Prepared by Dr. Anand Sharma (Assistant professor, ECED, MNNIT Allahabad) and Dr. Avaneesh Kumar Dubey (Faculty, ECED, MNNIT Allahabd)
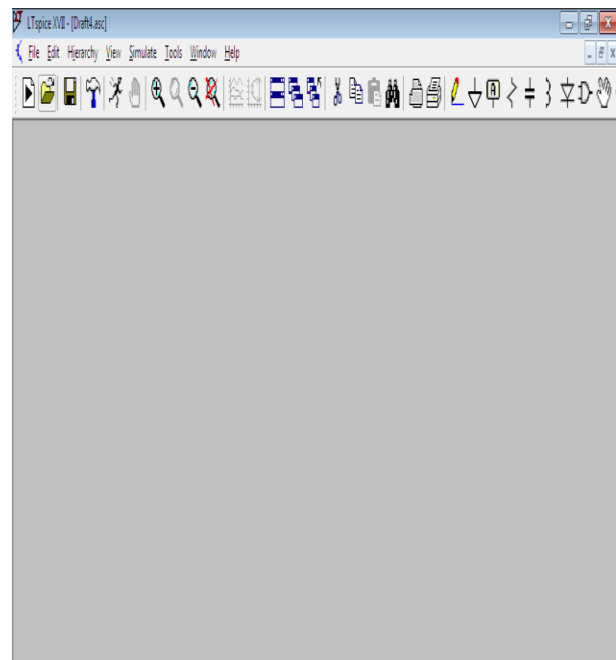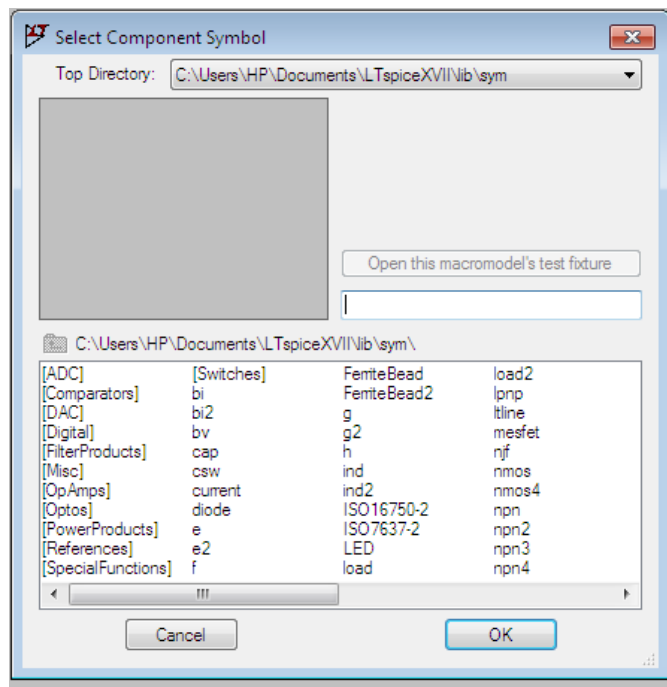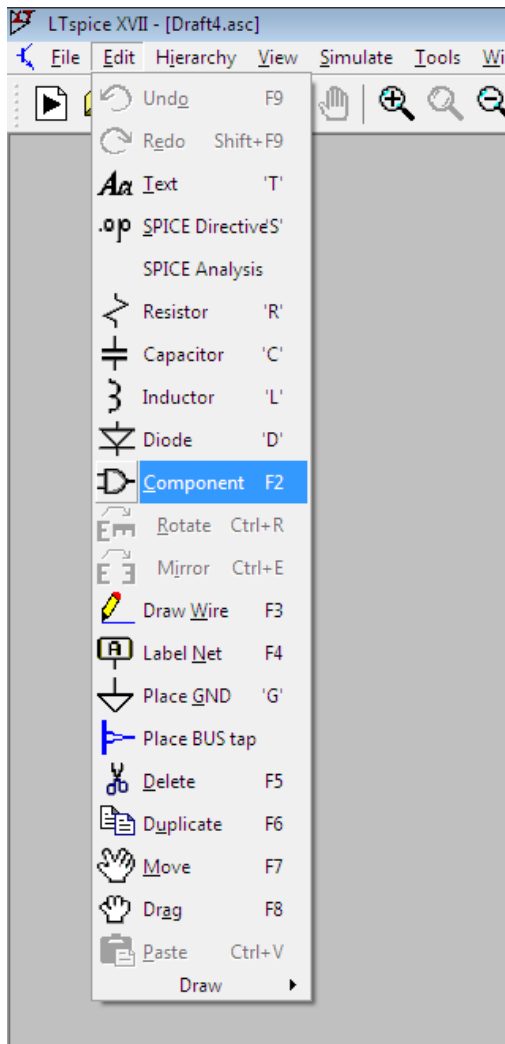
Demo LTspice

