ECE 4140/6240 – Inverter Tutorial: Schematic & Symbol Creation

Updated history: 4/15/21: Jerry Wu

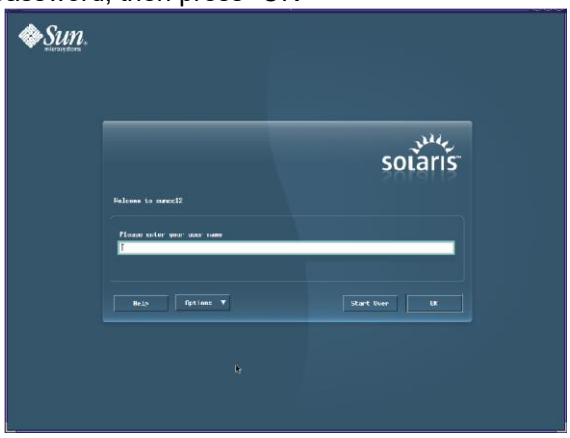
Updated history: 8/10/21: Jerry Wu

Tutorial adapted from:

Assumptions:
-You have configured your UNIX environment for Cadence in Lab 1

1. Log into a Sun Workstation: (TA will provide a new instruction for the updated environment)

Type user name and password, then press "OK"

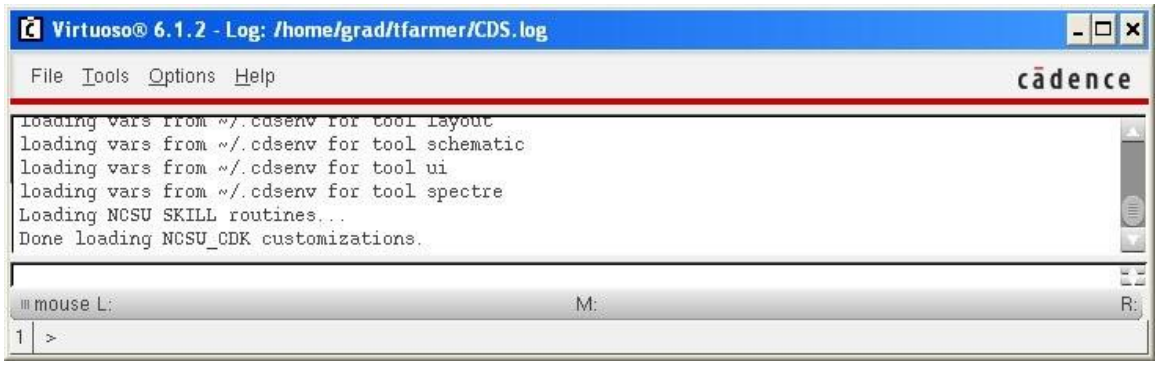


Once logged in, right click anywhere on the desktop and choose **Tools->Terminal**

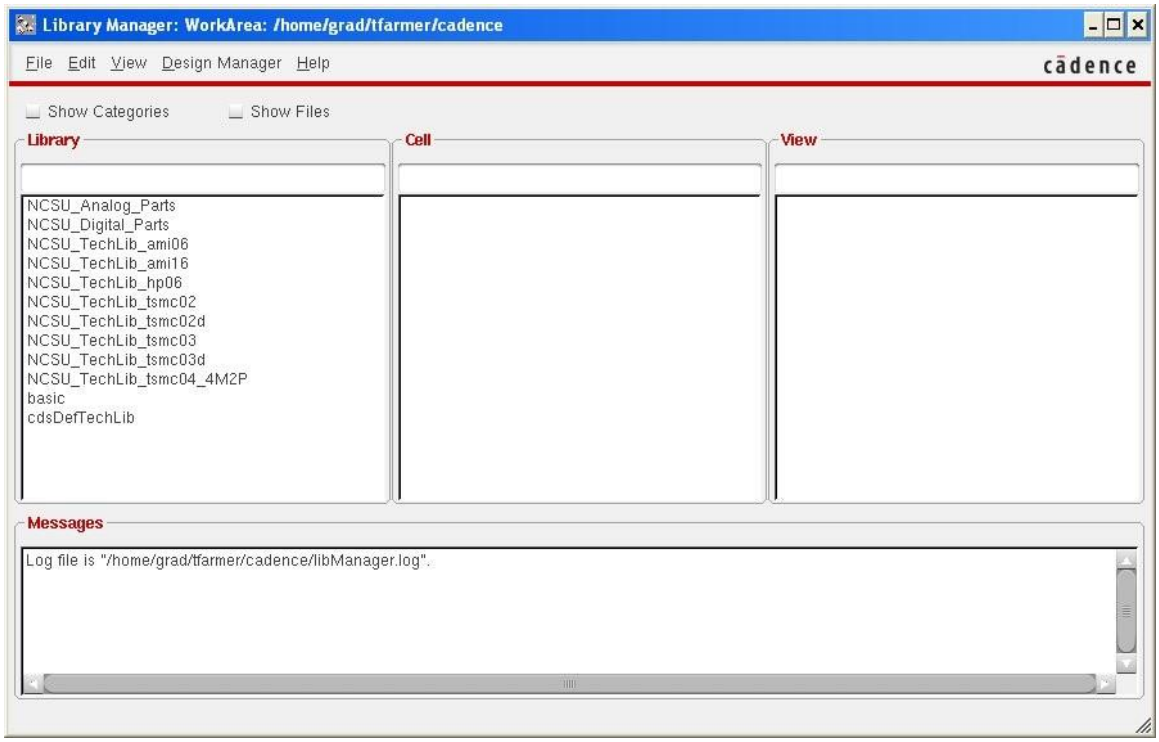
2. Start Cadence using the open terminal:

```
$ cd cadence
$ virtuoso &
```

