

SILICONCHIP TECHNOLOGIES

USER GUIDE

Cadence Tutorial

for Cadence version 6.1

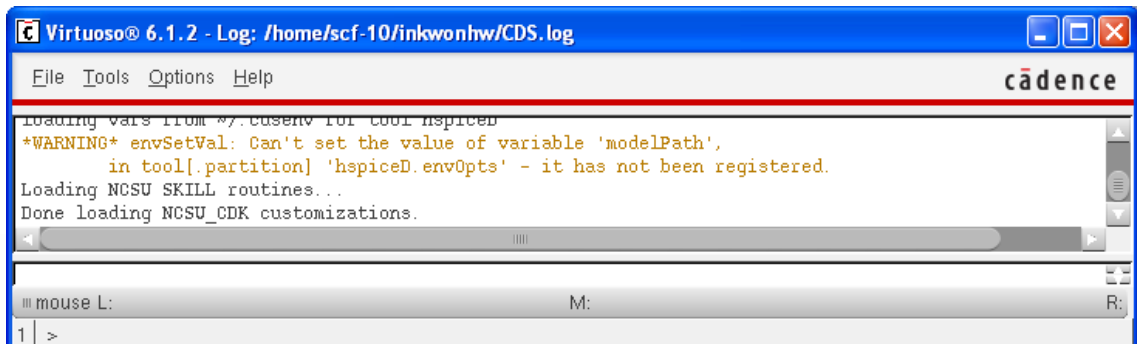
Inkwon Hwang

Feb, 2010

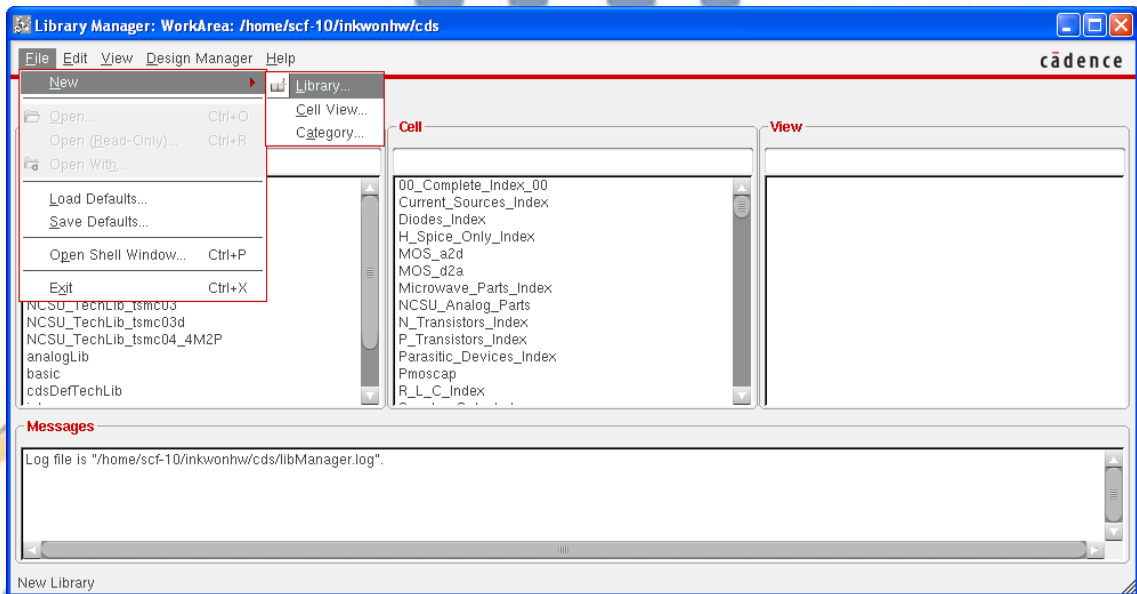
1. Create Library	2
2. Schematic.....	3
A. Create a cell view	3
B. Draw a schematic.....	4
3. Run Spectre simulation	9
A. Launch ADE (Analog Design Environment) L.....	9
4. Layout	10
A. Create a layout	10
B. Add an instance - nmos	11
C. Add more instances – pmos, ptap, ntap, and m1_ploy	12
D. Draw metal1	13
E. Run DRC	14
F. Add pins.....	16
G. Extract	18
H. Run LVS.....	19
I. Run Spectre simulation	20

1. Create Library

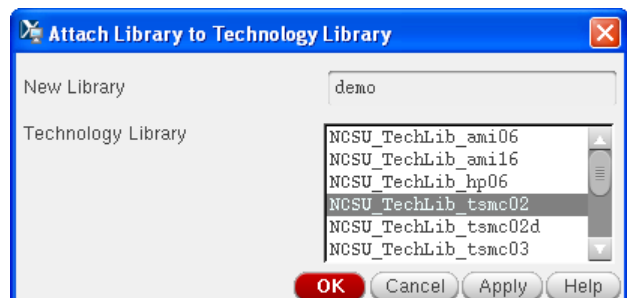
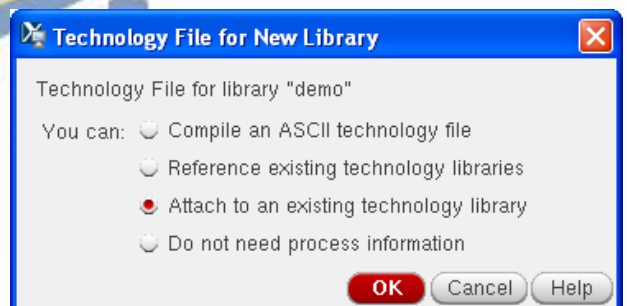
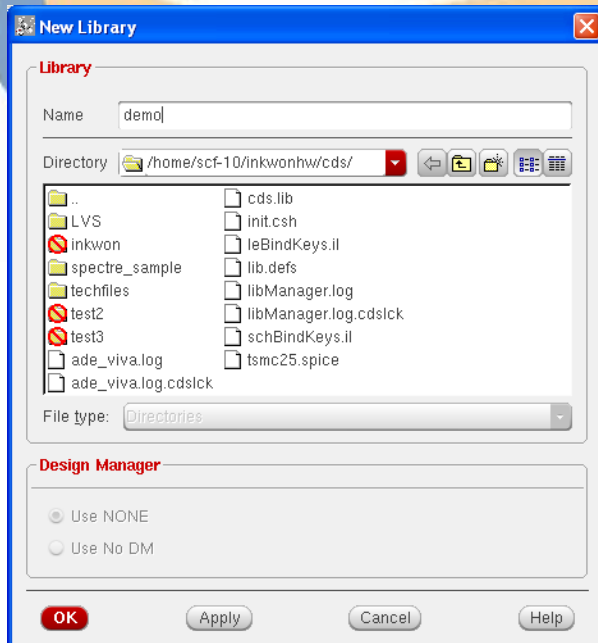
A. Tools → Library Manager



B. File → New → Library

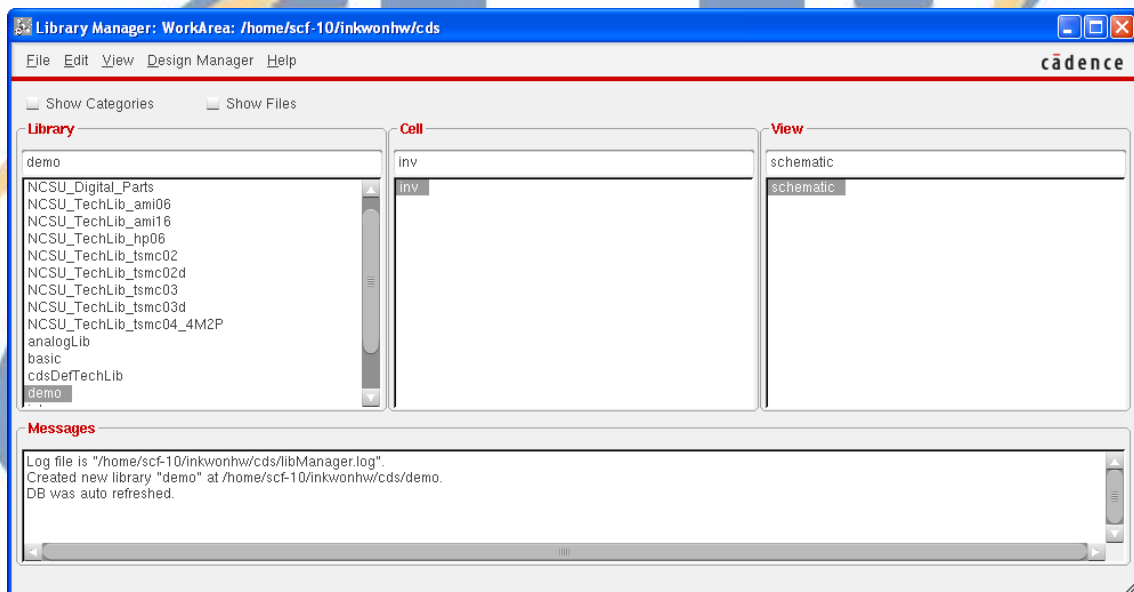
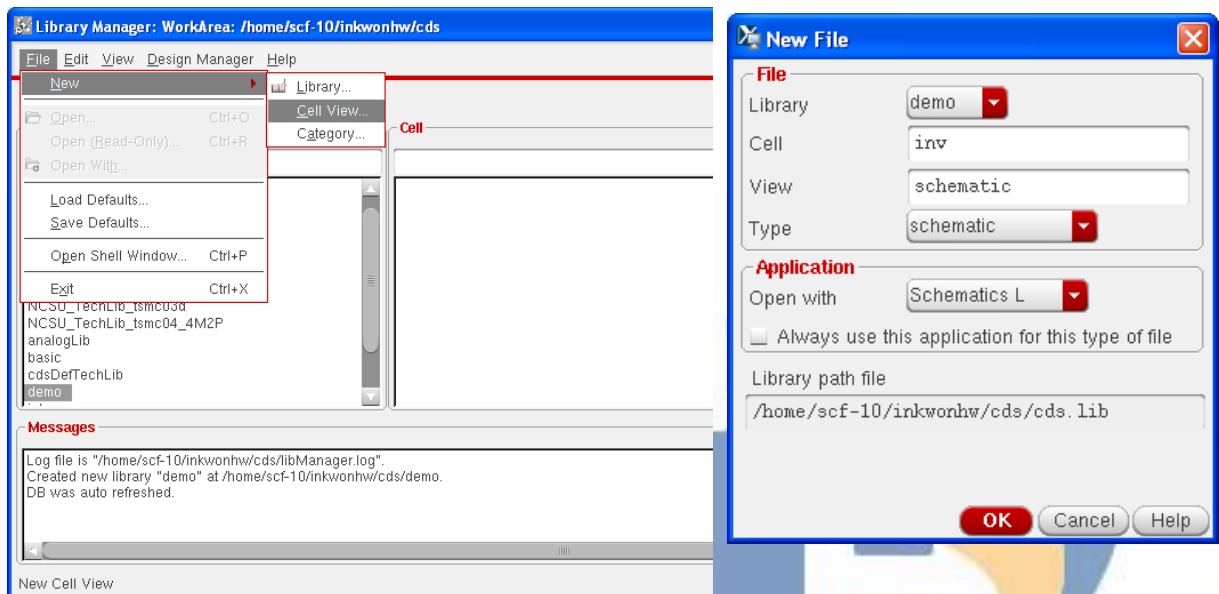


C. Give a name and attach it to a technology library



2. Schematic

A. Create a cell view



B. Draw a schematic

i. Add instances – pmos

You can modify Width of transistors. Don't modify length unless you have a special purpose.

You should select a NCSU_Analog_Parts library.

