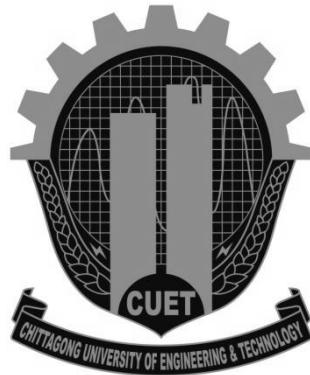


CHITTAGONG UNIVERSITY OF ENGINEERING & TECHNOLOGY
DEPARTMENT OF ELECTRICAL & ELECTRONIC ENGINEERING



LAB SHEET

**EEE490
VLSI TECHNOLOGY SESSIONAL**

Instruction

Before coming to the laboratory read the lab sheet carefully and understands the procedure of performing the experiments. Your performance in the class will be evaluated.

Department of Electrical and Electronic Engineering
Chittagong University of Engineering and Technology (CUET)
Course Title: VLSI Technology Sessional
Course No.: EEE490

Experiment List:

Experiment-01: Introduce with Linux operating system and cadence CAD tools.

Experiment-02: Perform simulation of IV characteristic of an NMOS device using the cadence CAD system.

Experiment-03: Characterization of a CMOS Inverter and create symbol from schematic using the cadence CAD system.

Experiment-04: Layout design of a CMOS Inverter using the cadence CAD system.

Experiment-05: Characterization of Two input NAND and NOR gate using the cadence CAD system.

Experiment-06: Layout design of a NAND and a NOR gate using the cadence CAD system.

Experiment-07: Realize a 2-input XOR gate.

Experiment-08: Realize a 1-bit full adder in CMOS schematic and simulate. Generate layout using tool option and simulate.

Experiment-09: 4 x 1 MUX using transmission gates.

Experiment-10: Realize a Boolean expression $Y=NOT((A+B)(C+D)E)$ in schematic and draw its layout and simulate using the cadence CAD system.

Department of Electrical and Electronic Engineering
Chittagong University of Engineering and Technology (CUET)
Course Title: VLSI Technology Sessional
Course No.: EEE490

Experiment-01: Introduce with Linux operating system and cadence CAD tools.

1. Start with “putty”

Run “putty” → load “cadserver1”→ Click “Open”

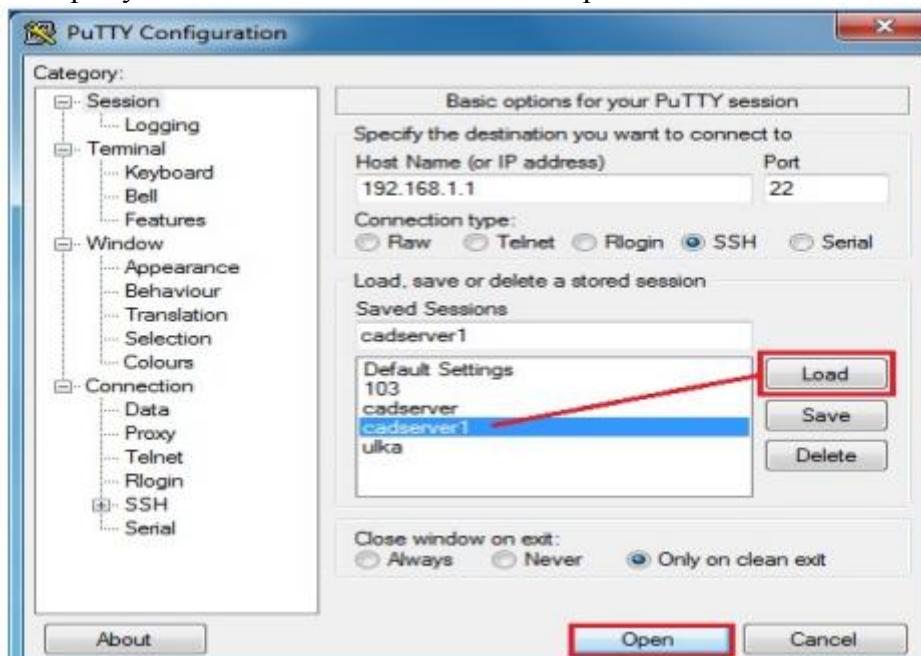


Figure 1: Load cadserver1 from putty.exe

Login as “cad1”→ password: “*****”→Press Enter



Figure 2: Login as cad1 to the server

Start “vncviewer” by the following command-

“vncserver :groupnumber-cc 3-depth 24-geometry 1280x768” →Enter

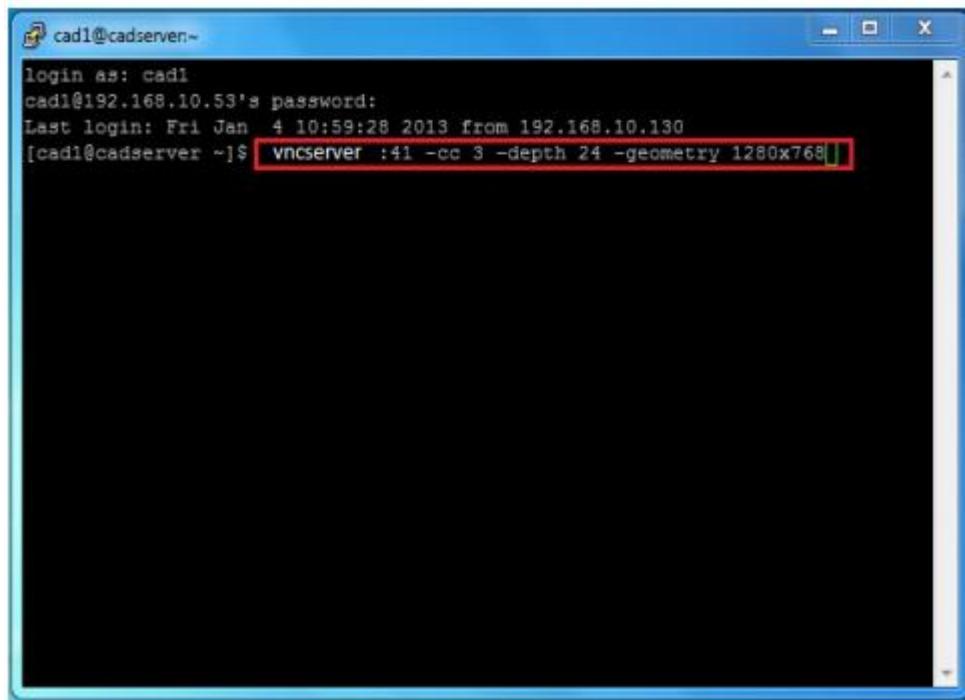


Figure 3: Command to create vnc session

2. Run vncviewer

Now run “vncviewer” → modify group no. → connect → continue if Unencrypted Connection → Enter Password “*****” → Ok



