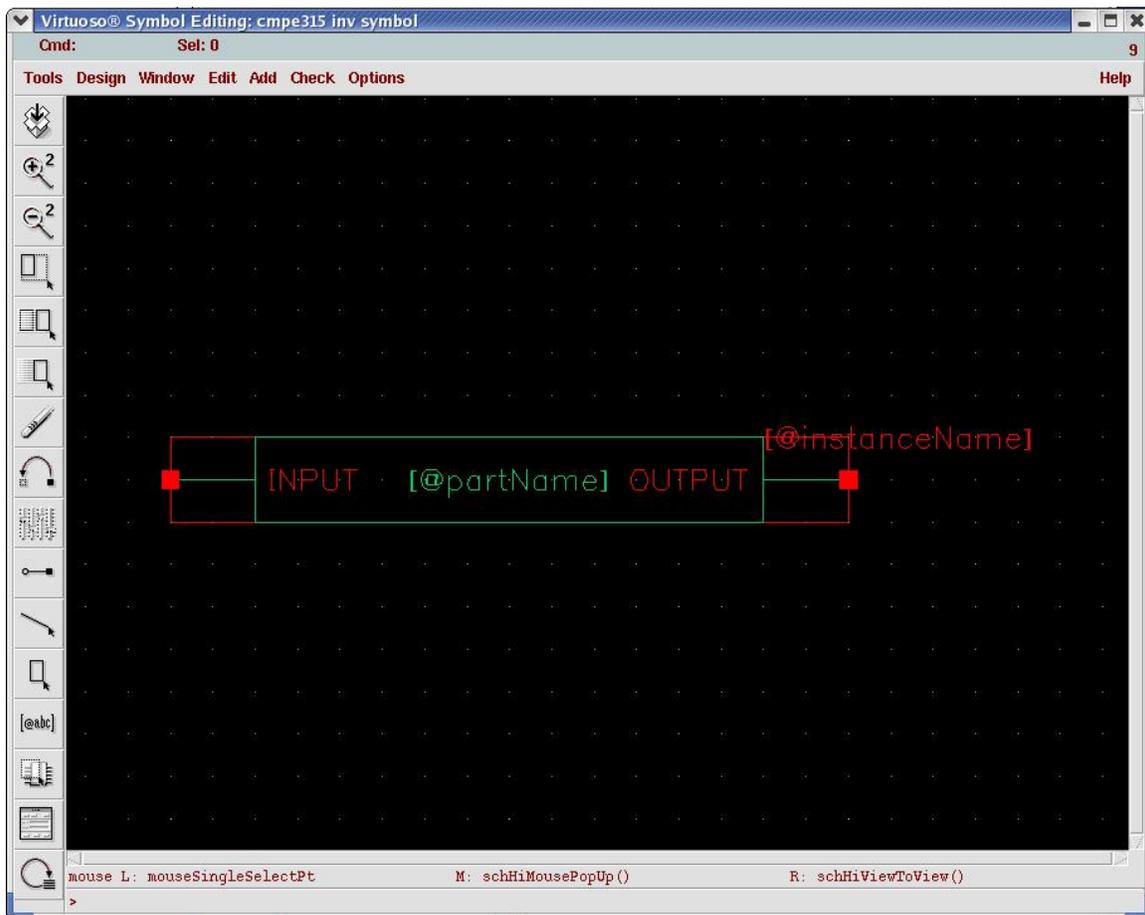
Figure 20: Create Cellview From Cellview

Figure 21: New created symbol view

Create Plot from Schematic

You can create plot of the schematic by go to **Design -> Plot -> Submit**, the Submit Plot dialog as shown in Figure 21 will appear. Click on **Plot Options**, the Plot Option dialog as shown in Figure 22 should show up. Select "**Send Plot Only To File**", type the file name (postscript file), Click OK to close Plot Options dialog and then click OK again on Submit Plot dialog.
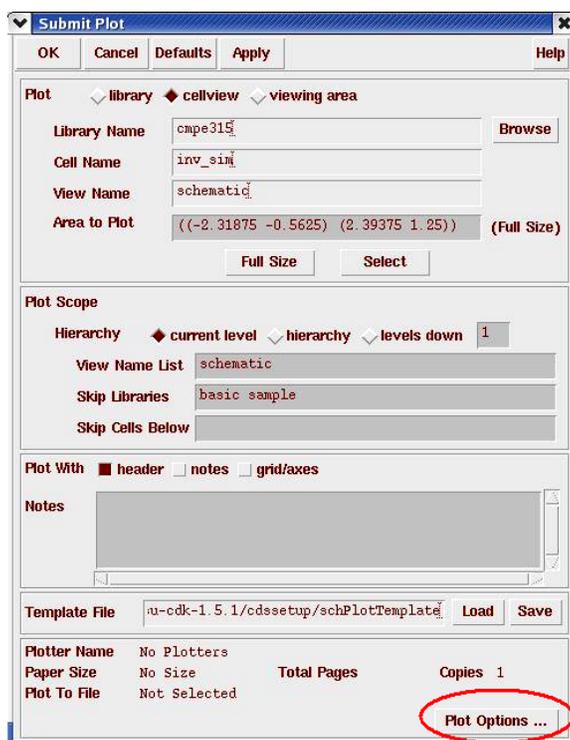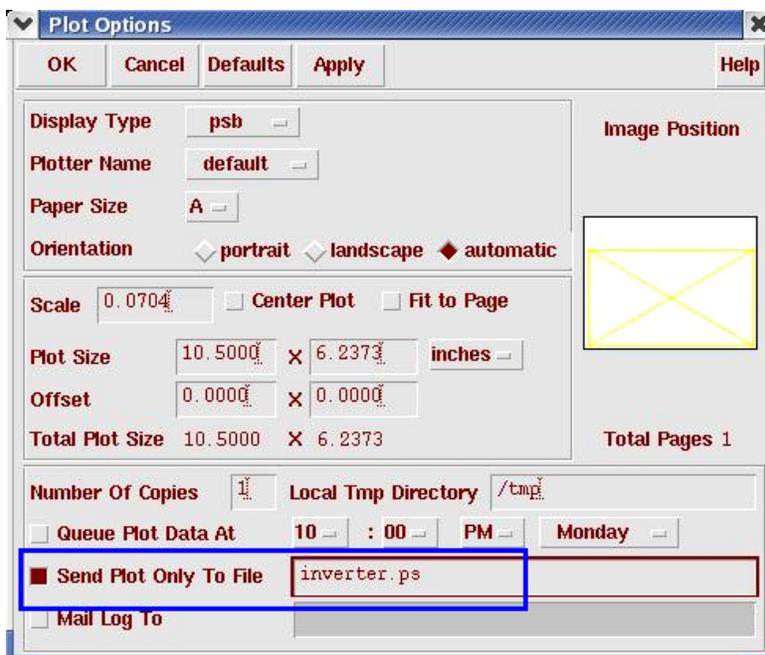
Figure 21: Submit Plot

Figure 22: Plot Options