The first screenshot shows the Virtuoso Schematic Editor with the 'Create' menu open and 'Instance...' selected. The second screenshot shows the 'Add Instance' dialog with 'NCSU_Analog_Parts' as the library and 'pmos' as the cell. The third screenshot shows the 'Component Browser' with 'NCSU_Analog_Parts' selected in the library list.

Virtuoso® Schematic Editor L Editing: demo inv schematic

Launch File Edit View Create Check Options Migrate Wind

Instance... I

- Wire (narrow) W
- Wire (wide) Shift+W
- Wire Name... L
- Net Expression...
- Pin... P
- Block... Shift+I
- Cellview
- Solder Dot
- Note
- Patchcord...
- Probe
- MultiSheet...

mouse L: 1(2) Instance... I

Add Instance

Library: NCSU_Analog_Parts Browse

Cell: pmos

View: symbol

Names:

Array: Rows: 1 Columns: 1

Rotate Sideways Upside Down

Model name: tsmc20P

Model Type: ☒ system ☐ user

Multiplier: 1

Fingers: 1

Width (grid units): 6

Width: 300n M

Width (minimum): 300n M

Length (grid units): 4

Length: 200n M

Length (minimum): 200n M

Drain diffusion area: 1.5e-13

Source diffusion area: 1.5e-13

Drain diffusion perimeter: 1.6u M

Source diffusion perimeter: 1.6u M

Drain diffusion res squares:

Source diffusion res squares:

Virtuoso-XL layout cell:

Drain diffusion length:

Source diffusion length:

Temp rise from ambient:

Estimated operating region: sat

Hide Cancel Defaults Help

Component Browser

Commands Help cadence

Library: NCSU_Analog_Parts

Flatten: NCSU_Analog_Parts

Filter: NCSU_Digital_Parts

Uncate: NCSU_TechLib_ami06

CONTENT: NCSU_TechLib_ami16

Current: NCSU_TechLib_hp06

Diodes: NCSU_TechLib_tsmc02

H_Spic: NCSU_TechLib_tsmc02d

Microw: NCSU_TechLib_tsmc03

Misc_P: NCSU_TechLib_tsmc03d

N_Tran: NCSU_TechLib_tsmc04_4M2P

P_Tran: analogLib

Parasi: basic

R_L_C: demo

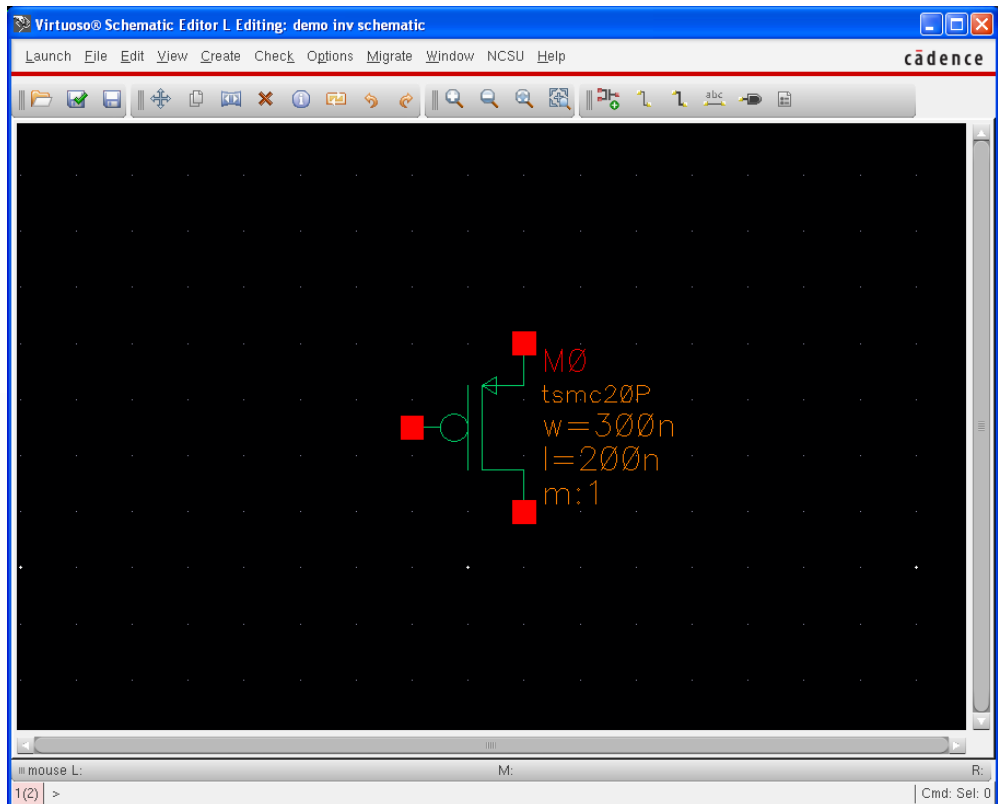
Spectre_Only: inkwon

test2

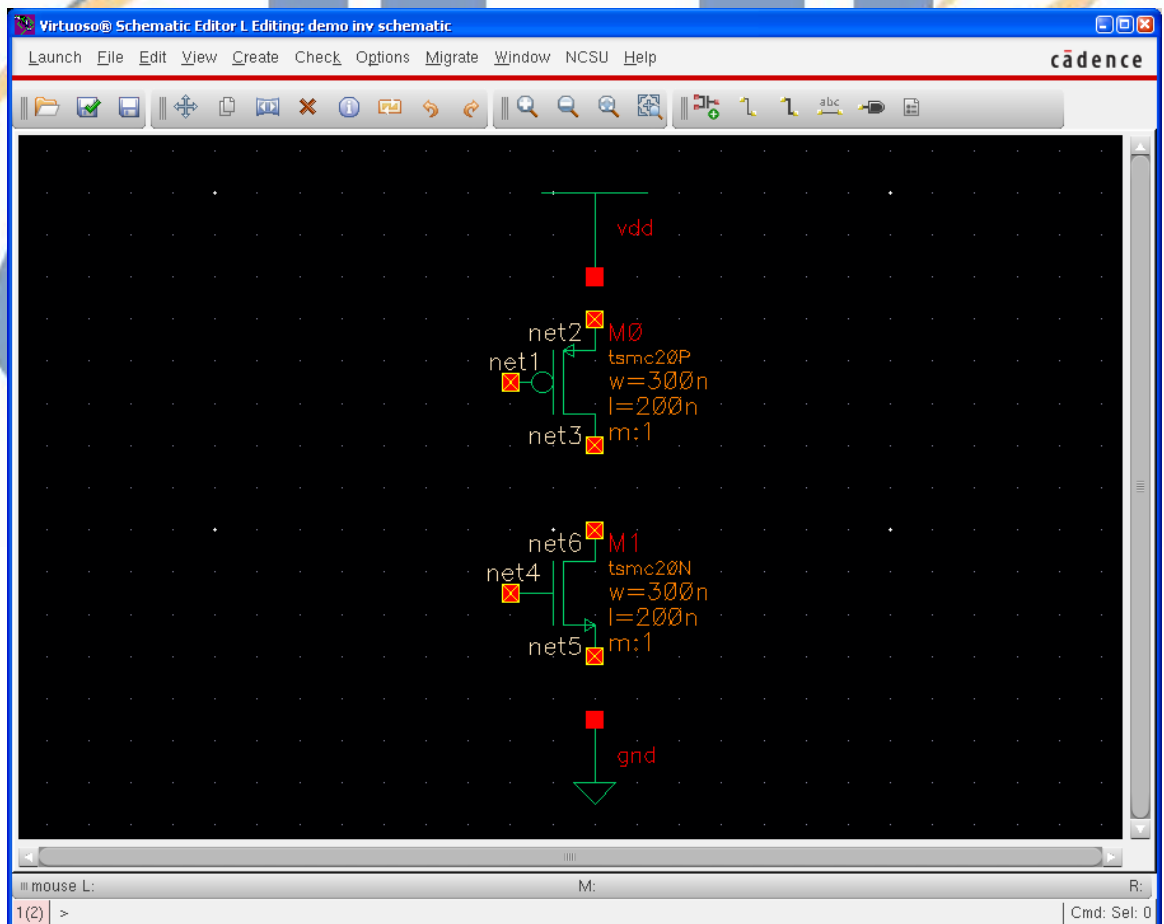
test3

All

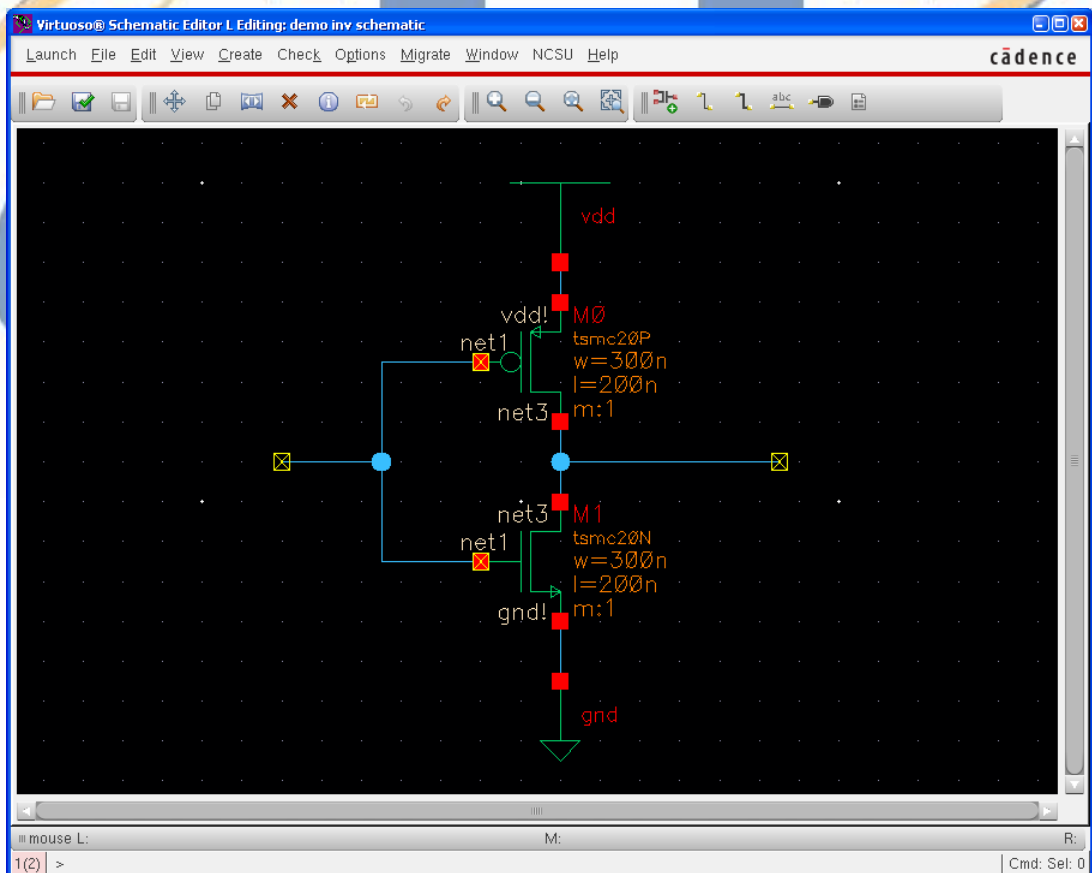
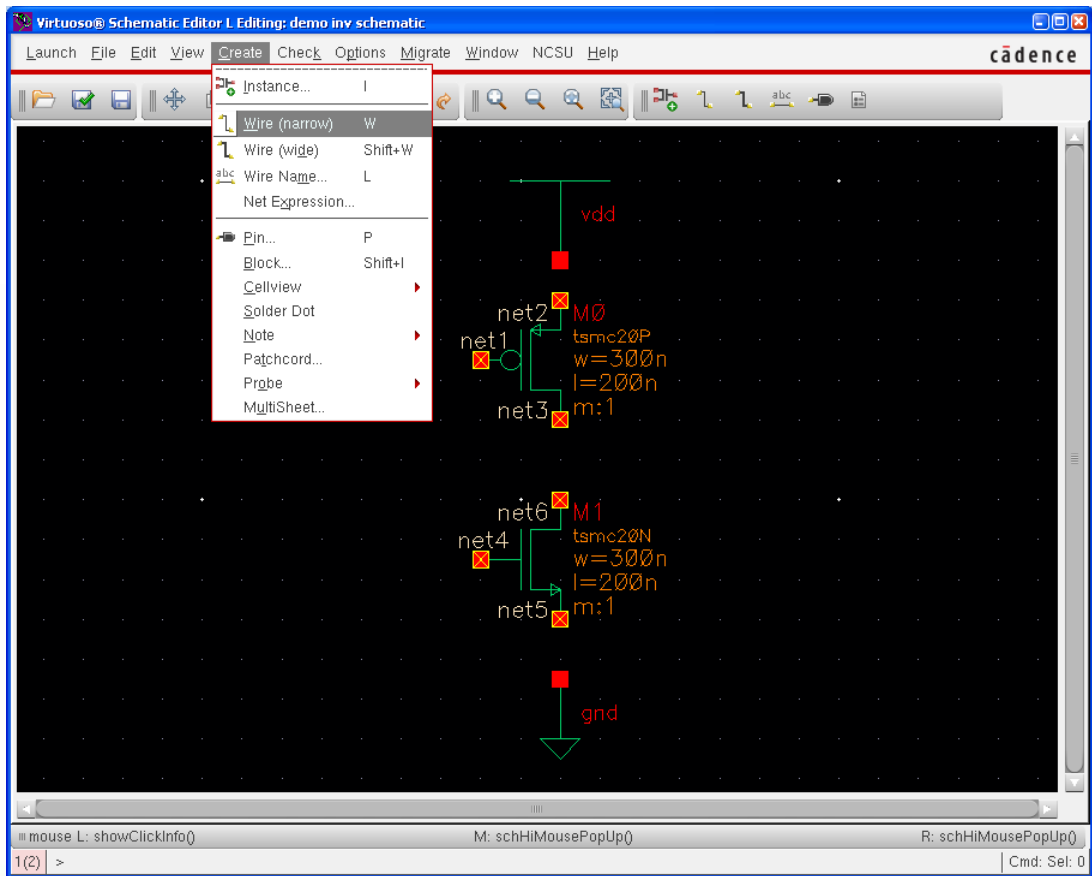
4