Figure 4: Connecting through vncviewer

3. Start with virtuoso

When the virtuoso command typed in the terminal, it appears the following-

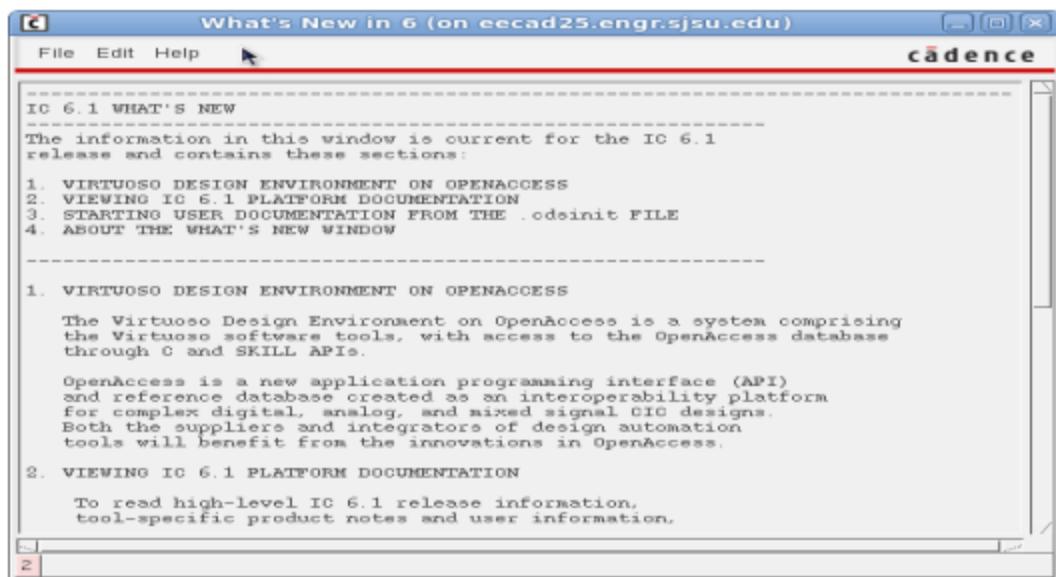


Figure 5: Release note of cadence 6

Close this window by clicking on <file> and then <close>

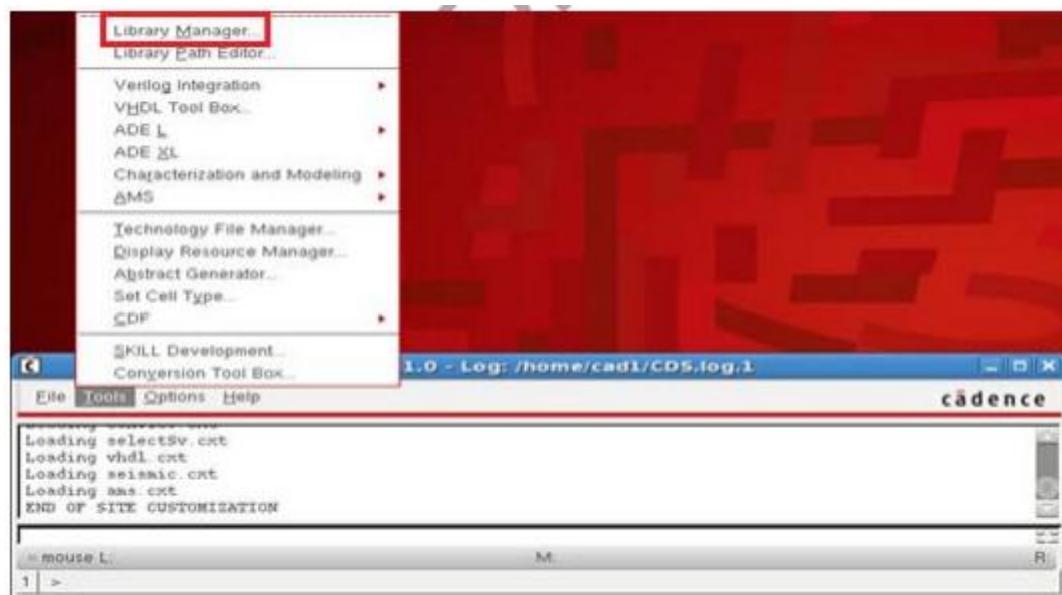


Figure 6: Load cadserver1 from putty.exe

Click on <tools> and then <library manager> to start the library manager

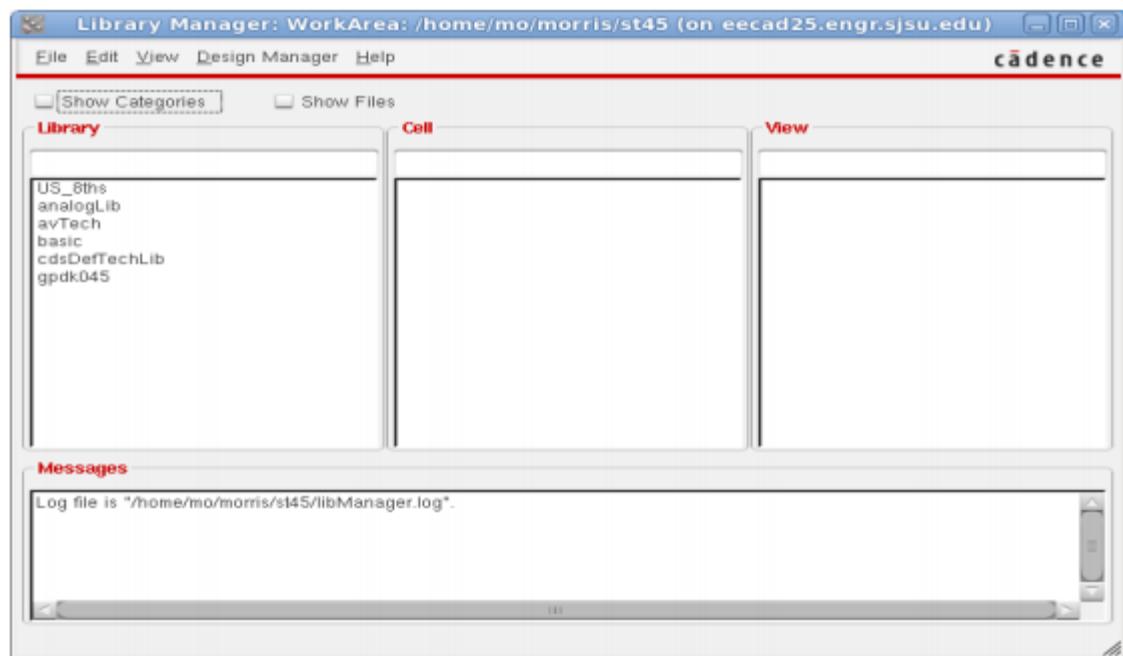


Figure 7: Library Manager

4. Create a new library

Click on <file> then cursor over to <new> then to <library> and click <library>

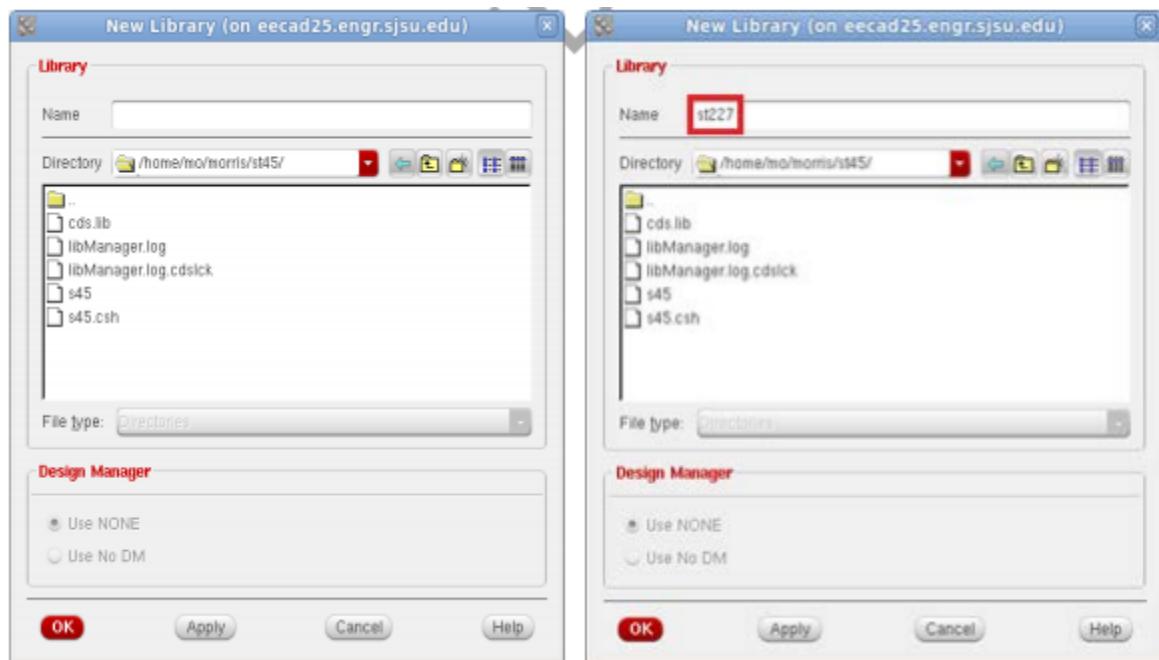


