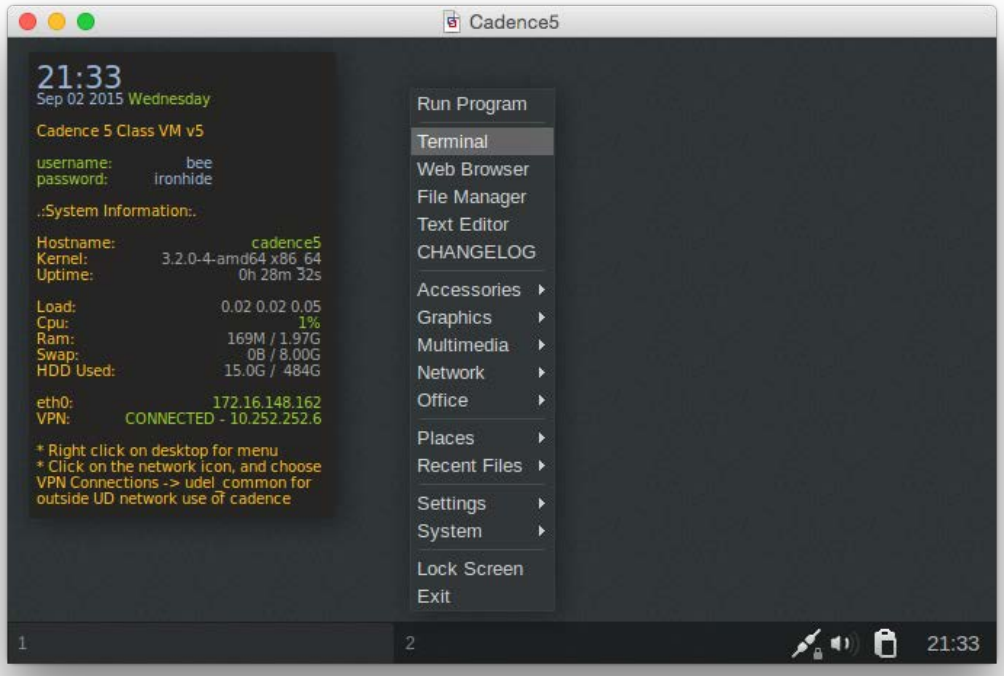
LAB1 – Inverter Tutorial: Schematic & Symbol Creation

Created for Peyman Barakhshan, CPEG460/660, Fall 2019, University of Delaware

Assumptions:
-You have downloaded Cadence virtual machine and installed VMWARE on your computer.

1. Start Cadence VM and left-click on the connection icon in the lower-left corner. Select “VPN Connections” and then select “udel_common”. This will connect you to the UDEL VPN server allowing you to run Cadence software.

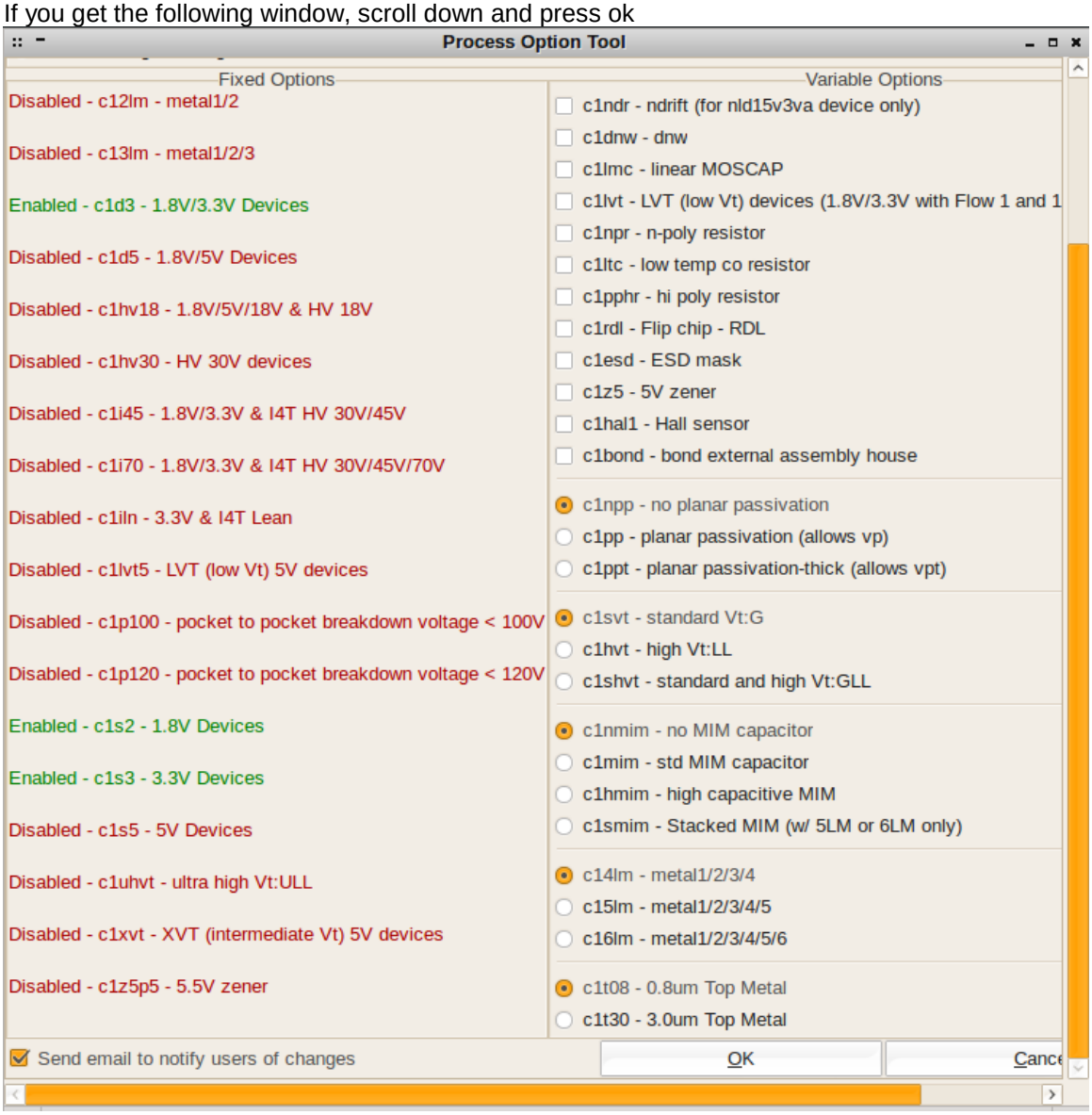
Once done, right click anywhere on the desktop and choose **Terminal**



