Composer Tutorial

Outline

- Full custom IC Design Flow

- Composer Tutorial
    Making library and cells
    Drawing schematics –an inverter example
    Opening existing cells
    Creating symbol

Full Custom IC Design Flow

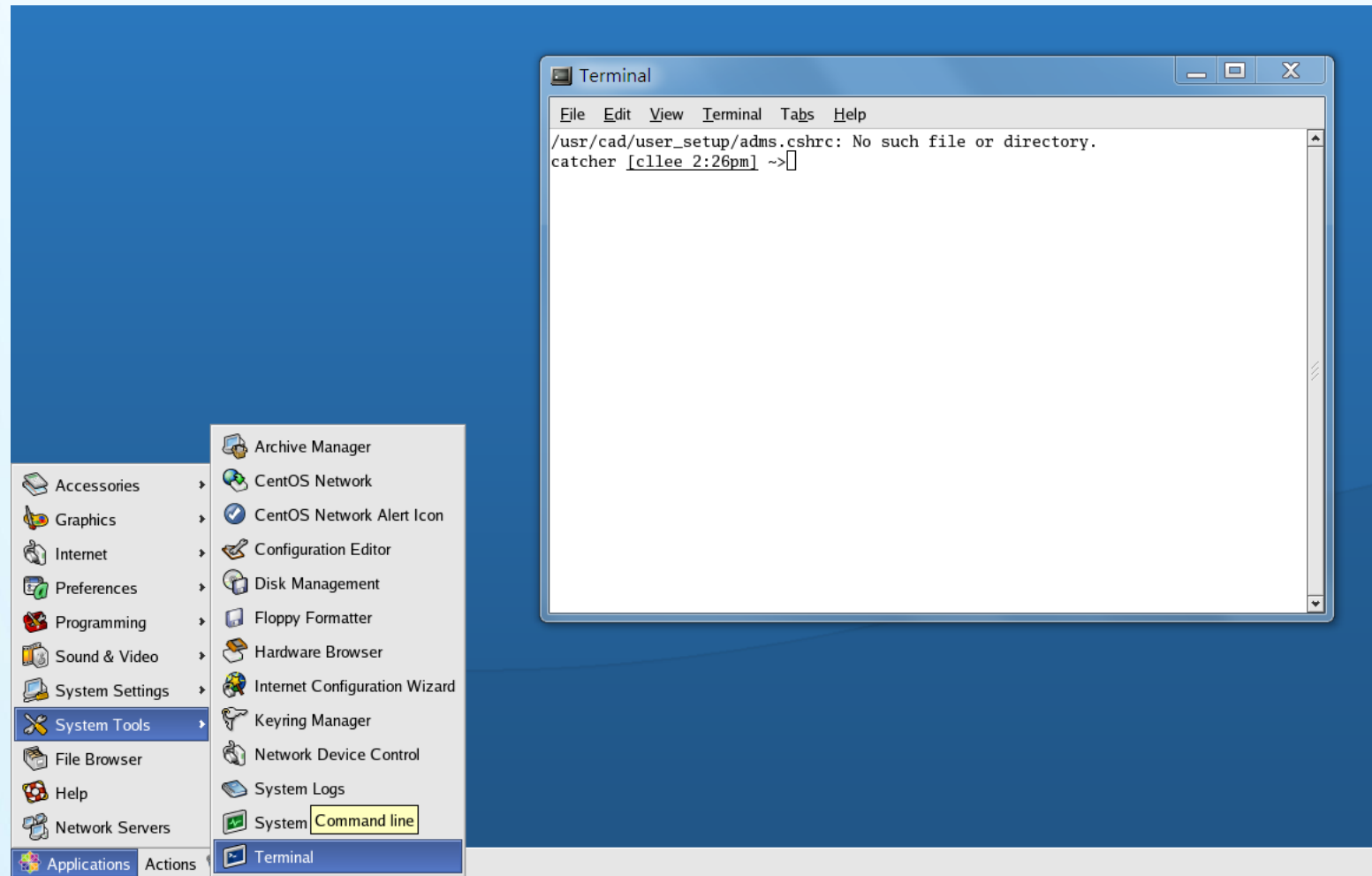|	
Circuit Netlist/ Schematic	*Composer*, Text Editor
Pre-layout simulation	Hspice, Spectre, Eldo...
Layout	Virtuoso, Laker
DRC, LVS	Calibre
Parasitic Extraction	Calibre
Post-layout simulation	Hspice, Spectre, Eldo...