Figure 8: Name of the library with directory

Enter the name of your library in the name filed (here used st227)

Click on OK at the bottom. You will get a small screen. It is very important that you click on “attach to an existing technology library”.

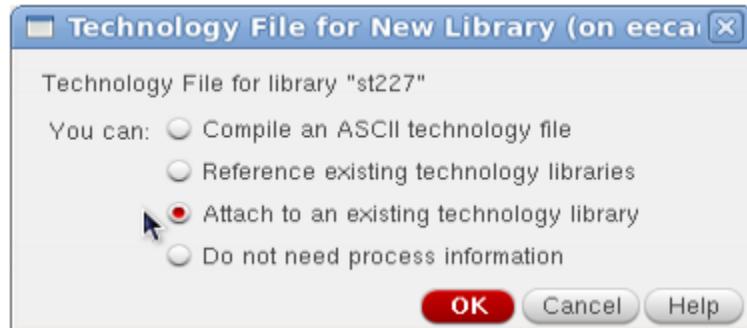


Figure 9: Attaching the technology file

Then Click on OK at the bottom. A screen will pop up asking which library to select.



Figure 10: Selecting the existing technology library

Select the gpdk045 option by clicking in the list. (If the screen is too small, you may need to scroll the list)

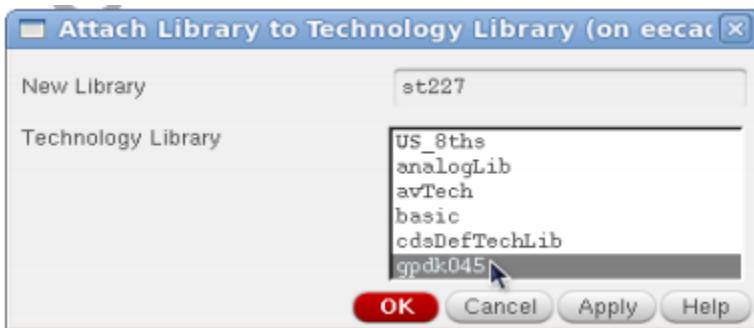


Figure 11: Selecting technology library of 'gpdk045'