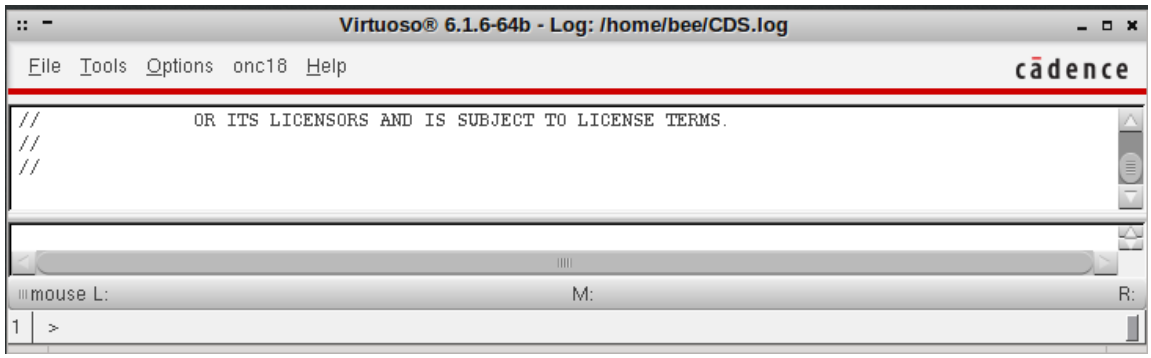
2. ***Start Cadence using the open terminal:***

```
$ virtuoso &
```

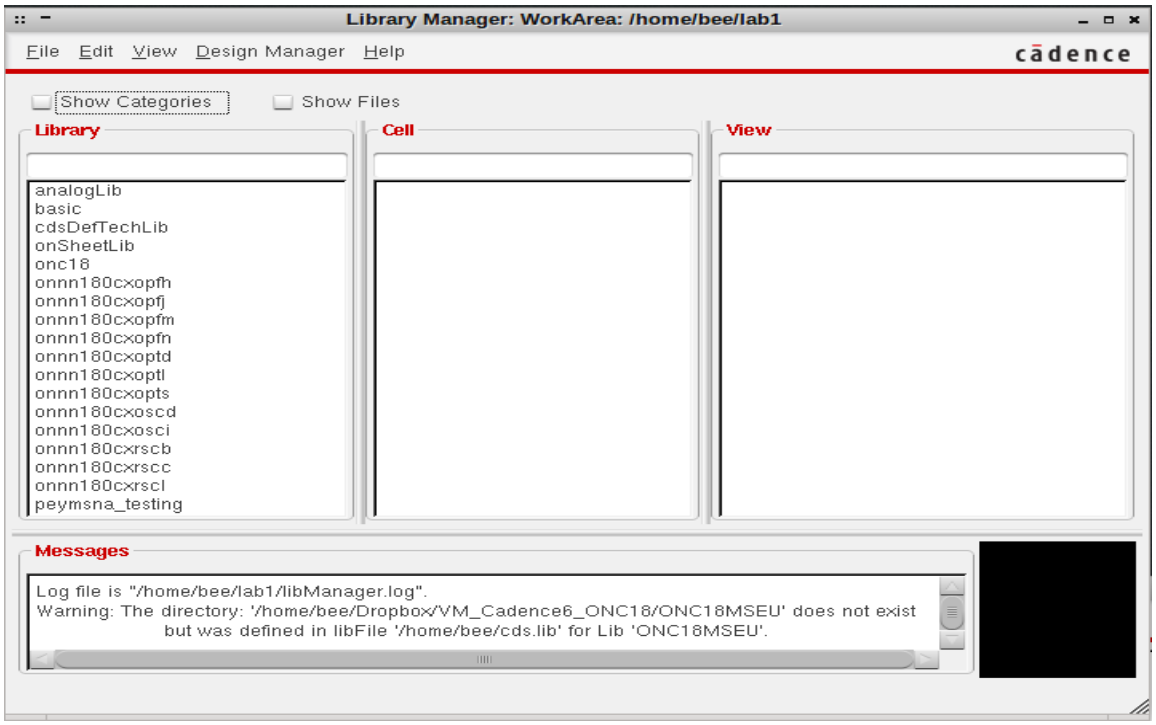
If you get the following window, scroll down and press ok



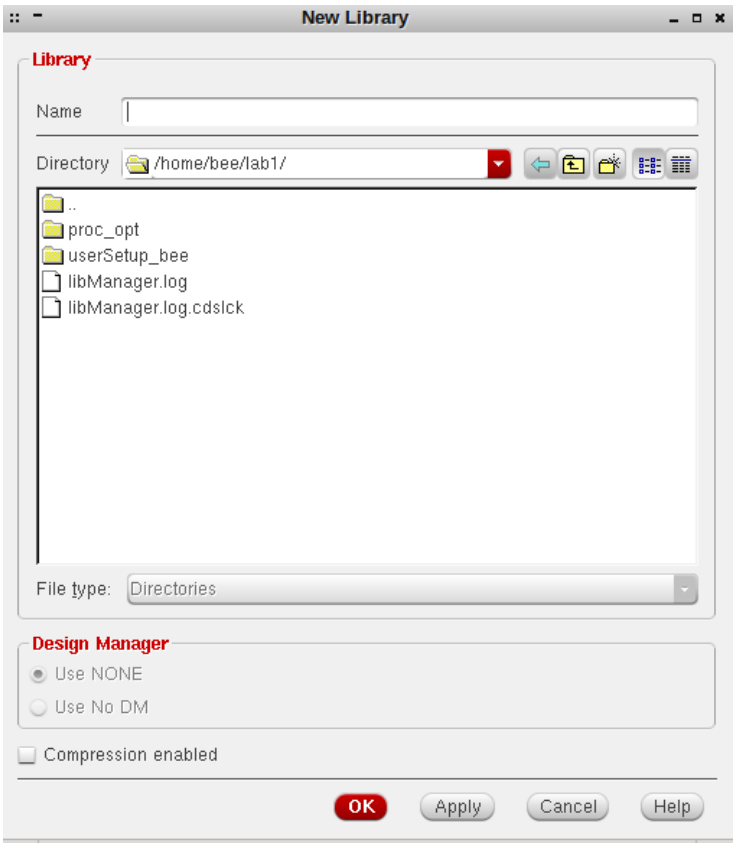
- After starting cadence, check the virtuoso window (screen shot is below). Double check the window and ensure that there are no error messages. Hint: Click on the scroll bar on the right to scroll up the window.
- If you see errors, or do not see the message above, ensure you have followed all of the steps above.



