

## Lab 2

### Create a Symbol of an Inverter and Use It in Simulation

#### Objectives:

In this lab, you will learn

1. How to draw the schematic of an inverter.
2. How to create a symbol for the inverter from the schematic.
3. How to use the symbol of the inverter in simulation.

Using symbols instead of schematics helps the final schematic containing a lot of digital blocks look clean, meaningful and compact. You can put these symbols in your schematic page and the simulator will consider it as the schematic associated with the particular symbol. The simulation results from the schematic page consider the blocks as ideal ones, and this is called pre layout simulation.

In the next Lab you will create the layout for the inverter, which will be used to create another cell view named “extracted”. The extracted view can be used to check if the layout and the schematic matches (LVS). It can be also used for post layout simulation, where all of the parasitic elements (mainly parasitic capacitors) in the layout are taken into consideration, and the simulation results follows the actual test results more closely.

#### **List of Useful Keyboard Shortcuts:**

1. I: to add instance.
2. F: to make the schematic fit the page.
3. Esc: to cancel the current command.
4. W: to activate wire.
5. L: to name a wire.
6. M: to move an instance.
7. S: to stretch an instance.
8. C: to copy an instance.
9. R: to rotate an instance (in the schematic window).
10. Q: to change the properties of an instance.
11. P: to add port (in the schematic window).
12. Delete: to delete an instance.
13. U: to undo an action.

14. Shift+Z: to redo an action.
15. Z: to zoom in.
16. Shift+Z: to zoom out.
17. X: to view the schematic diagram associated with a symbol.
18. B: to go back to the symbol from the schematic.

### **Part 1: Create the Schematic of the inverter**

1. From the library manager, create a library named “ECE433” using “*File* → *New* → *Library*”. Attach this library to the existing tech library “AMI 0.60u C5N” (the second one in the list). Select the library ECE 433 and then create a schematic using “*File* → *New* → *cell view*” and name it “Inv”. Make sure that the tool is selected as Composer-Schematic.
2. In your Schematic window press "I" to add instance. Add one NMOS (*N\_Transistors* → *nmos4*) and one PMOS (*P\_Transistors* → *pmos4*) from the library “NCSU\_Analog\_Parts”.
3. Change the width of the transistors. For p-type transistor, set  $W = 7.5$  micron, and for n-type transistor, set  $W = 3$  micron. So, the aspect ratio for PMOS becomes  $W/L = 7.5/0.6 = 12.5$ . For the NMOS, the aspect ratio is  $W/L = 3/0.6 = 5$ . Based on the inverter schematic, you need to connect the Gate and the Drain of the NMOS and PMOS together. Press “W” to activate wire. You also need to connect the body of NMOS and PMOS to their sources. (Do not forget the connection of NMOS and PMOS body.)

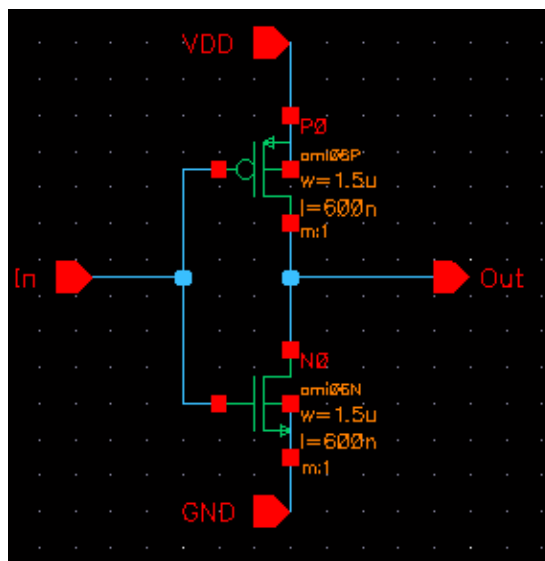


Figure 1: Schematic of the inverter with ports.

4. Ports are used to create accessibly to the nodes inside the schematic that we need after creating the symbol. To add a port press “P”. In the newly opened window, write “VDD” to be the Pin Name. Choose the direction of VDD to be input. Add “GND” and “In” port in the same way. For the pin “Out” choose the direction to be output. The final Schematic should look like fig.1.
5. Click on “save and check” button on the top of the left icons. You should have no error or warning.

