# Schematic Creation and Transient Simulation

Author: Jinhua Wang

This tutorial will guide you to build an inverter step by step.

1. Open Cadence Virtuoso 6.1.6 in **ECE Guacamole**
2. Open **Library Manager**
3. Create a new library (e.g.: ECE 4220)

**Note**: If you are not sure how to do 1, 2 and 3, please refer to "**00_ Environment_Setup**"

4. Under the library you just create, create a new cell view
   a. Click **ECE4220** under **Library**
   b. Click **File** in the tool bar on the top of the **Library Manager** window
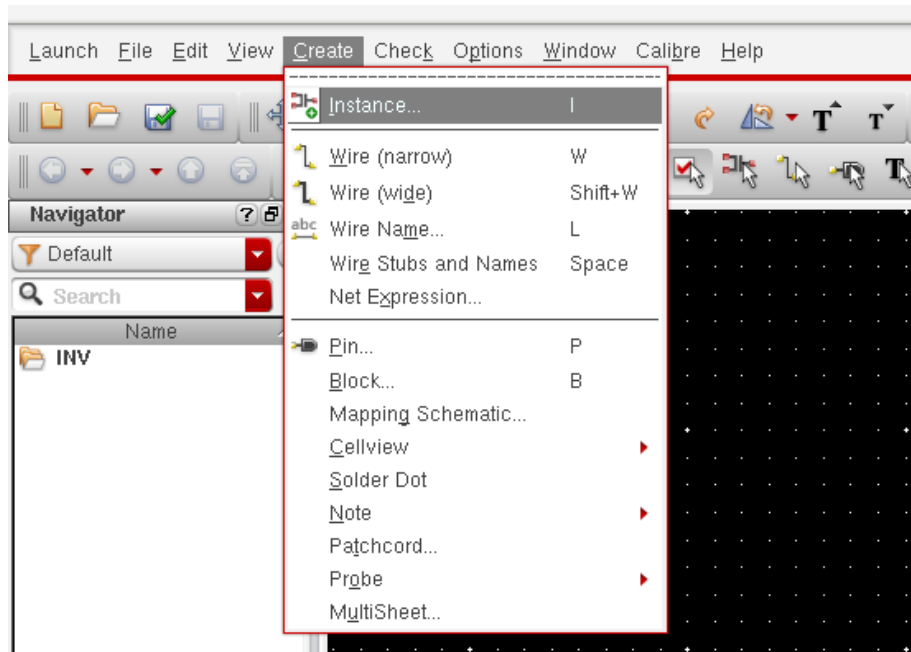   c. Click **New** and then click **Cell View**
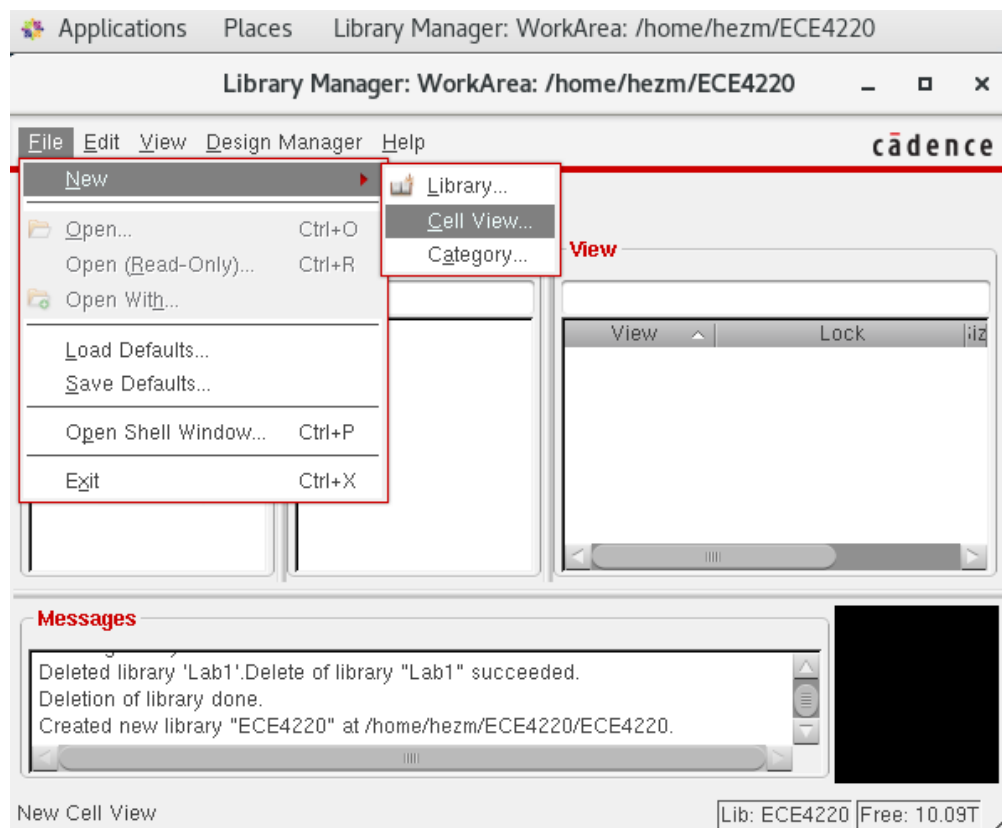
   d. A **New File** window pops up,
      i. For **Cell**, give a name your cell (e.g. **INV**)
      ii. For **View**, set it to **schematic**