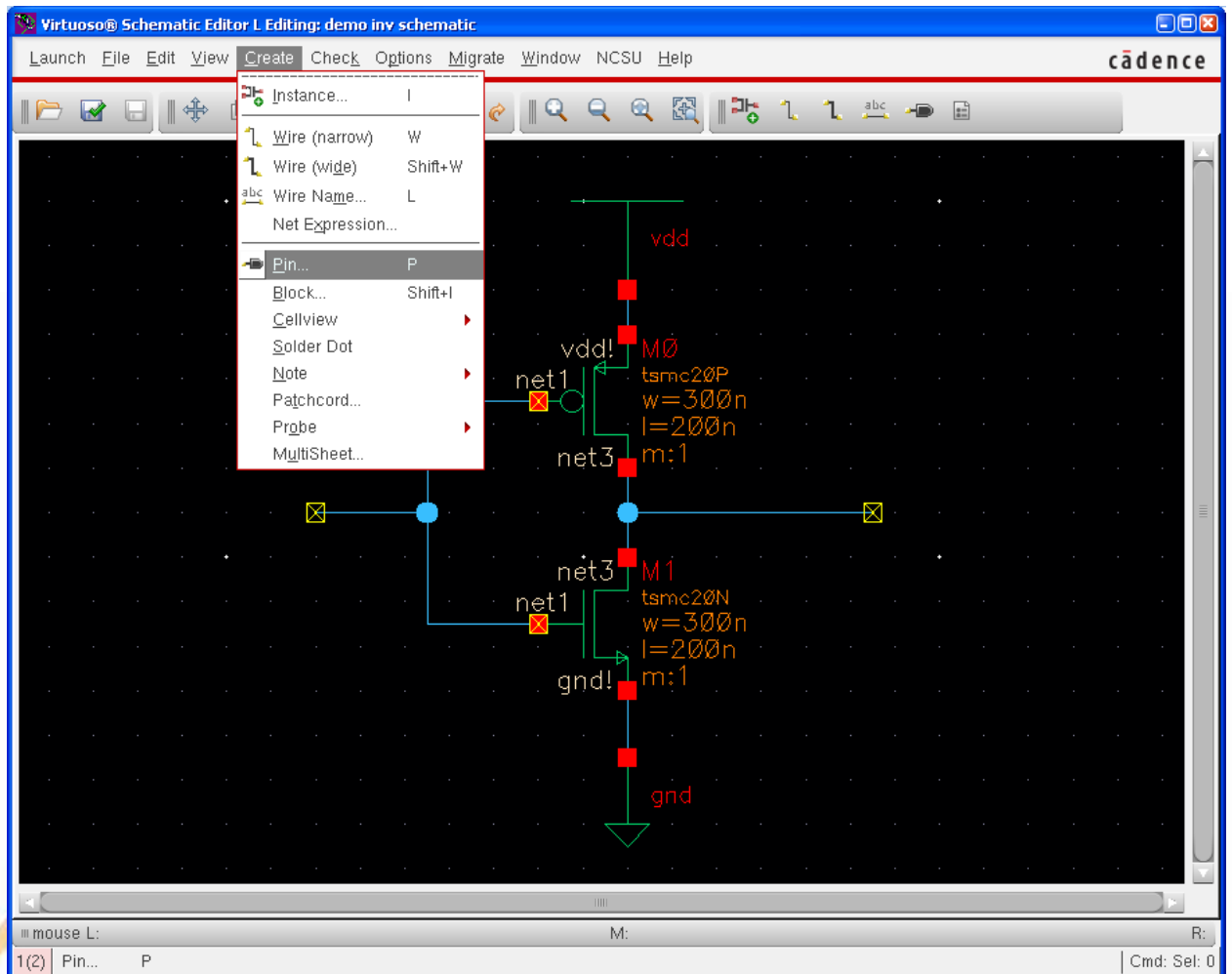
ii. Add instances – nmos, vdd, and gnd



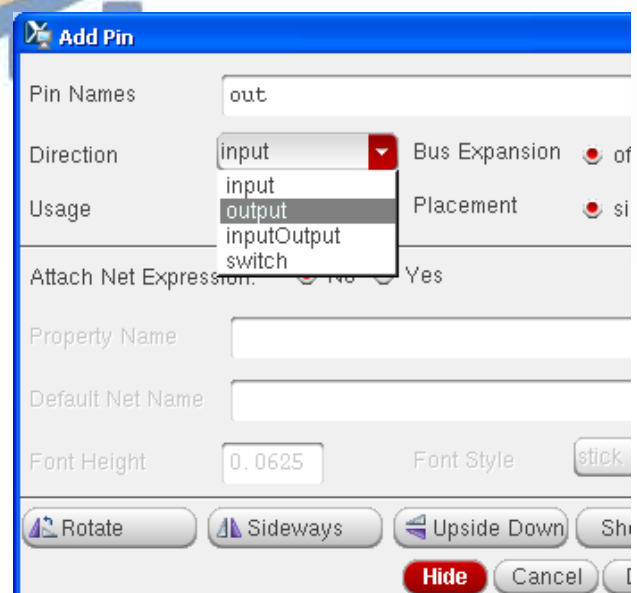
iii. Add wires: Create → Wire

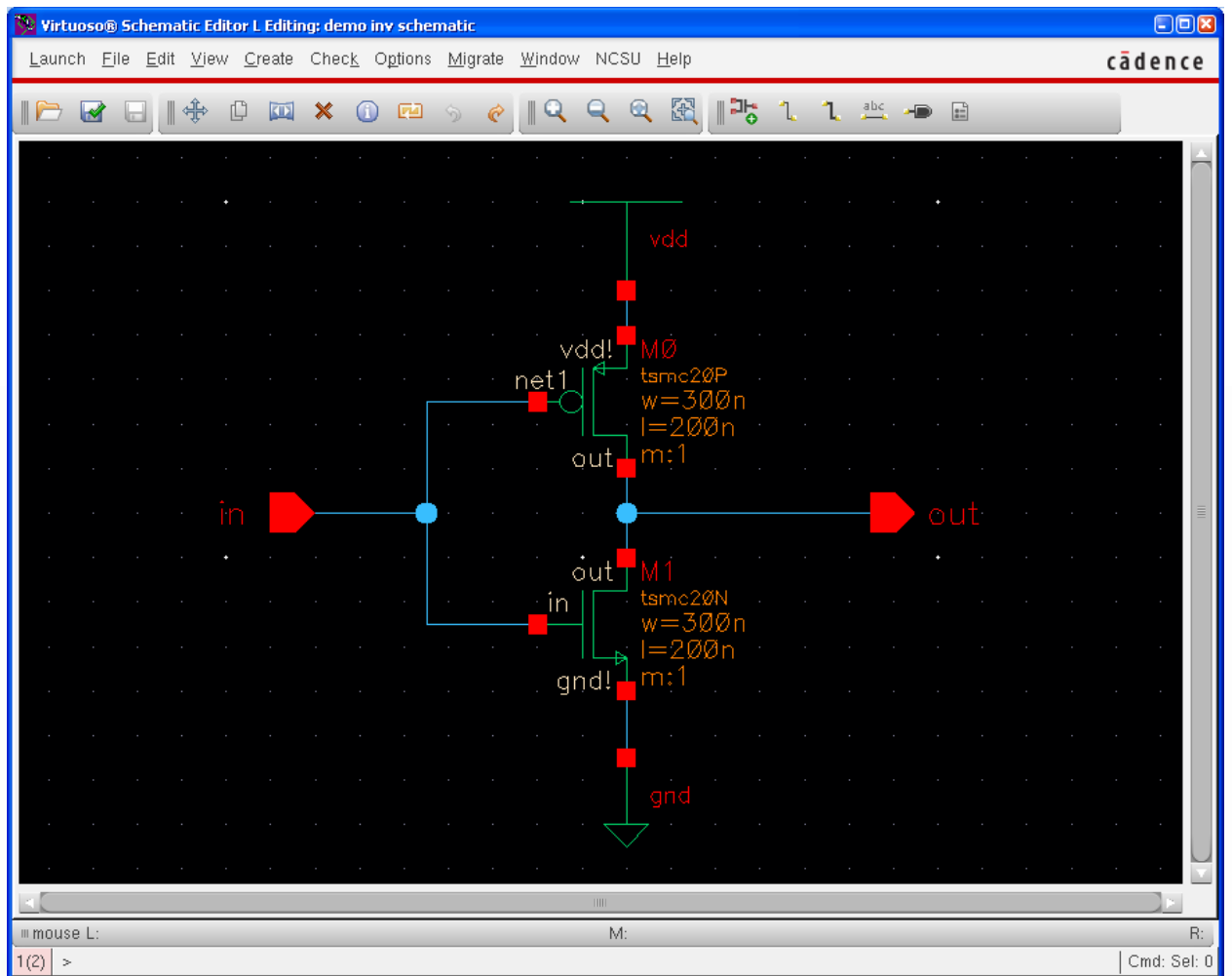


iv. Add pins: Create → Pin



We have for different types of direction. For schematics, we only use two types, input and output. InputOutput type is for supply changes, and it is necessary only for layout. We will discuss about this later.