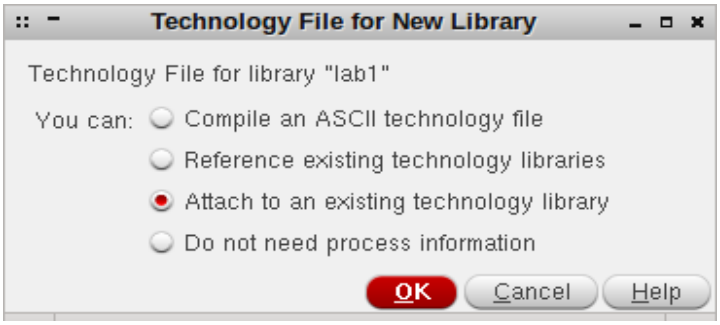
3. *Open the library manager window (Select Tools->Library Manager)*



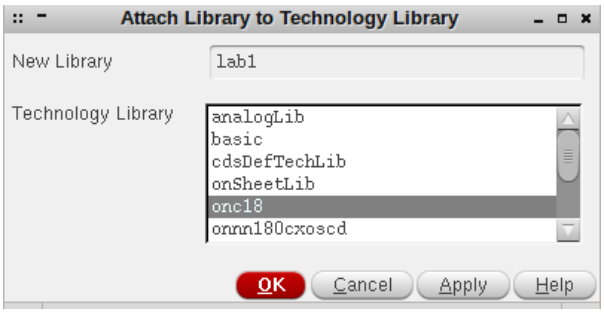
- From the menu, select **File -> New -> Library**
- You will see the following dialog box:



- In the "*NAME*" section type: **Lab1** and click **OK**



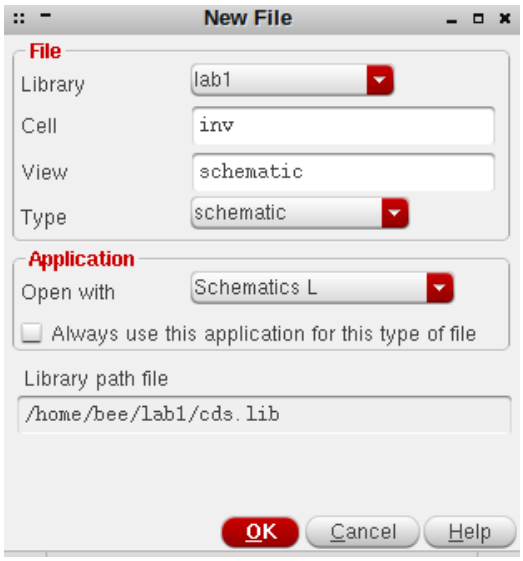
- In the next dialog box, select "*Attach to existing tech library*" section and click **OK**
- Confirm that the next dialog box is like below and click **OK**



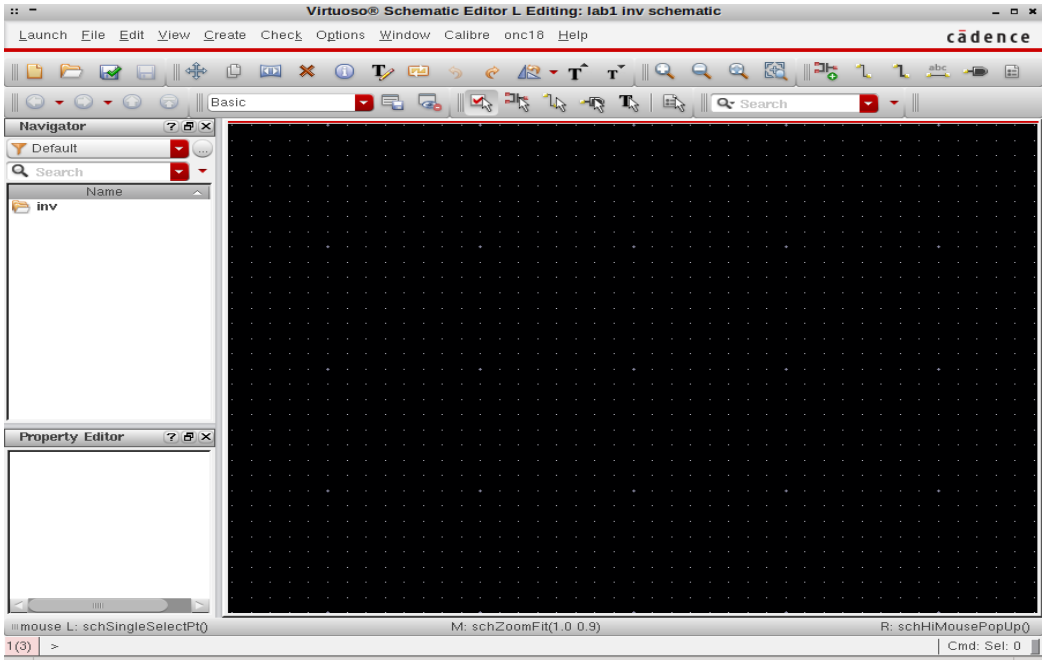
3. Creating a New Design:

- From the Library Manager, click on the “lab1” library you just created.
- From the menu, choose File → New → Cellview, and fill in the form as shown below in order to define a new schematic for our inverter.

- In the **Library** field, ensure lab1 is selected.
- In the **Cell** field, type “inv”
- From the **Type** selection drop down, choose “schematic”
- The **View** field will change to “schematic” automatically.
- Click OK

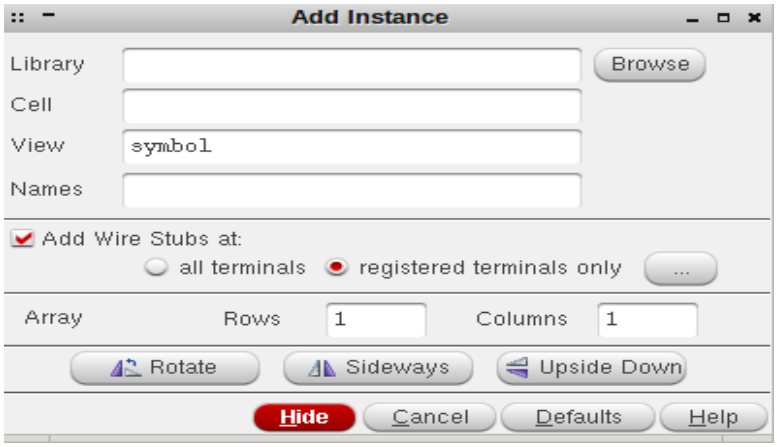


- The Cadence Virtuoso Schematic Editor will appear with a black background.

