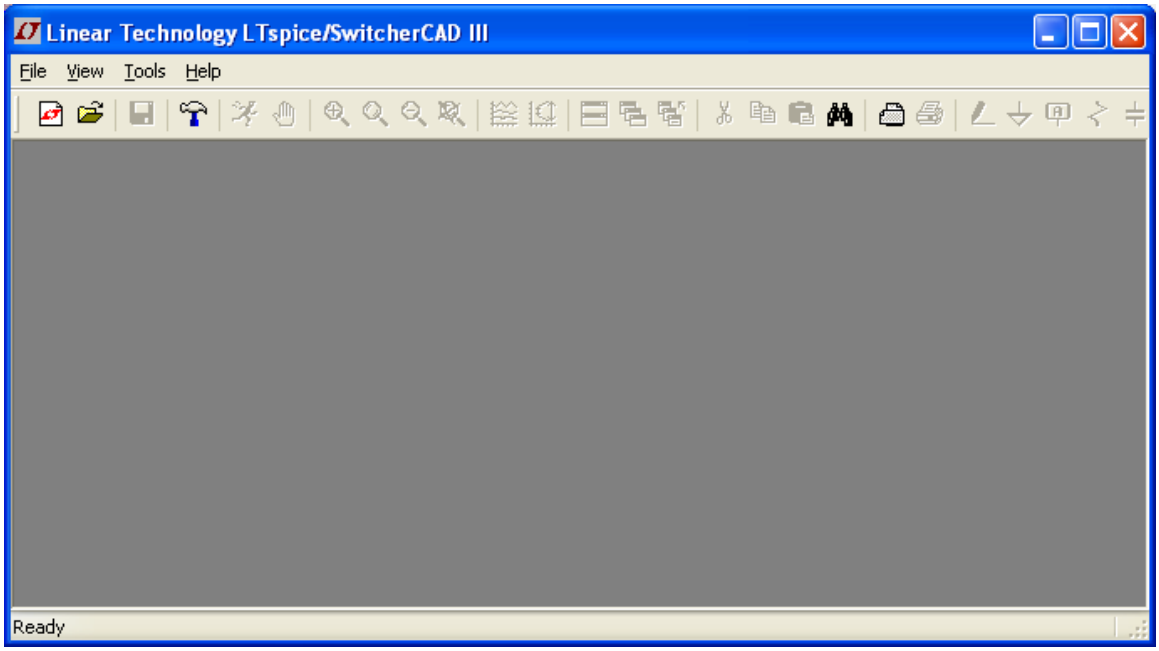
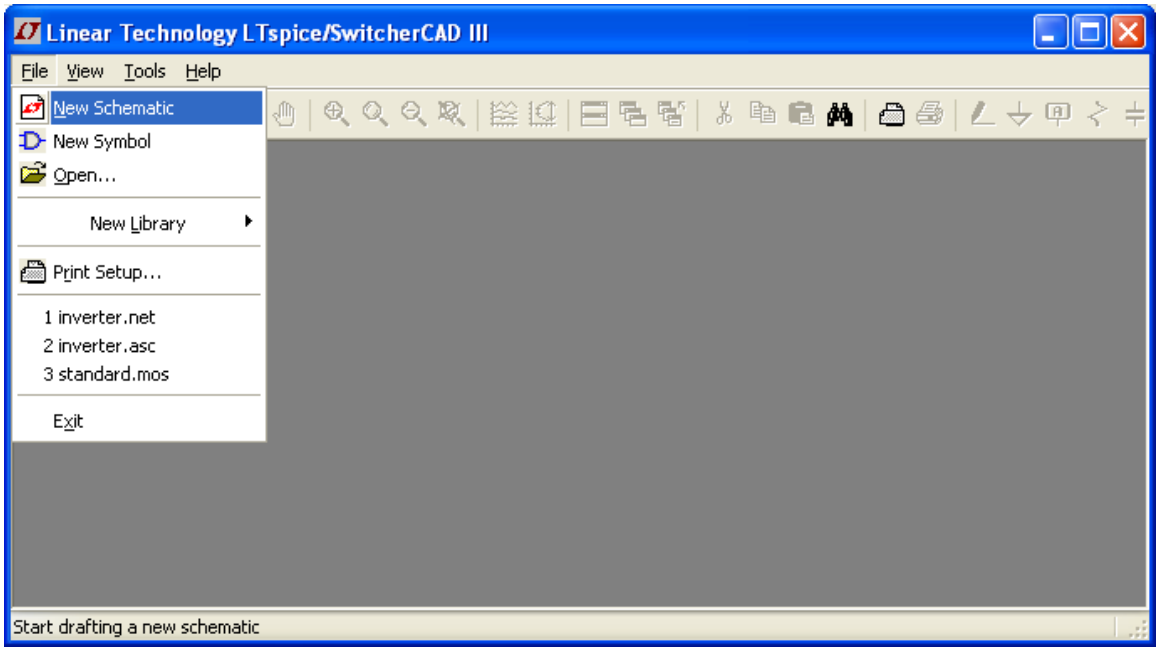
LTspice QuickStart

CMOS inverter schematic and simulation using LTspice

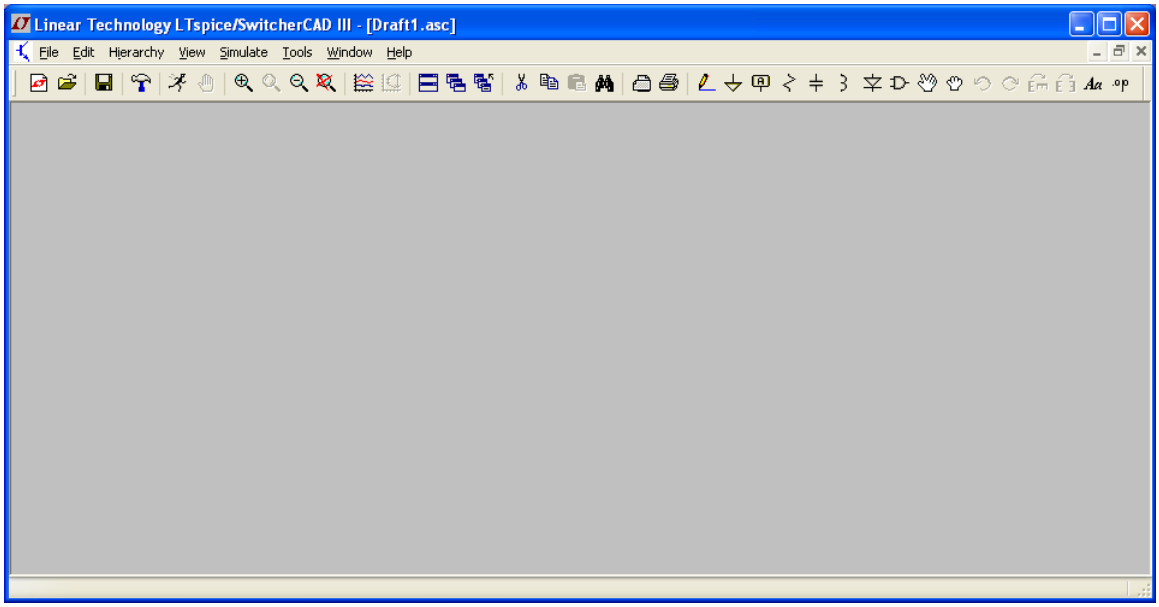
1. start LTspice either through the GUI (double click on the SWCAD III icon) or using the following command: `wine` *whateverpath_to*`/scad3.exe`