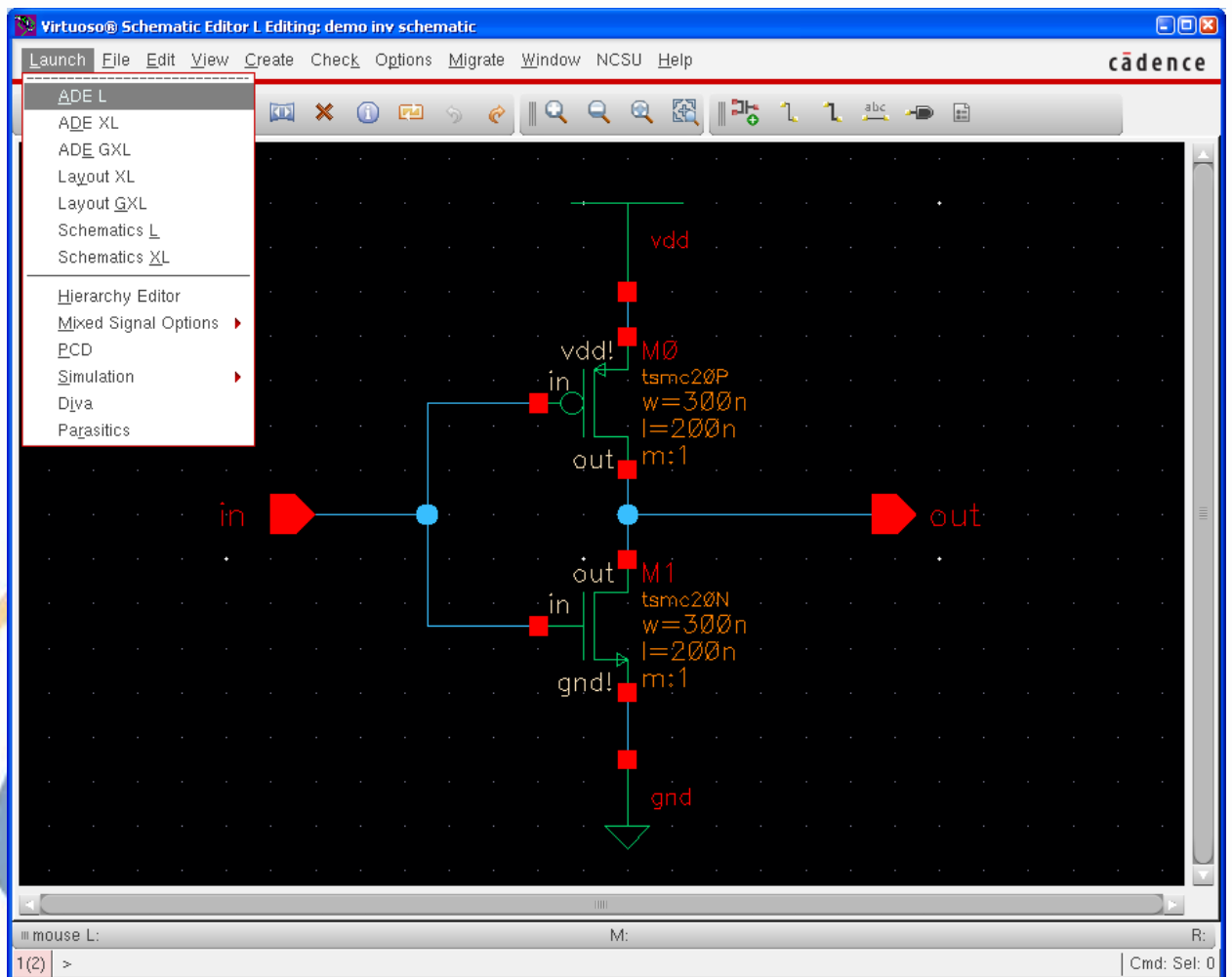
Now, we completed a schematic design. Let's move on the next phase.

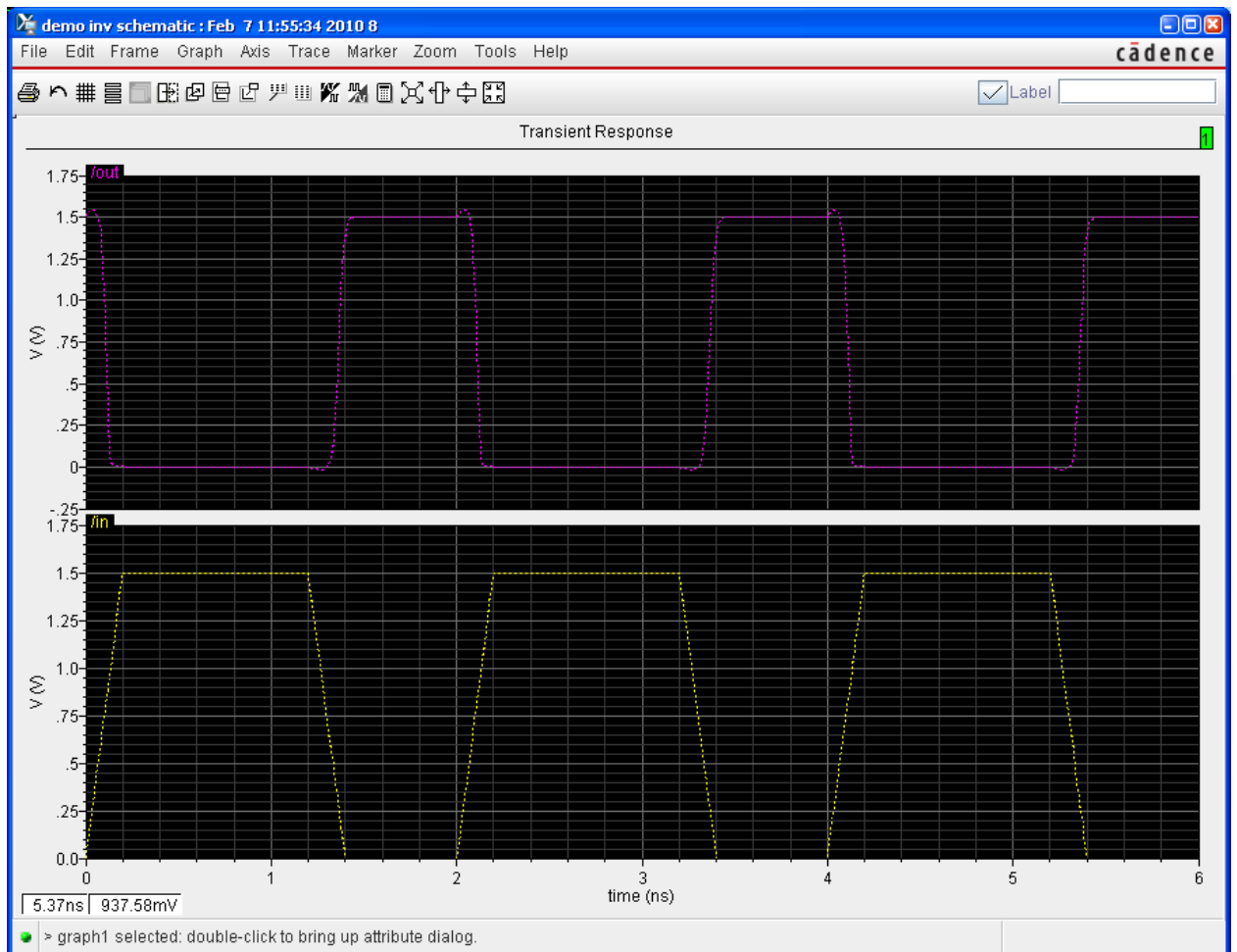
3. Run Spectre simulation

We will run spectre simulation. This section is for **both** schematics and layouts. I will show an example for a schematic. You can do the same thing for a layout.

A. Launch ADE (Analog Design Environment) L

Launch → ADE L





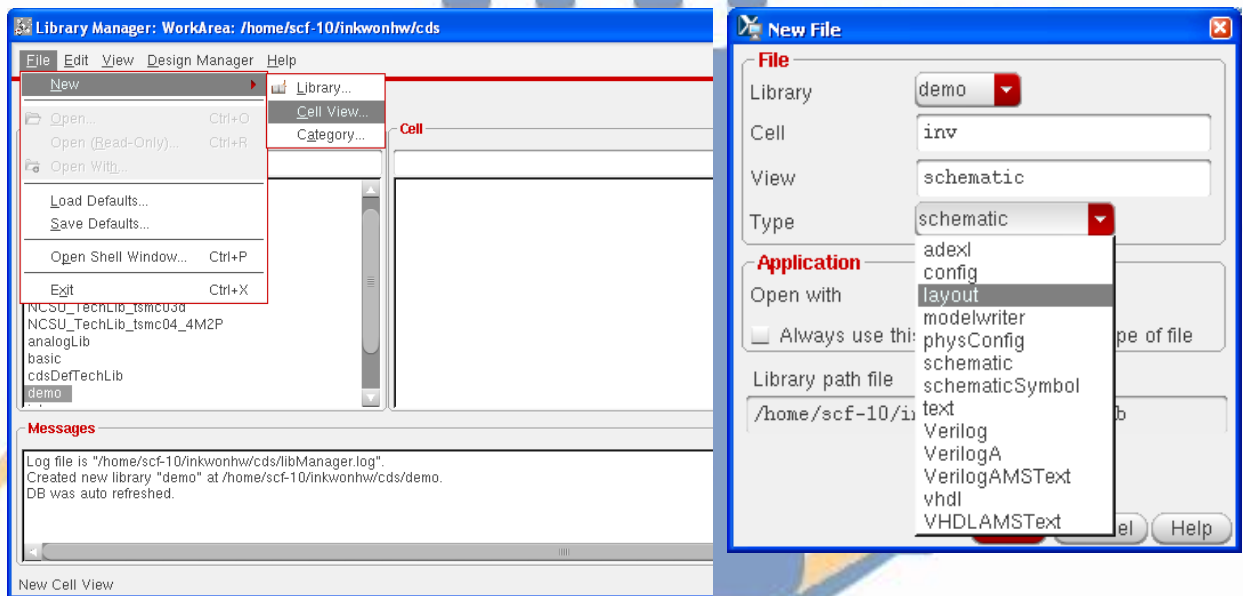
If you see a waveform like above picture, you followed every step properly.
Good job!.