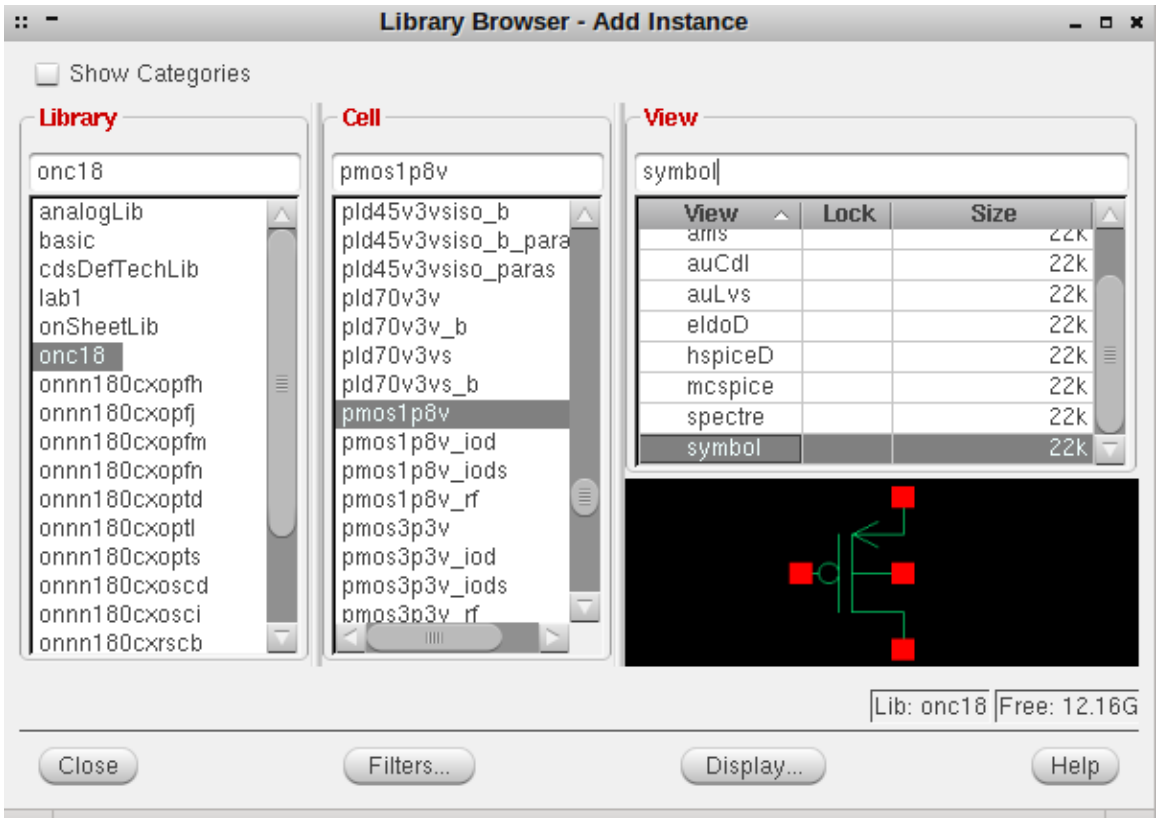


4.1 Instancing parts to create the inverter:

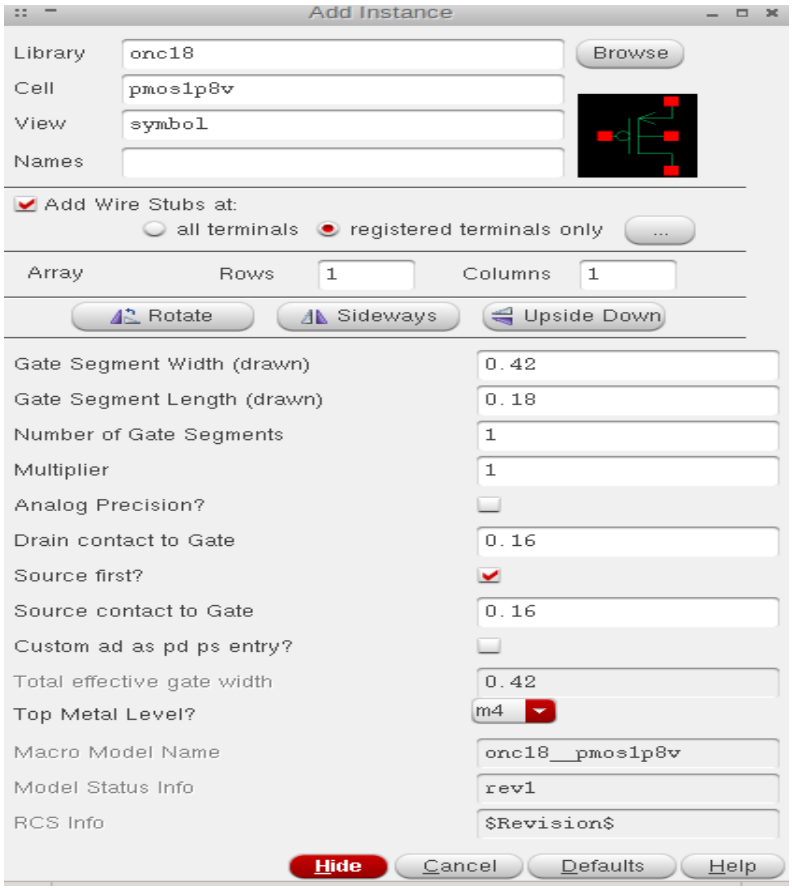
- From the menu choose: Create->Instance (or press the letter lowercase **i**)
- Click the ***Browse*** button in the form. The **Component Browser** form appears.