Open the Terminal Environment

Start Cadence Environment

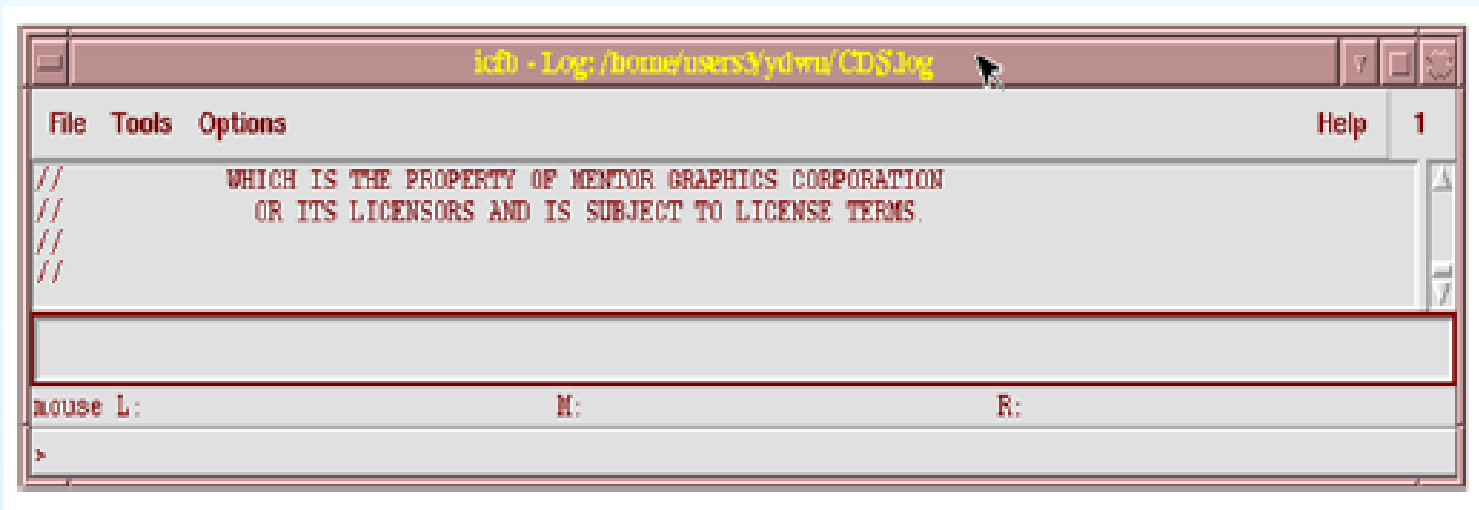Invoking *icfb &* in terminal

```
bigbird:~(m9761571)%
bigbird:~(m9761571)%telnet ws2
Trying 140.114.24.112...
```