4. Layout

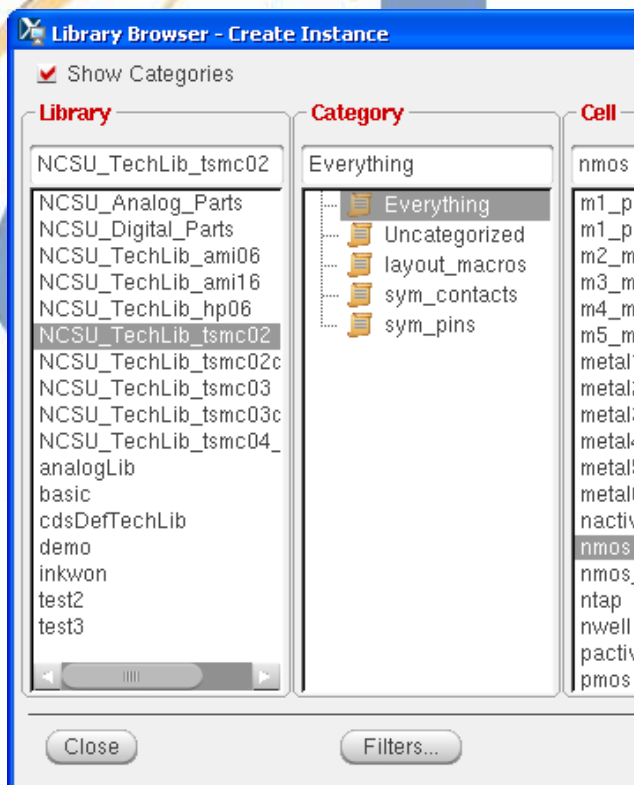
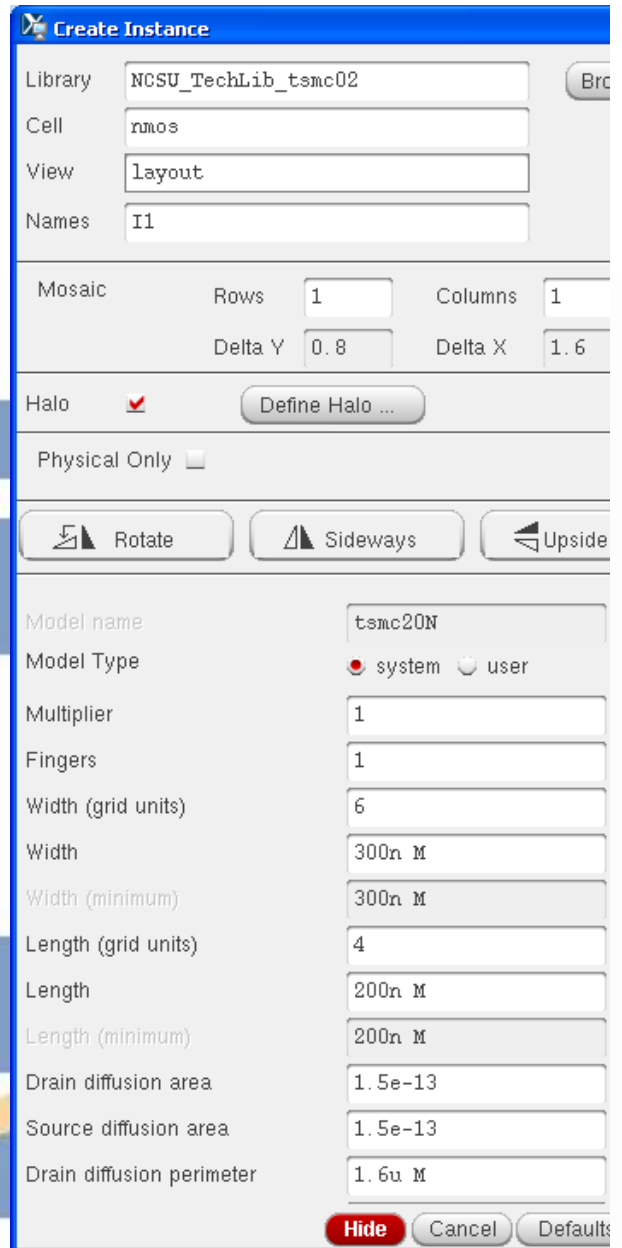
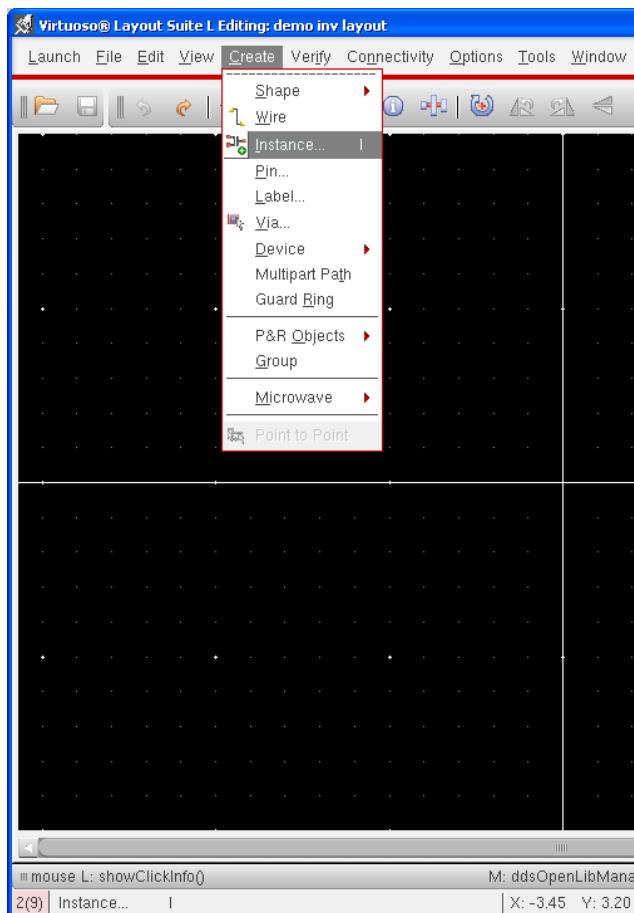
It's time to draw layout. Schematics are for verifying your design very roughly. They don't consider physical features like parasitic capacitances. After determining your design variables by schematics, you need to draw layouts.

Design flow of layouts is very similar to one of schematics, but it has additional step which is LVS check. It is for check if your layout is identical to the schematic or not. Hence, this step is very important. If your logic doesn't pass this step, you may lose significant points for that.

A. Create a layout

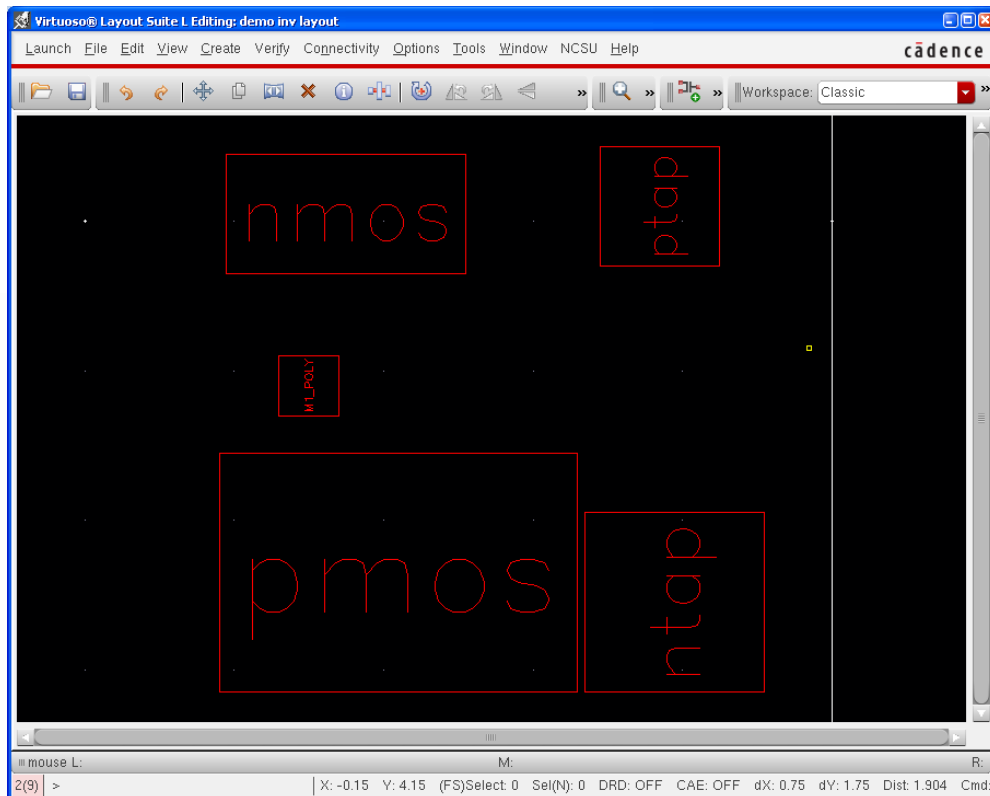


B. Add an instance - nmos

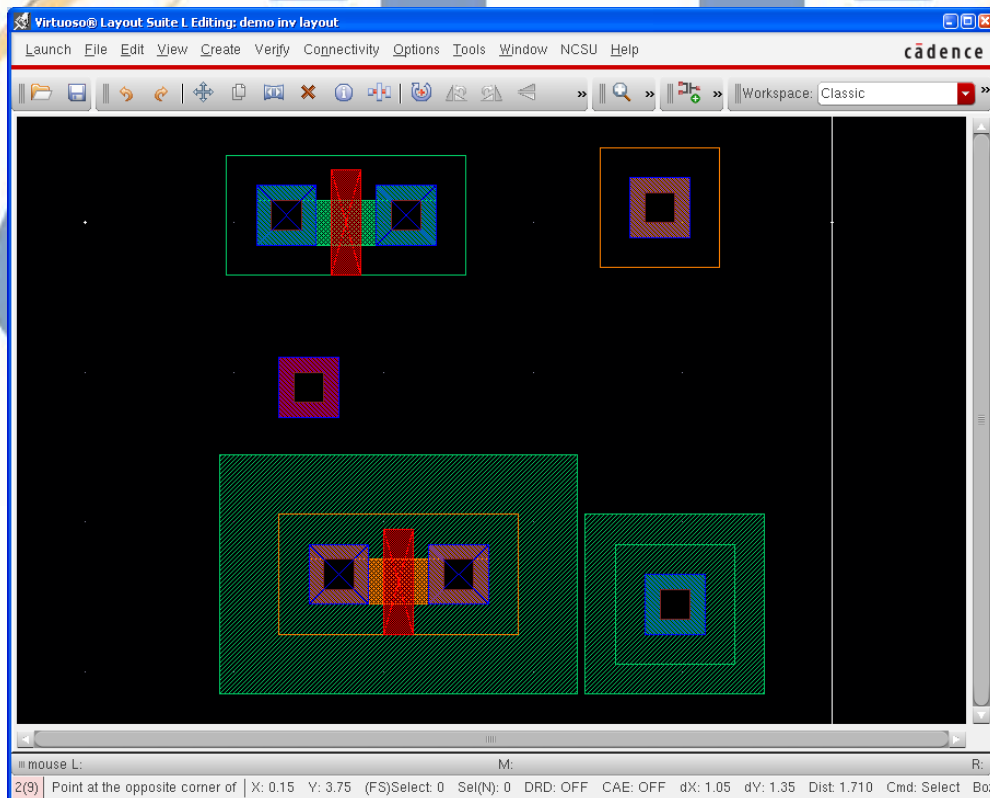


You can modify width of transistors.

C. Add more instances – pmos, ptap, ntap, and m1_ploy



You can select alternate view of a layout. Try 'Shift + f' and 'Ctrl + f'.