      iii. For **Type**, set it to **schematic**
      iv. For **Open with**, set it to **Schematic L** and click **OK**
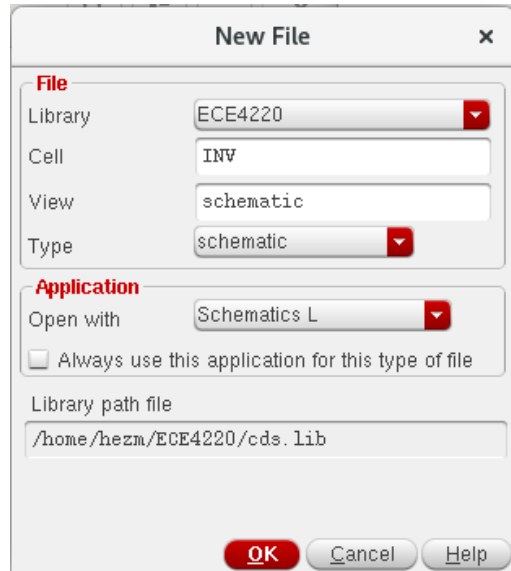
e. An empty schematic editor window should pop up
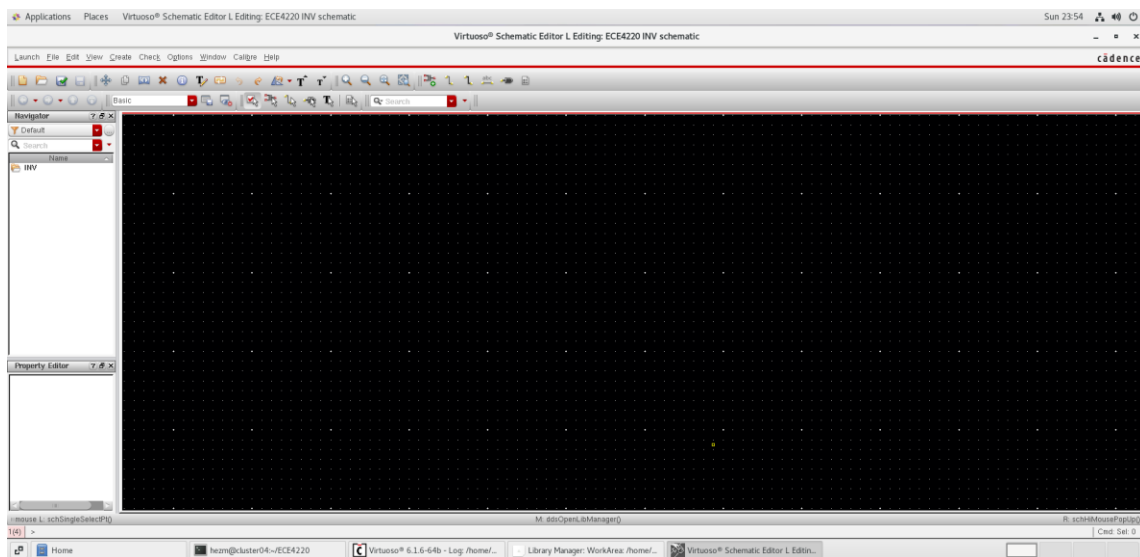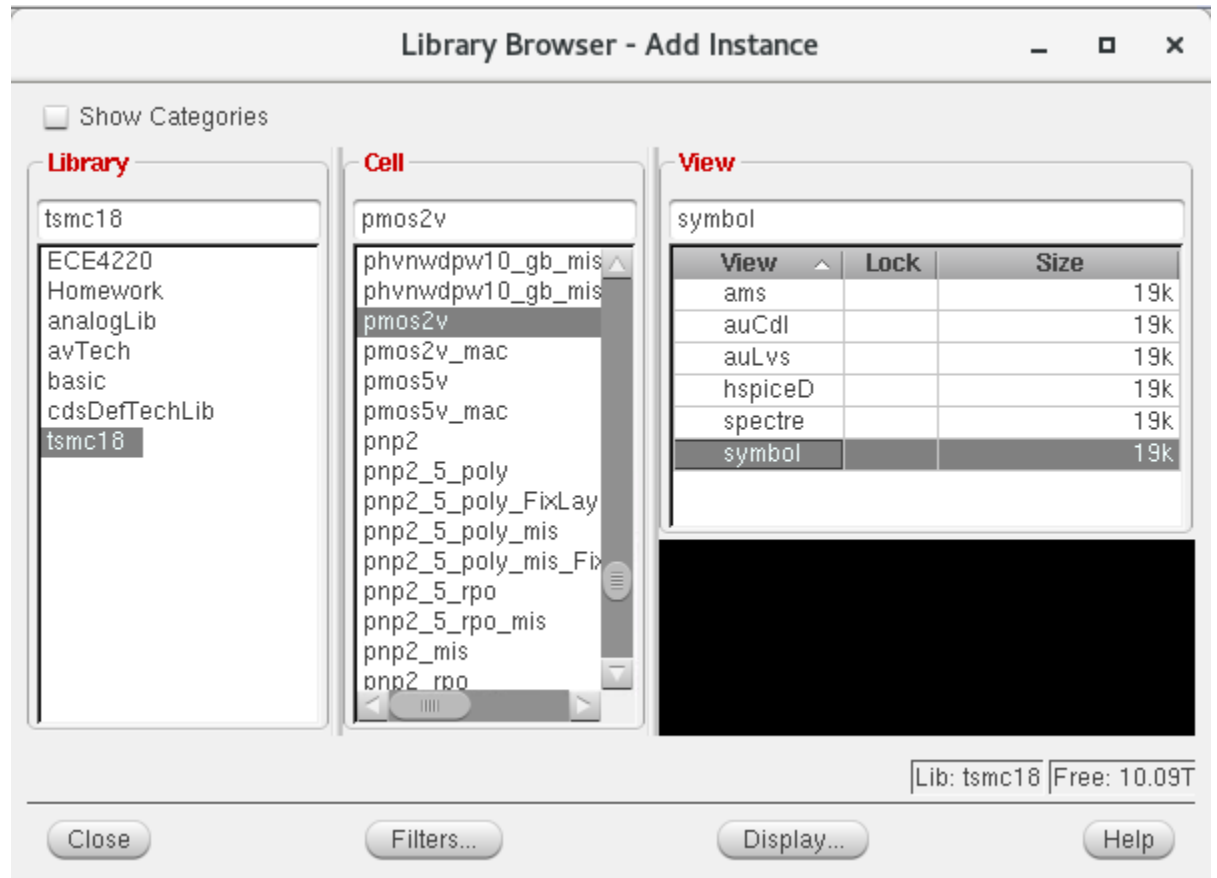
5. Place the components
    a. In the tool bar on the top of the editor, click **Create**, and then click **Instance**