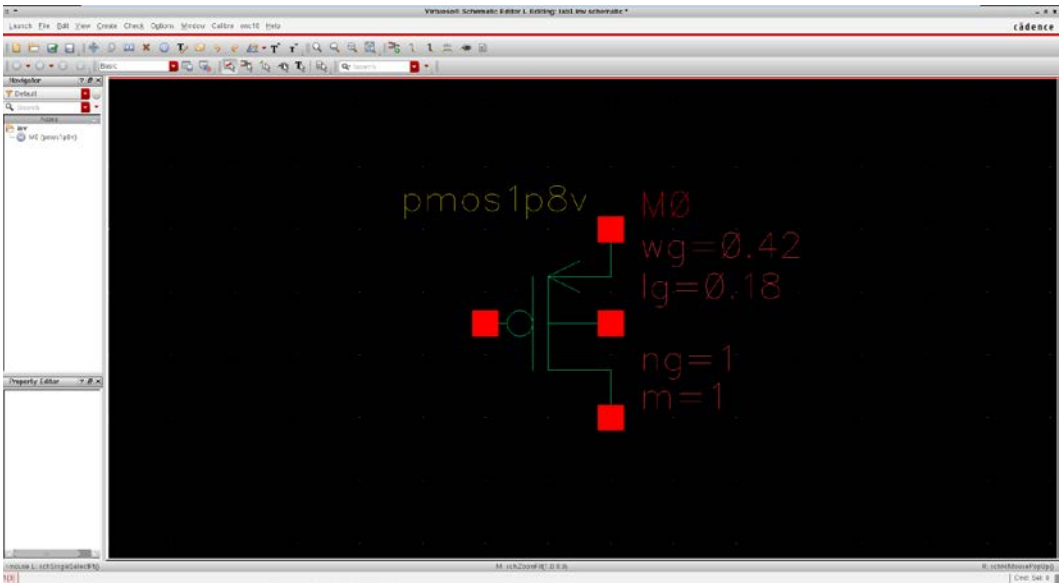
- Select the following:
  Under the ***Library*** column, select **onc18**.
  Under the **Cell** column choose: **pmos1p8v** (this is the PMOS transistor)
  Under the **View** column choose: symbol.



*Note that the **Add Instance** form has expanded to display various parameters for the pmos part. You can edit the pmos parameters (like it's width and length), but for now, do not change any parameters. We will use the default size for the 4 terminal PMOS, which is W/L = 0.42u/.18u*



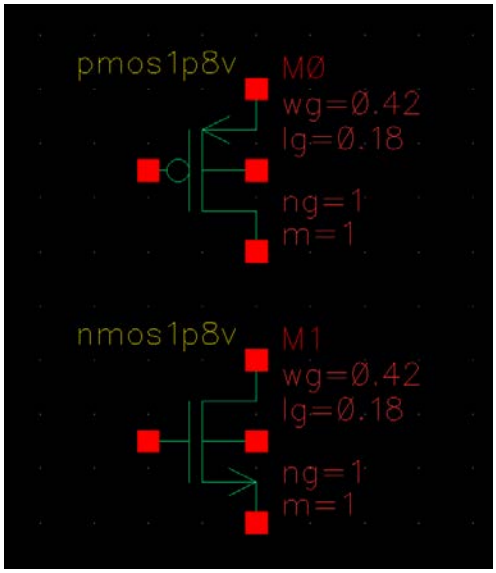
Click in the composer window to place the PMOS transistor. Your schematic should now look like this:



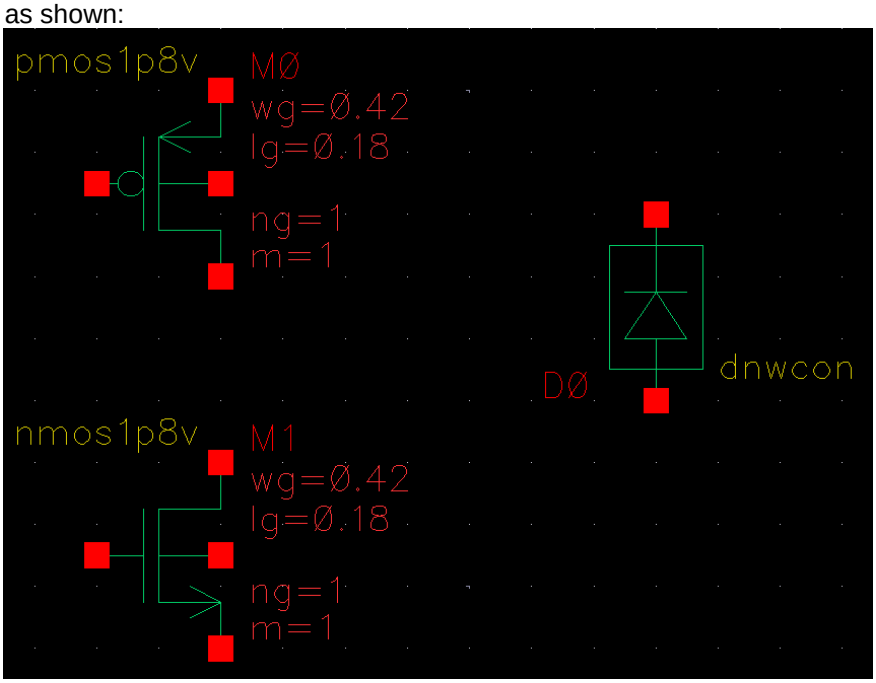
- After you have placed the PMOS transistor, press <esc> to end “placement” mode.

- Add and nmos1p8v part (this is an NMOS transistor) using the same procedure. Leave the default W/L, do not modify any of the default parameters.

- Place the NMOS transistor below the PMOS transistor as shown here:



- Add one other part from the onc18 as indicated below:
  - dnwcon
- Place it as shown:

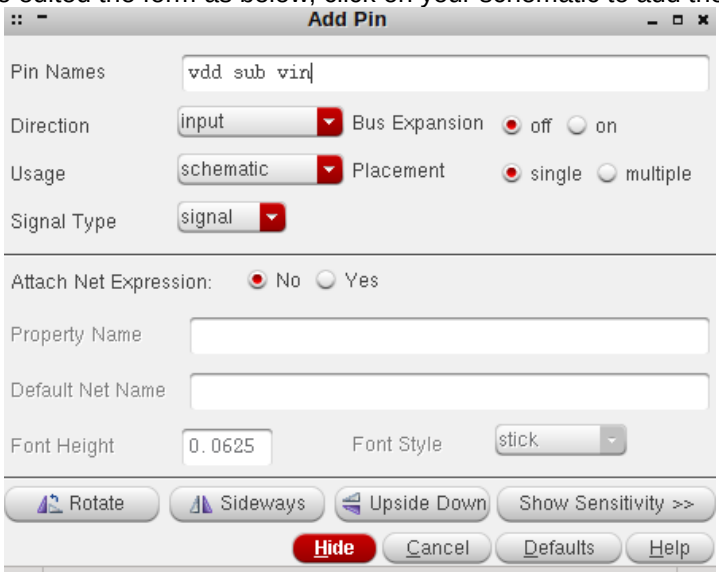


- Press <esc> to end “placement” mode.