### **Part 2: Create a Symbol from the Schematic**

In your Schematic window, go to “*Design → Create Cellview → from Cellview*” and click OK. A new window will open showing a box with all your ports connected to it. This is the default symbol created for your circuit. You can make it look like the familiar look of the symbol “Inverter”. You can use the drawing components that look like square or circle in the right side of the page. Final Figure should look like below: (Do not forget to save it)

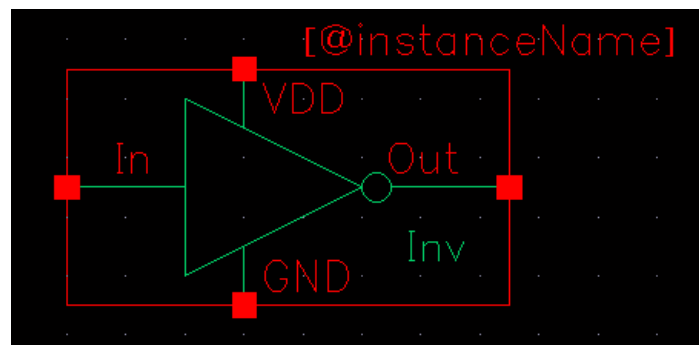


Figure 2: Symbol of the inverter

### **Part 3: Using your symbol for simulation**

1. In your Library create a new schematic and name it “test\_Inv”. In this schematic window, press "I" to insert the inverter you have just made. So, in the library part choose your own library “ECE433” and from this library choose “Inv” to see the symbol you have just made. Click on the schematic page to have your inverter there.

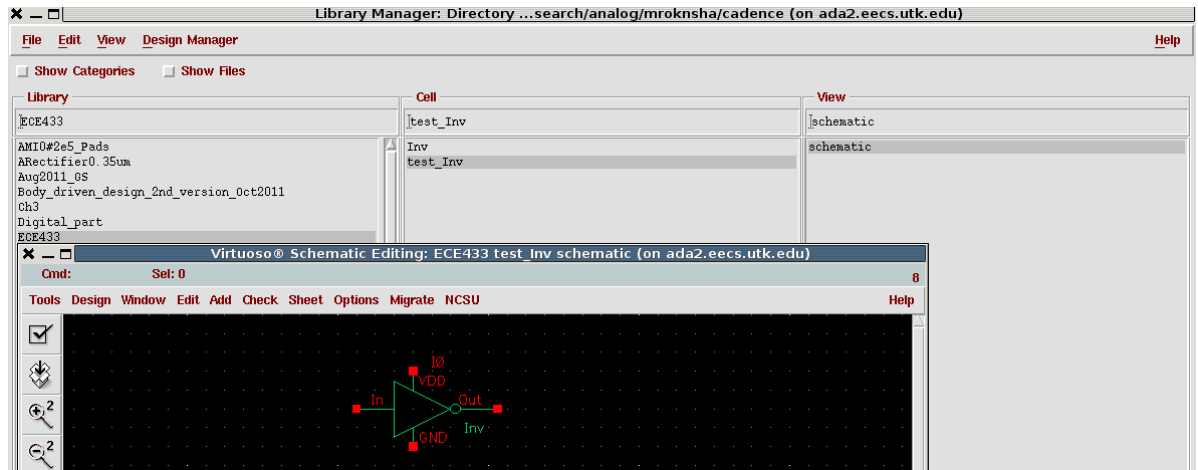


Figure 3: Selecting inverter symbol for simulation

To see the schematic of the inverter you designed, click on the inverter symbol and then press “X” and click OK. To get back to the “test\_Inv” schematic window press “B”.

2. To test the behavior of this circuit, we need a supply voltage, a pulse generator and a ground for the reference of our simulation. To get these parts, press "I" and select the “NCSU\_Analog\_Parts” Library. In this library click on the “Voltage\_Sources” and select “Vdc”. Choose 5V for the DC voltage. From the same library choose Voltage\_Sources → Vpulse and choose the parameters from figure below:

Voltage 1	0
Voltage 2	5
Rise time	100p
Fall time	100p
Pulse Width	5n
Period	10n
DC Voltage	0
Number of noise/freq pairs	0