    b. Click **Browse**, the **Library Browser – Add Instance** windows will pop up

c. Click **tsmc18** under **Library**, and select **pmos2v** under **Cell**, and select **symbol** under **View**, and click **Close**

d. Put the PMOS on the schematic
e. Do the same for NMOS (cell name: **nmos2v**)
f. Analog components such as voltage source, current source, ground can be found under the Library **analogLib**

g. For input, use "**vpulse**", right click on the component, click **Properties**, and set it up as below

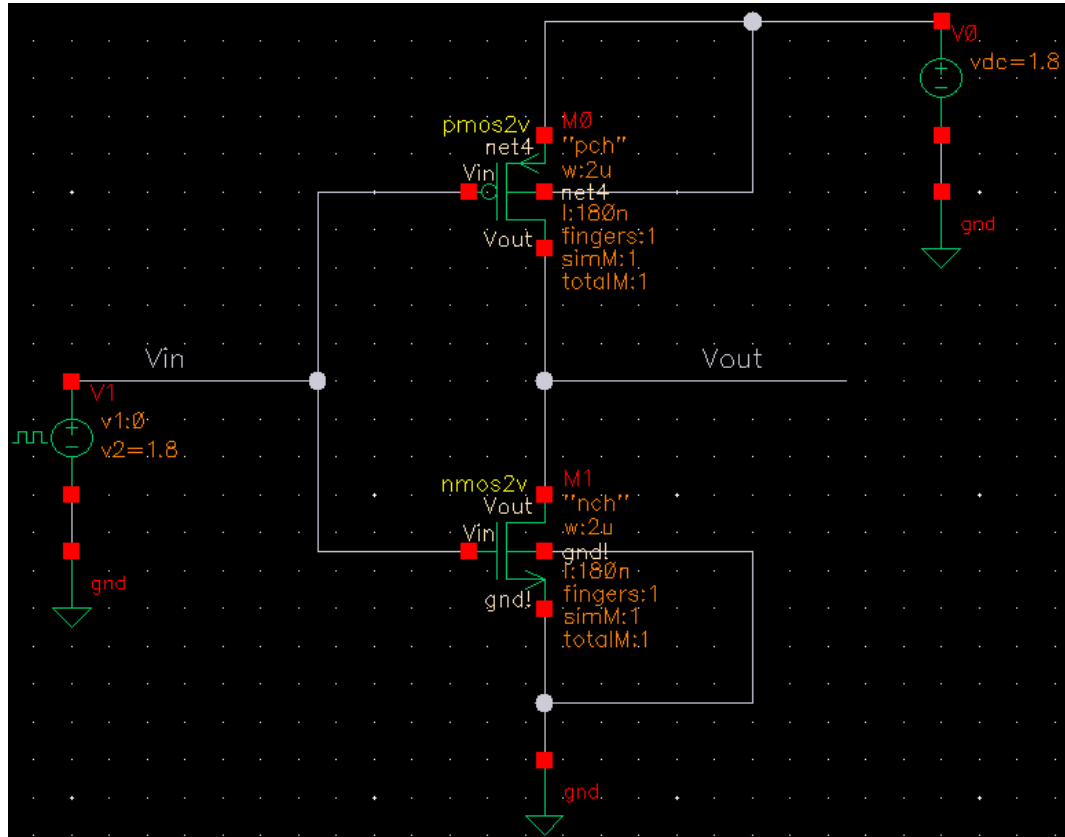i. For VDD, use "**vdc**", do the same as for "**vpulse**", and set **DC Voltage** to **1.8 V**.

