Alexandria University
Faculty of Engineering
Electrical Engineering Department

EE 431 Digital Integrated Circuits
Lab#1: Physical Design and Layout of CMOS Inverter

Objectives

Upon the completion of this Lab, you should be able to:

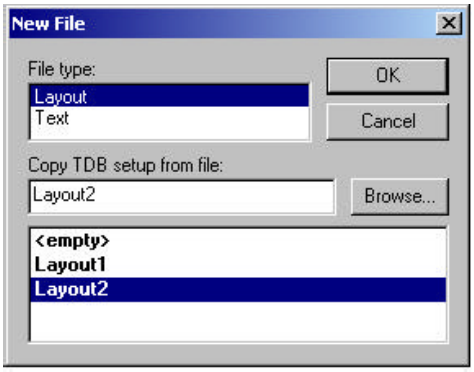
1. Use L-edit software to lay out basic CMOS digital circuits,
2. Use T-spice software to analyze static and dynamic characteristics of CMOS digital circuits.

Requirements

You are required to draw the vertical and horizontal layouts for the minimum sized symmetric CMOS inverter using the 2um MOSIS CMOS technology and compare between the two layouts in terms of the used area, power consumption, DC characteristics, and propagation delays. Both inverters should have the same dimensions. The following sections provide the detailed procedures to draw the layout of the vertical CMOS inverter using L-edit.

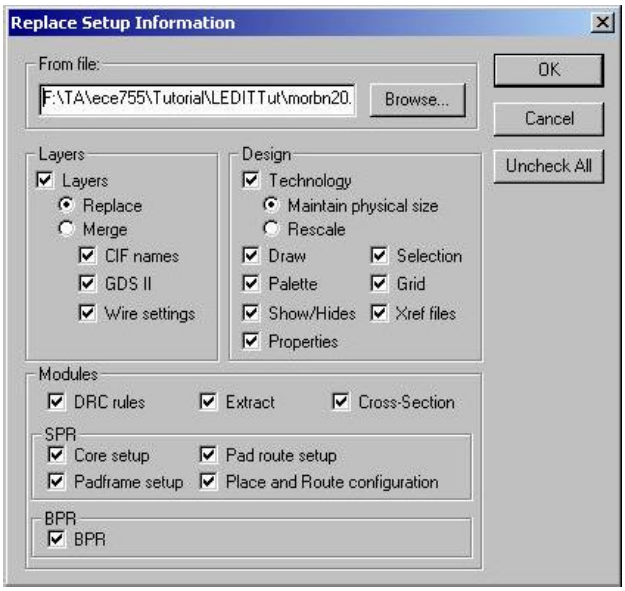
Manual Layout

1. Copy the following files into your directory
   Technology setup files for MOSIS/Orbit n-well 2.0 micron process. (Technology
   = SCNA, LAMBDA = 1.0 micron)
   C\tanner\ledit83\samples\tech\mosis\morbn20.
2. Launch L-Edit
3. Create New File. Create new files by choosing **File > New,** which opens the **New File** dialog:

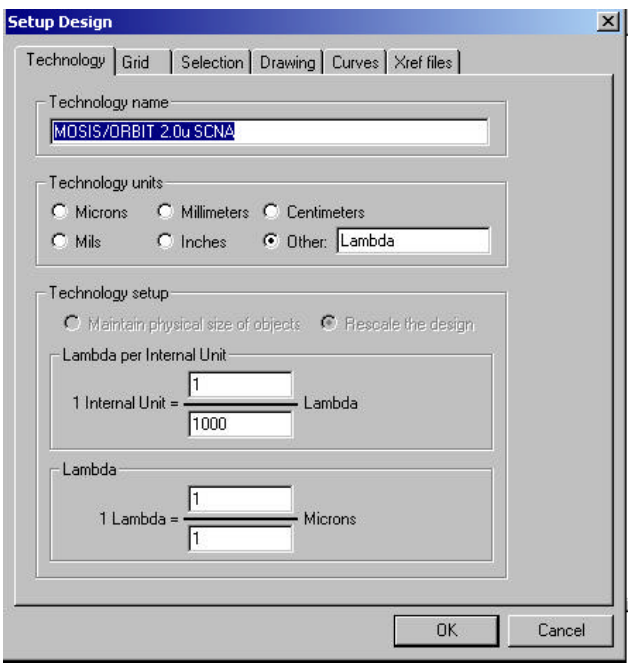


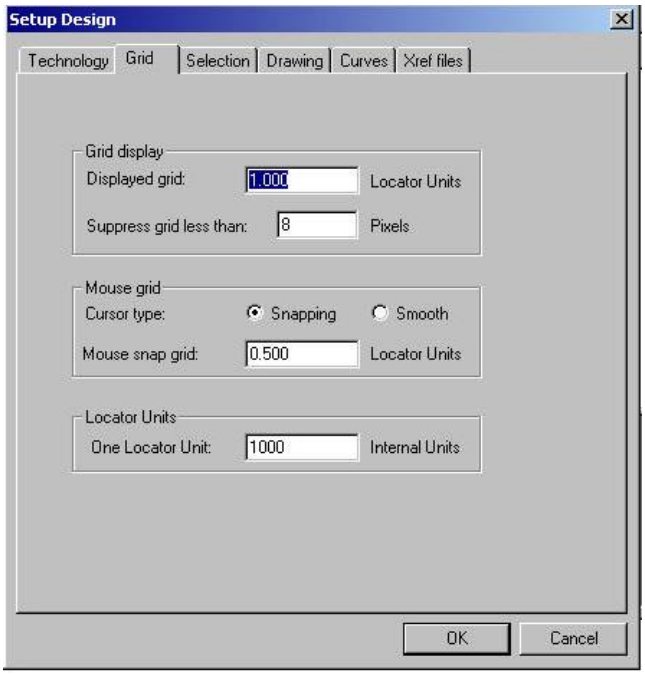
4. Replacing the Setup.

   File > Replace Setup transfers setup information from a file (the *source* file) to the current file (the *destination* file).



5. Under Setup -> Design
   The chosen technology units should be lambda It should be 1 internal unit for 1/1000 lambda The lambda value should be 1 microns
   Grid -> Grid Display -> Displayed Grid 0.5 locator units



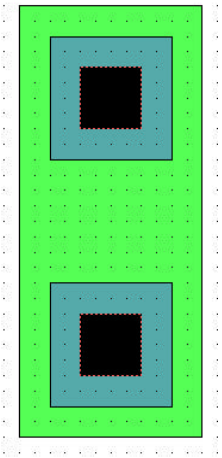


6. Create a new cell. Cell -> New, call it inv

7. Draw 14 x 6 λ **Active** Box.

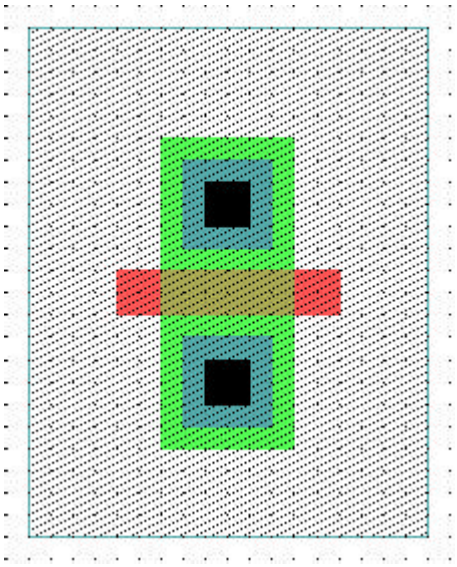
8. Draw two 4 x 4 λ **Metal1** box and put on both sides of Active box.