Click on OK

The library manager will now show the new library –

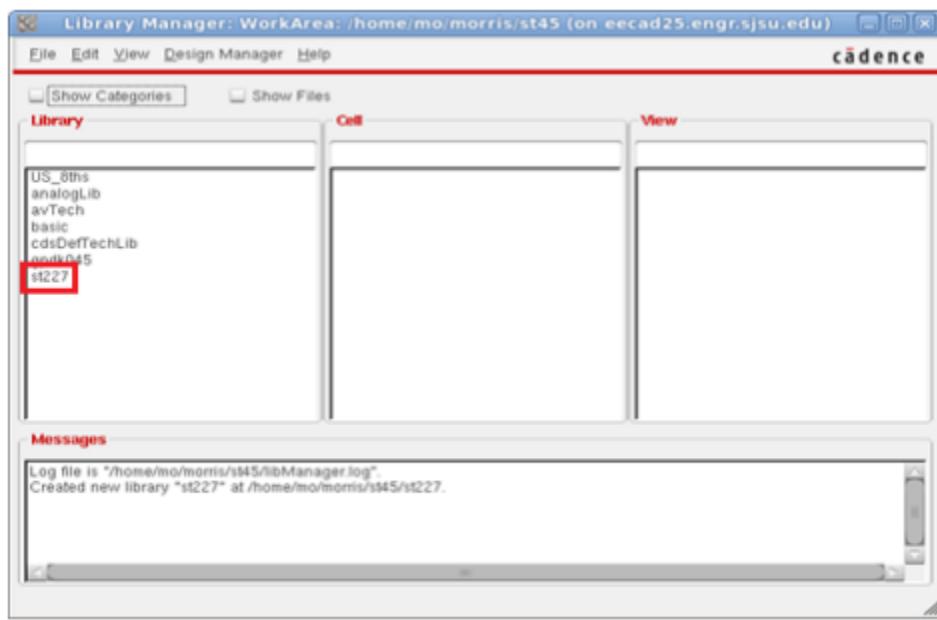


Figure 12: library manager with new library

5. Create a new cell of schematic

Select the new library by clicking on the name (st227 in this case). First create the schematic – click <file> from the library manager then cursor over to <new> then to <cell view>



Figure 13: Create a new cell named ‘inv’

In the field for cell, enter the name of the cell – “inv” is used in this case → OK. Is any license issues appear, click “yes”.

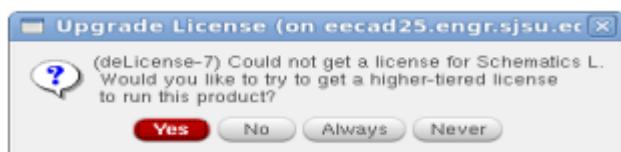


Figure 14: License issue with schematic L
You should now get the schematic editing screen.

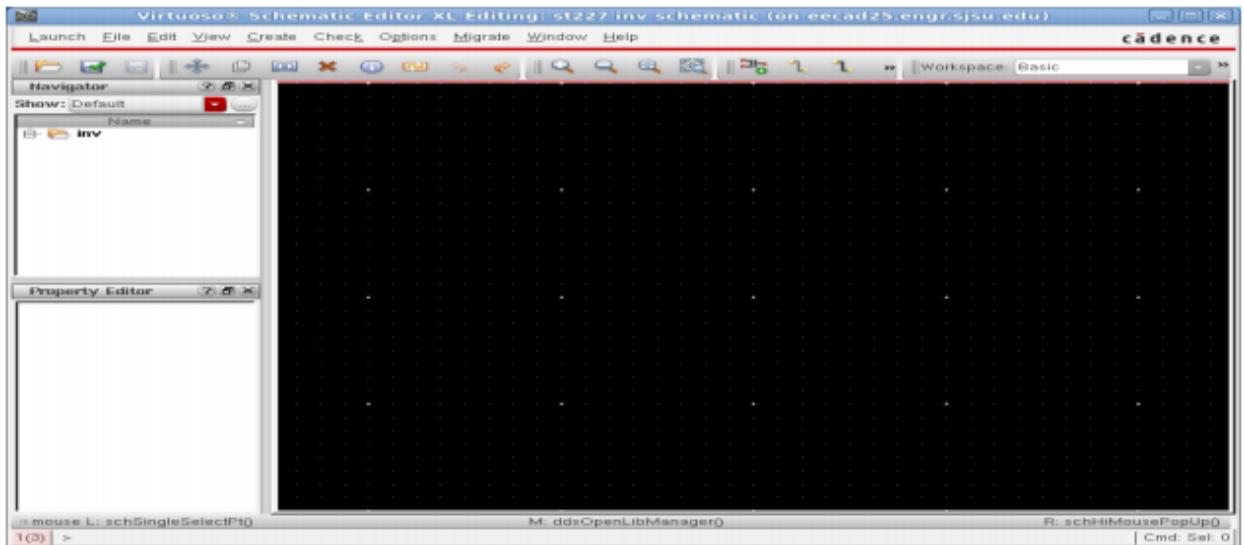


Figure 15: New Schematic window

6. Steps of drawing

a. Component placement

Select a component by clicking on the transistor symbol. (Highlighted below with a red circle) or press 'I' as a hotkeys for instance.

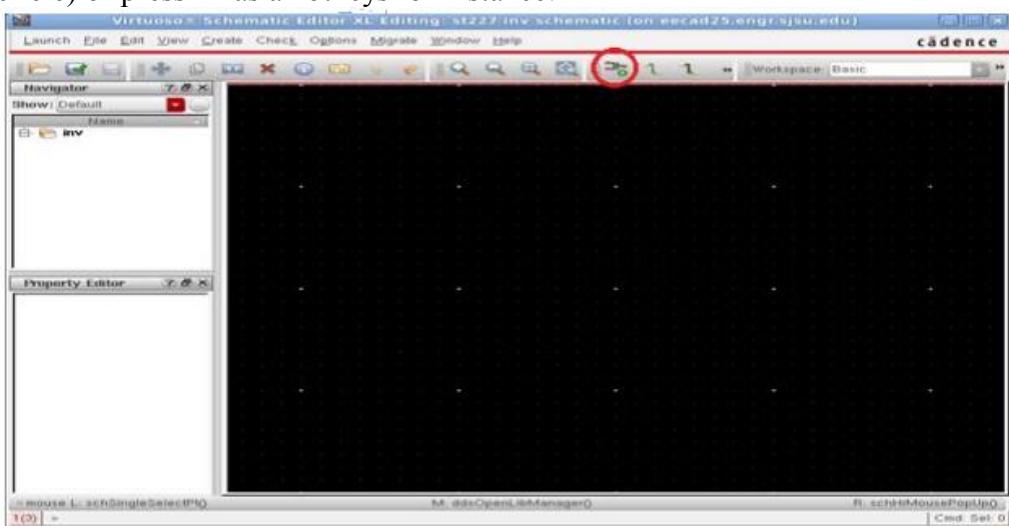


Figure 16: Instance insertion by highlighted transistor symbol

This gives a small selection screen.