- After starting virtuoso, check the CIW window (screen shot is below). Double check the window and ensure that there are no error messages, and ensure that the sentence: **"Done Loading NCSU_CDK customizations…"** appears.

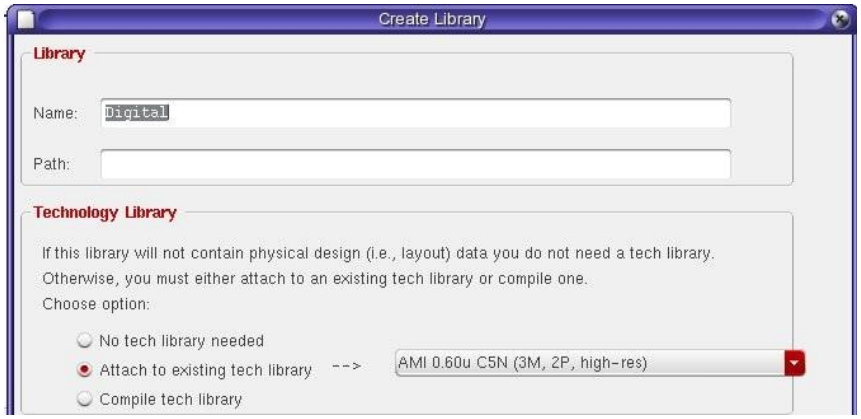


3. Go to the library manager window



- From the menu, select **File -> New -> Library**
  You may see one of two dialog boxes:

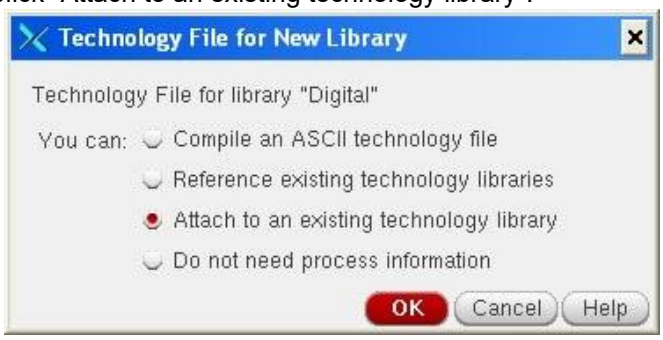
- If you see a dialog box that looks like this:



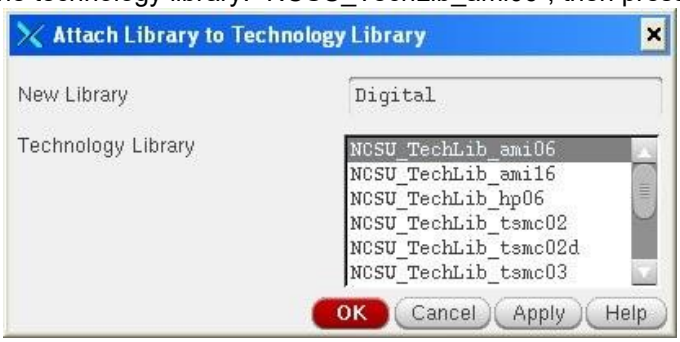
  - In the “NAME” section type: Digital
  - In the “*Attach to existing tech library*” section, select: **AMI 0.60u C5N (3M, 2P, high-res)**
  - **Press OK**
  - Double check the CIW window, ensure there are no errors,
  - **proceed to STEP 4**

- If your dialog box looks slightly different, and there is no “*Technology Library*” section, then:
  - Under Library name type “Digital” and press OK

- Another box will appear like the one below:
  - Click “Attach to an existing technology library”.