9. Draw two 2 x 2 λ **Active Contact** centered on each Metal 1 box It looks like.



10. Draw 10 x 2 λ poly box centered at the active.

11. Draw 18 x 23 λ N Select.



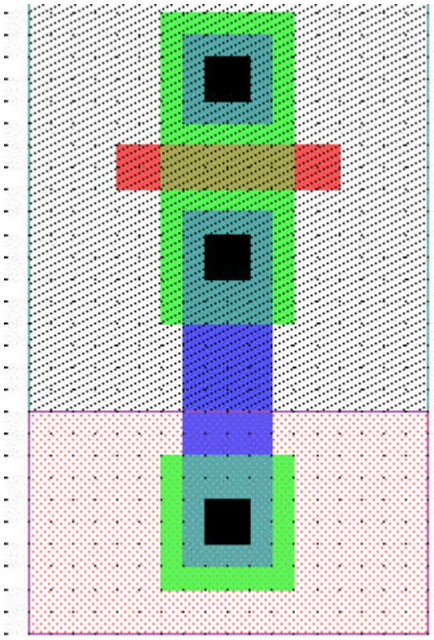
12. Copy the whole block above, select N Select and click Edit -> Edit Object, change
N Select to be P Select.

13. Draw 10 x 18 λ P Select below the Nselect of NMOS.

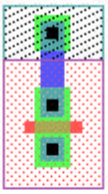
14. Draw 6 x 6 λ Active inside the Pselect.

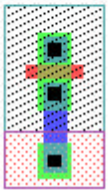
15. Draw 2 x 2 λ Active Contact inside the Active

16. Extend the Metal1 to Pselect area, it looks like:

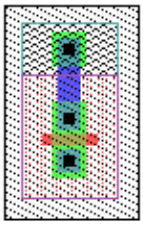


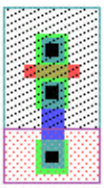
17. With the same method draw a Nselect on the top of PMOS



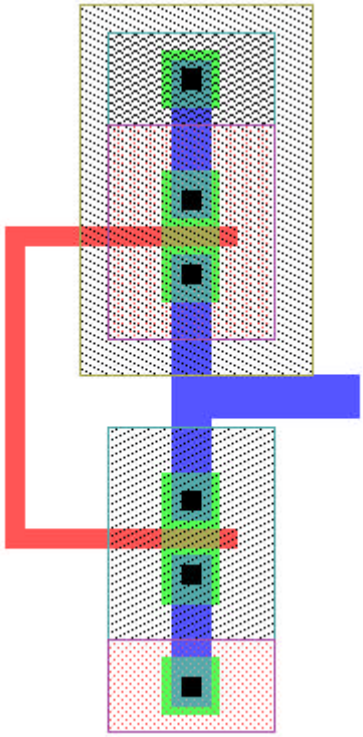


18. Draw 25 x 40 λ Nwell at PMOS place.





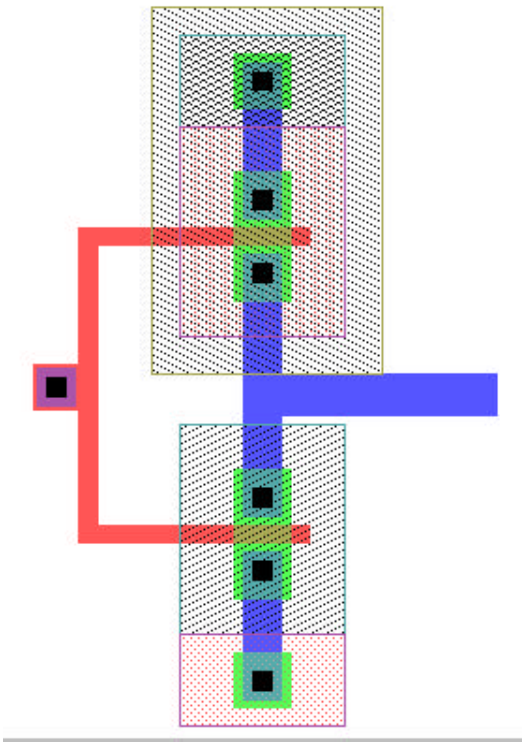
19. Connect following the inverter schematic.