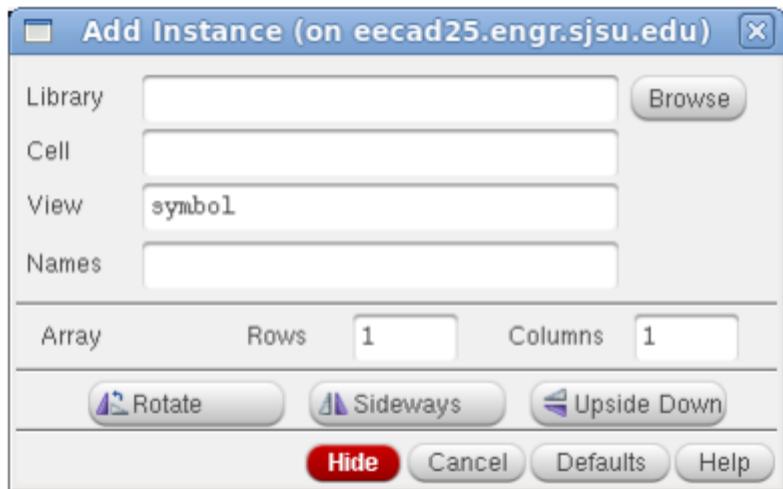


Figure 17: Instance insertion window

Click Browse to select the library.

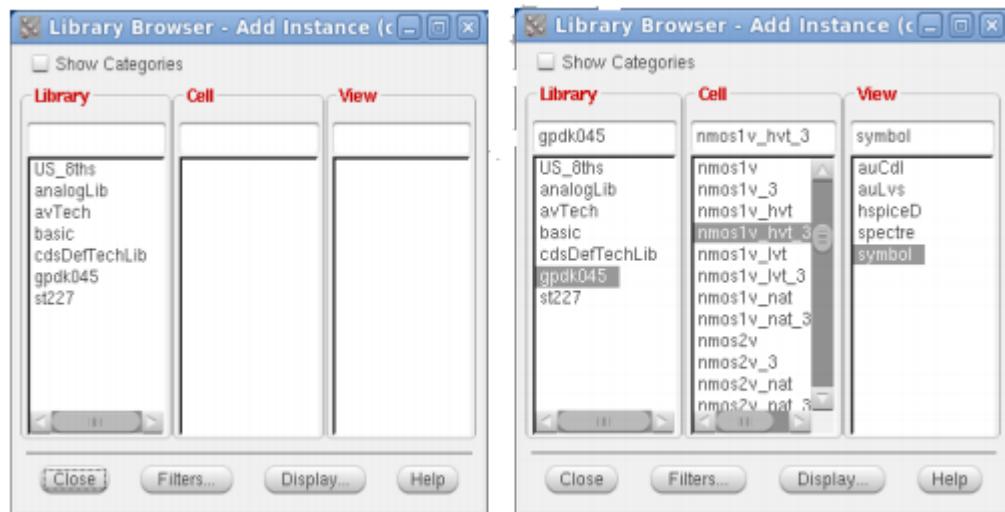


Figure 18: Browsing for the transistors

Now select gpdk045. In this case, we choose ‘nmos1v_hvt_3’. Then select the symbol for inclusion in the schematic.

You can click close, or leave this window open. The add instance window will now appear with all the device characteristics as below.

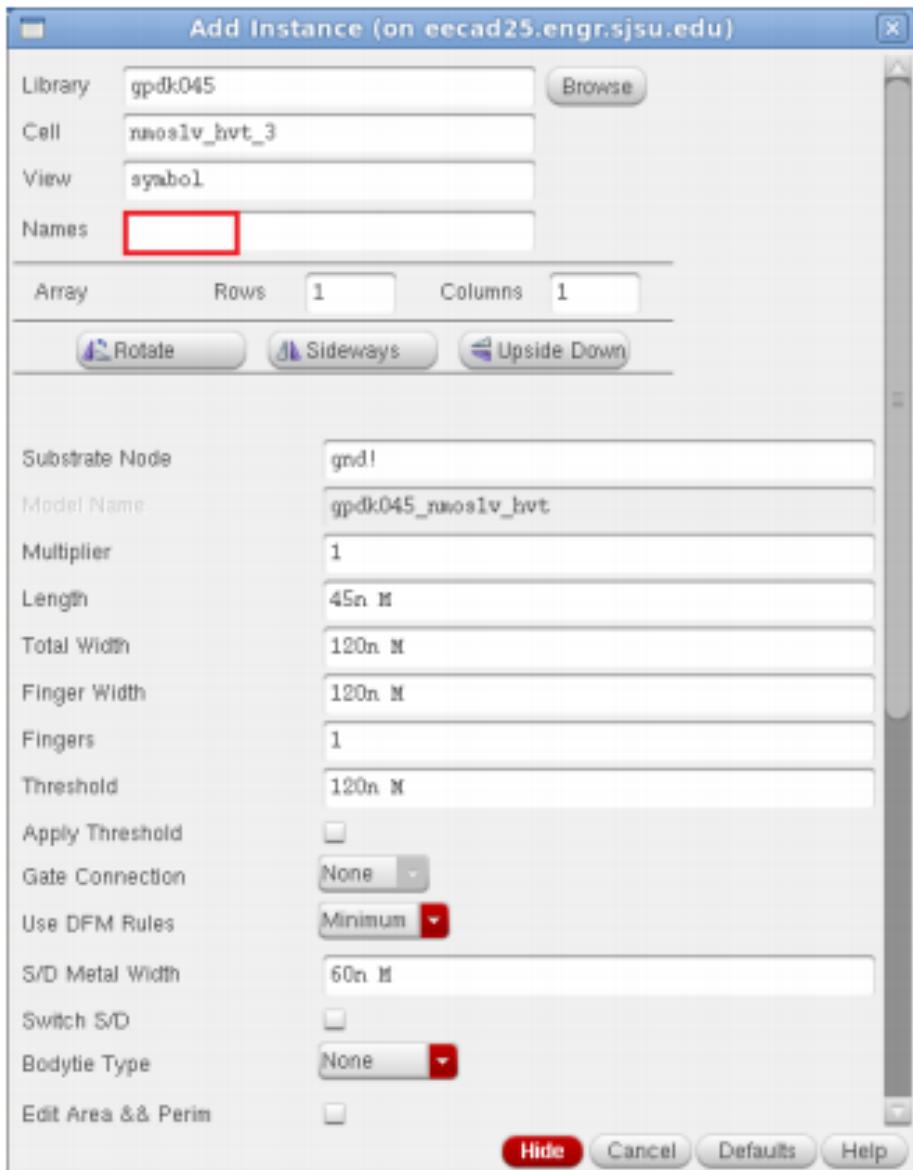


Figure 19: Inserting the name of the transistor

Add the name for this transistor (here used m1, mostly named automatically). You can change the device sizes now, or later. For this example, they will be left minimum size.