- Select the technology library: “NCSU_TechLib_ami06”, then press OK.

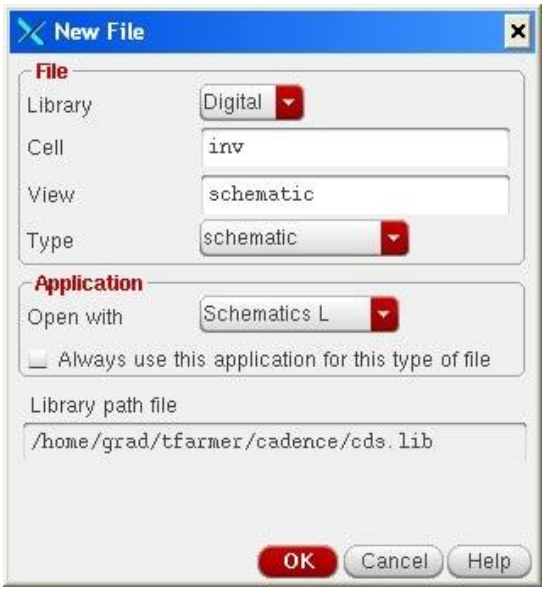


- Double check the CIW window, ensure there are no errors.

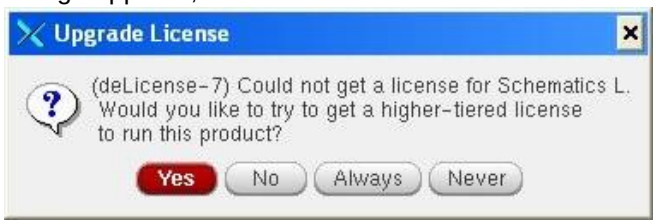
4. Creating a New Design:

- From the Library Manager, click on the “Digital” 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 Digital 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



- If the following message appears, click ALWAYS:

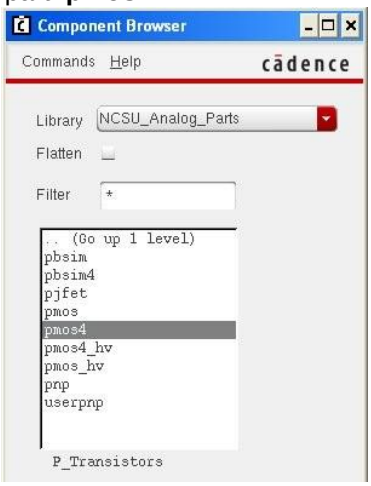


- The Cadence Virtuoso Schematic Editor will appear with a black background.

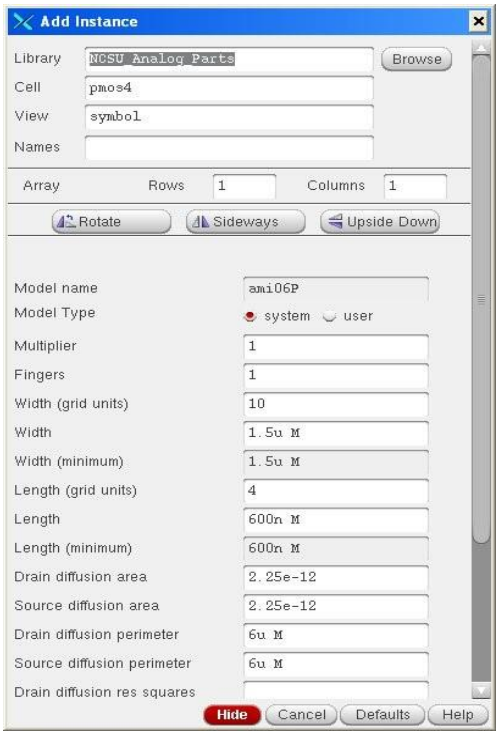
4.1 Instancing parts to create the inverter:

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

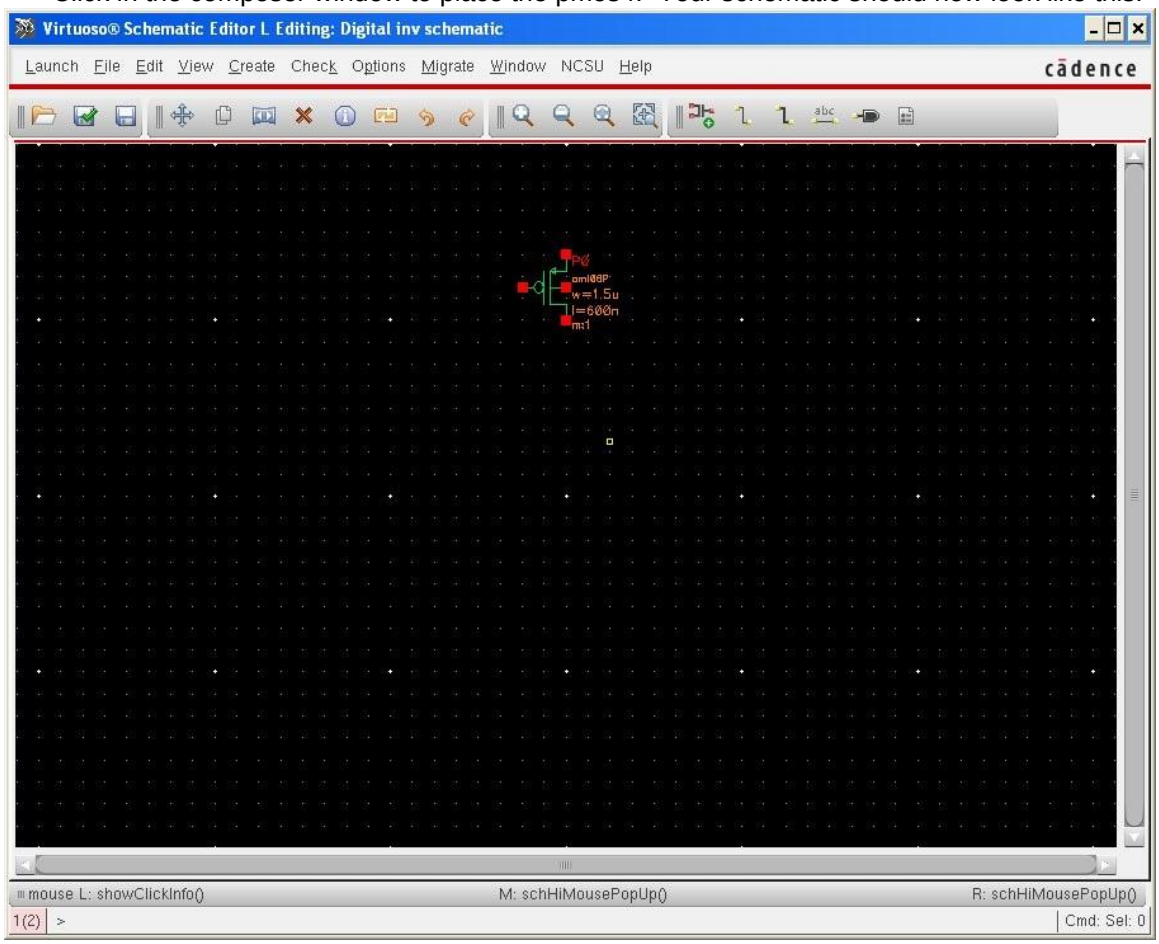
- Select the following:
  Under the ***Library*** column, select **NCSU_analog_parts**.
  In the list choose: **P_Transistors**
  From the new list choose the part: **pmos4**.



We will use the default size for the 4 terminal PMOS, which is W/L = 1.5u/.6u



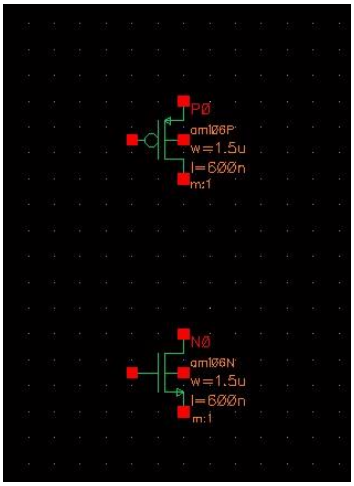
- Click in the composer window to place the pmos4. Your schematic should now look like this:



- After you have placed the pmos4, press <esc> to end “placement” mode.