h. To connect the components, select **Create Narrow Wire** in the tool bar
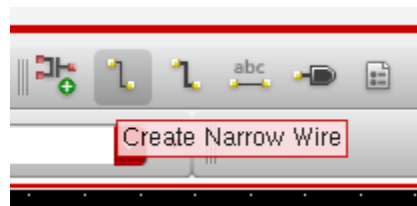
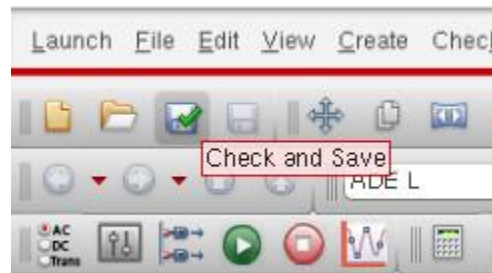i. You may name the wires (signals) by using **Create Wire Name**, and put the tag on the wires (signals)
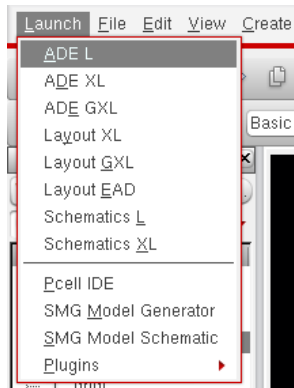