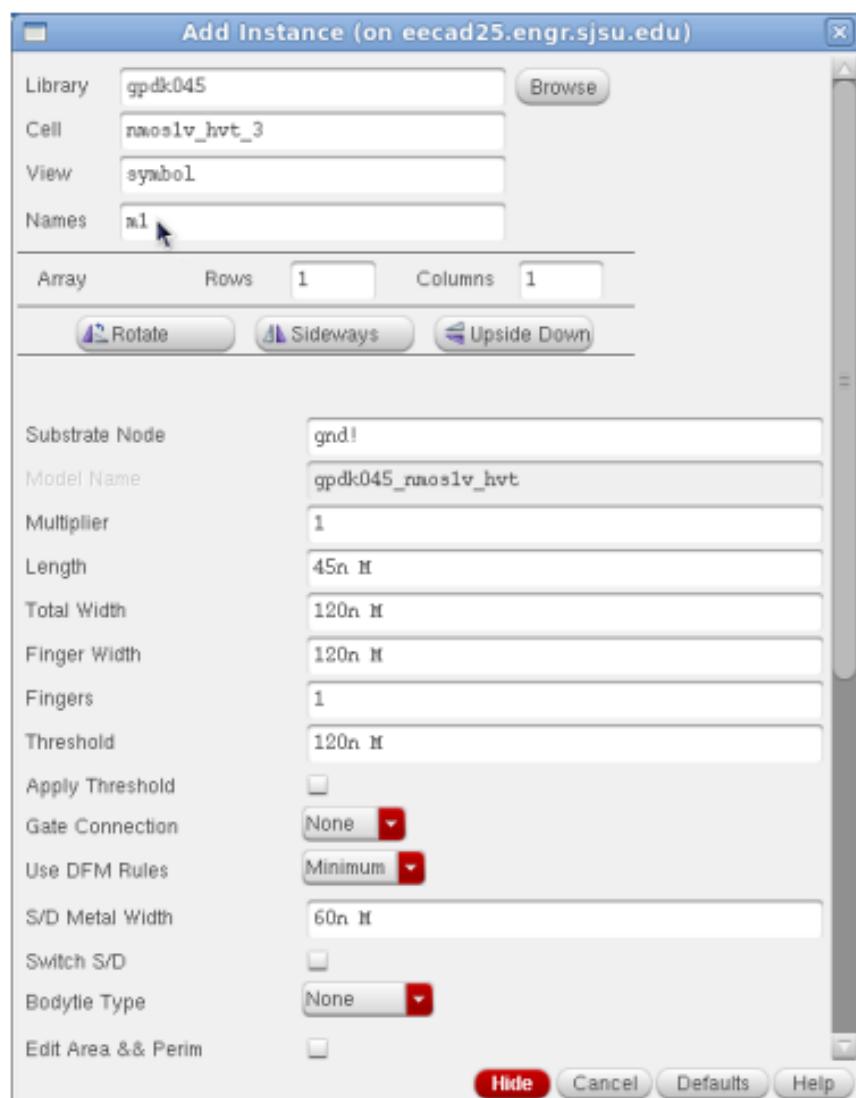


Figure 20: Parameter window of the transistor

Select Hide to minimize this screen, or select the schematic page to place a transistor on the schematic. Place the transistor by clicking on the schematic screen.

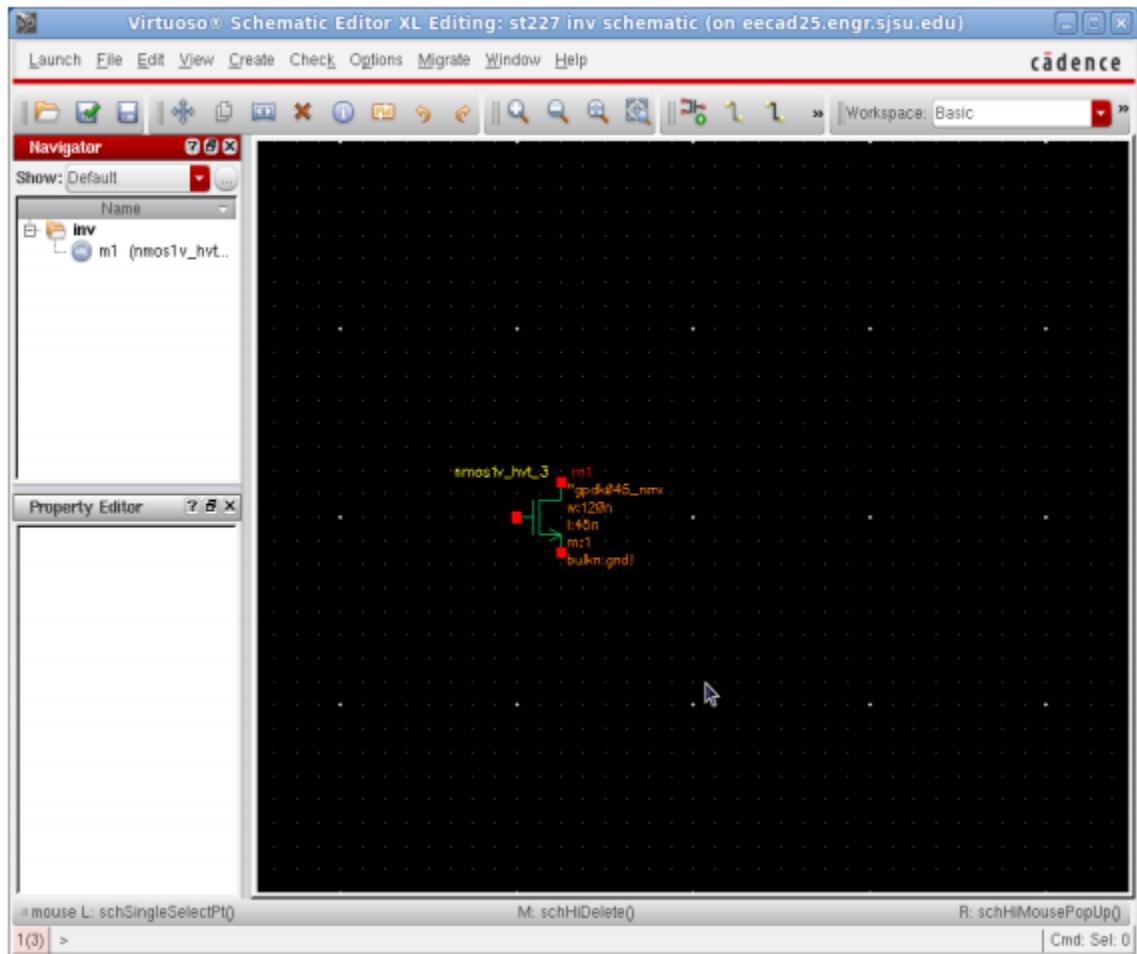


Figure 21: Schematic window with N transistor

Notice the cell name appears on the left in a tree list, and the property editor is below that. You can just click on the device name in the first list, and the properties will show up in the property editor.