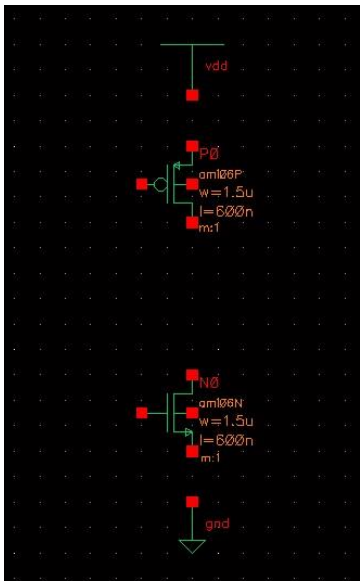
- Add and nmos4 using the same procedure.
- Its part location within the component browser is:
  NCSU_analog_parts\N_Transistors\nmos4
  Leave the default W/L, do not modify any of the default parameters.

- Place the nmos4 transistor below the pmos4 as shown here:



- Add two other parts from the NCSU_analog_parts as indicated below:
  - Voltage source (**NCSU_analog_parts\Supply Nets\ vdd**)
  - Ground (NCSU_analog_parts\Supply Nets\gnd)

- Place them as shown:



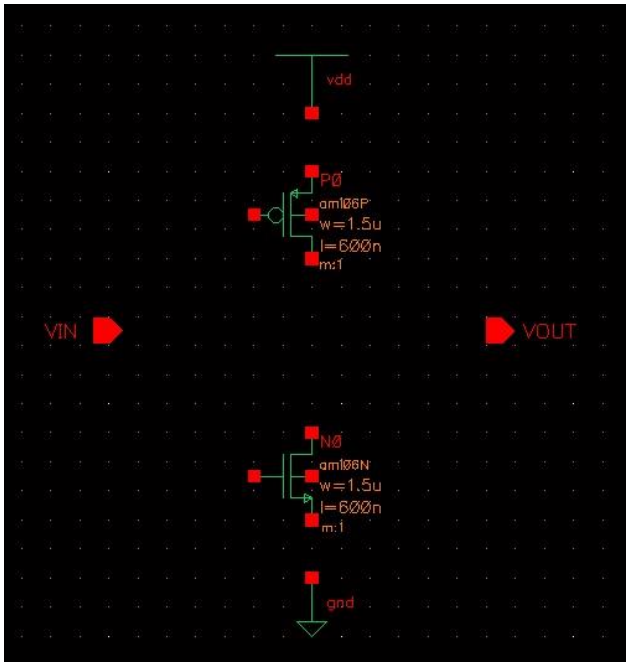
- Press <esc> to end “placement” mode.

4.2 Adding the I/O Pins

- To add the input and output pins, click on the Pin icon on the upper icon menu. Alternatively, you can 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: **VIN**

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

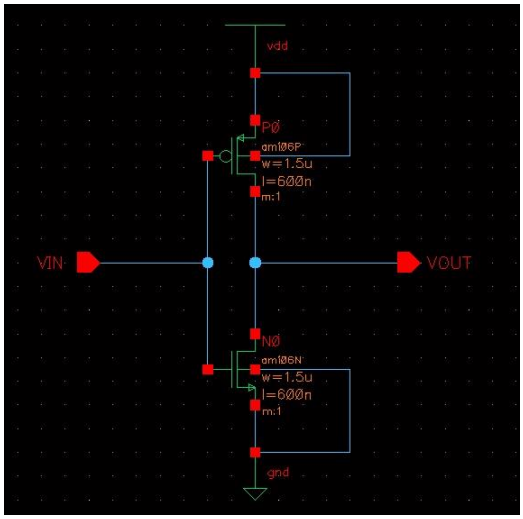
- Repeat the procedure to create an OUTPUT pin named: VOUT, see below (make sure you choose the DIRECTION to be “OUTPUT” on the add pin form:



4.3 Connecting Wires

- To connect wires, click on the icon: **Create Narrow Wire**, it in the top row of icons under the menu.
- 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.)