```
[m9761571@ws2 ~]$ mkdir vlsi_composer
[m9761571@ws2 ~]$ cd vlsi_composer/
[m9761571@ws2 ~/vlsi_composer]$ icfb &
```
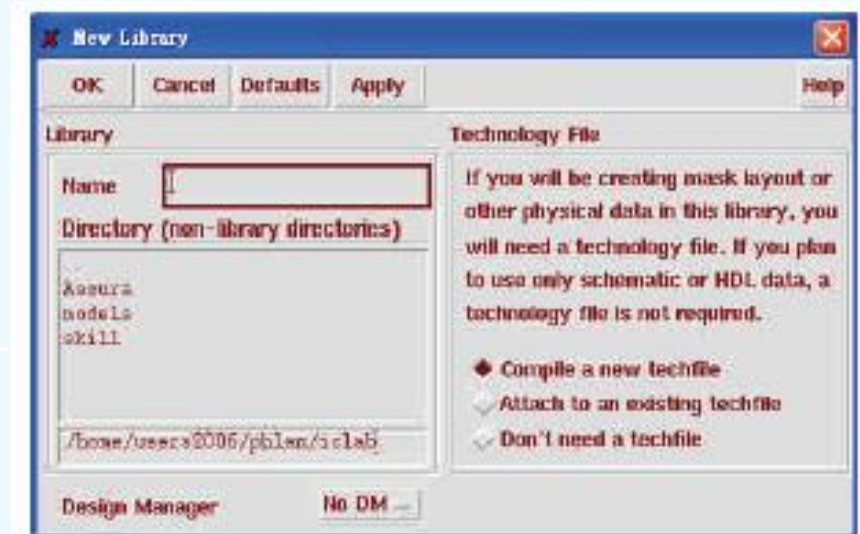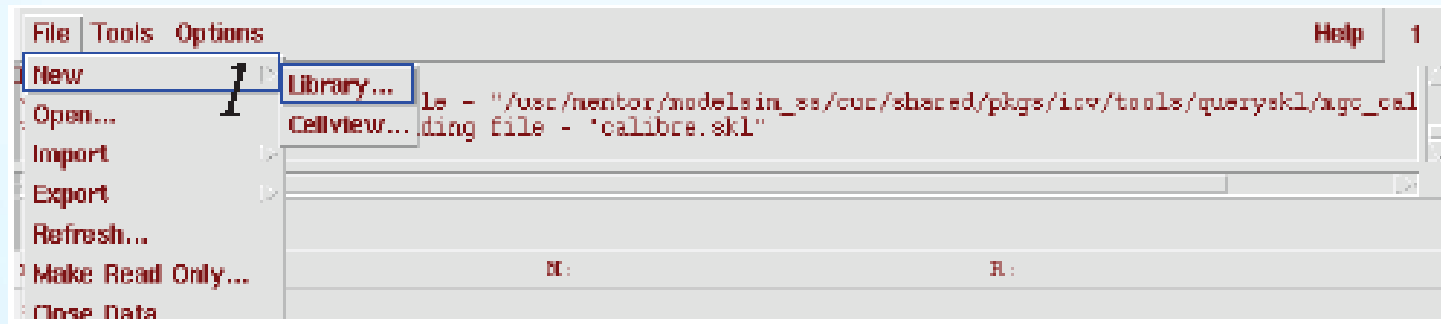
Main Window