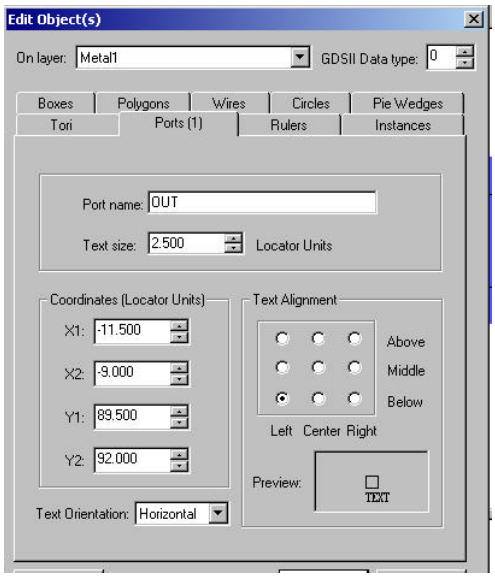
20. Draw 5 x 6 λ Poly beside the gate of two transistors.

21. Draw 4 x 4 λ Metal 1 centered in the poly.

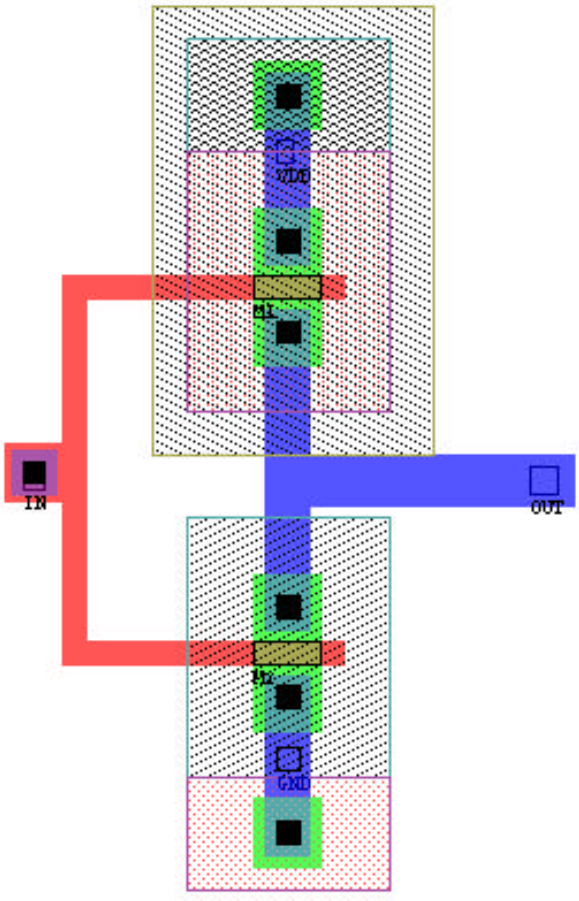
22. Draw 2 x 2 λ Poly Contact centered in the Metal1