Step1: Open LTspice
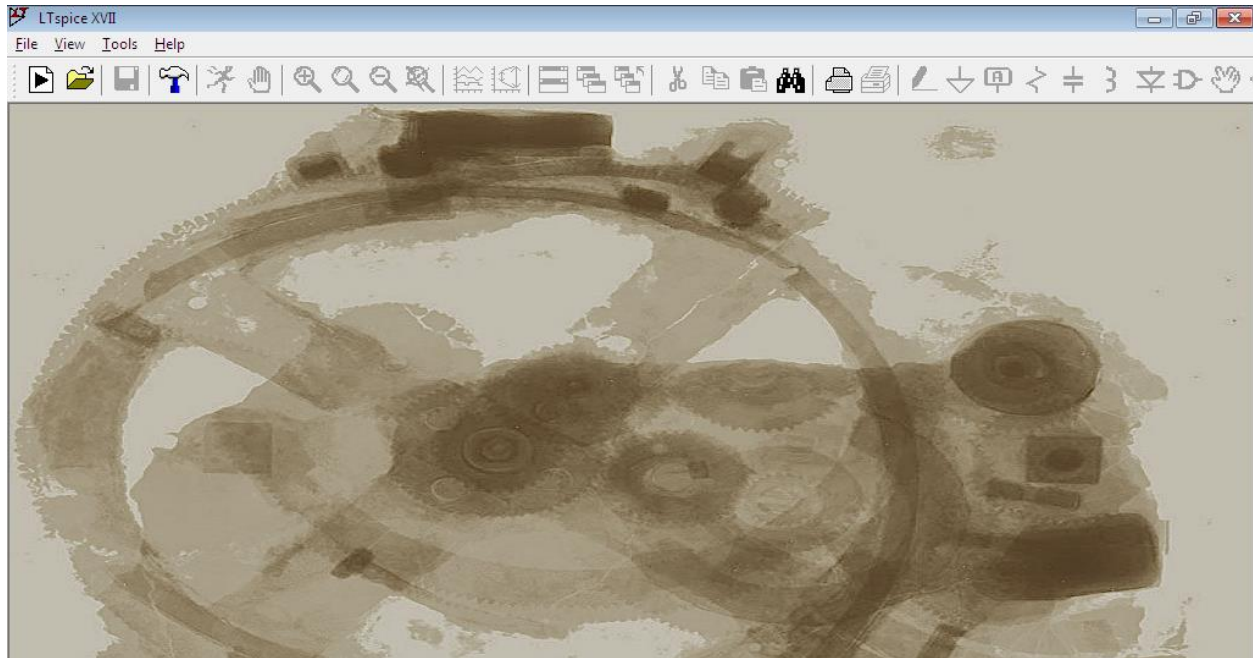
Step 2: go to file>New schematic (or click on icon in tool bar): schematic window will appear (in gray color).
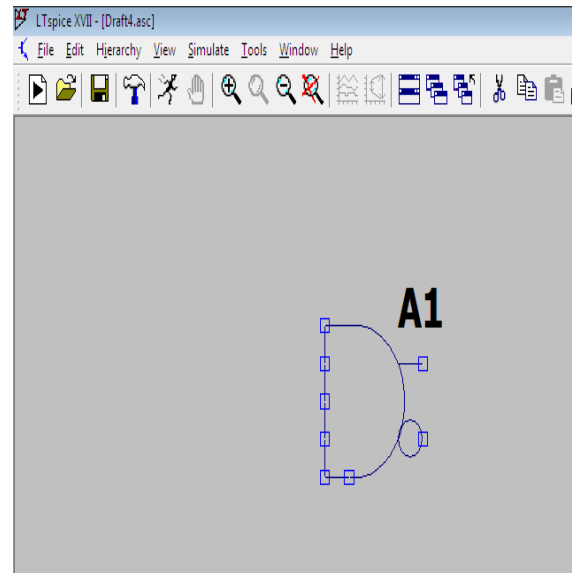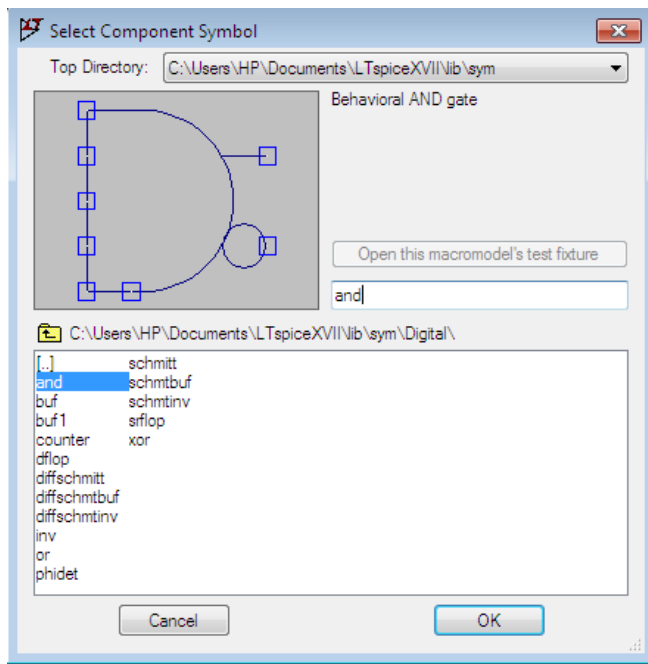
Step 3: go to edit>component (or click on ⊃⊃ icon in tool bar): new window will appear>> Select component symbol.
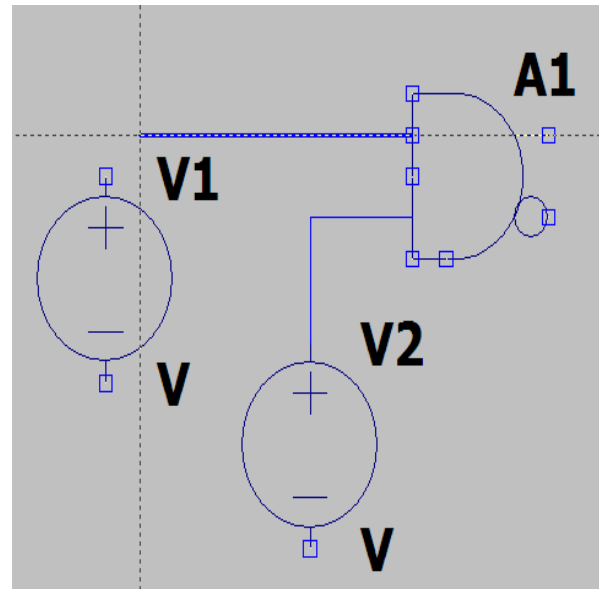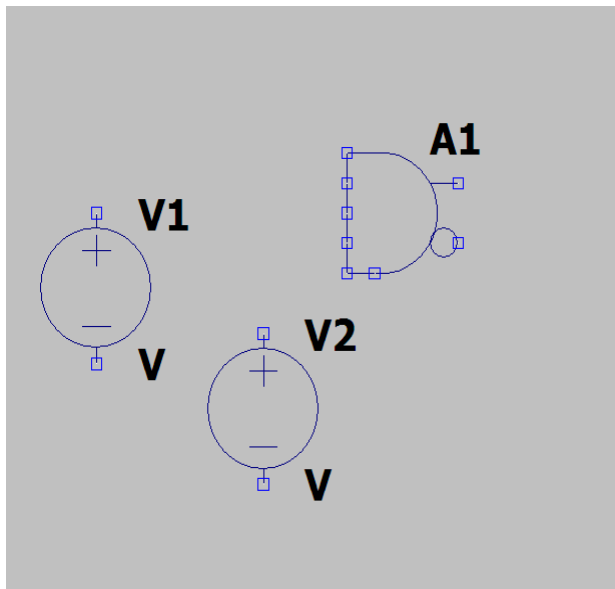

In the search box, type 'and' for and gate.