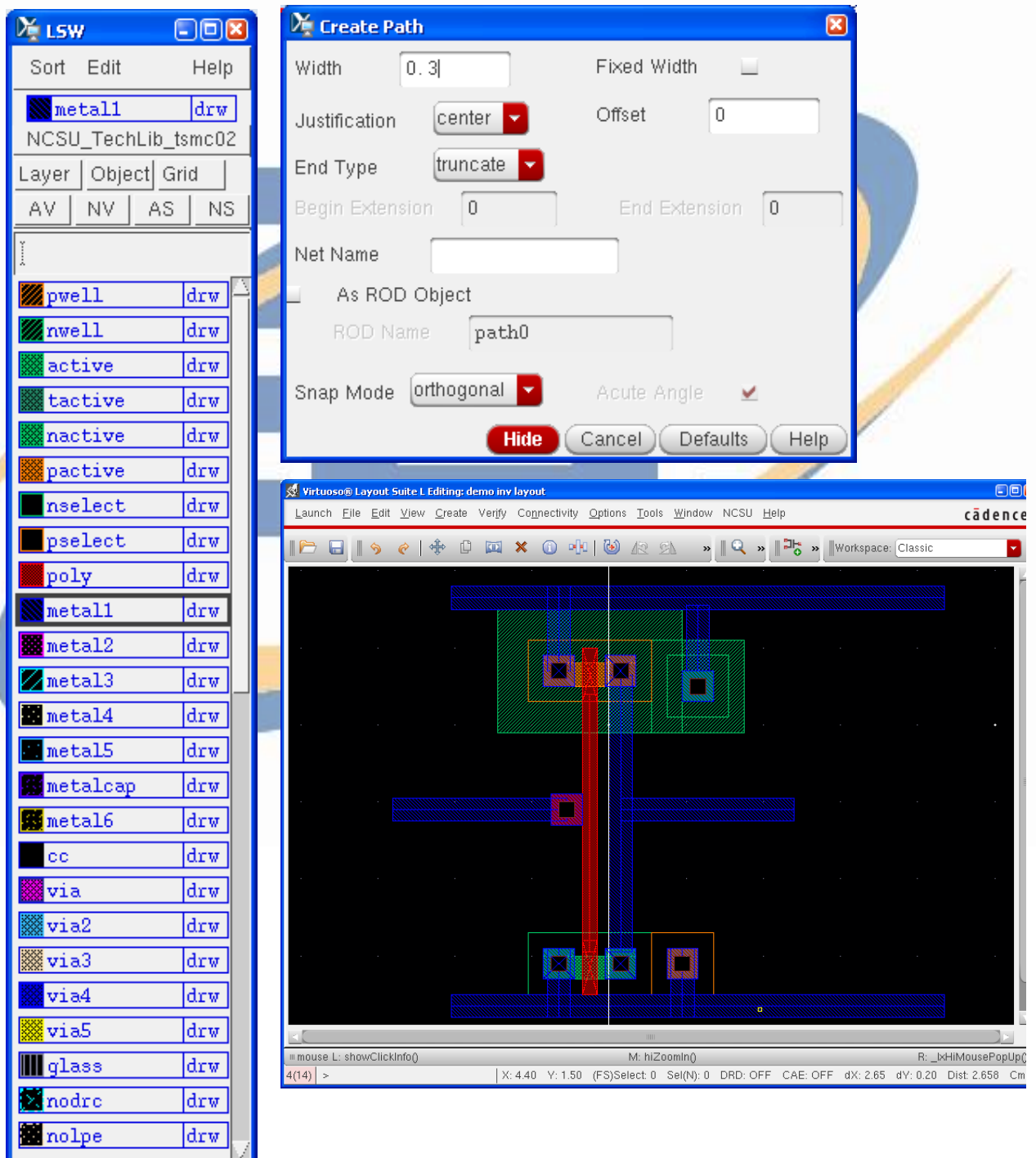


D. Draw metal1

There are few ways for drawing metal, but I recommend you use 'path'. It's quite convenience than others.

Create → Shape → Path

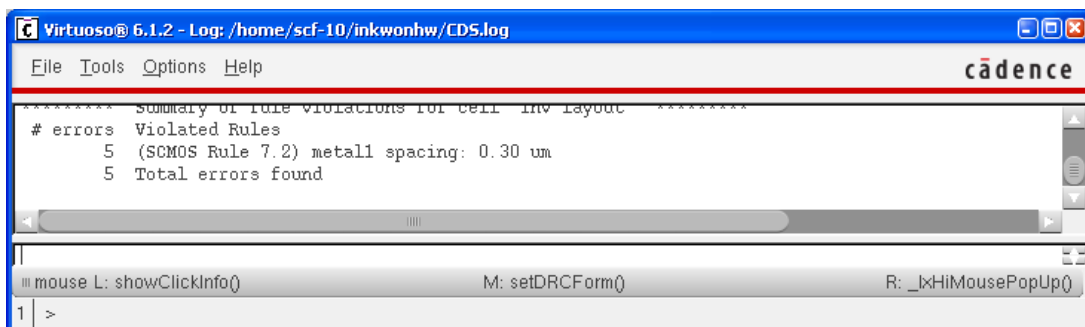
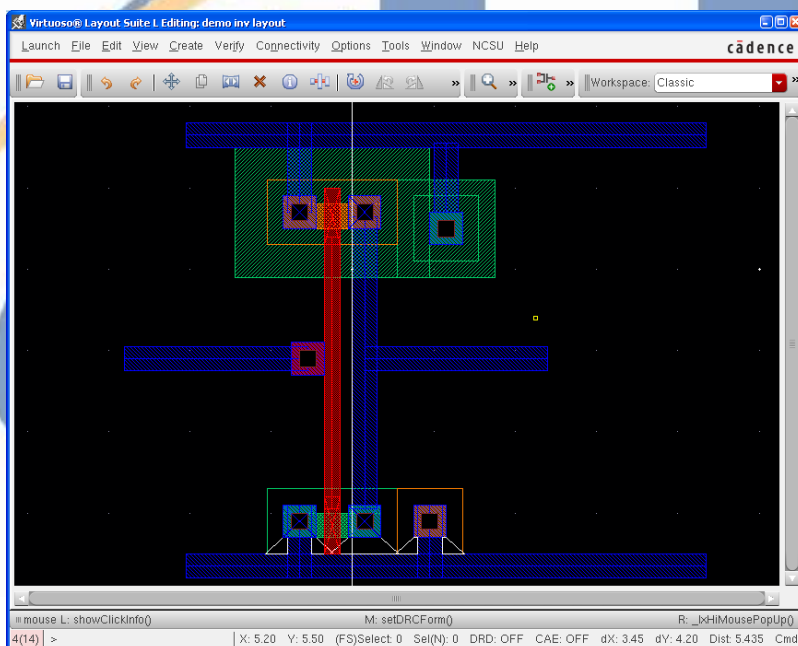
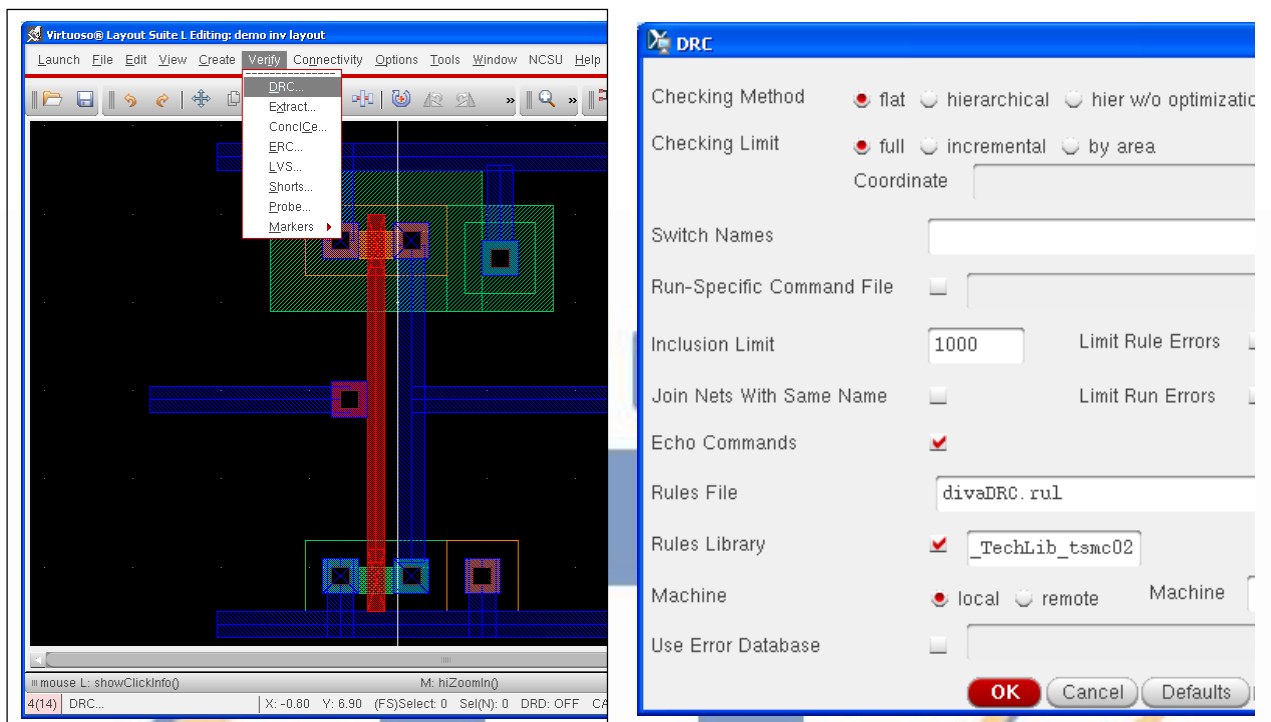
First of all, you should select metal1 on LSW window. Default width for metal1 is 0.3, which means 300nm (3 λ). You can draw metal layer simply by clicking



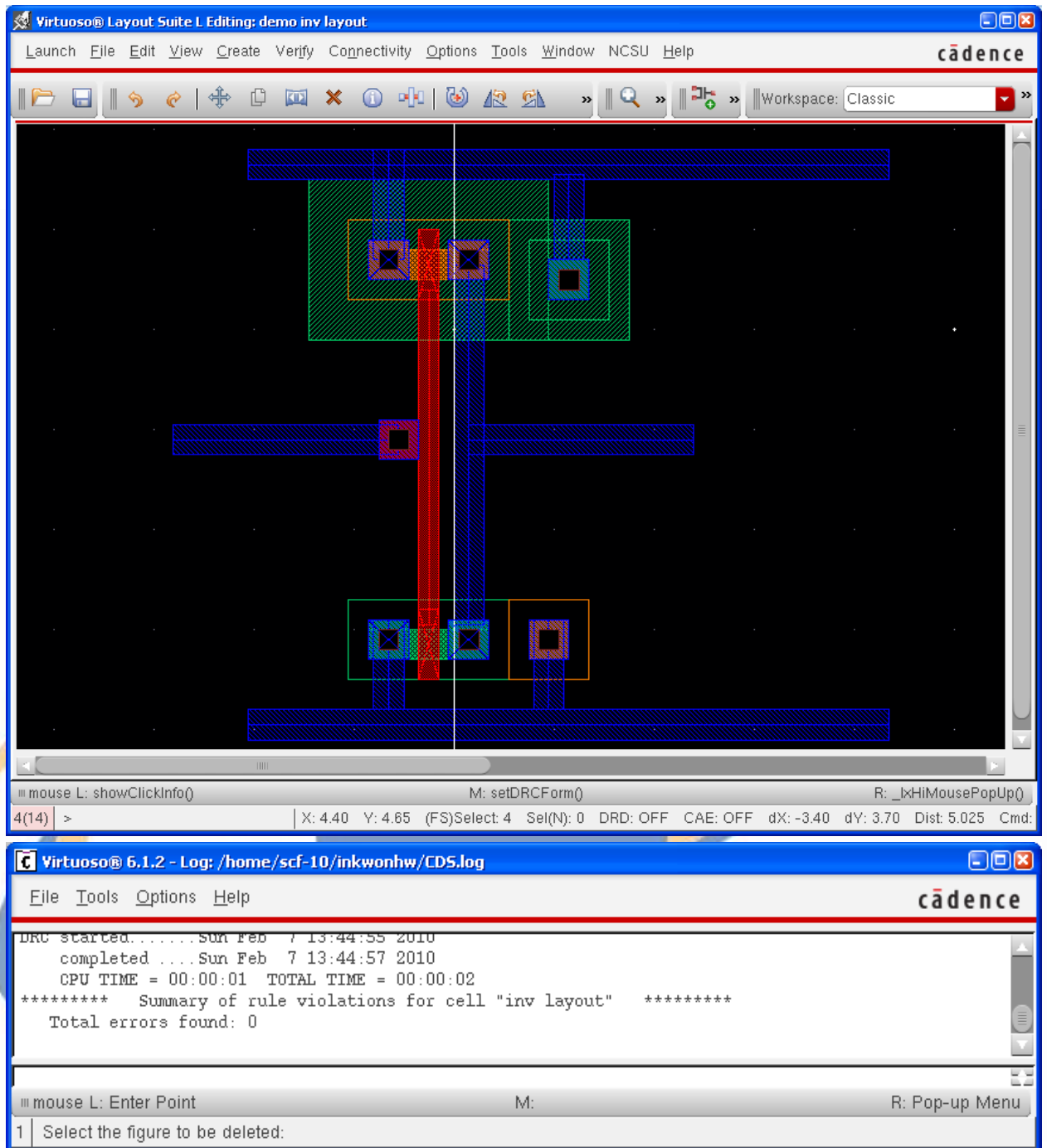
E. Run DRC

This step checks if your layout follows design rules.