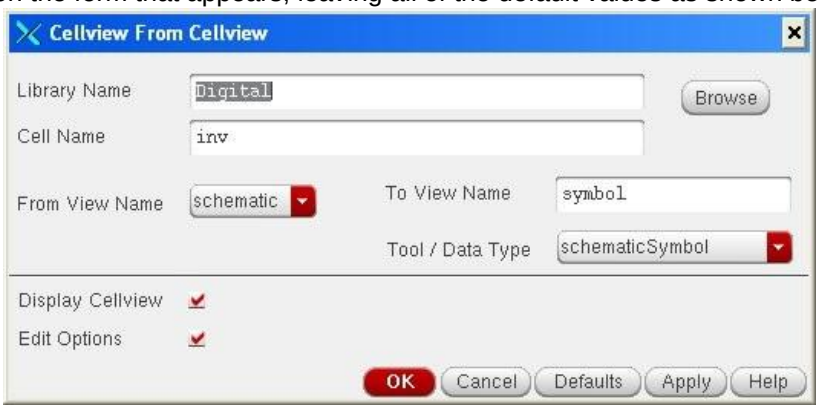
4.4 Checking and Saving

- 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.

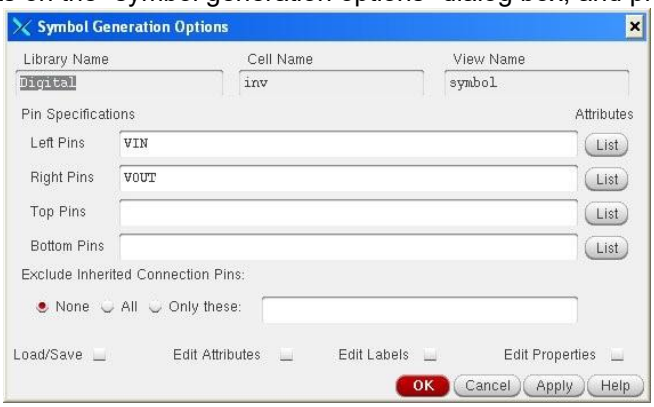
- If Warning/Error appears, go back to the schematic and fix the problem as necessary. Warnings are not as crucial as Errors.
- Repeat until you have no Errors.
- Look in the **CIW** window (at the bottom) to ensure there are no errors.

5. Creating the Symbol Cell 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:



- Leave the defaults on the “symbol generation options” dialog box, and press OK

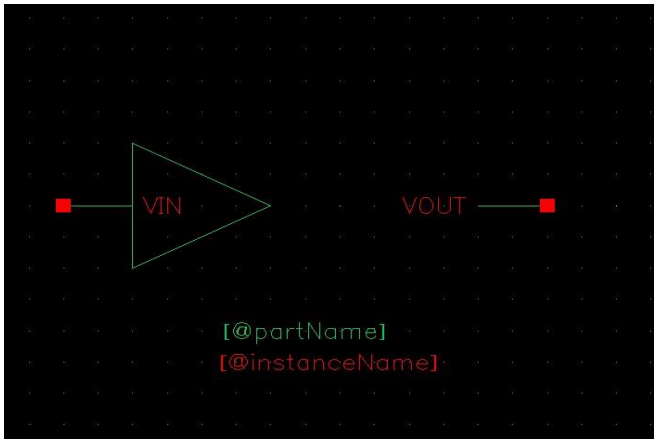


The "symbol editor" will open. You will see a green box surrounded by a red box. The input and output pins you defined in the schematic will appear as terminals. It is perfectly fine to use this rectangle as your symbol. But we will change the box into the symbol for an inverter, for use in future schematics.

- press the **delete** key on the keyboard.
- click on the lower left hand corner of the red box.
- click on the lower left hand corner of the green 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"

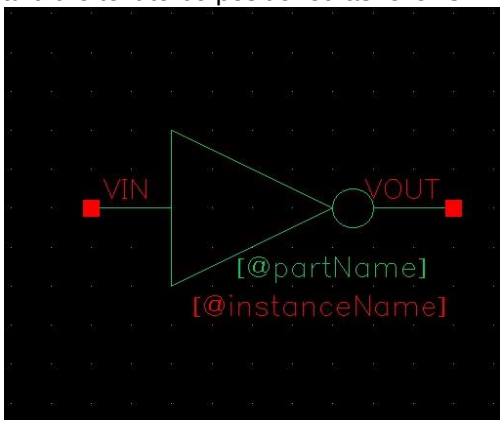
- click on the **create polygon** icon from top icon toolbar, or from the menu choose: create->Shape->Polygon
- draw a triangle as shown below (note: click where you want the verticies to be placed, do not "click and drag")



- press the <**esc**> key to end "polygon mode"

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