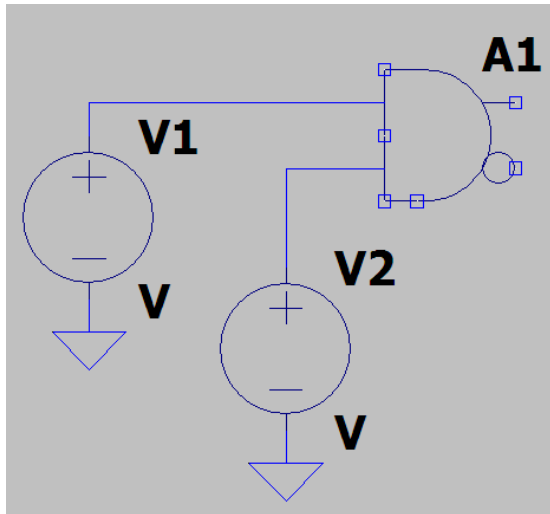Click ok, you will get 'and gate' symbol on schematic. Place it on schematic and 'esc'.

Step 4: Again click on component, get 'voltage' for inputs.
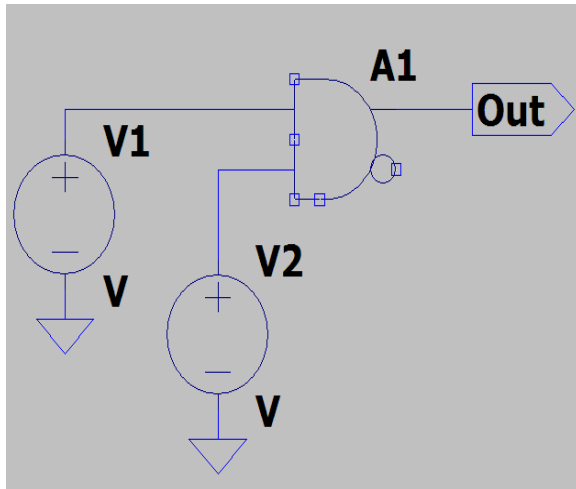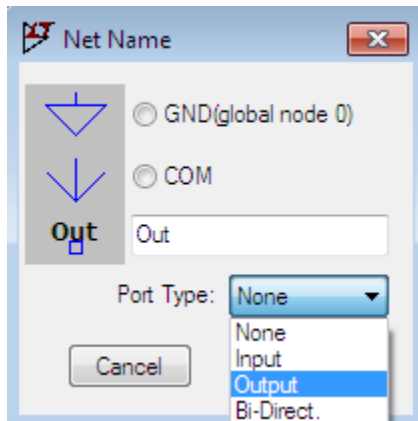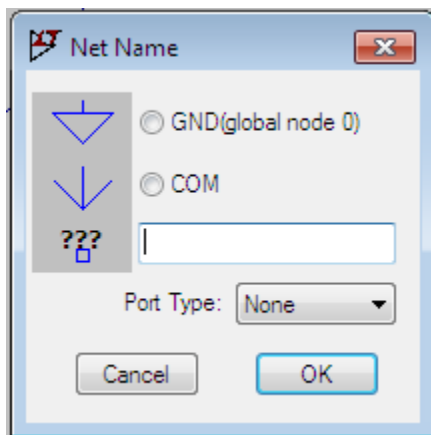

To connect with wire, click on icon, connect +ve terminal of voltage with input side.

And Connect –ve terminal of voltages with ground (click on icon).

3

Step 5: Click on icon, to get input/output port. New window will appear asking 'Net Name'. Write 'Out', select **port type** as 'output'.
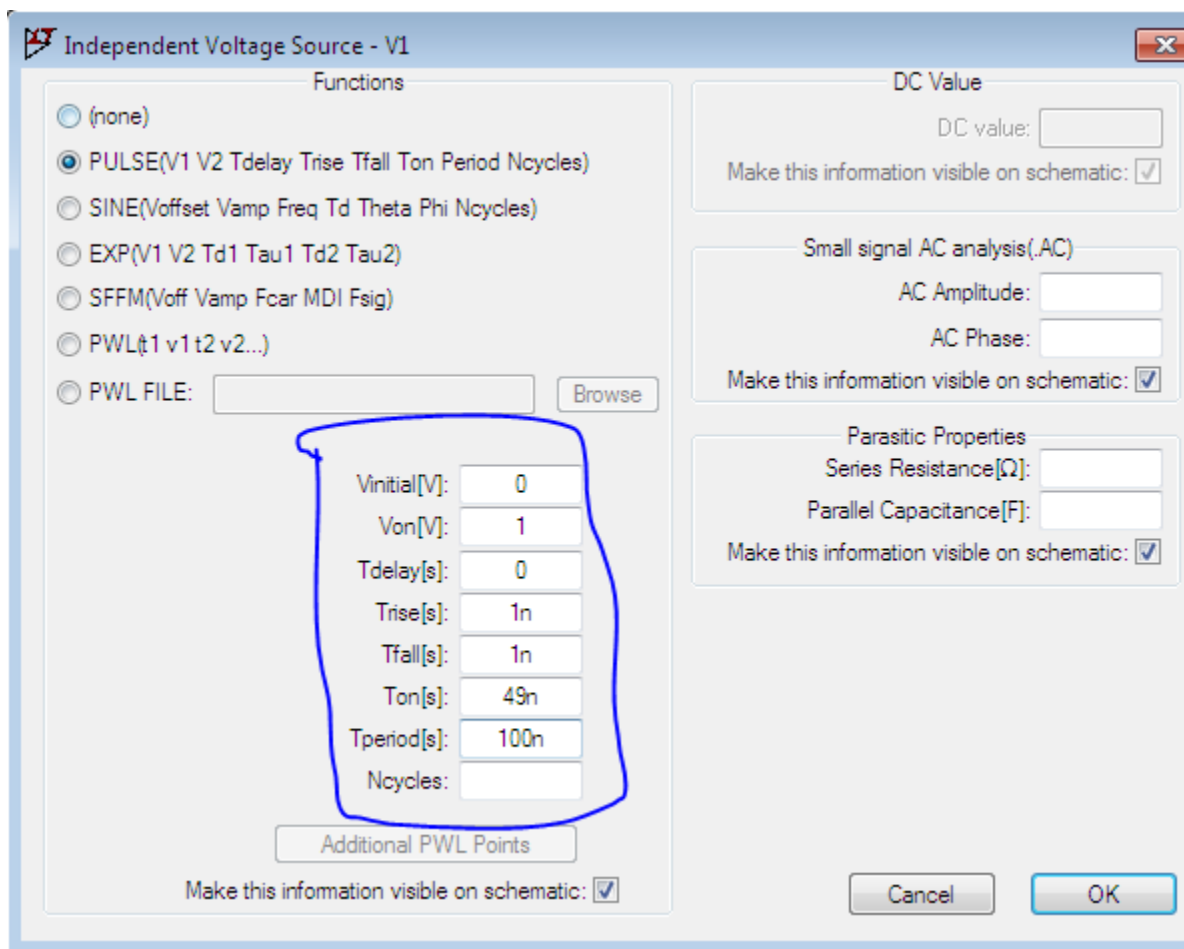