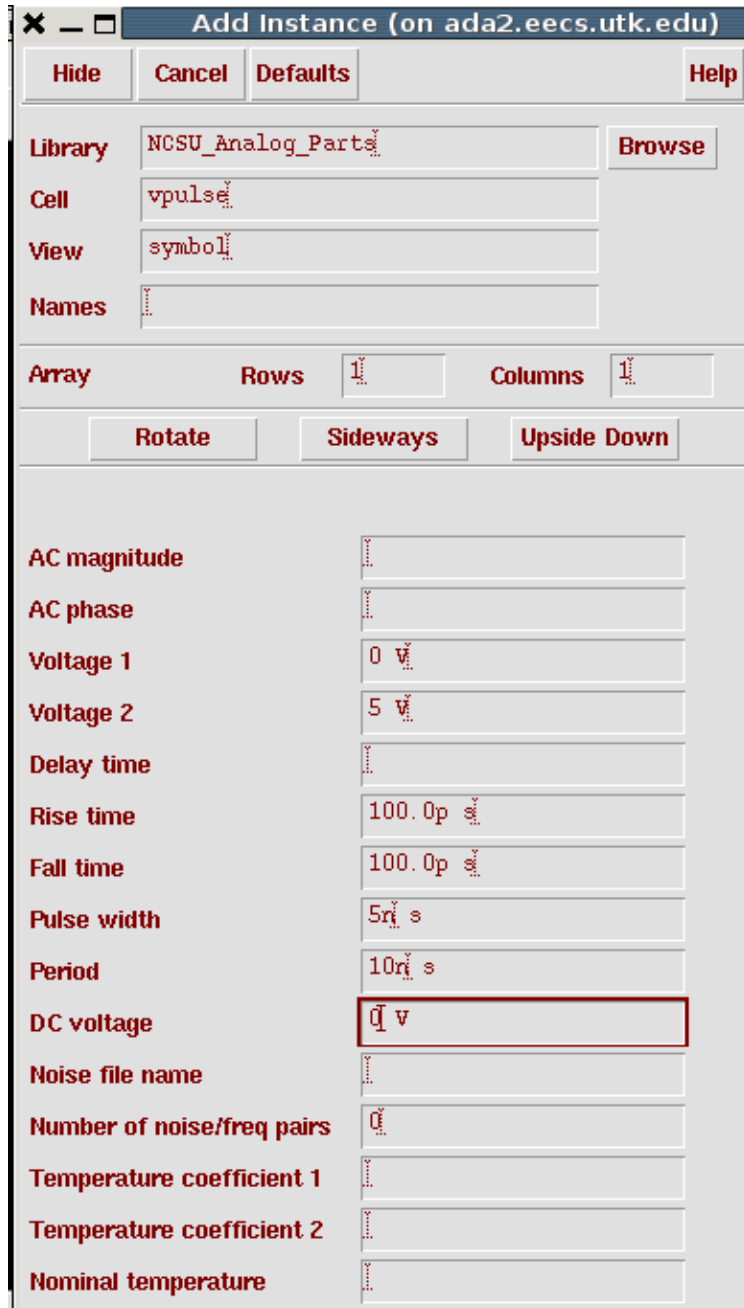


Figure 4: Parameters for the pulse generator.

3. To define the nets VDD and GND in the “NCSU\_Analog\_Parts” Library choose “Supply\_Nets” → vdd. Add this symbol on top of the inverter and the DC voltage. Then from this library choose “gnd” and connect it to all of the grounds in your schematic. Click “ESC” to close the window. You can press “W” to start wiring. To name the wires, press “L” and write the name of wire. Your final figure should look like below: (Do not forget to save it)