The same steps will be followed for the P (pmos1v_hvt_3) transistor with different name. Place it on the schematic by clicking above the N transistor location.

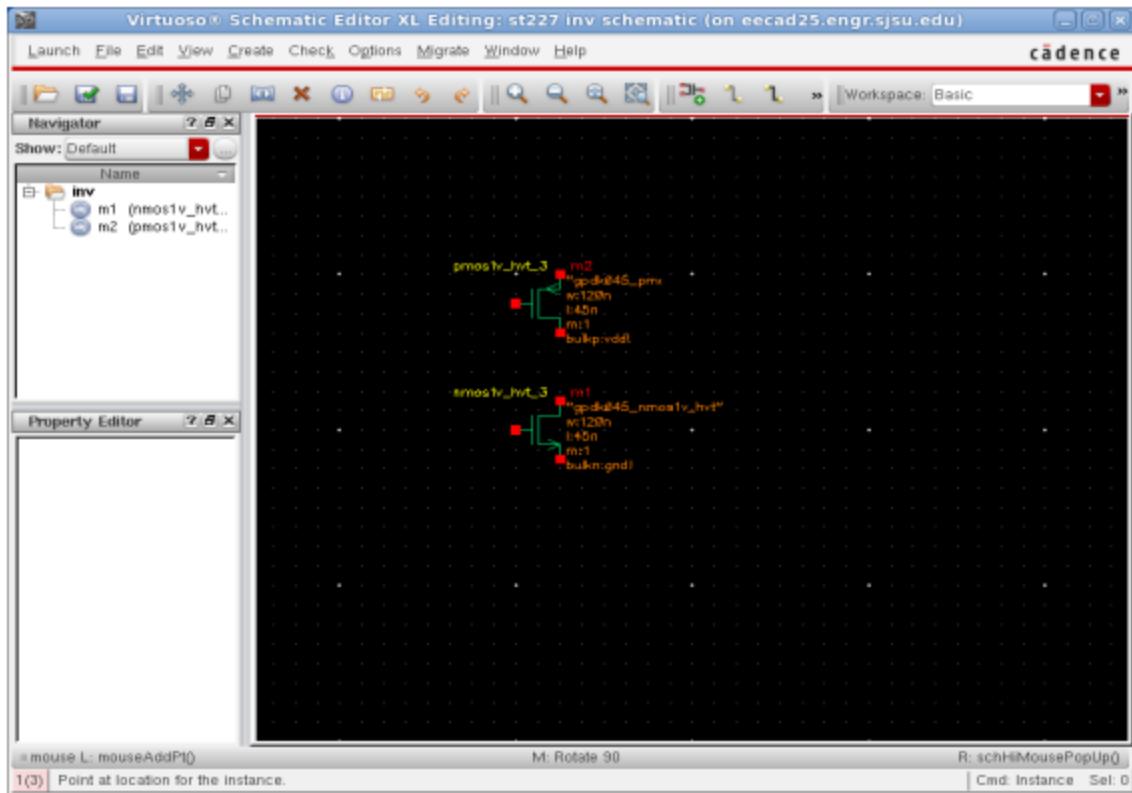


Figure 22: Schematic window with N & P transistor

Other different types of component are found in analogLib or in basic etc.



Figure 23: Placing vdd and gnd from ‘basic’ library

b. Create Pin

Select the symbol next to create pin. This may already be visible on your terminal, or you will need to click on the >> symbol. Alternately, you could click on the create and then pin items in the menu. The tool bars generally allow single click selection.

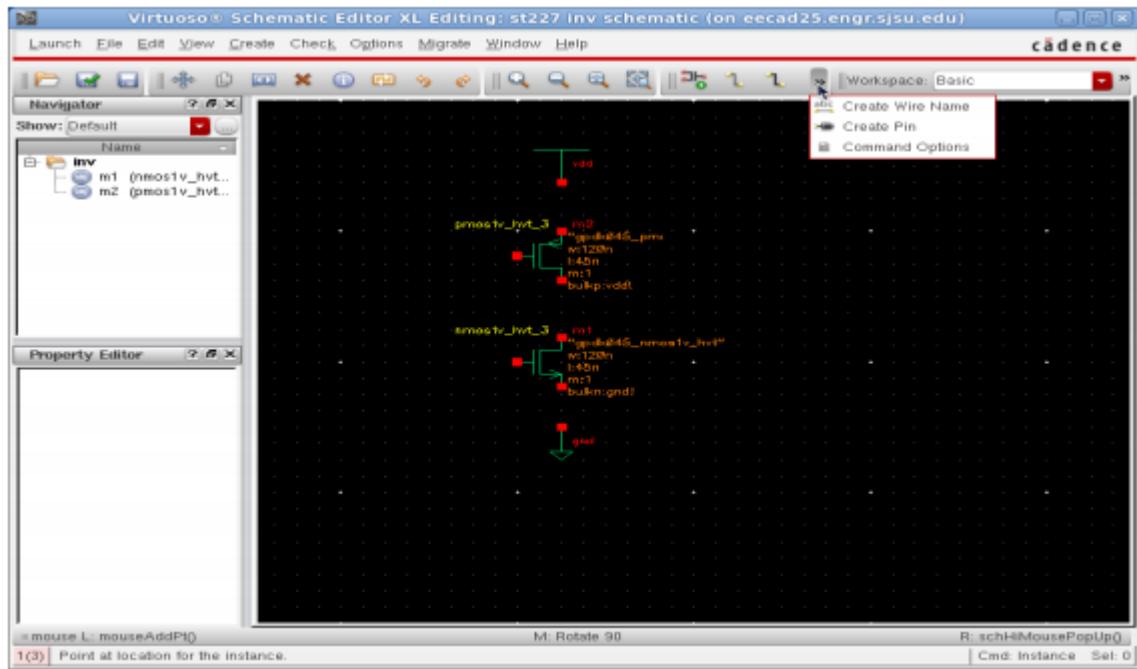


Figure 24: Schematic window after placing vdd & gnd



Figure 25: Adding 'Pin'

Give the pin a name and select the desired direction. In this case A for the input pin with input direction. More than one pin of the same type can be entered at a time. Just put spaces between the names. Place the pin on the schematic. Next, create an output pin called Z and place it on the schematic.