Note: To delete something from schematic, use icon.

4

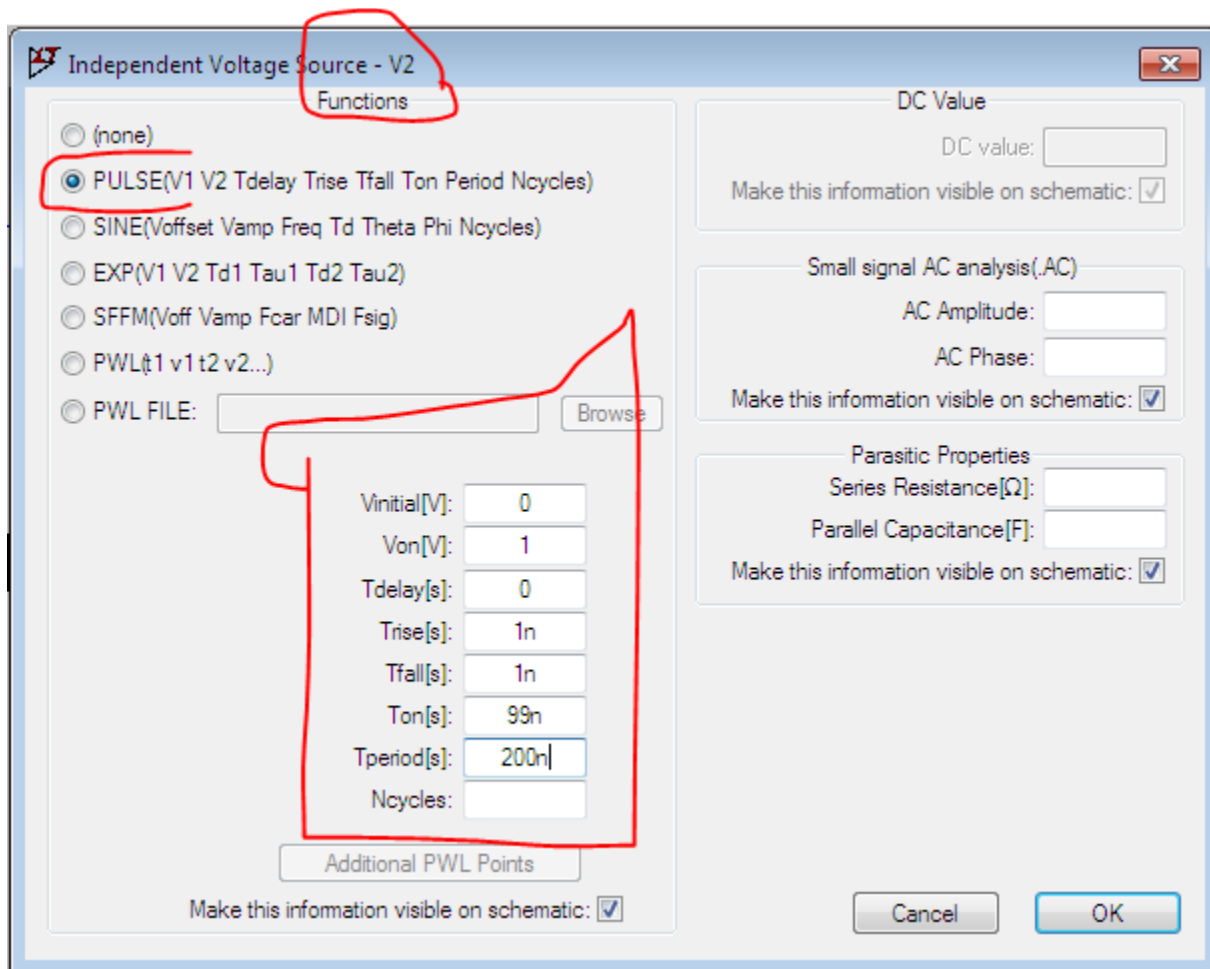**Step 6:** Now edit the properties of 'voltage' source to convert it into 'Pulse' signal.

