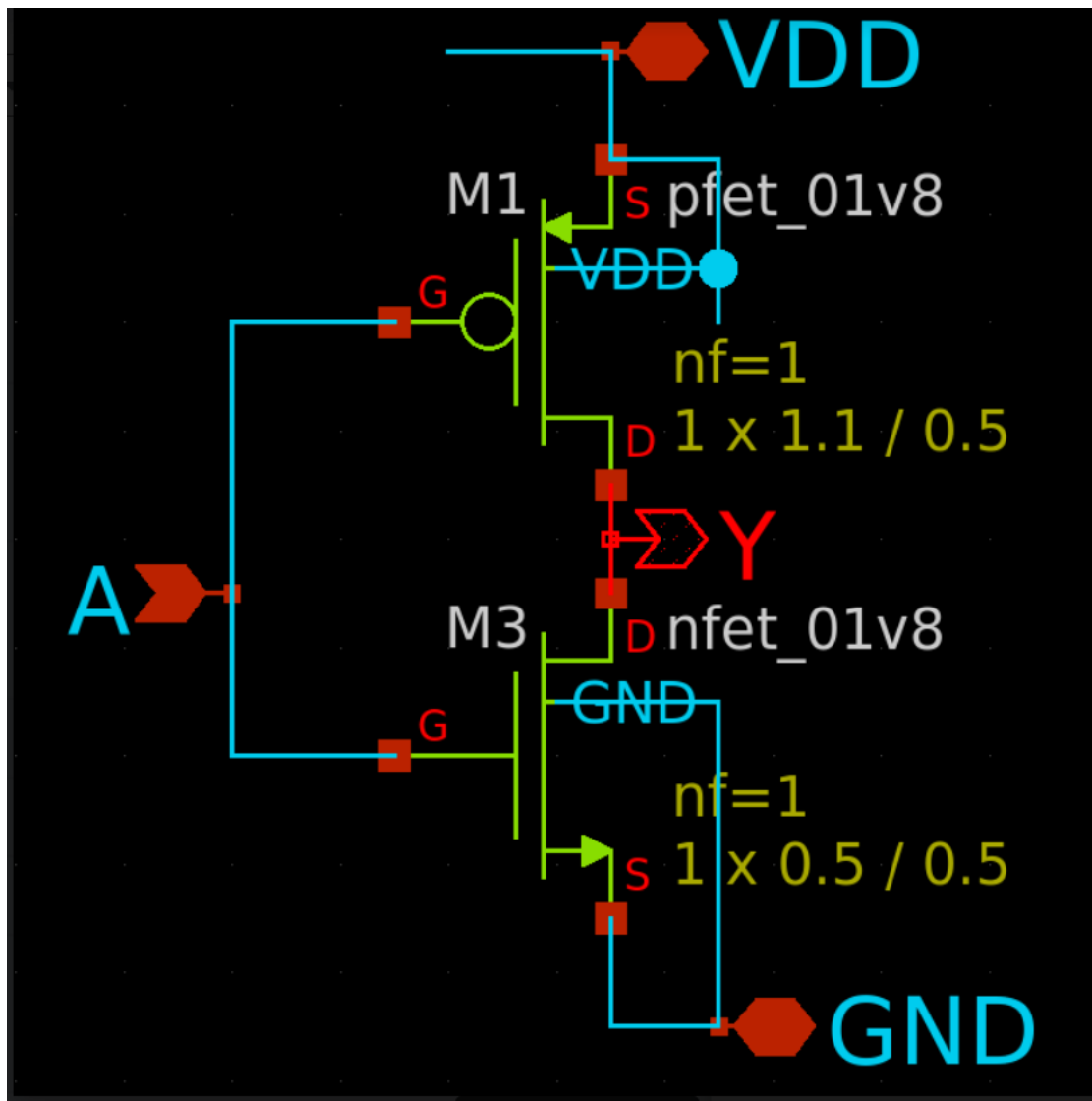
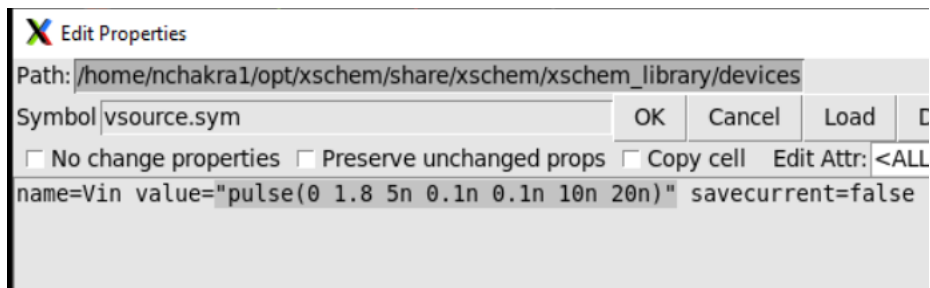


Designing the schematic and the layout of an inverter using Skywater 130nm process