23. Click Port  on the Toolbar. And draw OUT port. The dialog jumps out. On Layer: Metal1
Port name: OUT

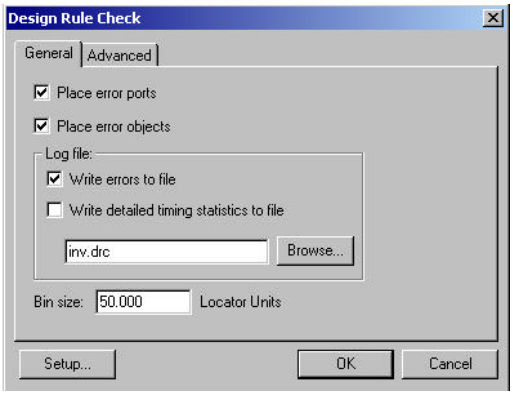


24. With the same method draw port for IN, VDD, GND. On Layer: Metal1, Port name:IN/VDD/GND. It looks like



25. Run DRC check 

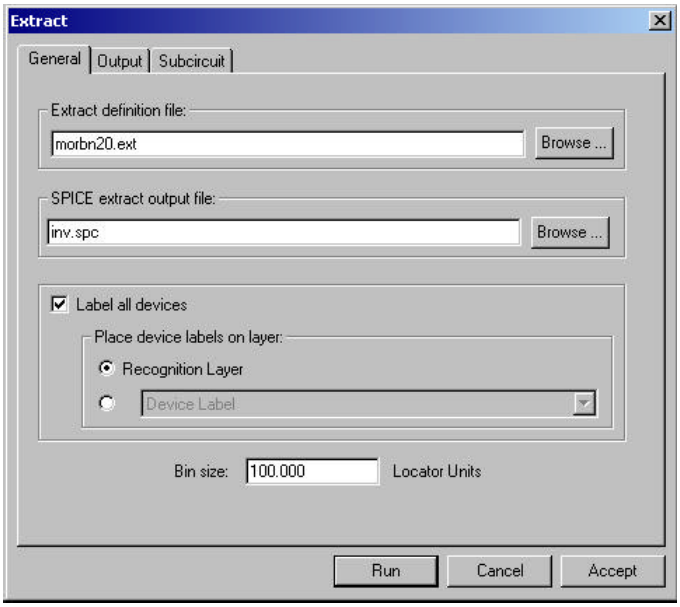
A dialog box jumps out.

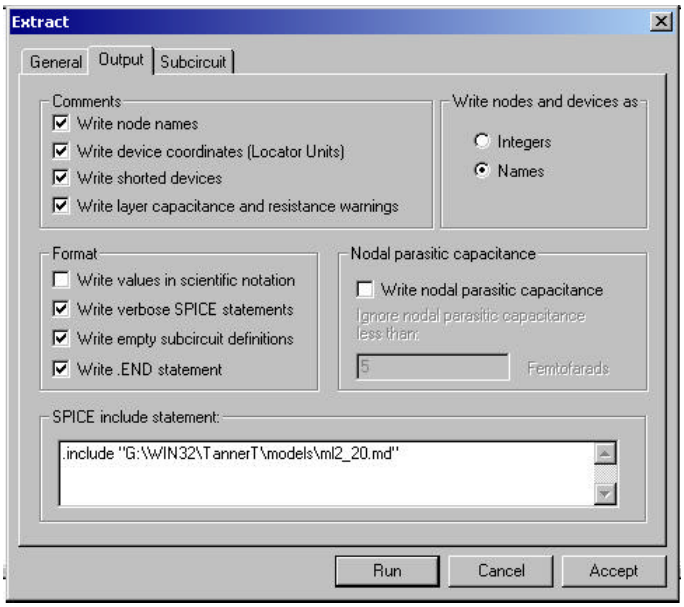


26. If No DRC error, go to next step. If there is any, open the file inv.drc or click to see the error. If you don't like the error displayed on the screen, click  on the bar.

27. Extract the file to be SPICE file. Click 

on the toolbar. A dialog jumps out. Fill the form as following then click RUN.