To edit, get the cursor on voltage source "V1", and right click, new window will appear. And click on 'Advanced'.
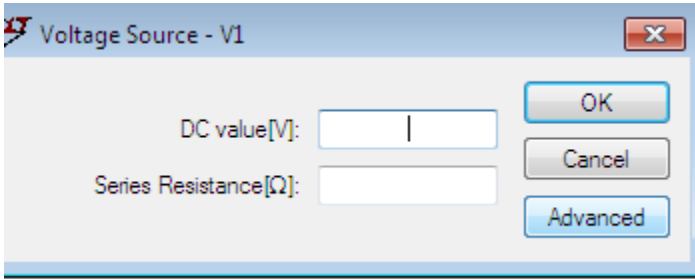
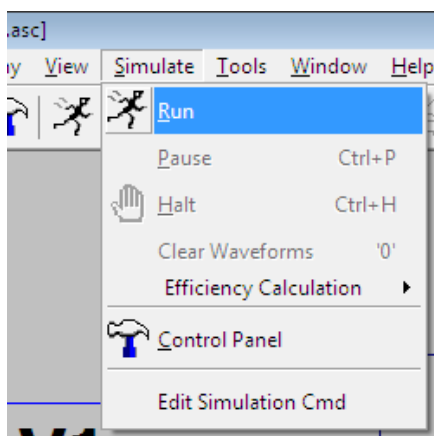Select PULSE, and fill following values to the properties. And click on 'OK'

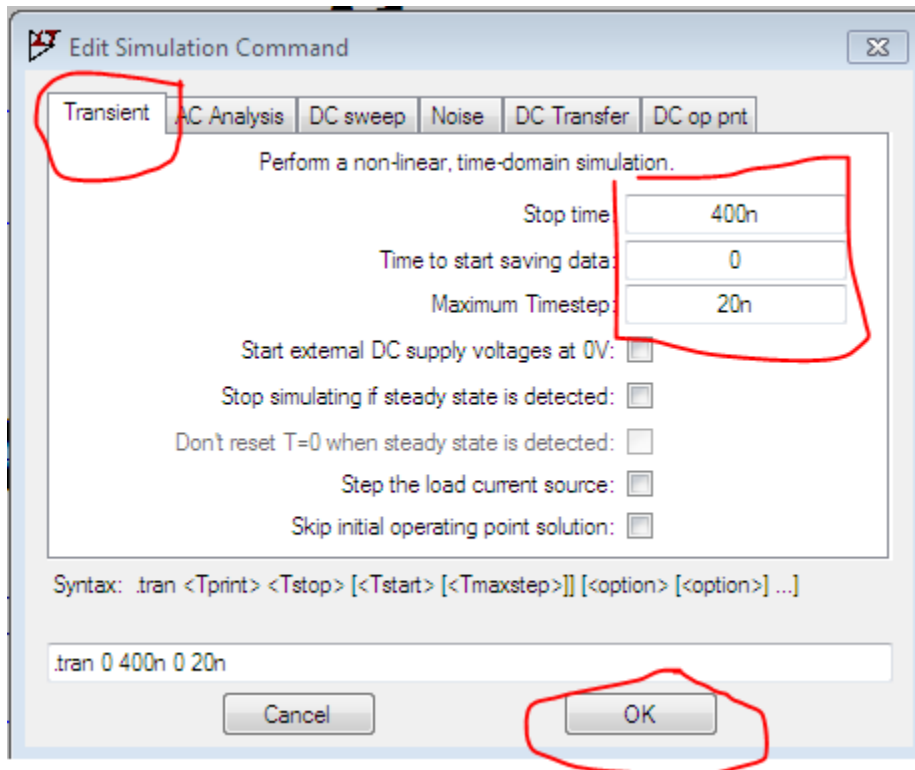Similarly, change the properties of voltage V2, with the following properties, and click OK.
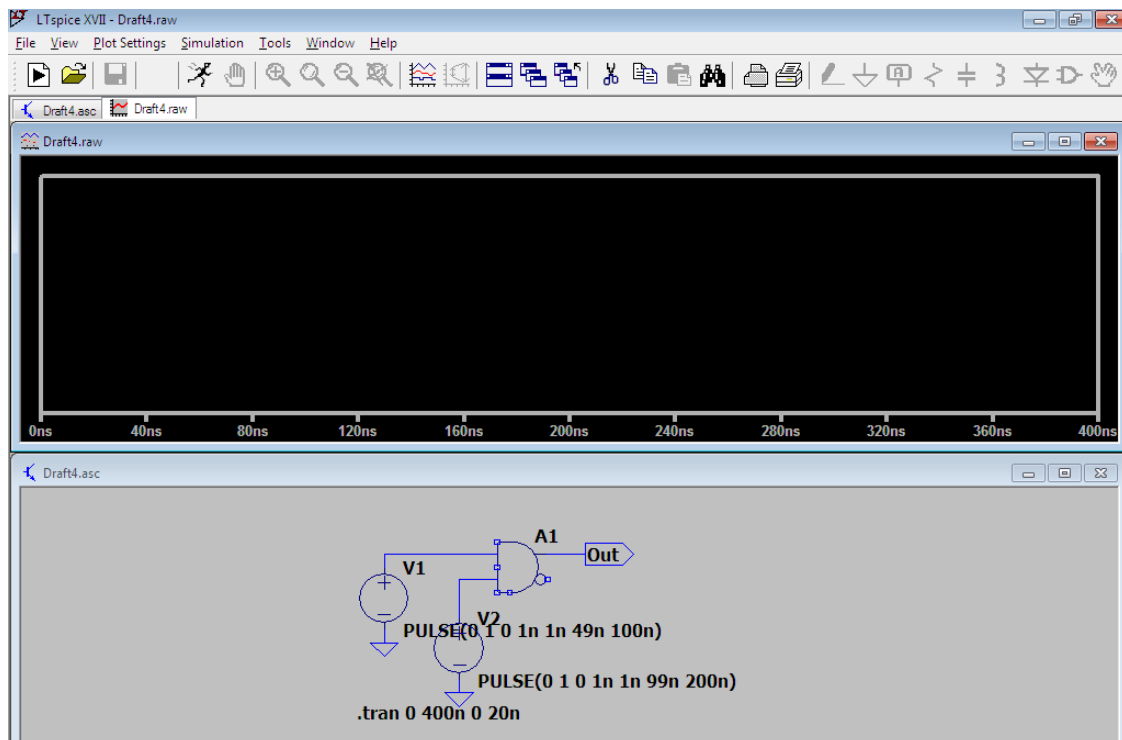

Now save ![save icon] your design.

Step 7: Go to Simulate>Run (or click on ![run icon] icon to run).