2. Open: File > New Schematic

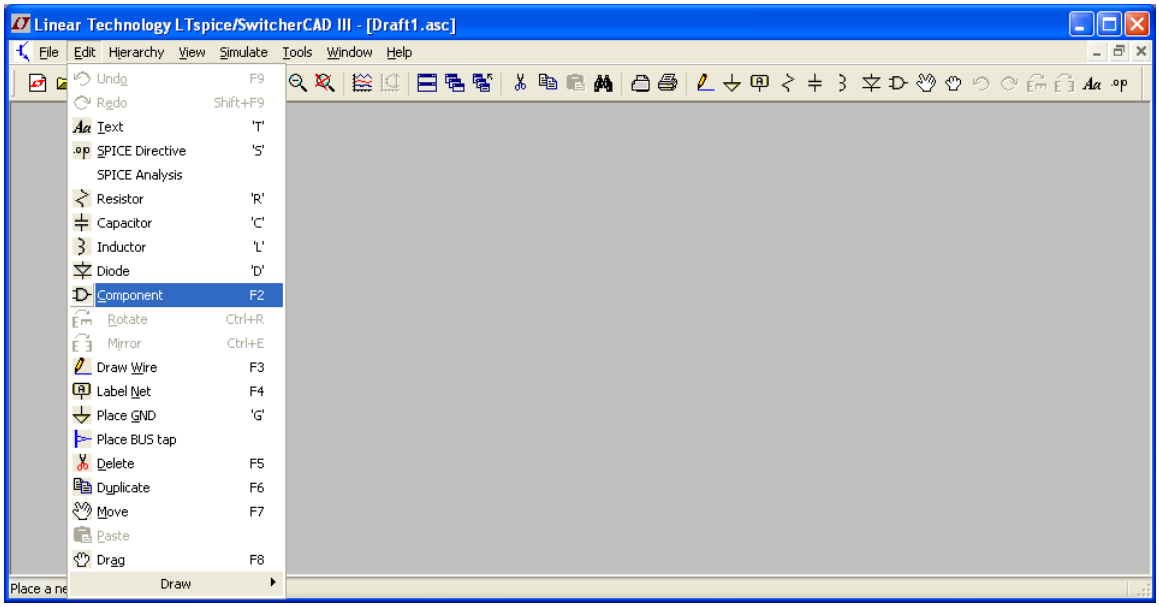


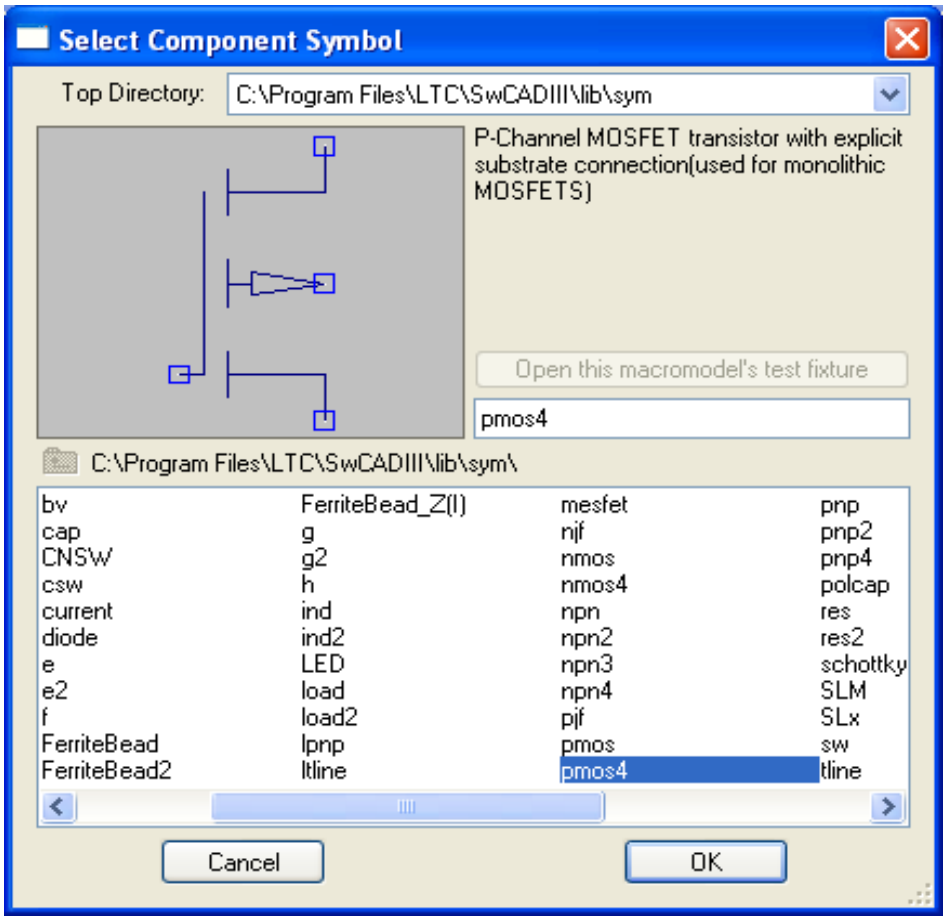
A new schematic file (Draft1.asc} will open for you to edit.



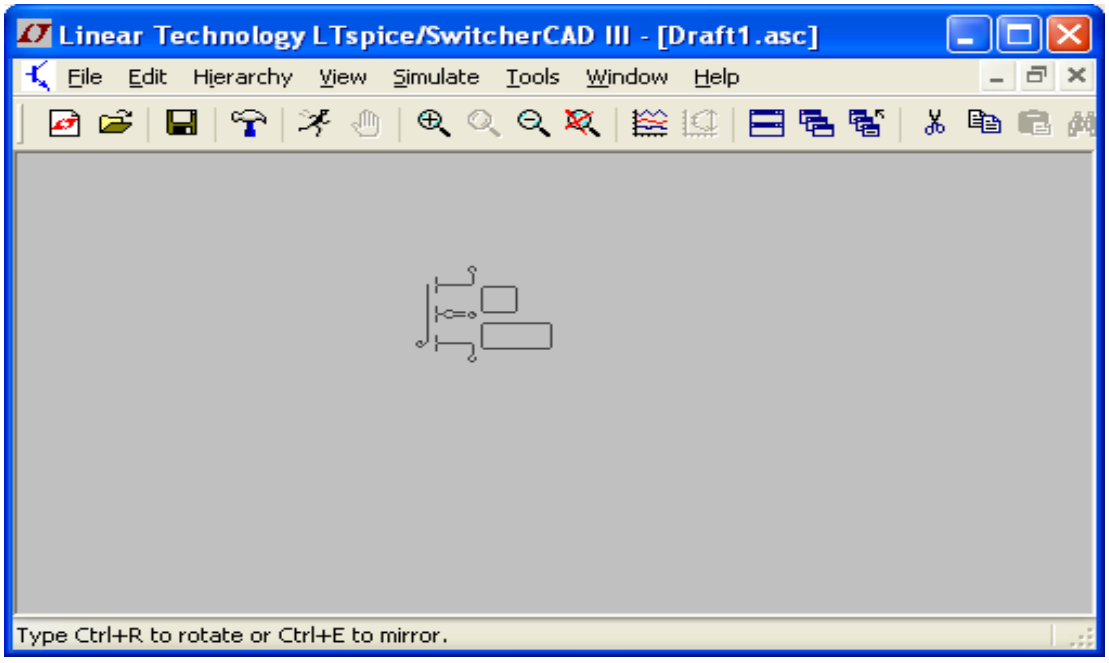
3. Select, place, edit and connect together the necessary circuit elements.

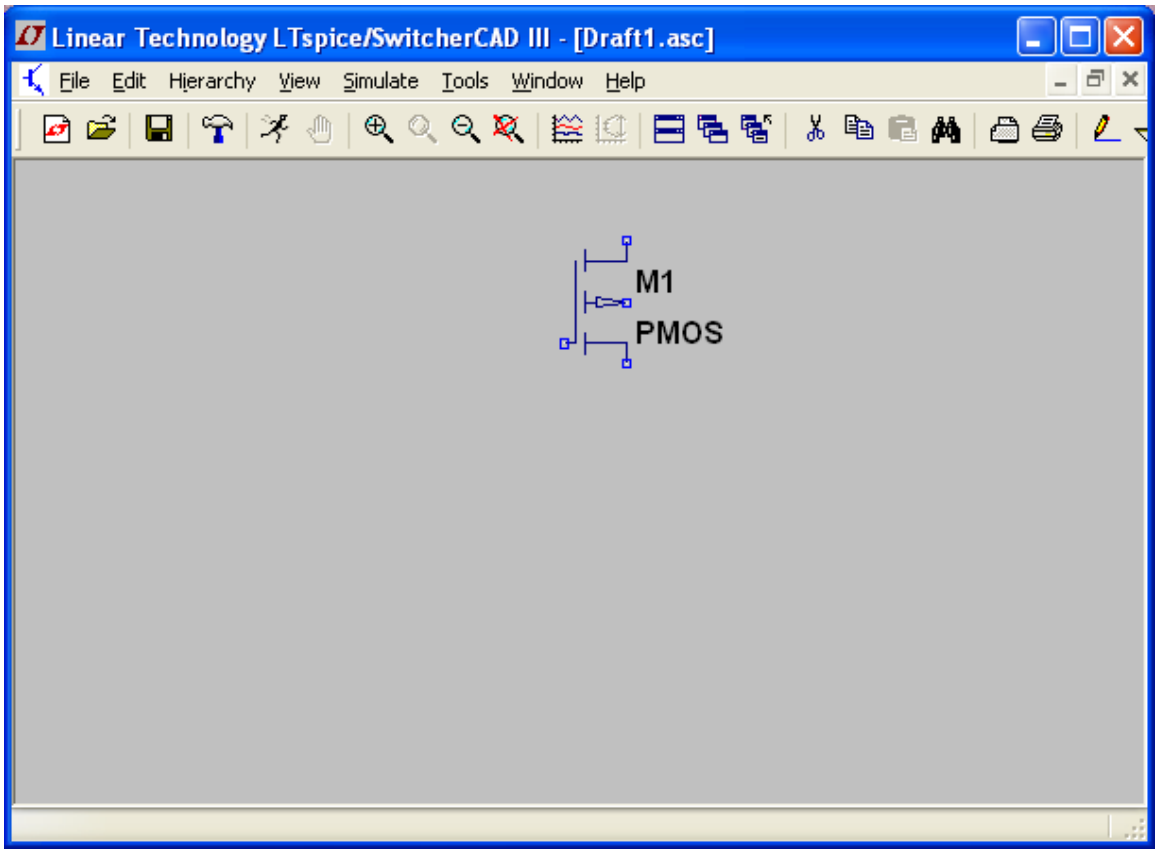
Select the PMOS device:



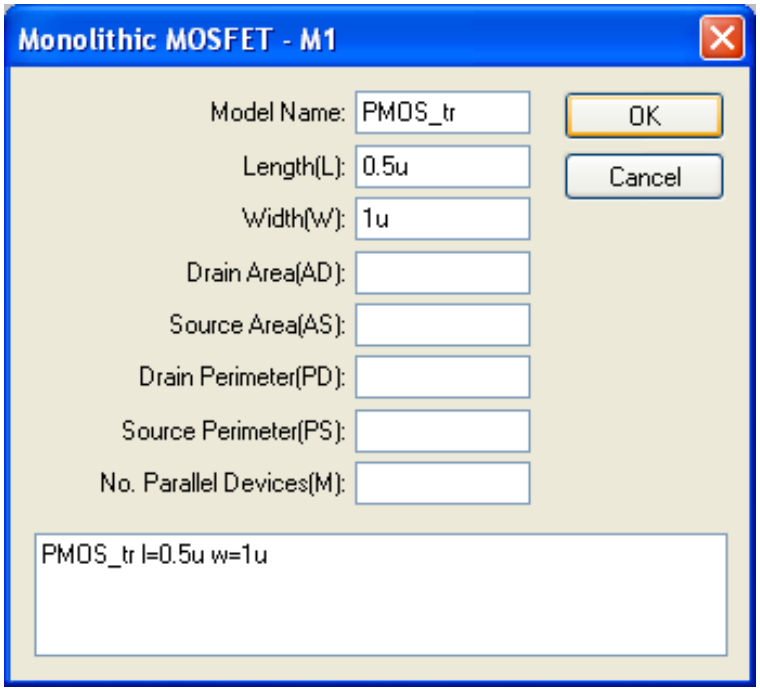


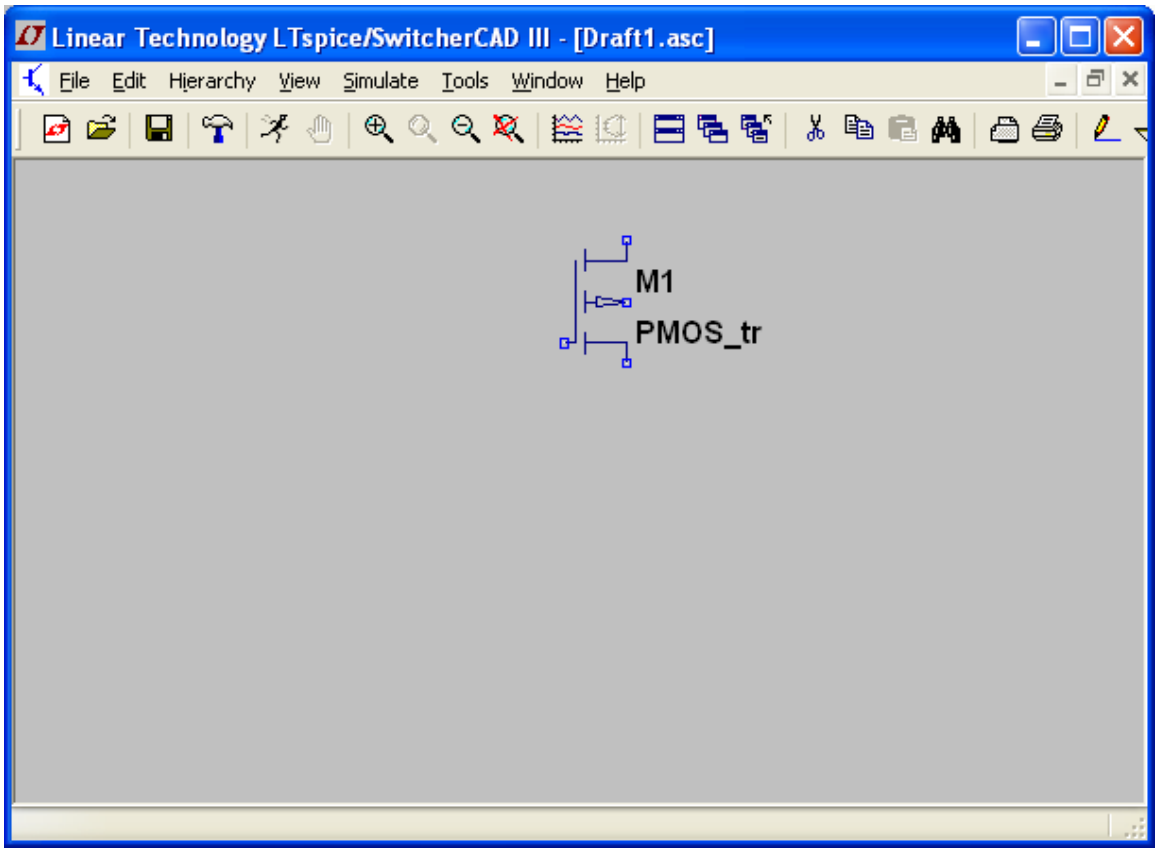
Click OK to confirm the selection. Drag the mouse to the position where you want to place the device and then left click.



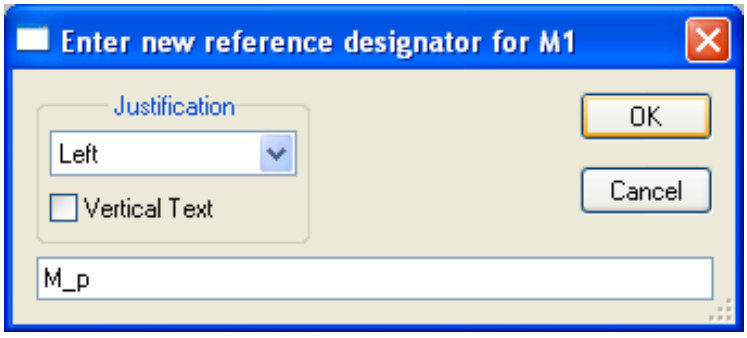


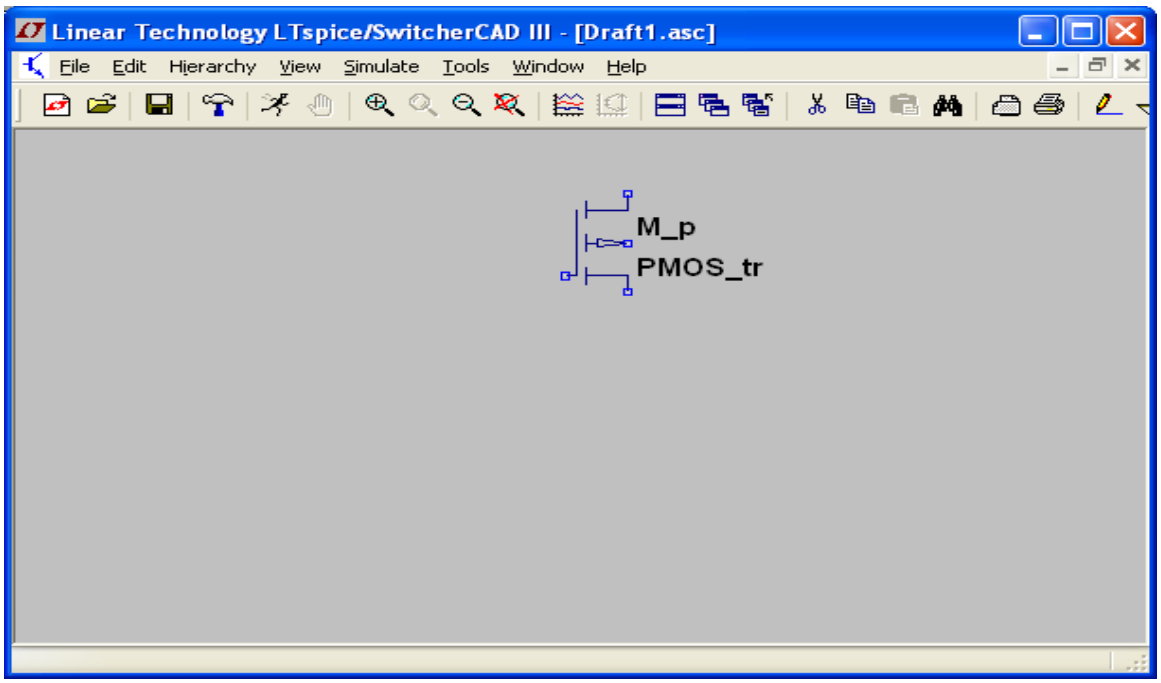
Type ESC and then right click on the symbol to edit the device parameters.