Verify → DRC



We have five errors. It is because a gnd metal layer is too close to an nmos transistor.
After modifying layout, run DRC again.

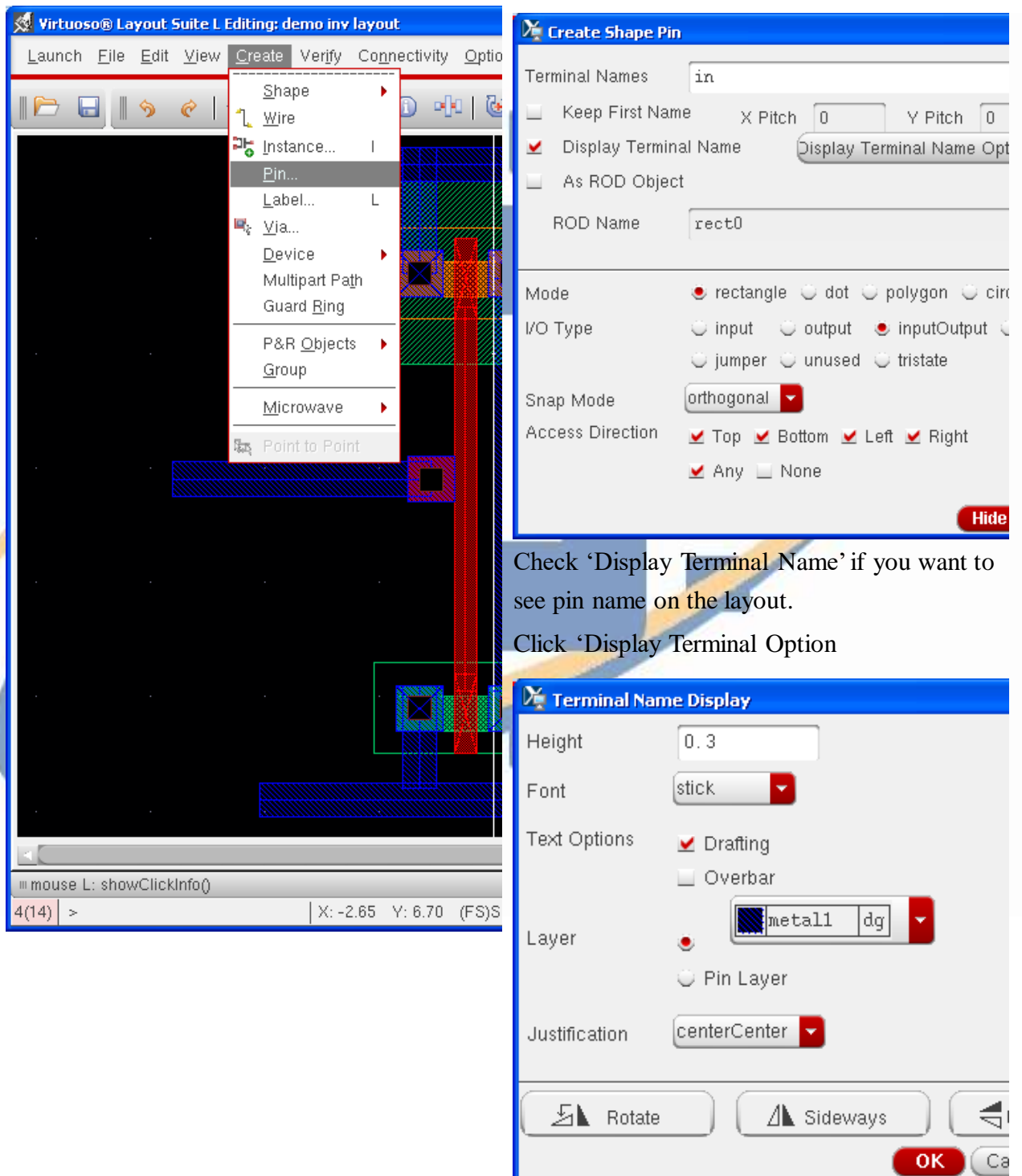


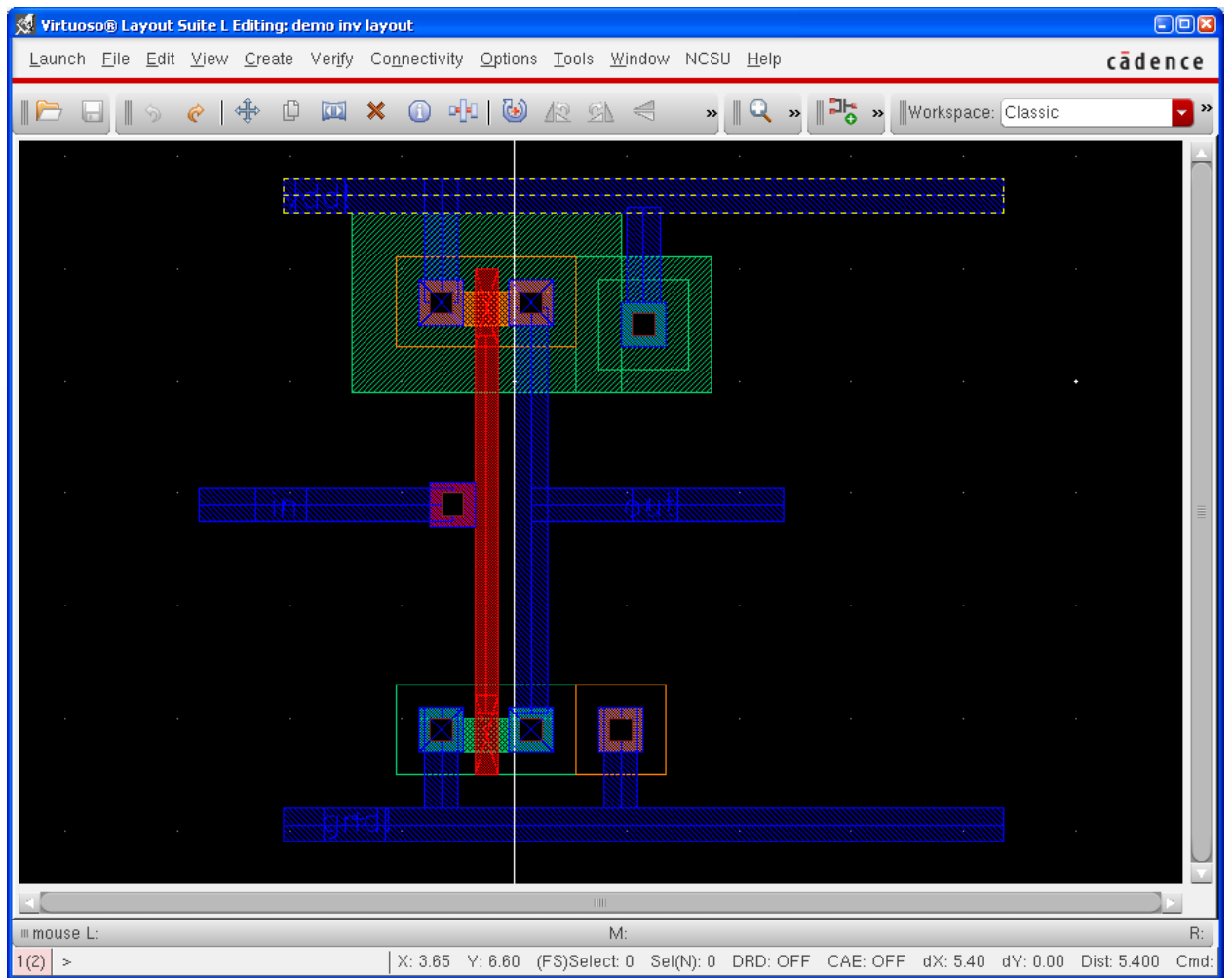
There is no error!!

F. Add pins

We had two pins on a schematic, which are 'in' and 'out'. Pins are for assigning signals to physical device, so we assign voltage level of gnd and vdd by using pins. Hence, we have 4 pins for the layout, which are 'in', 'out', 'gnd!', and 'vdd!'.