New window will appaer, asking 'Edit simulation command', choose 'transient', and fill following values as shown bellow. Click OK.
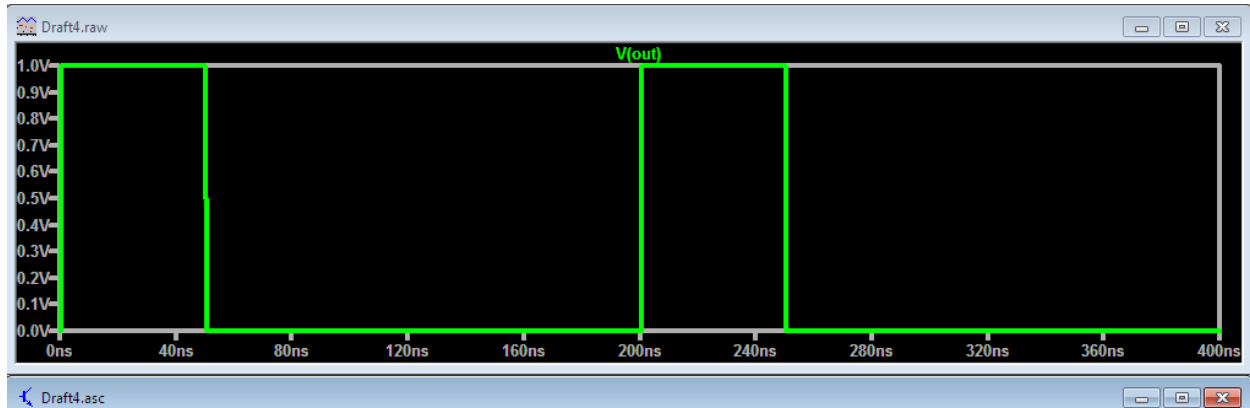

Following Windows will appear.

Step 8: Go to the schematic, keep cursor near to input and output port wires, the cursor will change in probe ![probe symbol] symbol, click on wire to get the node voltage.
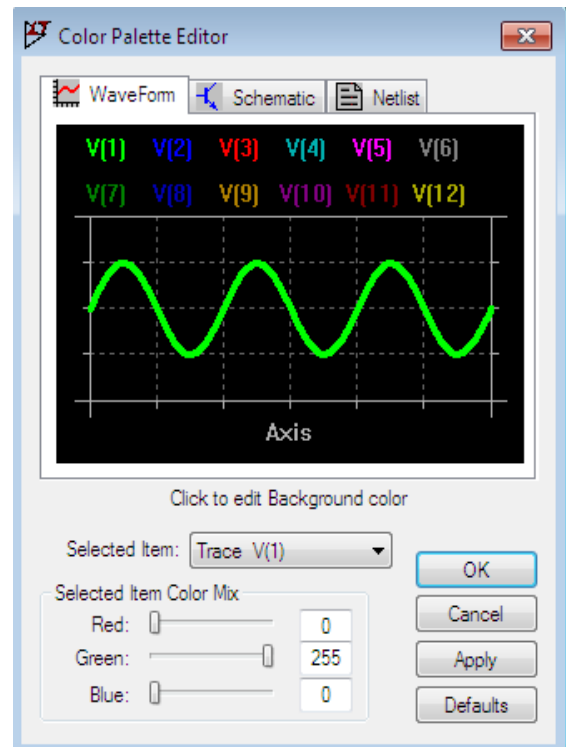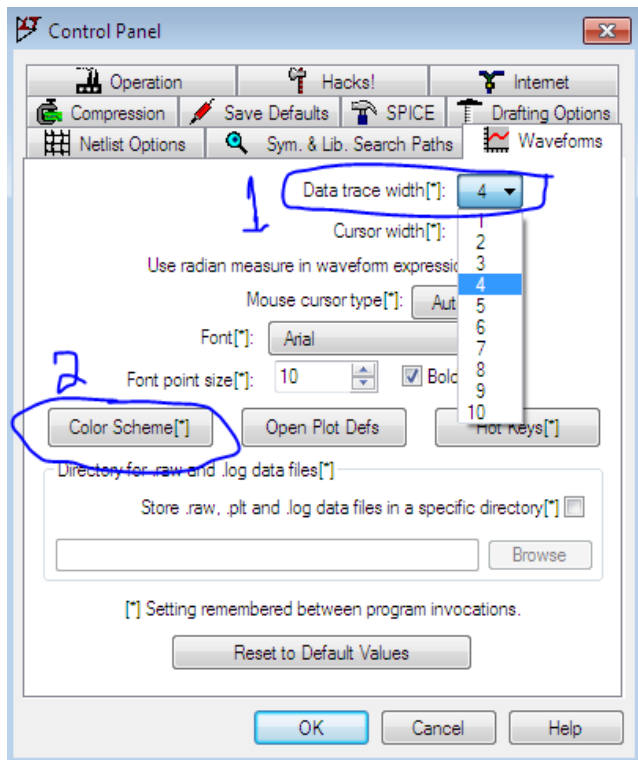
The waveforms will appear in the black window.

Step9: Changing of the background color and waveform width.