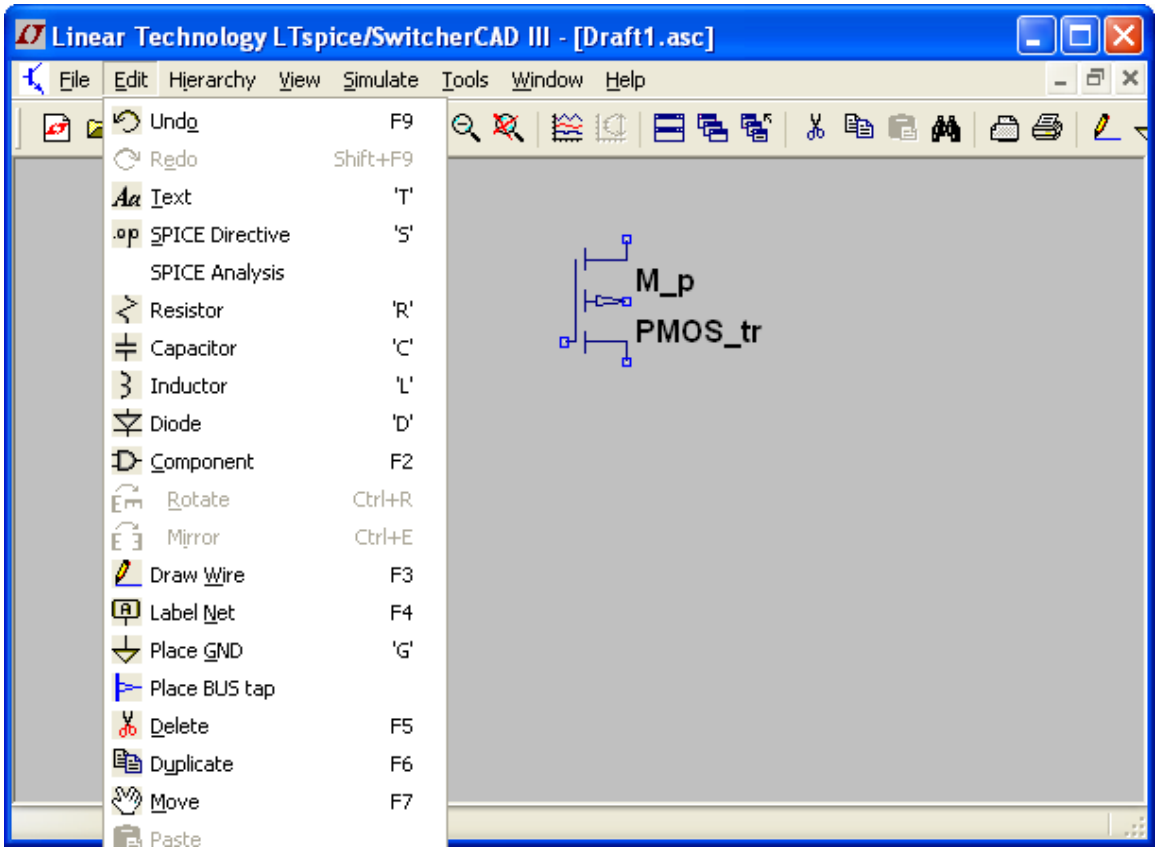


If you want to change the name (M1) of the transistor or the location of the name, right click on the name of the transistor.

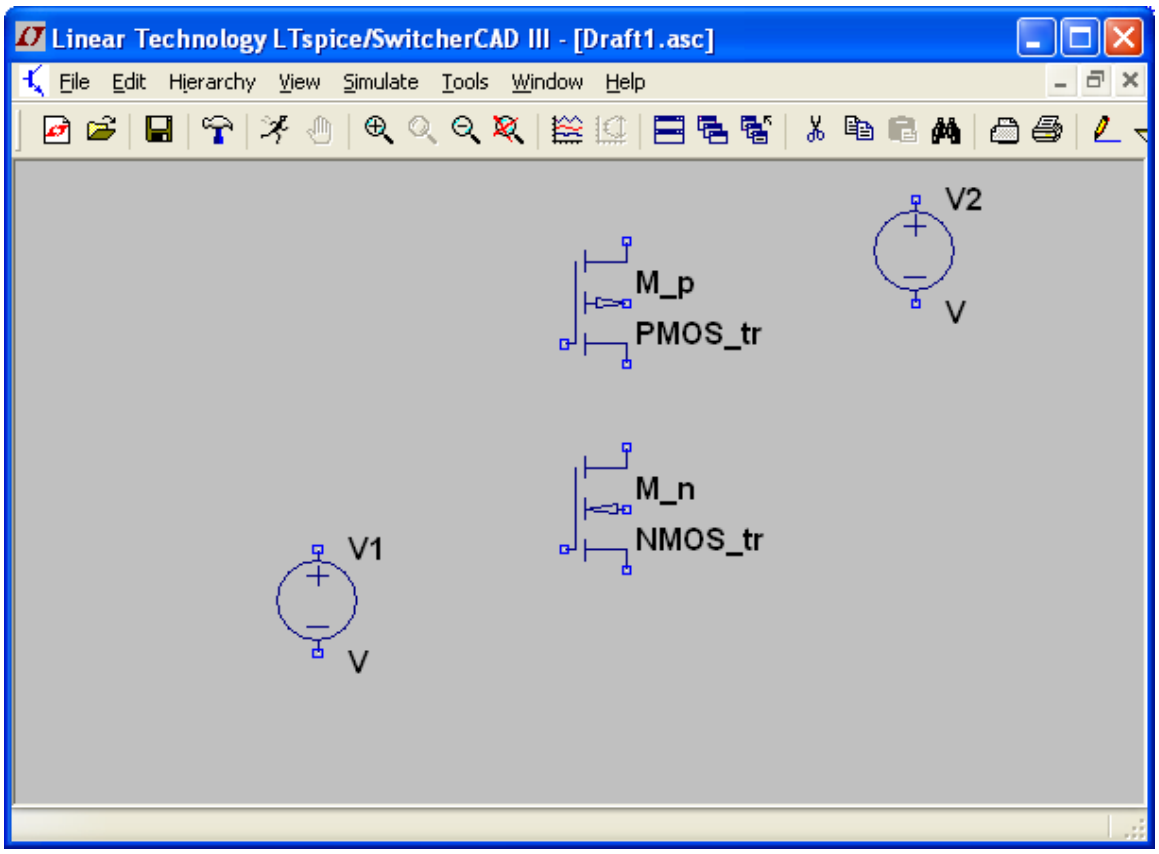




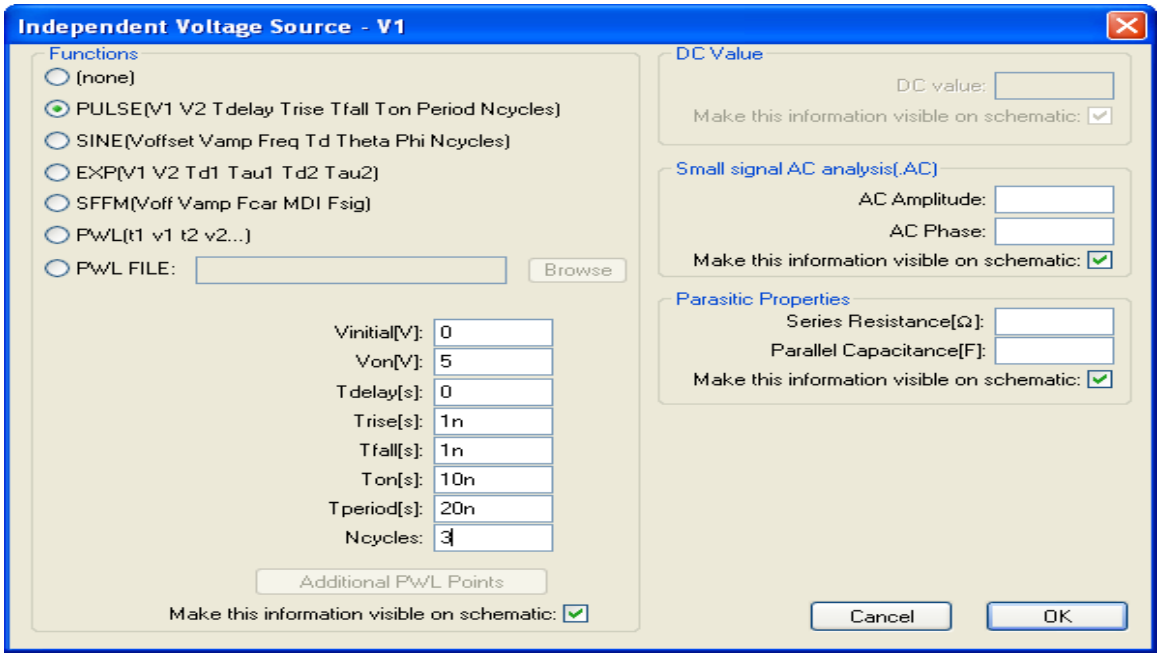
The appearance of the circuit can be modified at any time by using the Edit Menu (or the corresponding toolbar icons)