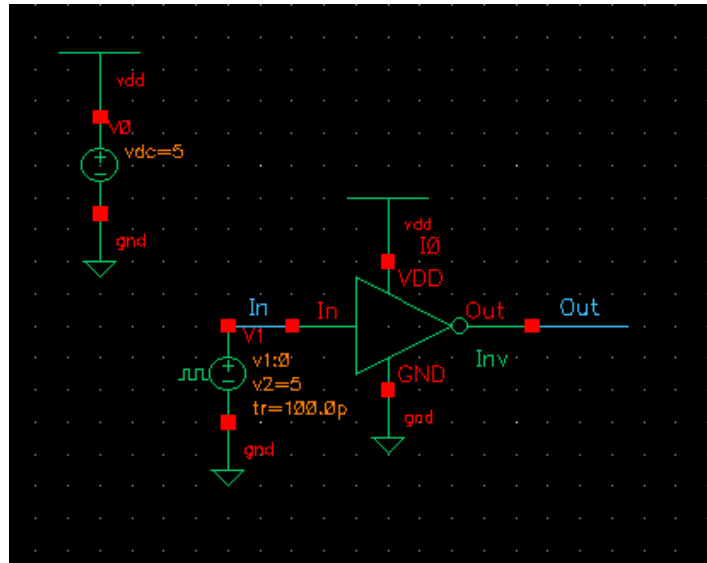


Figure 5: Final test circuit ready for simulation

#### Part 4: Simulation

1. To start Simulation choose “*Tools → Analog Environment*”. In the newly opened window click on “*Setup → Simulator/Directory/Host*”. Choose “spectre” as your simulator.

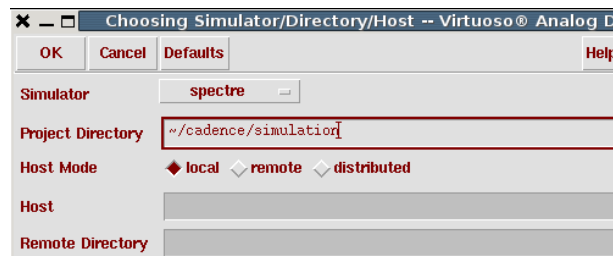


Figure 6: Selecting the simulator

2. Now you need to add model libraries. Click on “*Setup → Model Libraries*”, then “Browse” to find “ami06N.m” and “ami06P.m”. These two files are the ones that you copied to the folder “model” in the first lab. When you click “browse” the “cadence” folder opens, therefore you need to go one folder back to your home folder (the first option in the list is going one folder back) then you can find the folder you created in lab 1 named “model” and then click on “ami0N.m” and then click “OK” and then in the “model library” page click “Add”. To add “ami06P.m” click “Browse” again and add “ami06P.m” library in the same way and click “OK”.

- Click “*Analysis → Choose*” and select the parameters as follows:

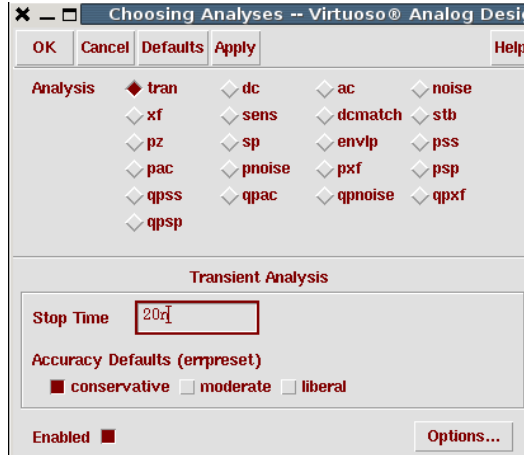


Figure 7: Choosing analysis parameters

- To choose the nodes of which you want to see the response click “*Output → To Be Plotted → select on Schematic*” and then click on the wire input and output (nodes In and Out). Your Simulator window should look like below: (pay attention that if you click on nodes it will draw a circle around that node showing that you want to plot the current passing that node. But in this lab you need to plot the voltages and therefore you need to click on wires and not nodes).