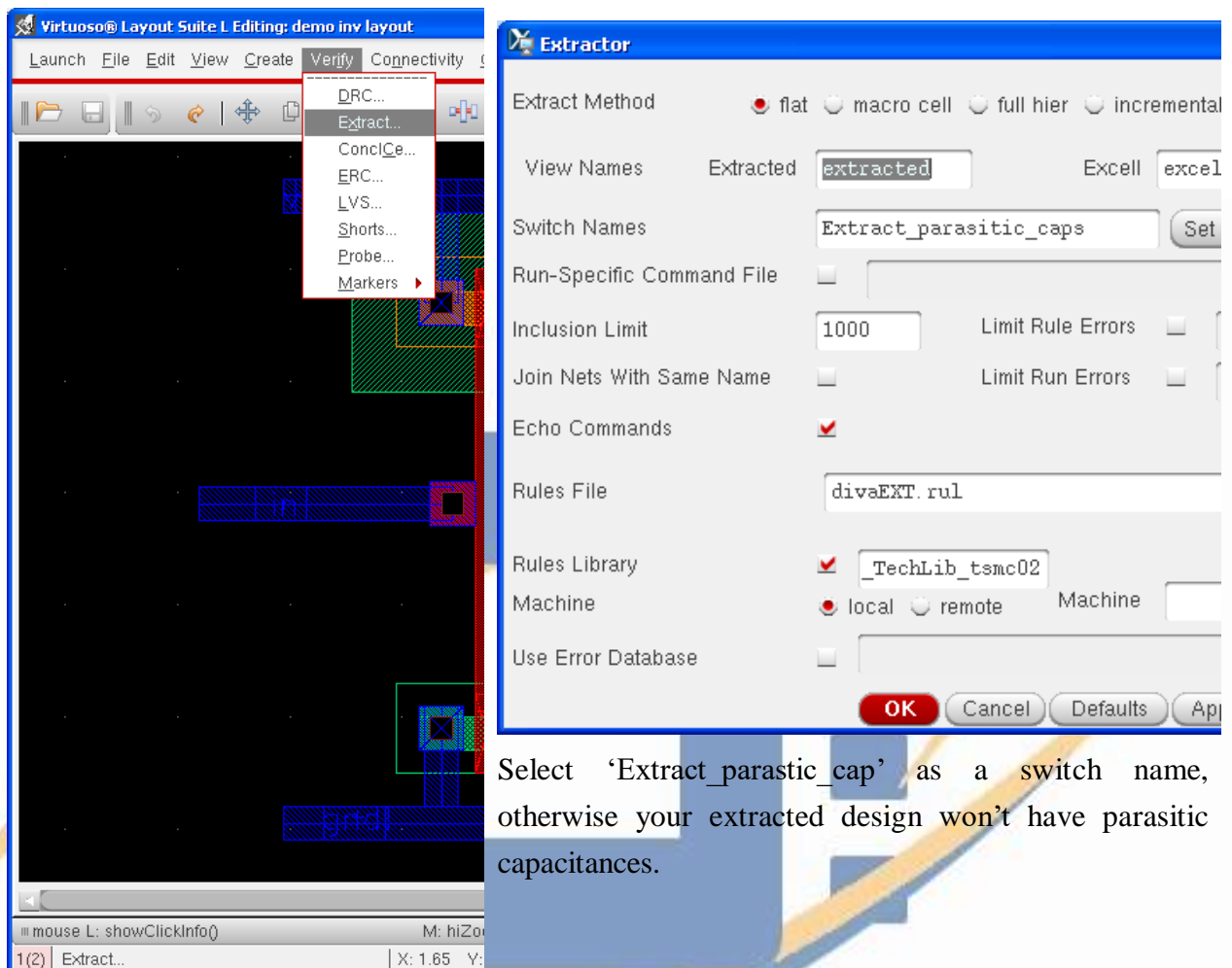
Create → Pin



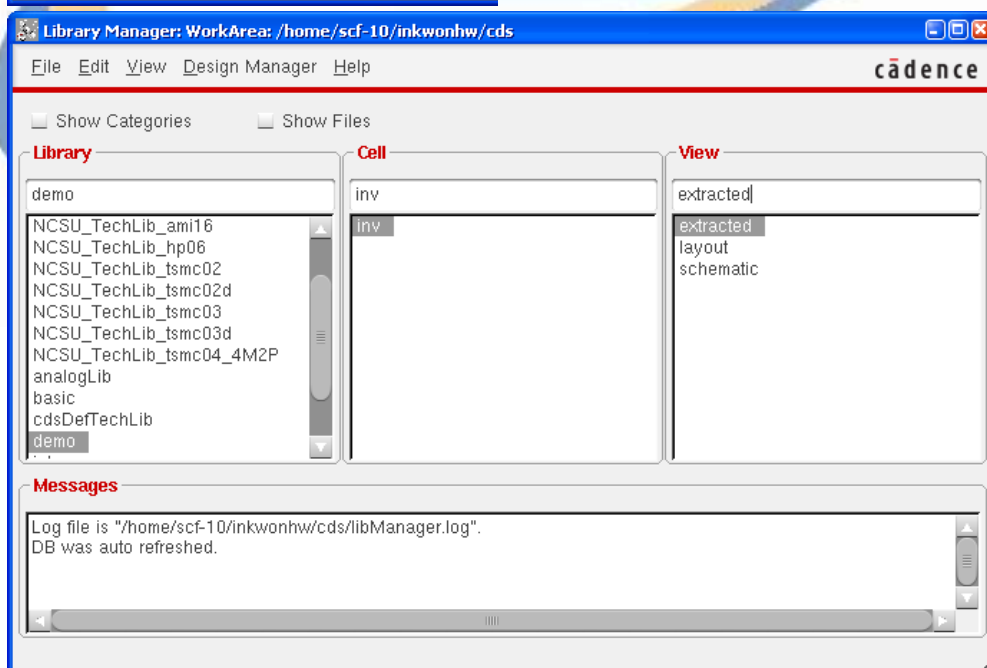


G. Extract

A layout is just a picture. If you need to run simulation using the layout, you should convert it to the other format. It is done by extracting. It's something like compiling a code.



Select 'Extract_parasitic_cap' as a switch name, otherwise your extracted design won't have parasitic capacitances.

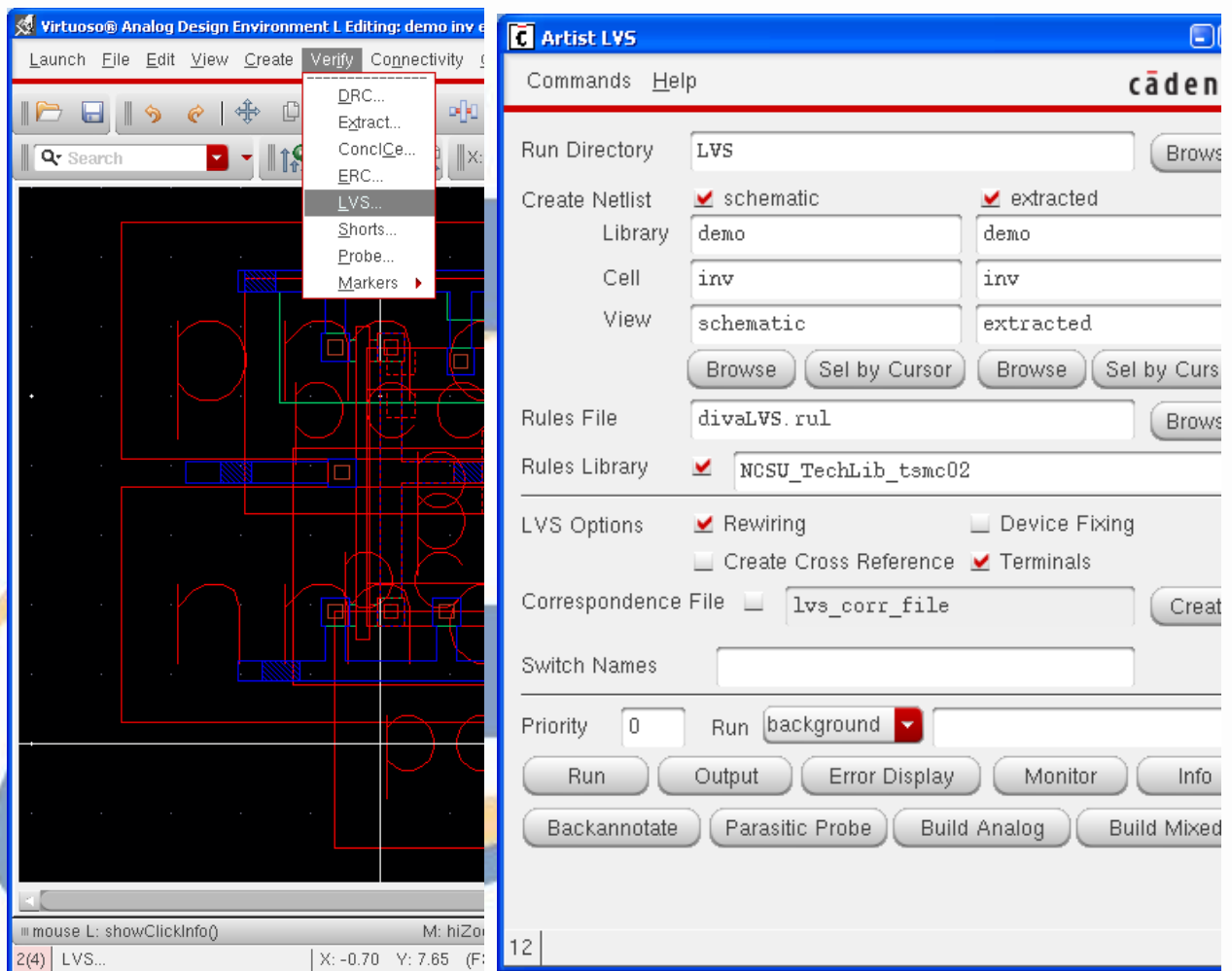


H. Run LVS

As I mentioned before, this step is very important for your grading. More complicated design, more time will be required for debugging LVS.

Verify → LVS

Keep in mind. You **SHOULD** compare your schematic with **EXTRACTED**.



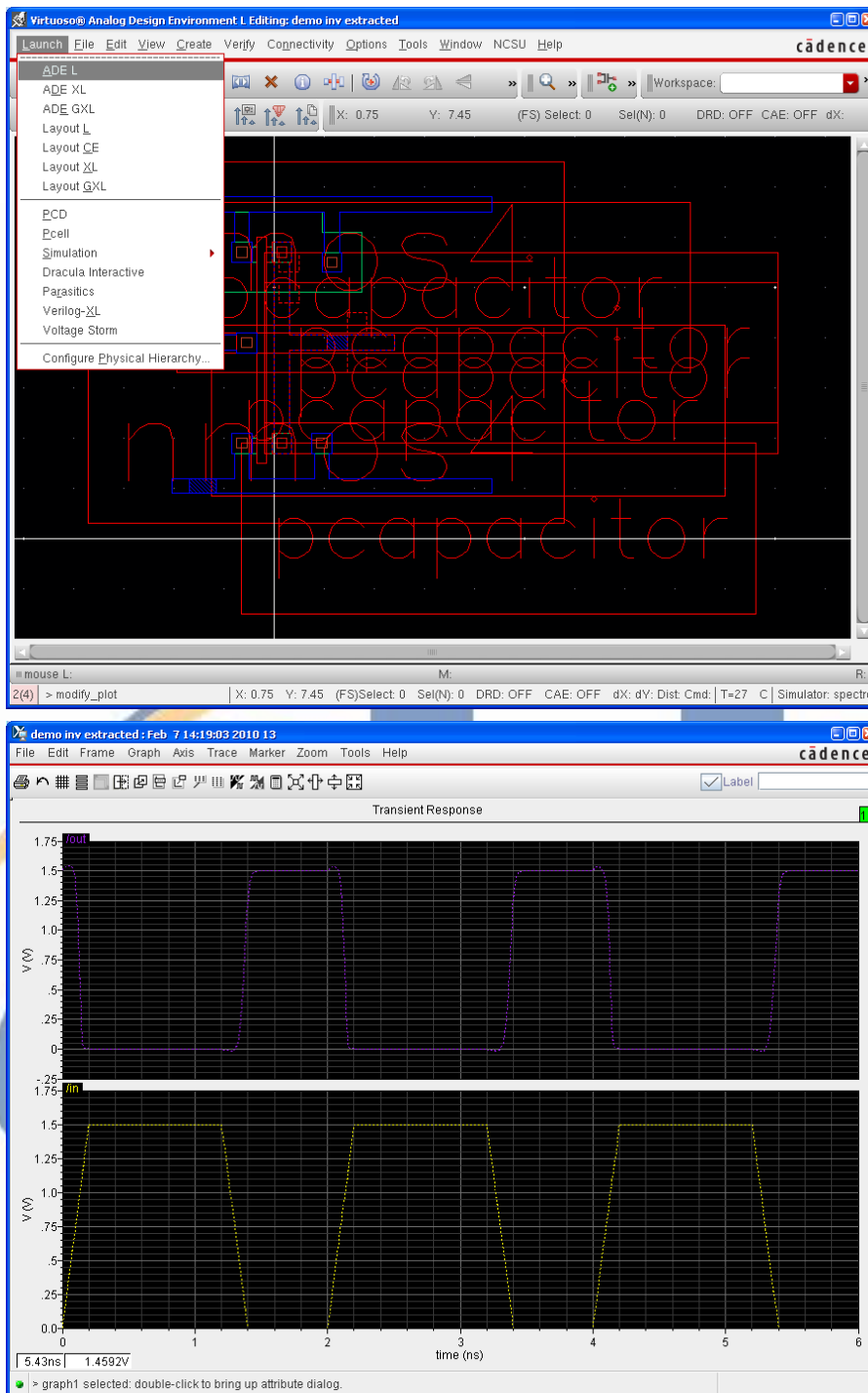
I hope all of you will see the following window.



If there are errors, you can check the results by clicking 'Output' button. 'Error Display' also might be helpful.

I. Run Spectre simulation

It is same as schematics. Go to step '4. Run Spectre simulation'.



Congratulations!!

You followed all steps I prepared. Let's do the same thing for more complicated designs.