j. Below is the complete schematic for an inverter

k. Click **Check and Save** after you finish you schematic.
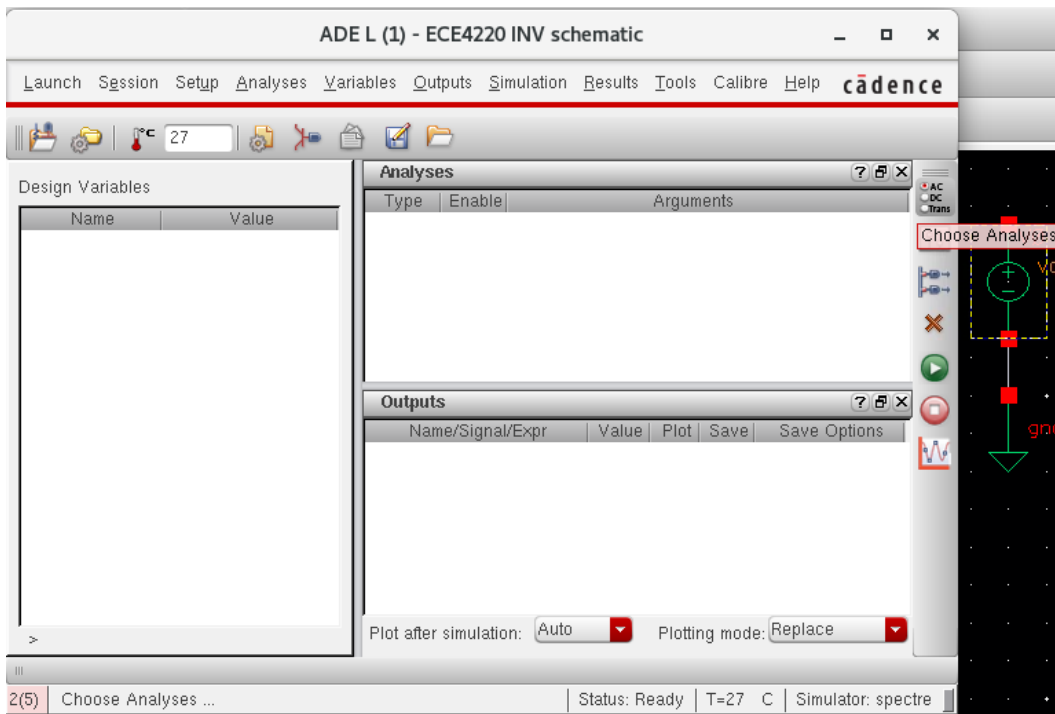
Note: Make sure you do so **every time** after you edit your schematic, otherwise errors will occur when you do simulations
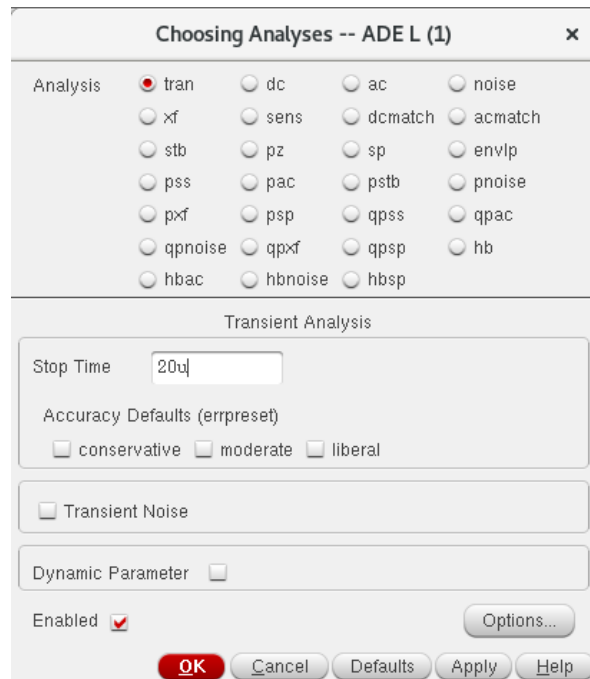
6. Run a transient simulation
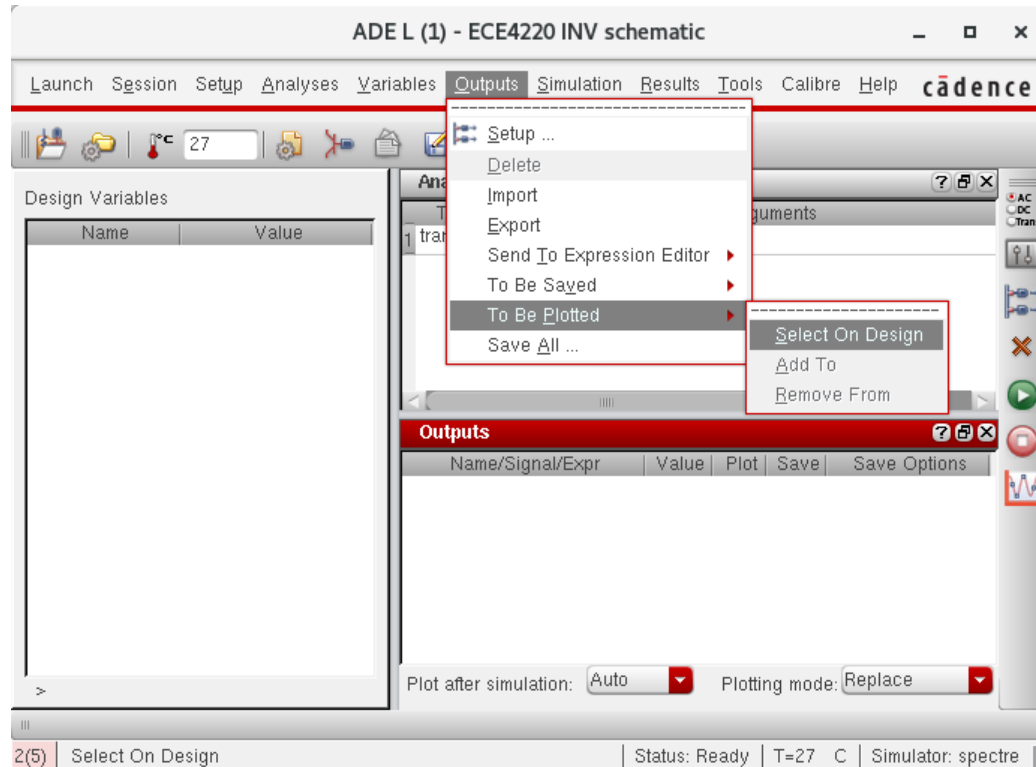   a. Click **Launch**, and then click **ADE L**

   b. In the **ADE L** window, click **Choose Analyses**

c. Set **Stop Time** to **20u**, check **Enabled**, and click **OK**

d. In the **ADE L** window, click **Outputs**, then **To Be Plotted** and then **Select On Design**

e. Click the wire (signal) where you want to plot, then the selected wires will
    blink in different colors

f. Click **Netlist and Run** in the **ADE L** window

g. The plot window will pop up automatically