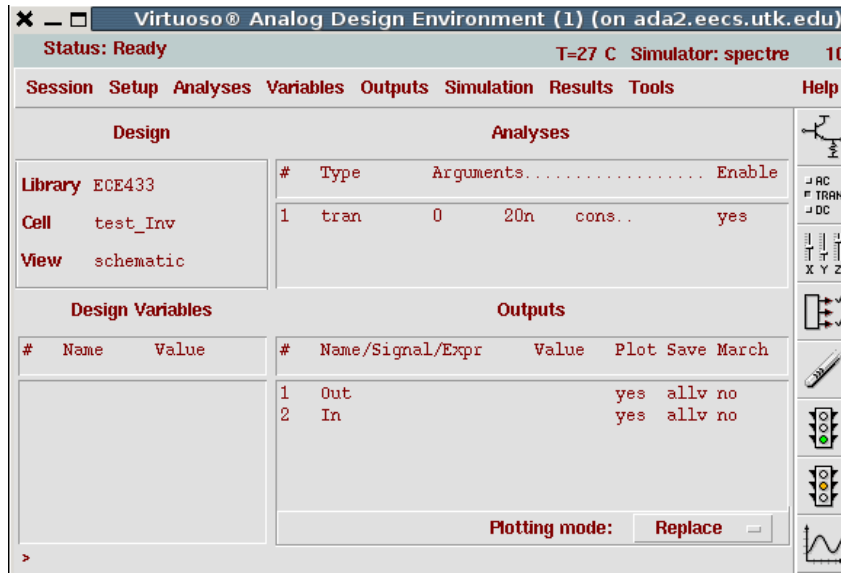


Figure 8: Selecting outputs to be plotted

5. Now Click “*Simulation* → *Netlist and Run*”. You should see the input and output in one window like below, you can also change the color of the background and plots by double clicking on them.

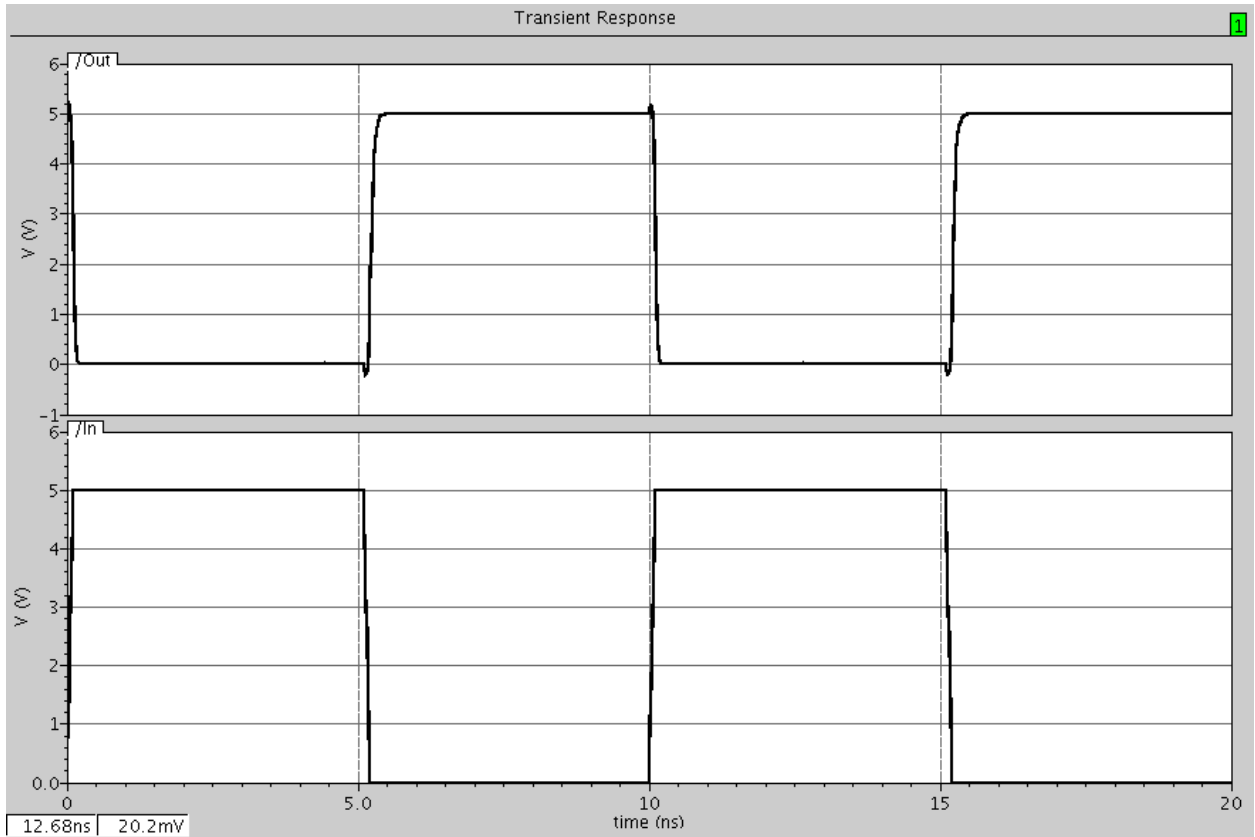


Figure 9: Input and output of the simulated inverter.