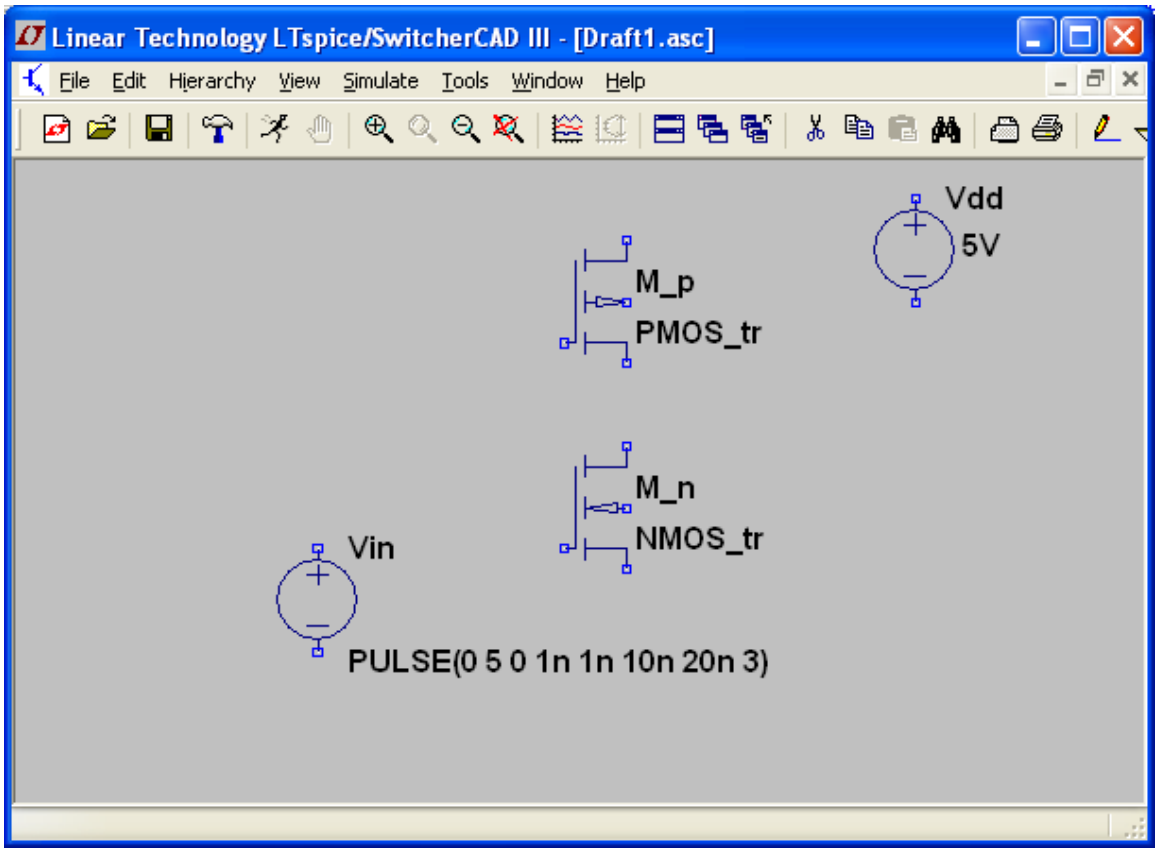


Add the NMOS device and the voltage sources (Edit > Component)

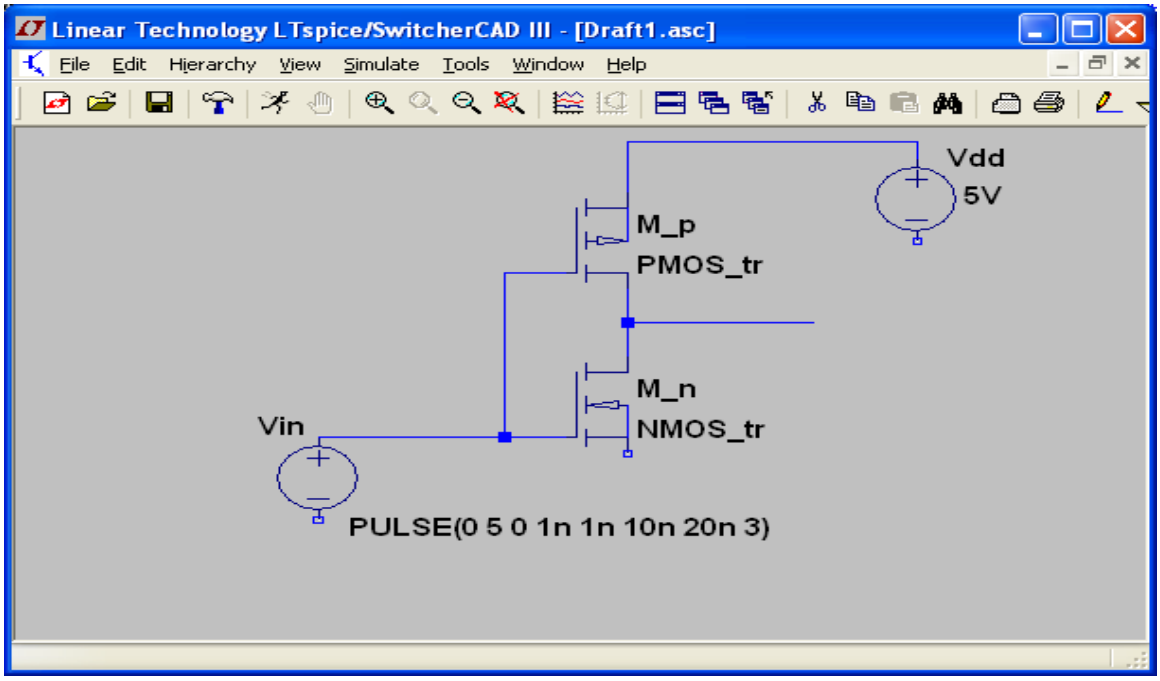


Right click on the NMOS device and the voltage sources to enter their parameters. To set the parameters of the input voltage use the advanced button.

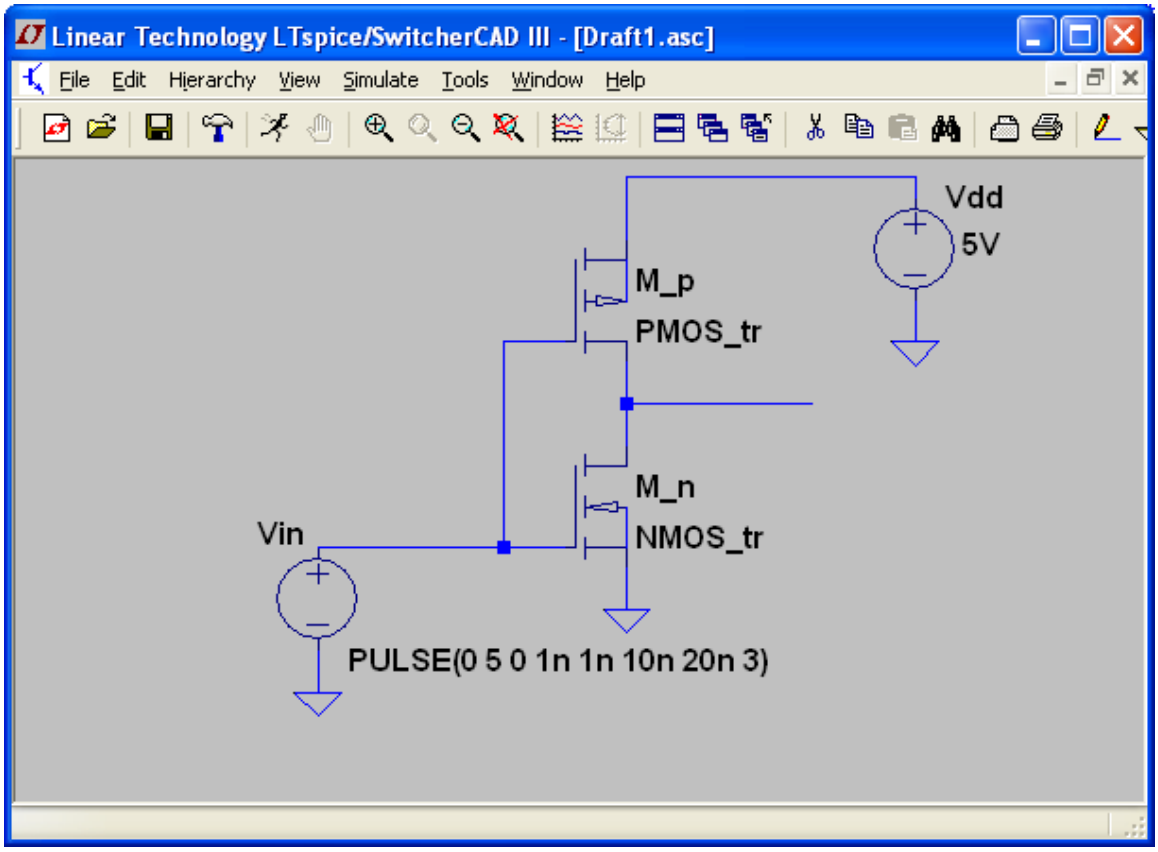




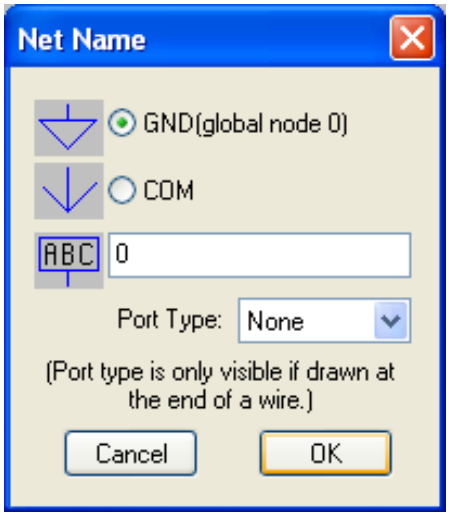
All components necessary to build the inverter are now on the schematic. Next step requires to connect the component together using Edit > Draw Wire.