28. Launch T-Spice to run simulation. The Spice file looks like

```
* Circuit Extracted by Tanner Research's L-Edit Version 8.30 / Extract Version 8.30 ;
* TDB File:  F:\TA\ece755\Tutorial\LEDITTut\inv.tdb
* Cell:  inv   Version 1.24
* Extract Definition File:  morbn20.ext
* Extract Date and Time:  09/28/2001  - 10:26

.include "G:\WIN32\TannerT\models\ml2_20.md"

* Warning:  Layers with Unassigned AREA Capacitance.
*   <P Base Resistor>
```

```
*   <N Well Resistor>
*   <P Diff Resistor>
*   <N Diff Resistor>
*   <Poly2 Resistor>
*   <Poly Resistor>
* Warning:  Layers with Unassigned FRINGE Capacitance.
*   <Poly1-Poly2 Capacitor>
*   <P Base Resistor>
*   <N Well Resistor>
*   <P Diff Resistor>
*   <N Diff Resistor>
*   <Poly2 Resistor>
*   <Poly Resistor>
*   <Pad Comment>
* Warning:  Layers with Zero Resistance.
*   <PMOS Capacitor>
*   <NMOS Capacitor>
*   <Poly1-Poly2 Capacitor>
*   <Pad Comment>

* NODE NAME ALIASES
*       1 = IN (-56.5,90)
*       2 = OUT (-11.5,89.5)
*       3 = VDD (-34,119)
*       4 = GND (-34,65)


M1 OUT IN VDD VDD PMOS L=2u W=6u AD=36p PD=24u AS=36p PS=24u
* M1 DRAIN GATE SOURCE BULK (-36 107  -30 109)
M2 GND IN OUT GND NMOS L=2u W=6u AD=36p PD=24u AS=36p PS=24u
* M2 DRAIN GATE SOURCE BULK (-36 74.5  -30 76.5)

* Total Nodes: 4
* Total Elements: 2
* Total Number of Shorted Elements not written to the SPICE file: 0
* Extract Elapsed Time: 0 seconds
.END
```

Pay attention to the sequence of the MOSFET connection to ensure it is drain, gate, source and base

29. Add the running command and run simulation.

```
* Circuit Extracted by Tanner Research's L-Edit Version 8.30 / Extract Version 8.30 ;
* TDB File:  F:\TA\ece755\Tutorial\LEDITTut\inv.tdb
* Cell:  inv   Version 1.24
* Extract Definition File:  morbn20.ext
* Extract Date and Time:  09/28/2001  - 10:26

.include "G:\WIN32\TannerT\models\ml2_20.md"

* Warning:  Layers with Unassigned AREA Capacitance.
*   <P Base Resistor>
*   <N Well Resistor>
*   <P Diff Resistor>
*   <N Diff Resistor>
*   <Poly2 Resistor>
```

```
*   <Poly Resistor>
* Warning:  Layers with Unassigned FRINGE Capacitance.
*   <Poly1-Poly2 Capacitor>
*   <P Base Resistor>
*   <N Well Resistor>
*   <P Diff Resistor>
*   <N Diff Resistor>
*   <Poly2 Resistor>
*   <Poly Resistor>
*   <Pad Comment>
* Warning:  Layers with Zero Resistance.
*   <PMOS Capacitor>
*   <NMOS Capacitor>
*   <Poly1-Poly2 Capacitor>
*   <Pad Comment>

* NODE NAME ALIASES
*       1 = IN (-56.5,90)
*       2 = OUT (-11.5,89.5)
*       3 = VDD (-34,119)
*       4 = GND (-34,65)


M1 OUT IN VDD VDD PMOS L=2u W=6u AD=36p PD=24u AS=36p PS=24u
* M1 DRAIN GATE SOURCE BULK (-36 107  -30 109)
M2 GND IN OUT GND NMOS L=2u W=6u AD=36p PD=24u AS=36p PS=24u
* M2 DRAIN GATE SOURCE BULK (-36 74.5  -30 76.5)

Vin IN GND PULSE (0 5 0 1n 1n 100n
200n) Vdd VDD GND 5

.tran/powerup 5n 500n method=bdf
.print tran v(IN) v(OUT)
* Total Nodes: 4
* Total Elements: 2
* Total Number of Shorted Elements not written to the SPICE file: 0
* Extract Elapsed Time: 0 seconds
.END
```