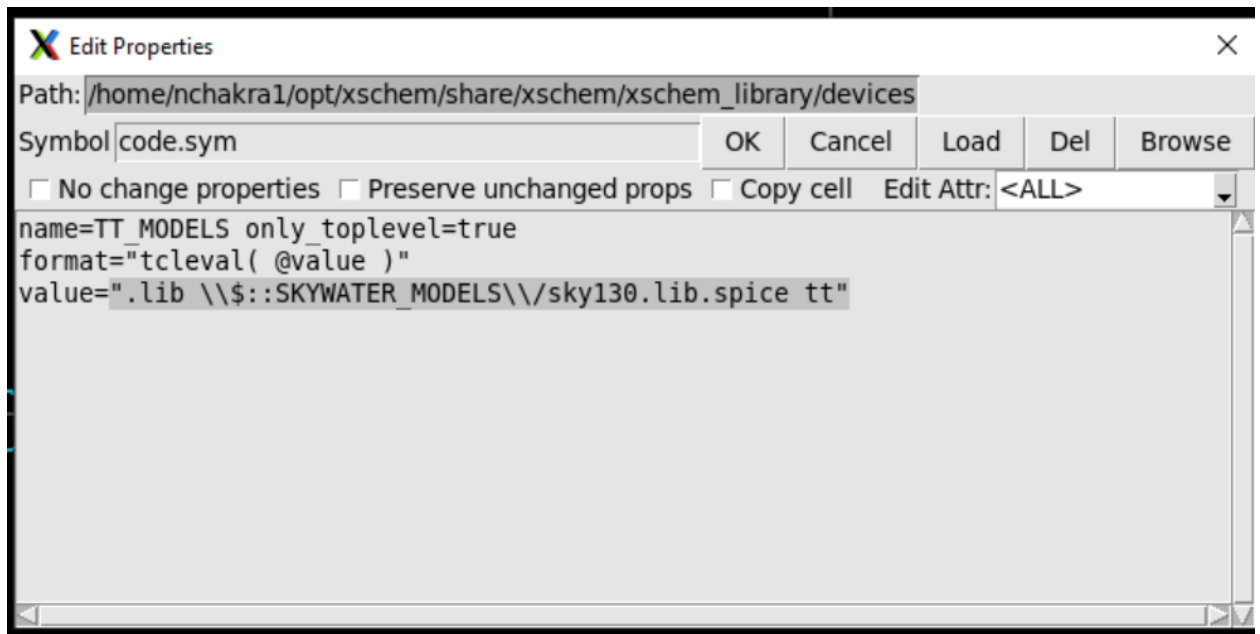
1. Create a working directory using: `mkdir <dir_name>`
2. Go to the directory using: `cd <dir_name>`
3. Copy in the example directory the xschem initialization file available in the PDK
`cp ~/share/pdk/sky130A/libs.tech/xschem/xschemrc .`
4. create an ngspice initialization file `.spiceinit` with the following lines:
set ngbehavior=hs
set ng_nomodcheck
set color0=white
set color1=black
set xbrushwidth=2
5. Start xschem using: `xschem &`
6. Create the schematic and the symbol of the inverter using xschem. To do that, click on the '+' sign on the tab where the current file is displayed.
7. To find components use shift+i. Or use tools->insert symbol
8. For the inverter, use `pfet_01v8` and `nfet_01v8`. You can find those in `xschem_library/sky130_fd_pr`(something like this).
9. Double click on the writings to change the length and width of the mosfets. You can use nmos length 0.5um, width 1um, pmos length 0.5um, and width 2um.
10. Add pins using tools->insert symbol. The input pin should be `ipin`, output pin is `opin`, and for VDD and GND, you should use `iopin`. Connect everything using 'w'



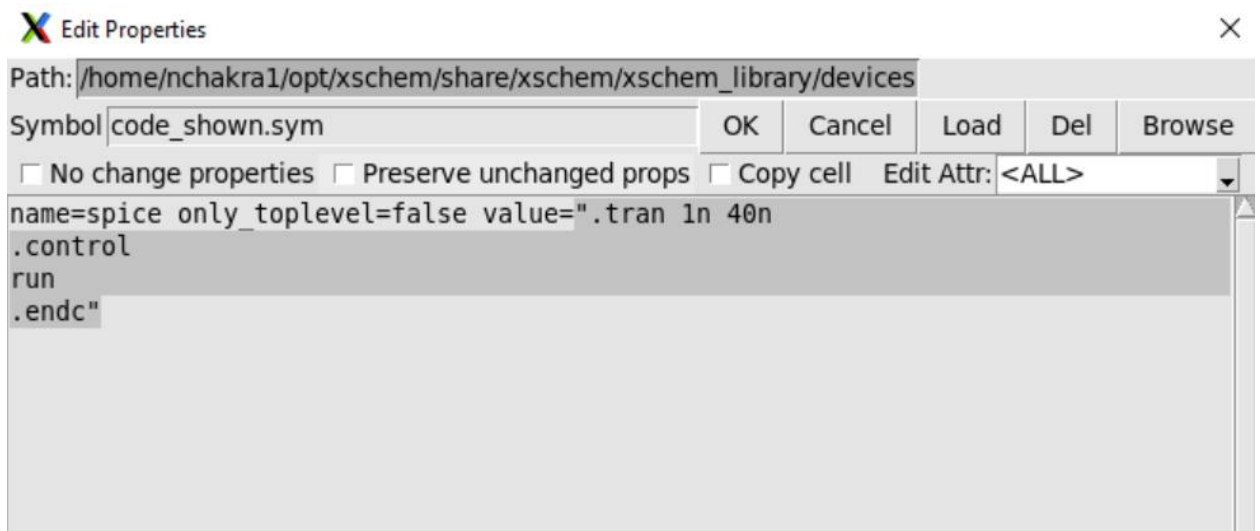
11. Save the schematic. Then create symbol using Symbol > Make symbol from schematic
12. You will see an automatically generated symbol. You can edit it to look like an inverter symbol.
13. Next, simulate the inverter with ngspice. To do that, create another schematic file. Use your inverter to make a test bench. You can use 'vsource' from the symbols. To use it as a pulse, edit it like following:



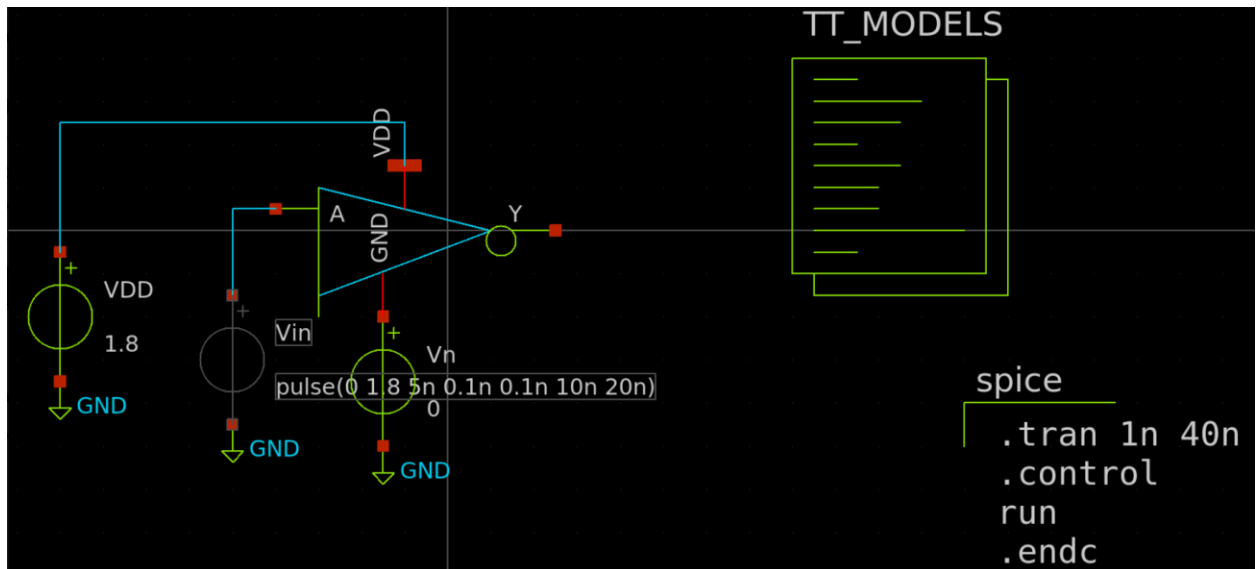
14. Add a TT_Models file from symbols library. Use code.sym. Then edit the content like following by double-clicking on it.



15. Also add another code_shown.sym and edit it to include transient analysis.



16. Your final testbench will look like following:



17. Save the file. Then click on 'Netlist', after that click on 'Simulate'. Once simulation is completed, you will see a window like following:

```

ngspice
ngspice 1 -> display
Here are the vectors currently active:

Title: ** sch_path: /home/nchakra1/example_xschem_sky130/inv_test.sch
Name: tran1 (Transient Analysis)
Date: Sat Oct 19 21:02:27 2024

m.x1.xm1.msky130_fd_pr__pfet_01v8#body: voltage, real, 104 long
m.x1.xm1.msky130_fd_pr__pfet_01v8#dbody: voltage, real, 104 long
m.x1.xm1.msky130_fd_pr__pfet_01v8#sbody: voltage, real, 104 long
m.x1.xm3.msky130_fd_pr__nfet_01v8#body: voltage, real, 104 long
m.x1.xm3.msky130_fd_pr__nfet_01v8#dbody: voltage, real, 104 long
m.x1.xm3.msky130_fd_pr__nfet_01v8#sbody: voltage, real, 104 long
net1      : voltage, real, 104 long
net2      : voltage, real, 104 long
net3      : voltage, real, 104 long
net4      : voltage, real, 104 long
time      : time, real, 104 long [default scale]
vdd#branch : current, real, 104 long
vin#branch : current, real, 104 long
vn#branch  : current, real, 104 long
ngspice 2 -> plot net3
ngspice 3 -> /home/nchakra1/tmp/hc3758_0.ps: No such file or directory

```

18. You can type plot 'net_name' to see the output.

19. Once you are done with this part, start working on the layout using magic. Make sure to use the exact nmos and pmos dimensions as the schematic in your layout.