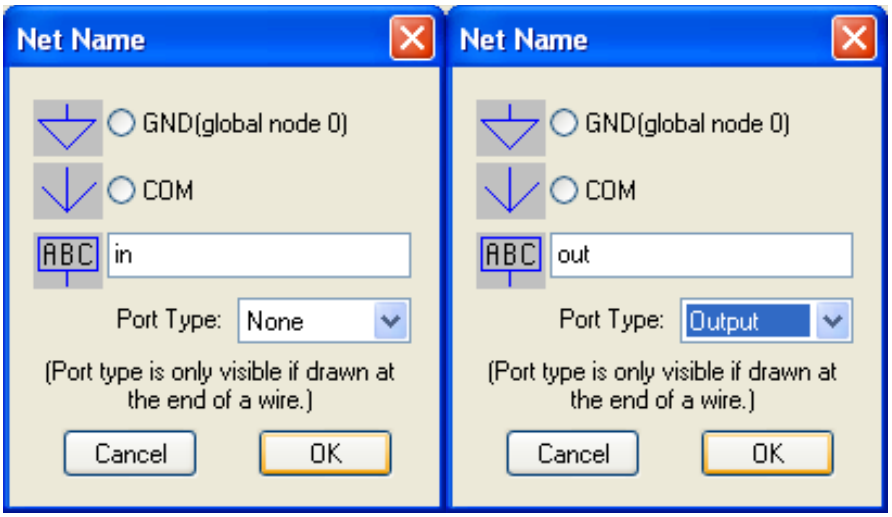
Let's now complete the circuit adding grounds (Edit > Place GND) where needed.

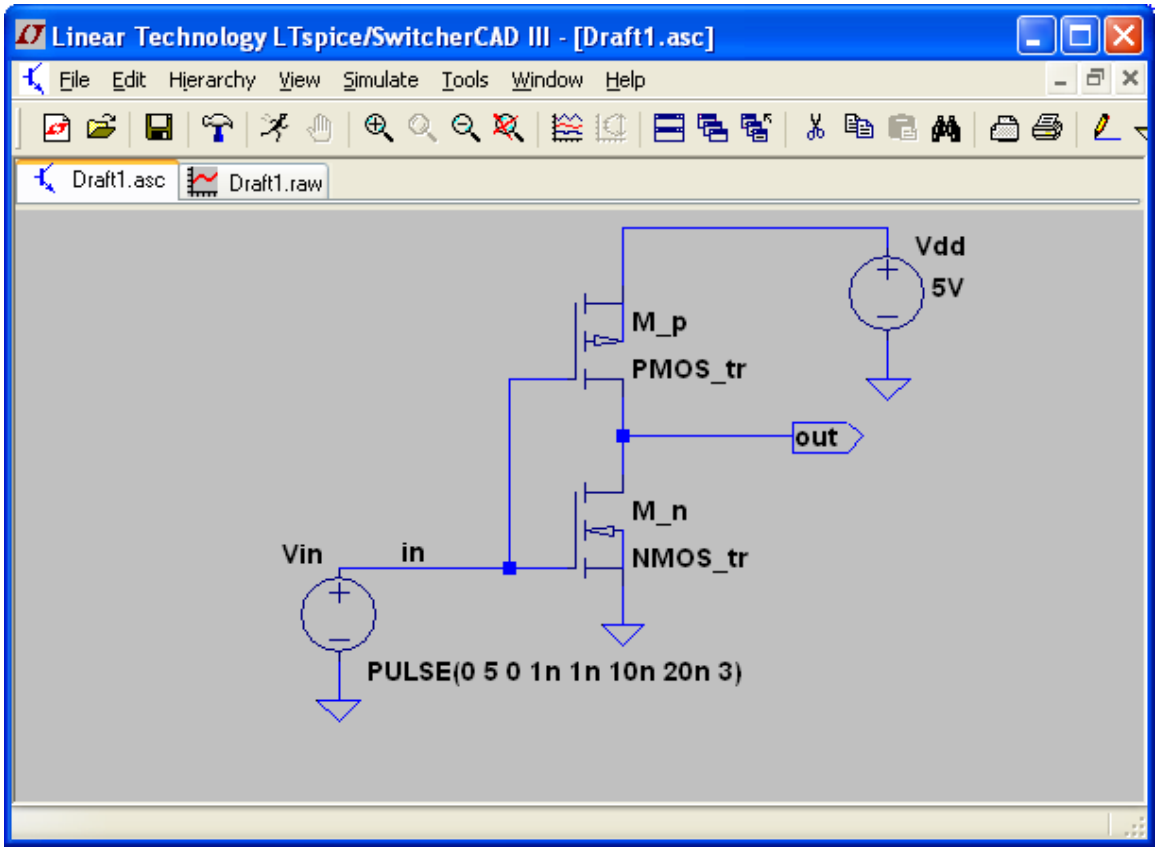


Right click on each ground symbol and make sure that all grounds are common.



Let's label a couple of nodes of the circuit with convenient names (for example the node where the input signal is applied and the node where the output of the inverter is taken) by using Edit > Net Name

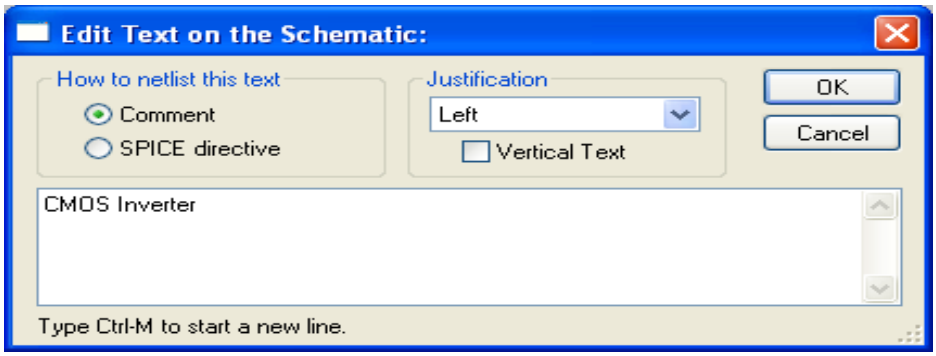


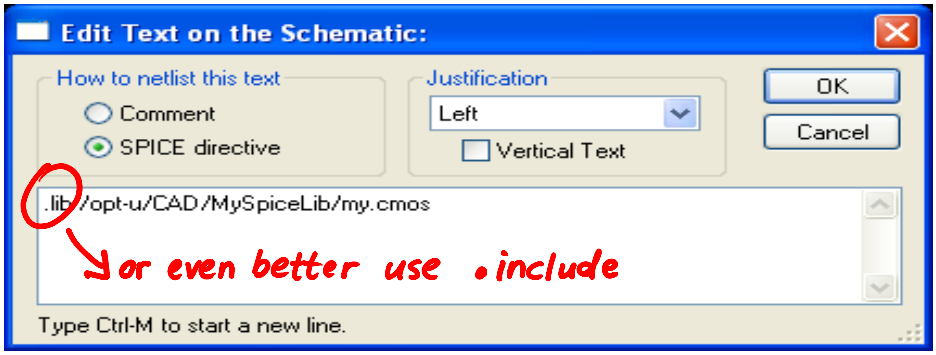


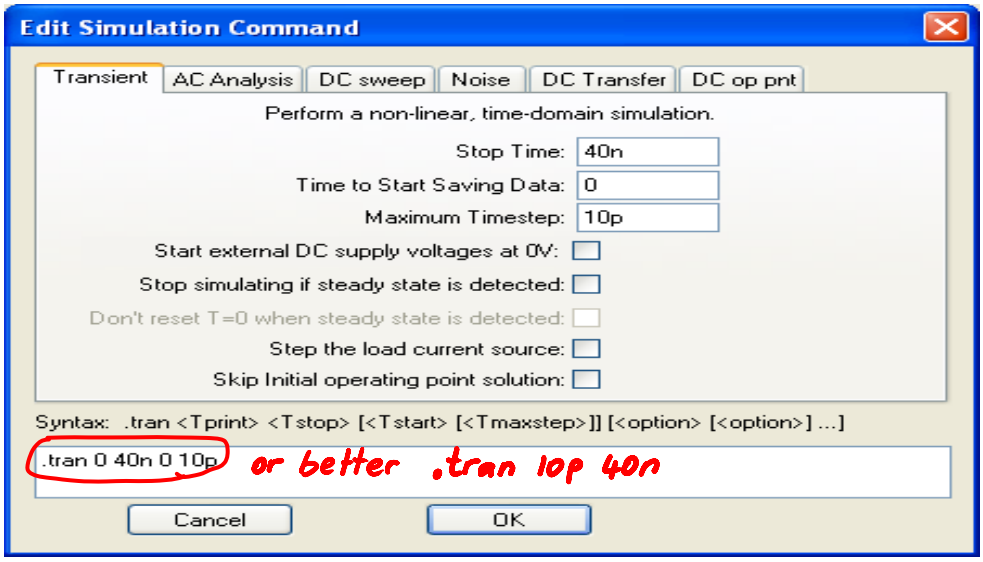
At this point all we have to do is to specify the SPICE analysis we would like to run and where the models for the transistors are located.
Let's put the spice models in a file called `my.cmos` located in the directory: `/opt-u/CAD/MySpiceLib`