Go to the simulate>control panel (or click on ![hammer icon] icon). New window will appear. Select 'waveform'.
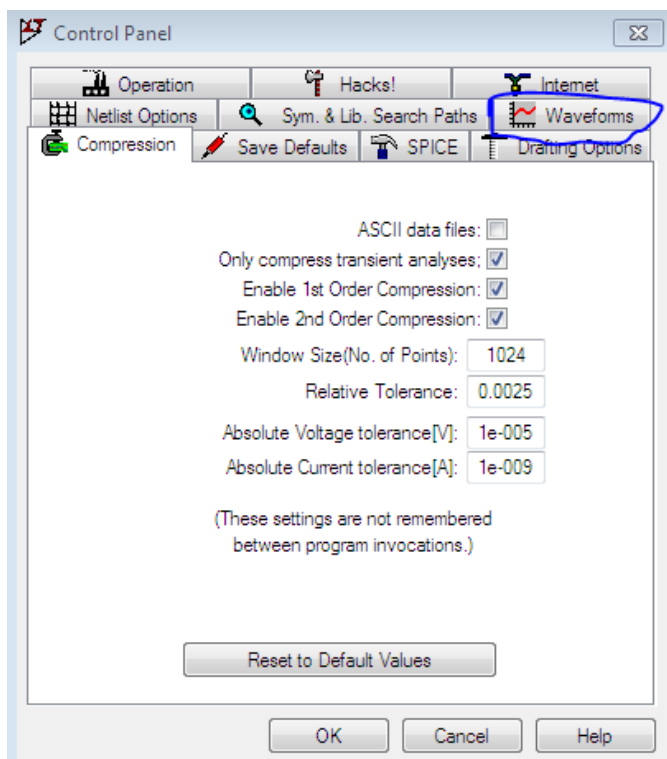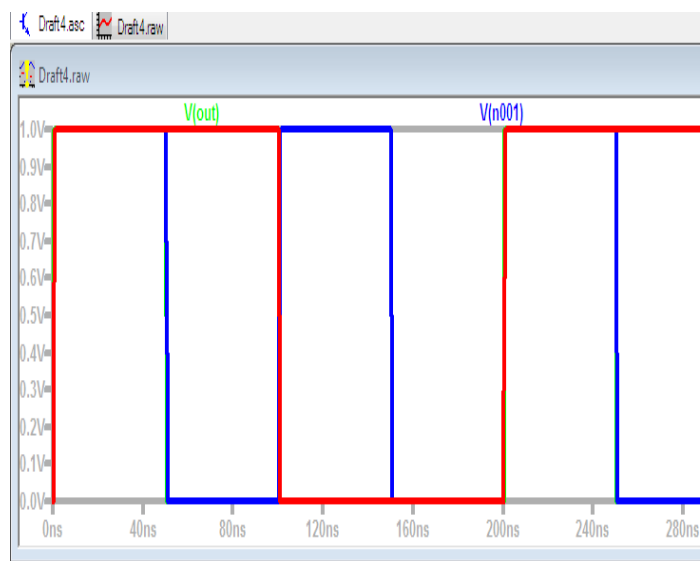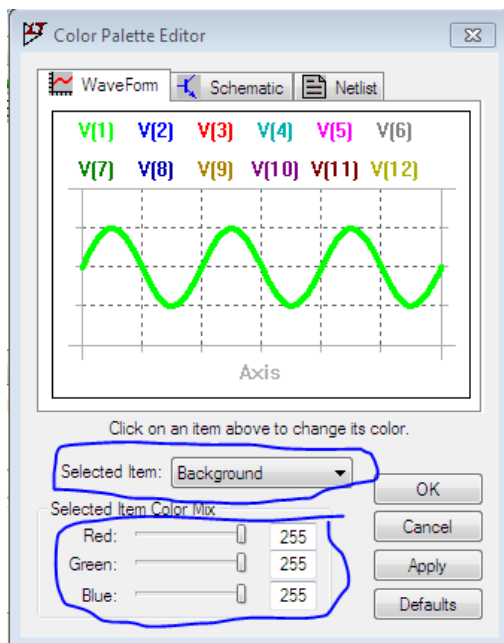
1: change 'data trace width' to 4.

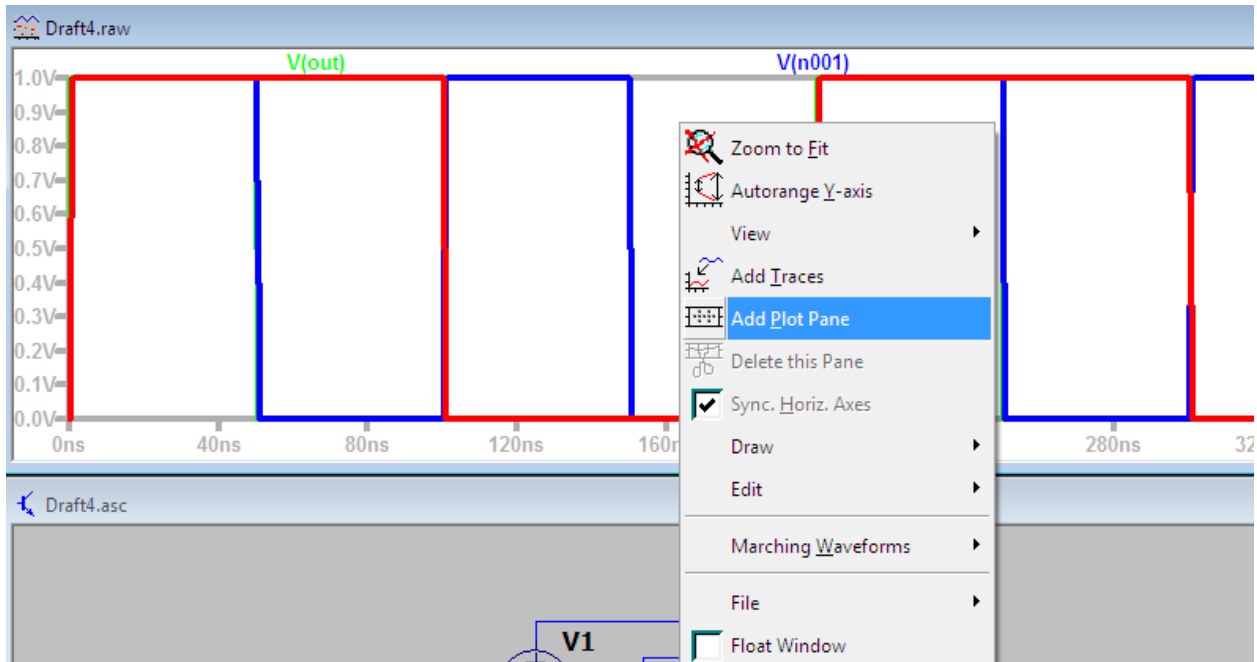2: click on 'color scheme[*]'. New window will appear as 'color palette editor'.