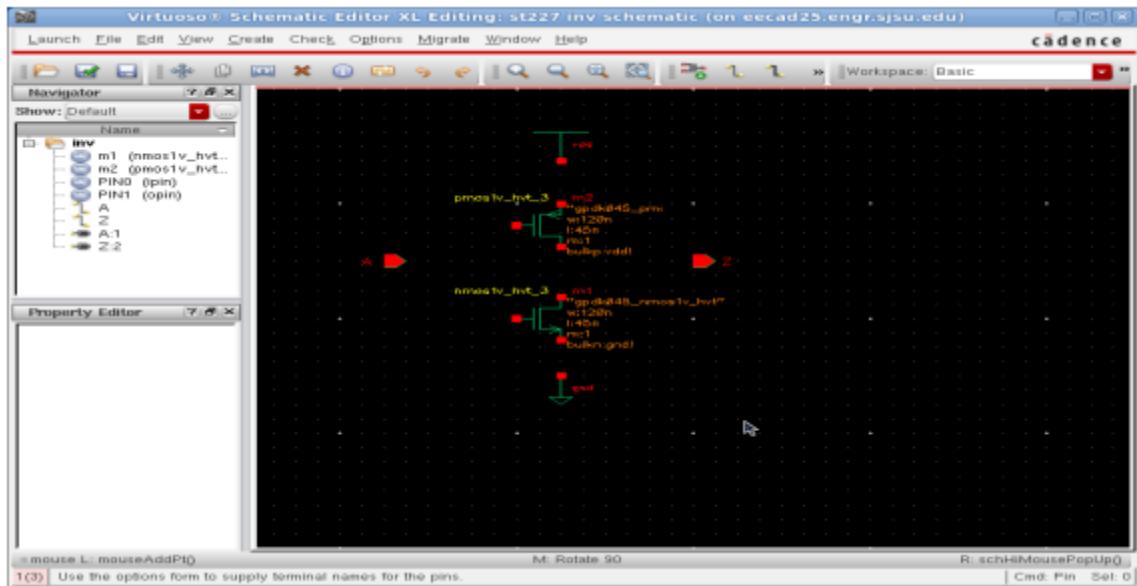


Figure 26: Schematic window with Pin's

c. Routing

Now time to add some wire to the schematic. The single width wire is just to the right of the transistor symbol. Click on it, and then move the cursor to the first point to be wired. It should be highlight with a yellow diamond.

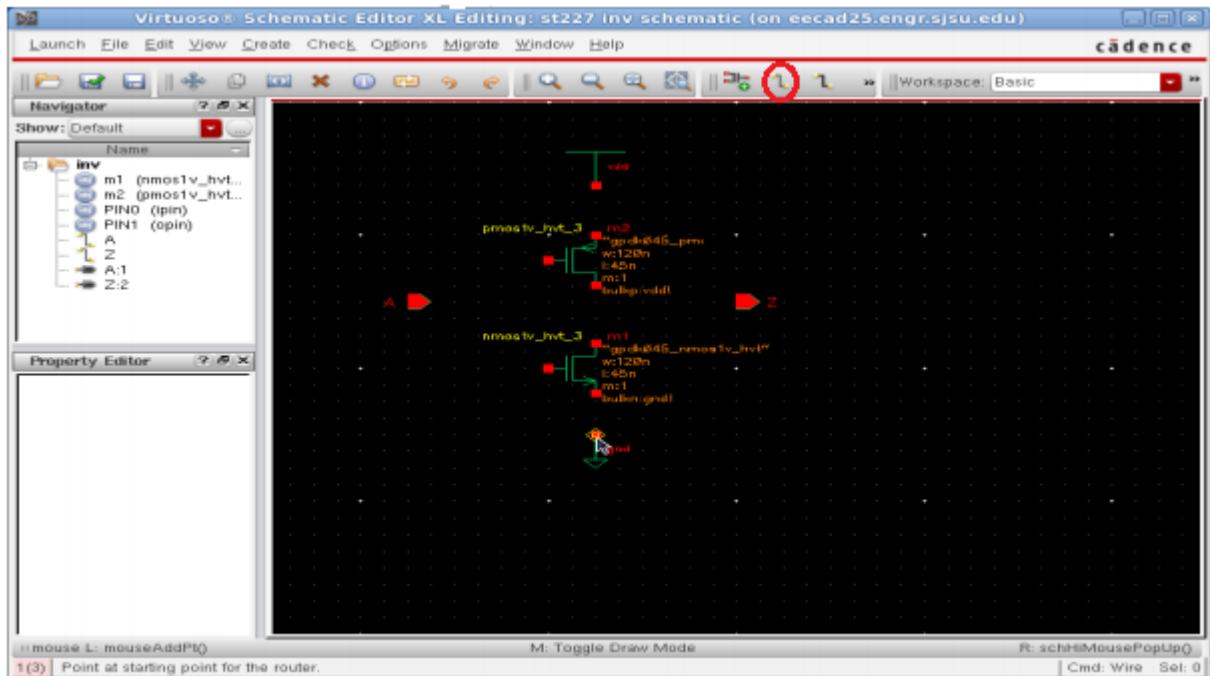


Figure 27: Click a point to draw a wire

Some notes, cadence doesn't allow 4 connections at a point. If you make a mistake, then press the escape key. Click on the wire in error if any is left, and then click on edit then delete. Now, click on the floppy disk with the green arrow. This will check the design and save it. If all goes well, it should work.

d. Bind keys for Schematic Capture

Hot Key/bind Key	Functions	Details
i	Instance call	Insertion of any kind of instance in schematic capture window
u	Undo	Go back to your previous work
Shift + u	Redo	Go forward to your work if available
w	Wire	Drawing wire for connecting points. Press 'w', then click one to start and double click to stop or press 'esc'
l	Label	Add a name to a wire. Press 'l' then enter a name of label, then click on the wire you want to add the label
]	Zoom in	Zoom in the screen of the capture
[Zoom out	Zoom out the screen of the capture
q	Properties	To view all properties of any kind of instance/pins/wire
p	Pin	To add pin as a input/output notation
f	Fit Screen	Fit the whole capture to the screen
m	move	To move from one place to another. Press 'm' then click on the instance you want move
E	Descend Edit	To view in a symbol what the symbol already captured for descend edit
e	Descend Read	To view in a symbol what the symbol already captured for descend read only
c	Copy	To copy any instance/wire/pin etc. Press 'c' then click on the instance you want to make a copy
Delete/del	Delete	To delete anything

Table 1: Bind keys for Schematic Capture