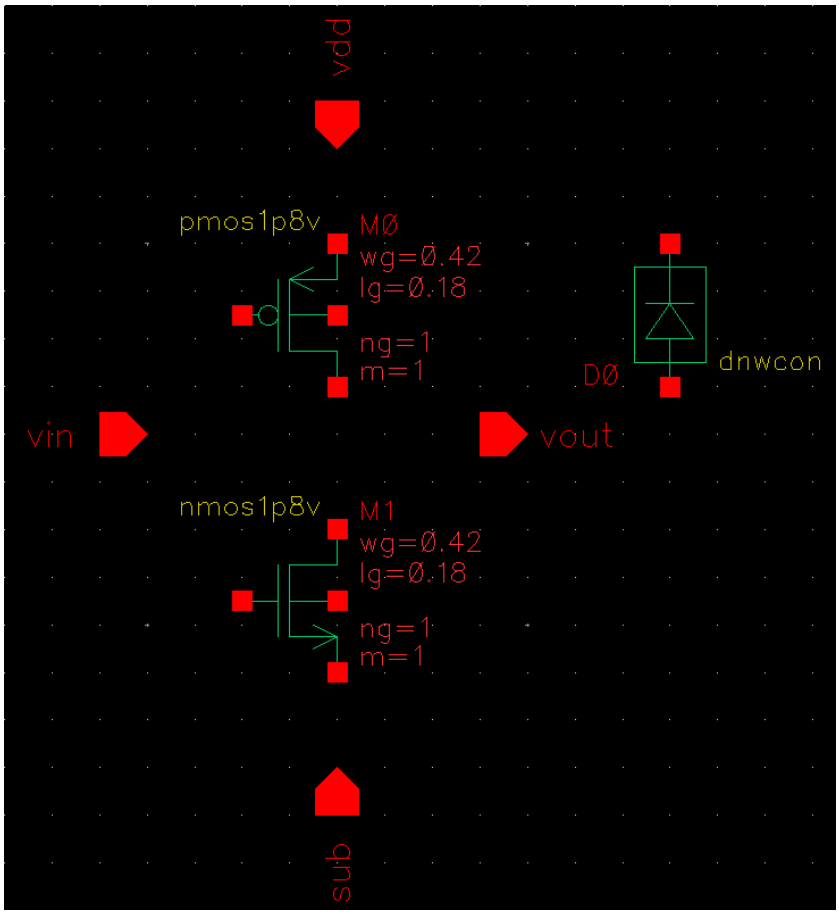
4.2 Adding the I/O Pins

- To add the input and output pins, choose: Create->Pin from the menu, or press the P key
- The **Add Pin** form appears.
- Under Pin Names, type the name of the pin: **vdd sub vin**

- Choose the Direction to be **input**
- Once you have edited the form as below, click on your schematic to add the pin.



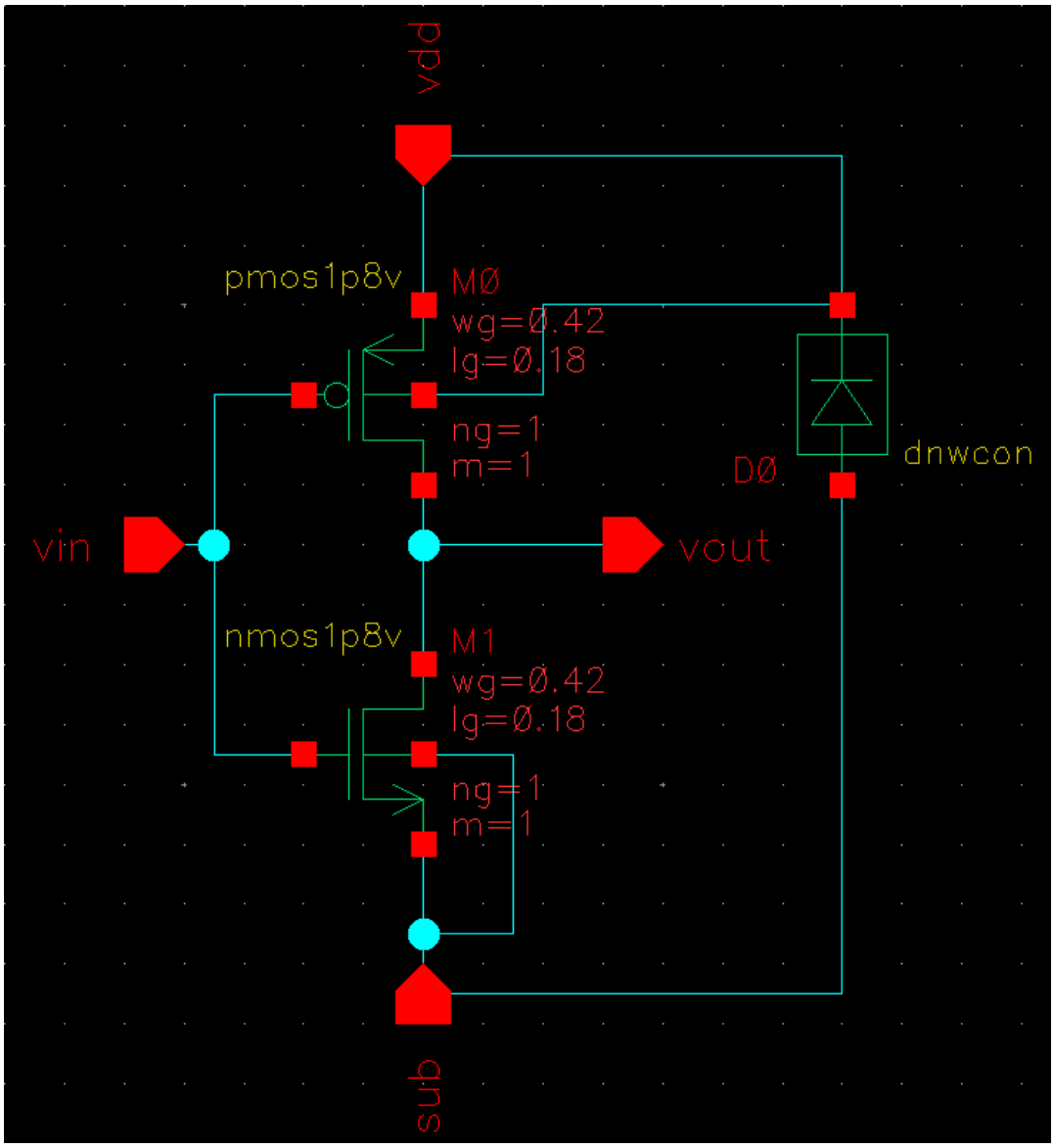
- Repeat the procedure to create an OUTPUT pin named: vout, see below (make sure you choose the DIRECTION to be “OUTPUT” on the add pin form:



4.3 Connecting Wires

- To connect wires, select **Add->Wire(narrow)** or press the w key.
- Click once where you want a wire to begin, click a second time where you want the wire to end.
- Wire the components as show below.

(Don't forget to wire the PMOS' body terminal to VDD and the NMOS' body terminal to GND.)

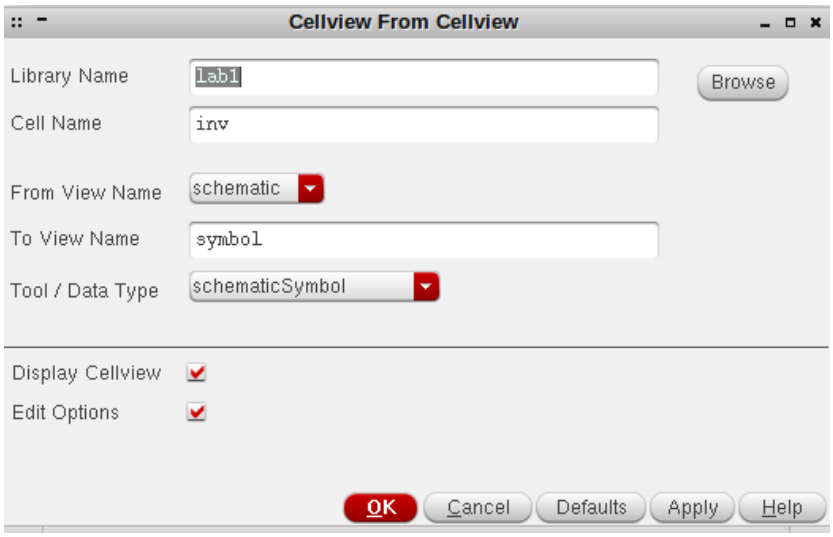


4.4 Checking and Saving

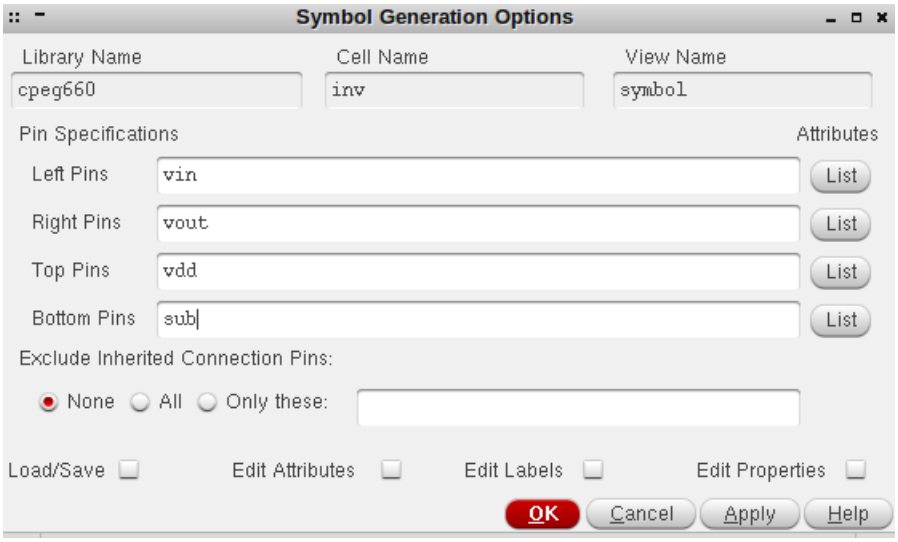
- To check and save the schematic, from the menu choose: **File->Check and Save**.
- Alternatively, you may click the checkmark icon on the left/top of the schematic editor.
- If Warning/Error appears, go back to the schematic and fix the problem as necessary. Warnings are not as crucial as Errors.
- Repeat until you have no Errors.
- Look in the **virtuoso** window (at the bottom) to ensure there are no errors.

5. Creating the Symbol Cell view

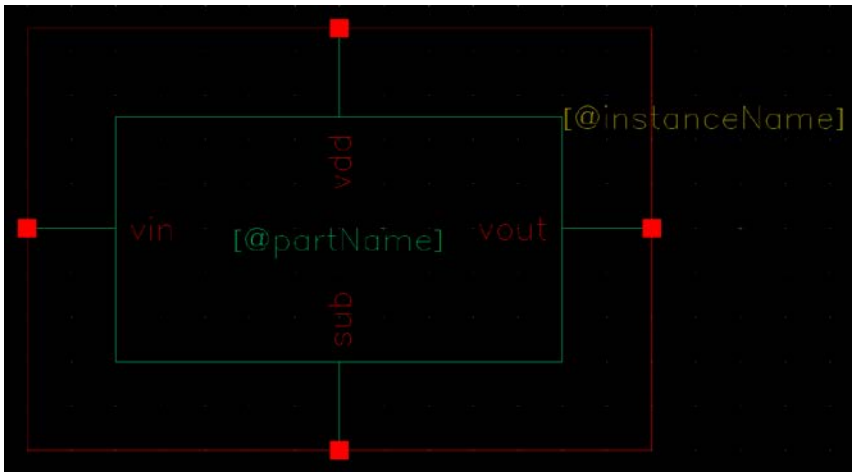
- Now we'll create a symbol (black box) to represent our circuit. The symbol Cellview will be created based on the already-available schematic Cellview. This is called *creating a Cellview from the schematic view*.
- From schematic editor's menu choose: **Create→Cellview→From Cellview**.
- Click OK on the form that appears, leaving all of the default values as shown below:



- Do the following changes in the dialog box, and press OK



The "symbol editor" will open. You will see a green box surrounded by a white box. The input and output pins you defined in the schematic will appear as terminals. It is perfectly fine to use this rectangle as your symbol. But we will change the box into the symbol for an inverter, for use in future schematics.