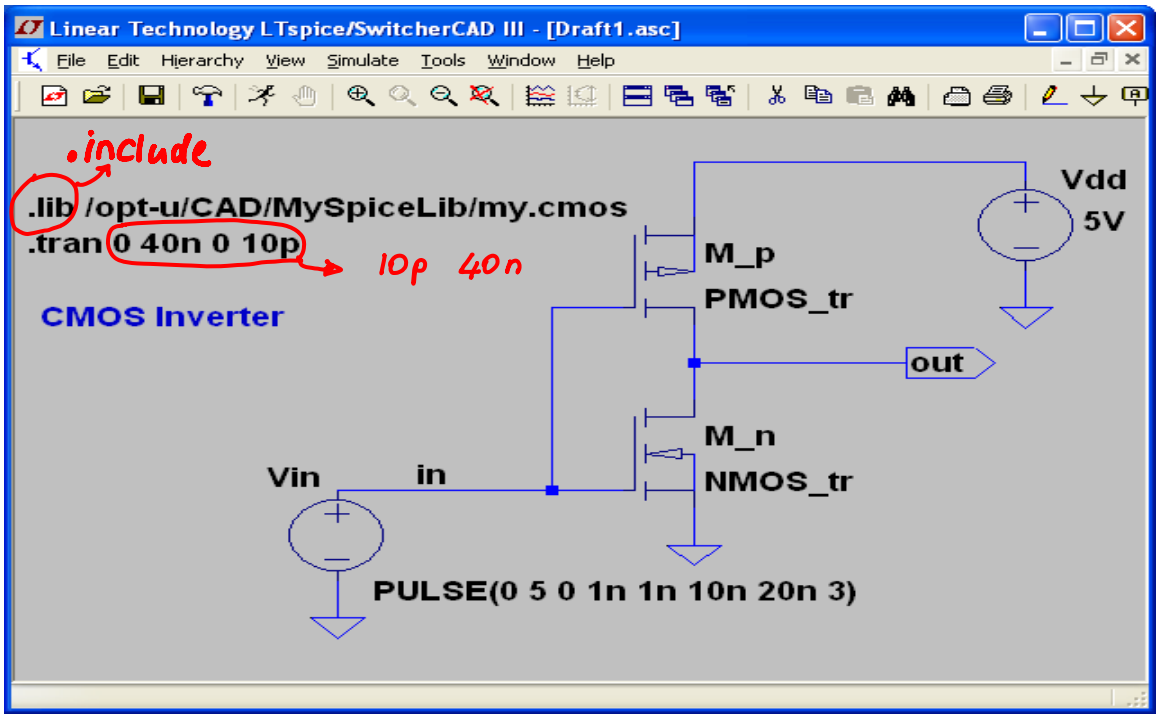
```
* Spice Models Library. Filename: my.cmos

.model NMOS_tr NMOS(kp=0.25m Vto=1 lambda=0.02)
.model PMOS_tr PMOS(kp=0.25m Vto=-1 lambda=0.02)
```

To add to the schematic the necessary spice directives and analysis we use:
Edit > SPICE Directives and Edit > SPICE Analysis.