Change selected item as background and change selected item color mix to maximum. As shown bellow… The background will change to 'white' color. Press OK and return to waveform.
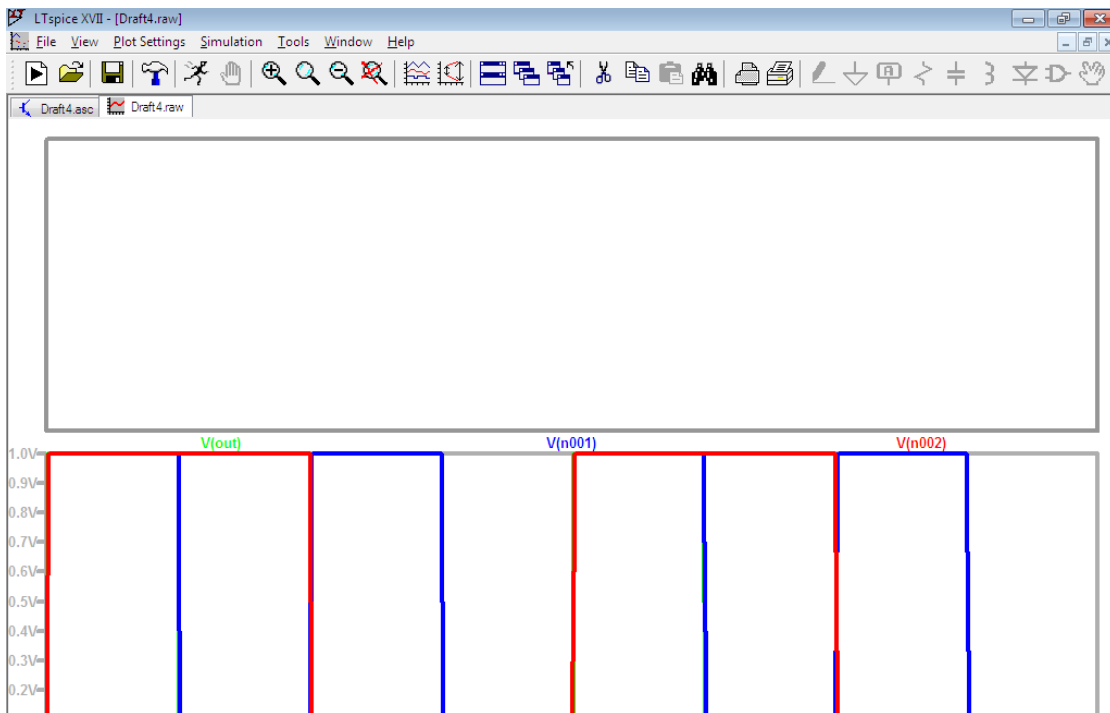
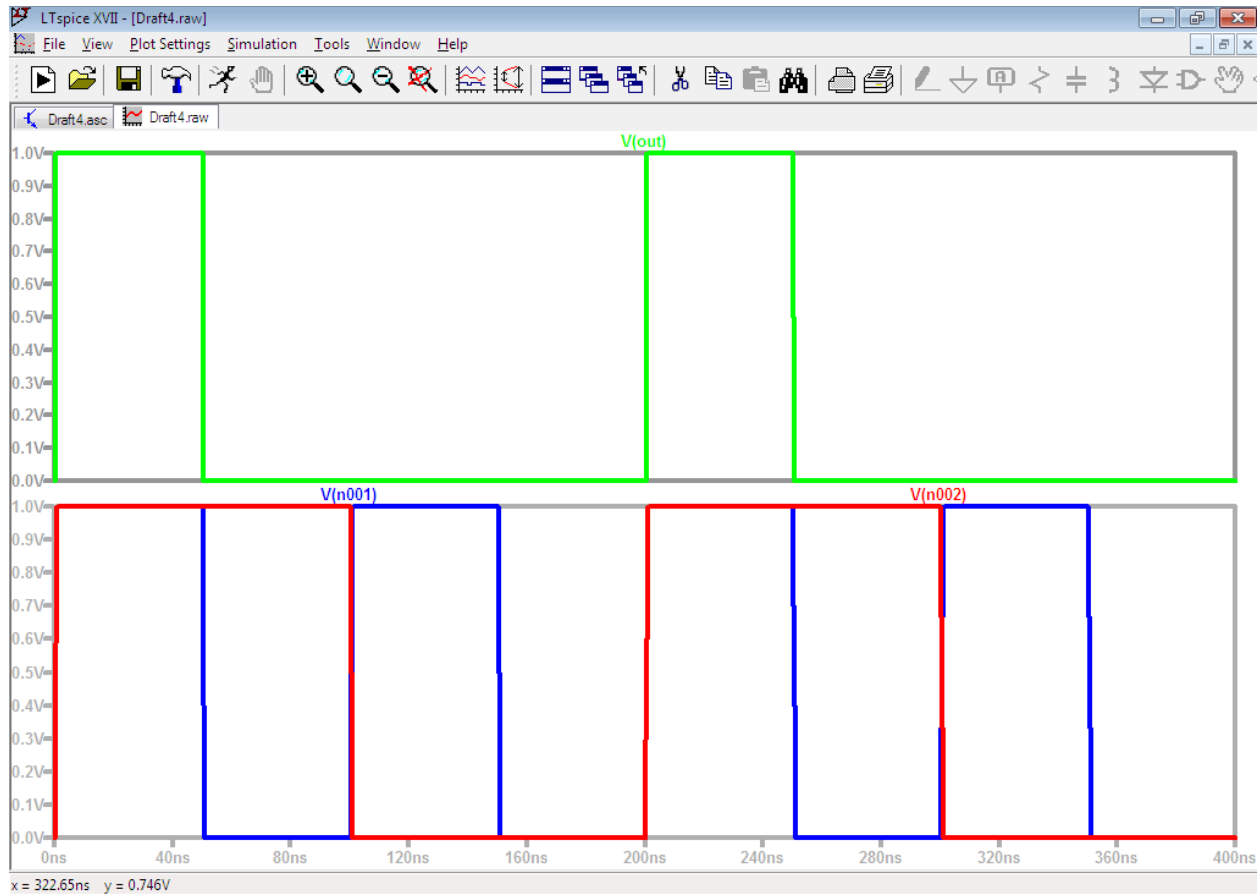**Step 10: Separating multiple waveform.**

Maximize waveform window. Right click on waveform window, and choose 'Add plot Pane'.

A blank white window will appear.

Hold any waveform (for ex. Hold V(out), green color), and take it to blank window and leave it.

By using the similar steps, you can again separate another waveform.