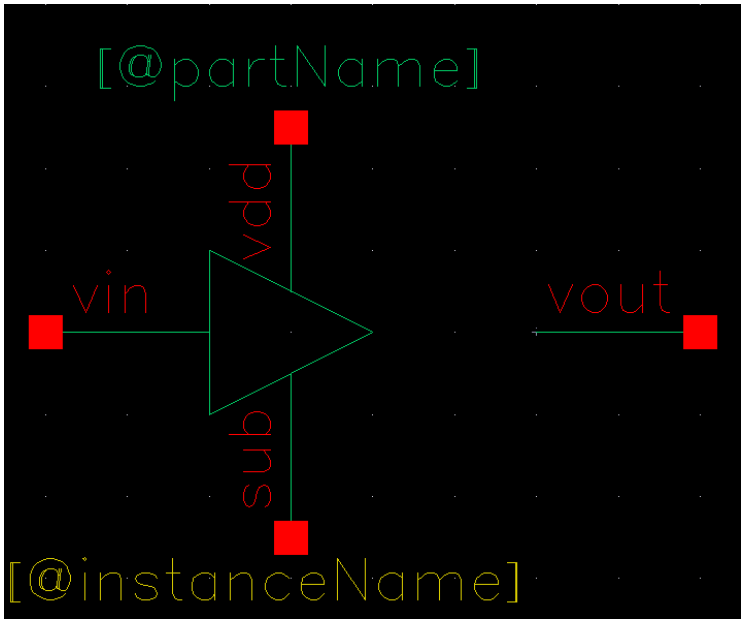


- press the **delete** key on the keyboard.
- click on the lower left hand corner of the green box.
- click on the lower left hand corner of the red box.
- press <**esc**> to end “delete mode”

- press the **m** key to enter “move mode”
- click on the **[@partName]** text and move it to the bottom of the drawing
- click on the **[@instanceName]** text and move it to the bottom of the drawing
- press <**esc**> to end “move mode”

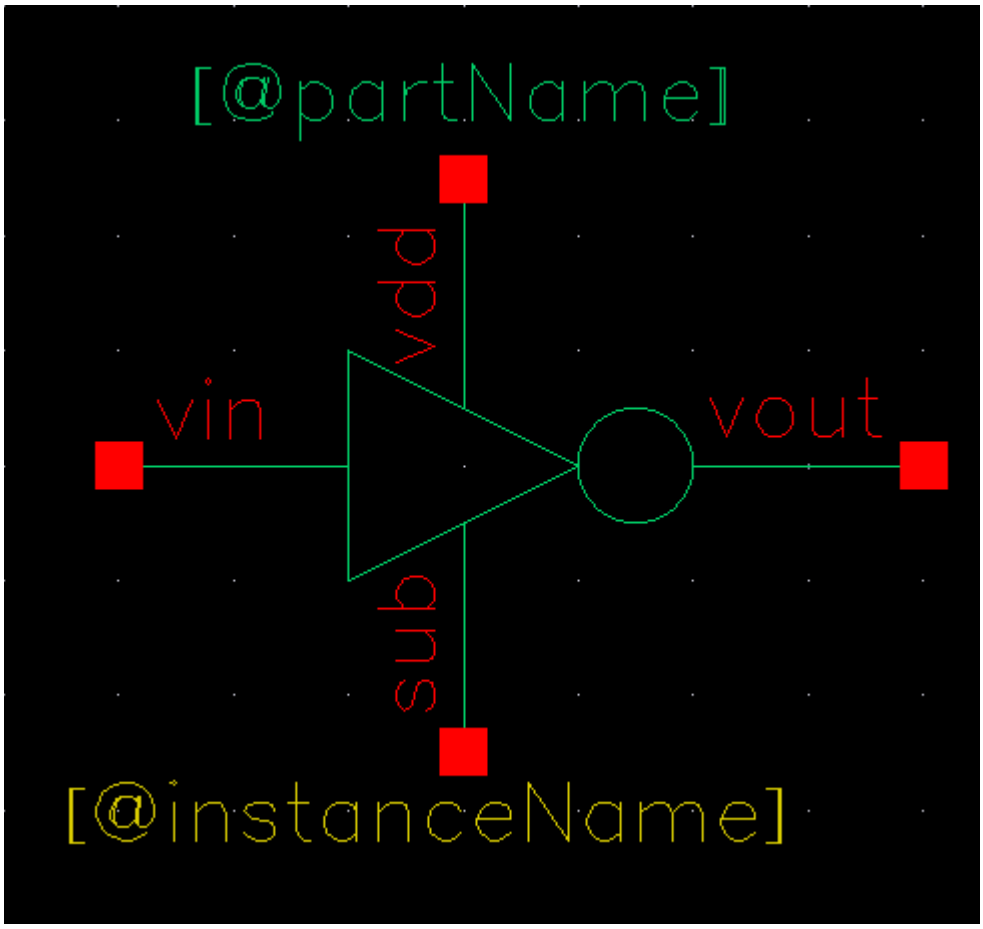
- Select **Create->Shape->polygon** from the menu.

- draw a triangle as shown below (note: click where you want the verticies to be placed, do not “click and drag”)



- press the <**esc**> key to end “polygon mode”
- Select **Cretae->Shape->circle** from the menu

- click on the schematic, where you wish the **center** of the circle to appear (see graphic below):
- click again when you have the proper **radius**.
- press the <**esc**> key to end "circle mode"
- move the red terminals and the text to be positioned as follows:



- click on the "**save**" icon from the top left hand toolbar
- close the symbol editor
- close the schematic editor, saving any changes.

***Note**: If for any reason you have accidentally "deleted" one of the Red Pins, you must delete your symbol and start over again. You can delete the symbol from the Library Manager, by right clicking on the 'symbol' view for the "INV" cell, and choosing delete.*

By creating the symbol, you have created a logical connection between the input/output pins of your schematic and this symbolic representation. In the next lab, we will use this symbol to test your schematic in a test bench.

What to turn in: Two pages – screen shot of your schematic and of the symbol.