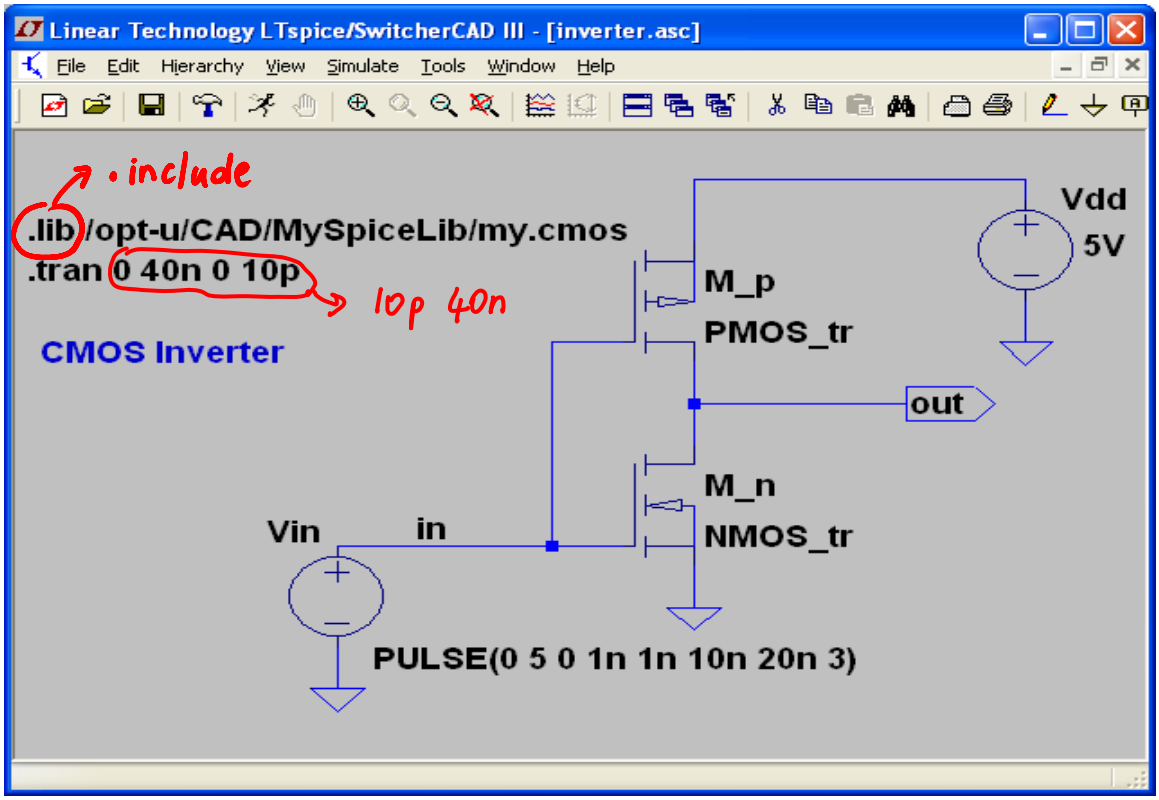




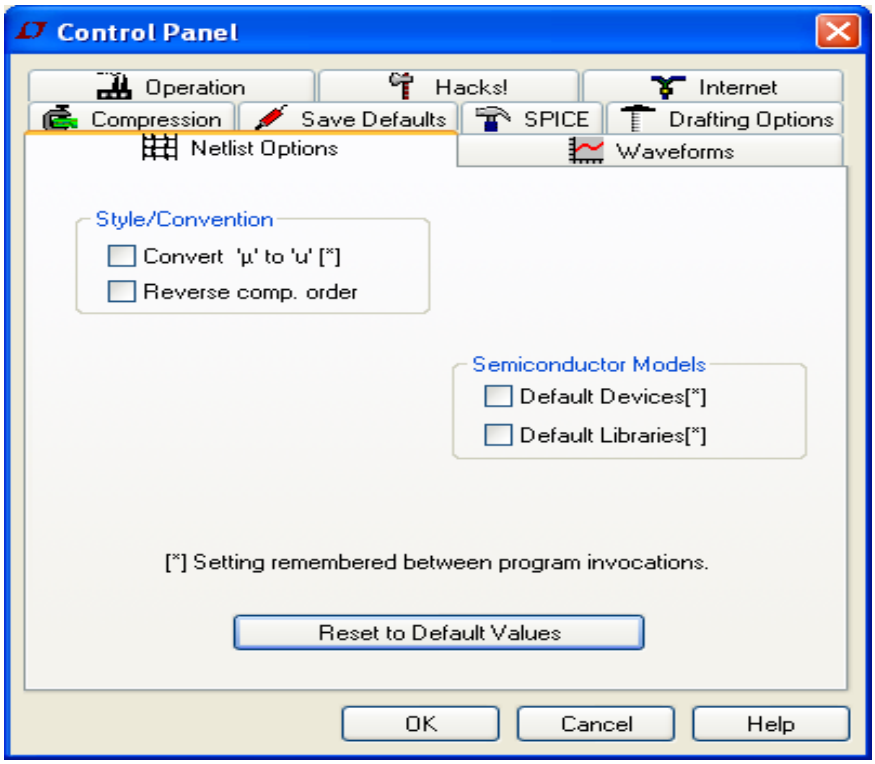




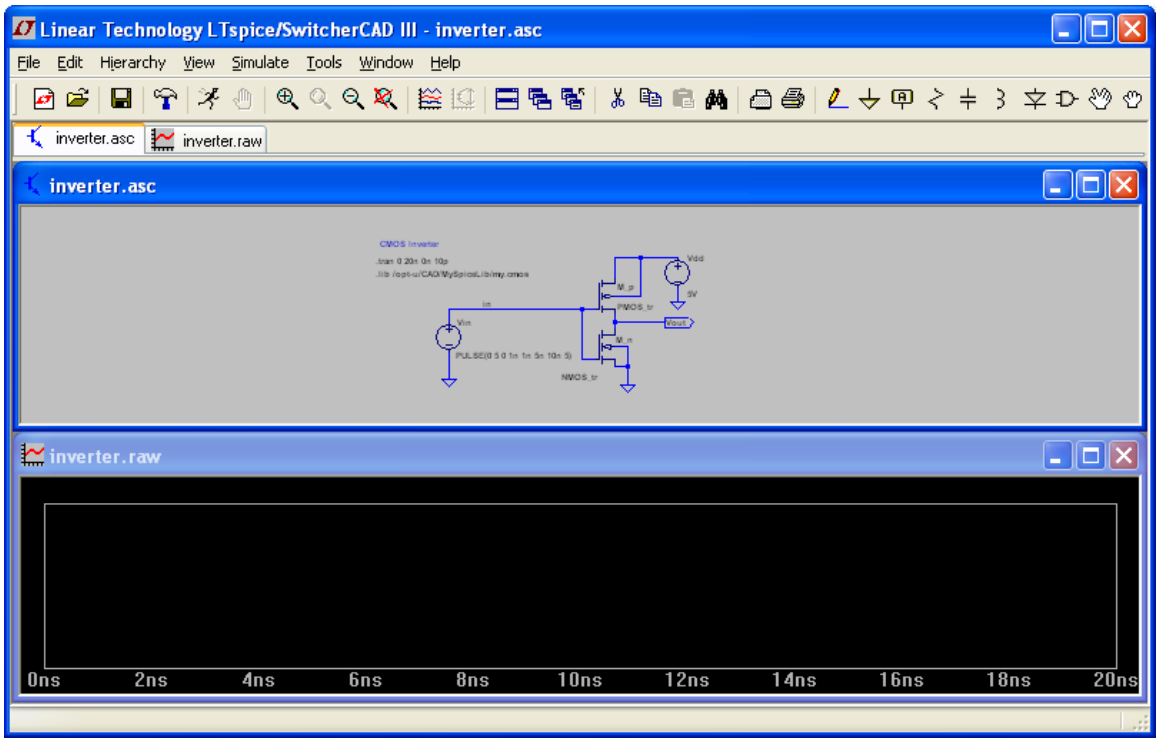
4. Save the schematic As: inverter.asc



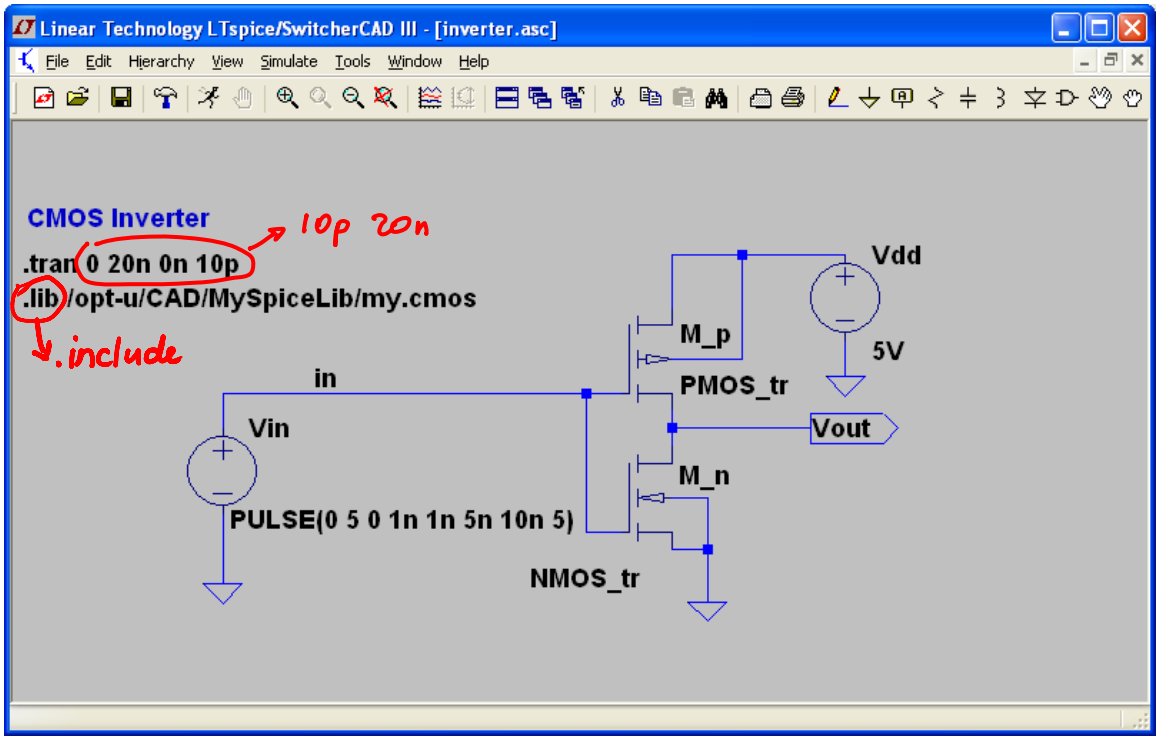
5. Set up the SPICE netlist generation preferences using:
   Tools > Control Panel > Netlist Option



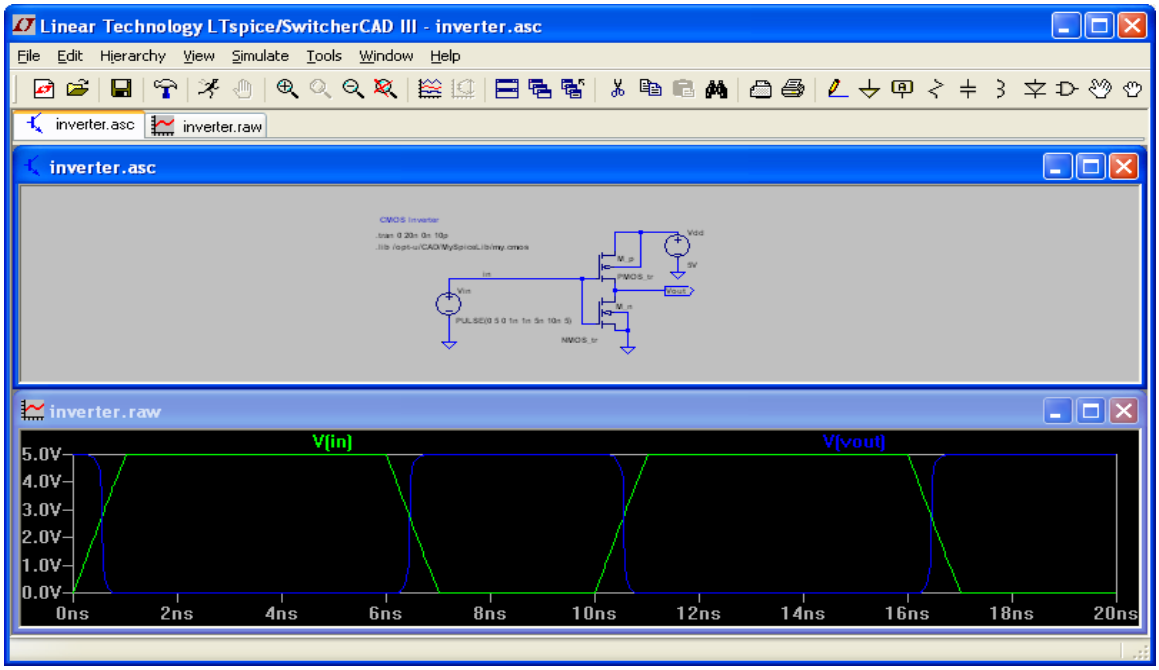
6. Run the Spice Simulation (Simulate > Run)



7. Make sure there are no errors by checking View >SPICE Error Log
   Hops!! Here we notice the bulks are not properly connected, so we 1) fix the schematic, 2) run again the simulation and 3) check again for errors



8. In order to observe the traces we are interested in 1) right click on the inverter.raw window, 2) click on Add Trace or Visible Traces and 3) make your selections. Alternatively you can also click on the nodes of interest on the schematic window.



9. The spice netlist is automatically saved as inverter.net and can be opened using: View > Spice Netlist

```
* /home/ctalarico/MySpiceCir/inverter.asc
M_p N001 in out N001 PMOS_tr l=0.5u w=1u
M_n out in 0 0 NMOS_tr l=0.5u w=0.5u
Vdd N001 0 5V
Vin in 0 PULSE(0 5 0 1n 1n 5n 10n 5)
* CMOS Inverter
.tran 0 40n 0n 10p
.lib /opt-u/CAD/MySpiceLib/my.cmos
.backanno
.end
```