After successfully checking license, the main window shows

Making library

From main window, *File → New → Library…*
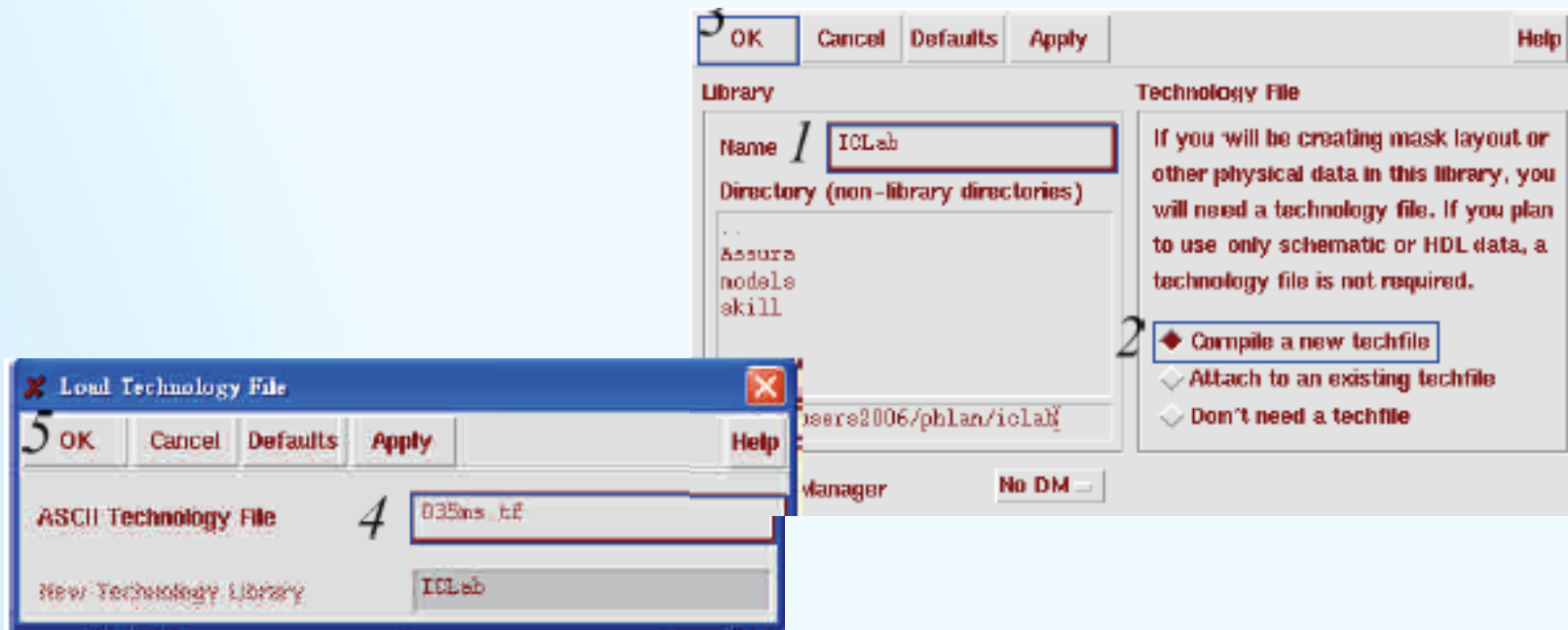A new window called *New Library* pops up

48um

Making Libraries

Enter the library name in the **Name** field
If you have techfile (018ms.tf), you can choose the option of
Compile a new techfile option, and press **OK.**
or , you choose the option "**Don't need a techfile**" .
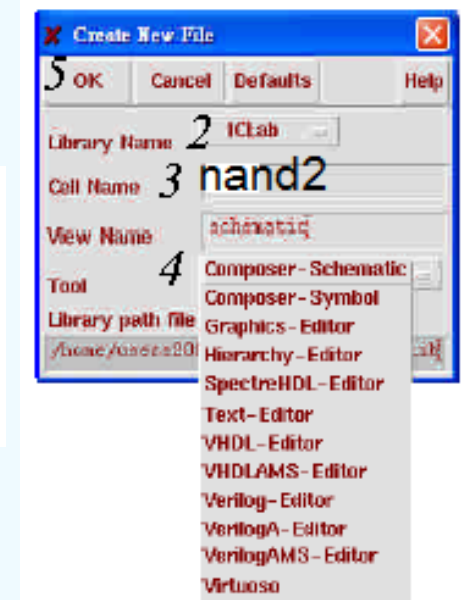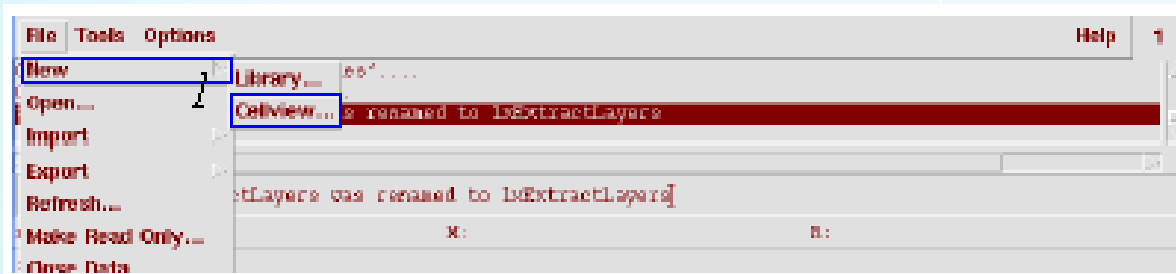
Making Cells

From main window, **File → New → Cellview…**
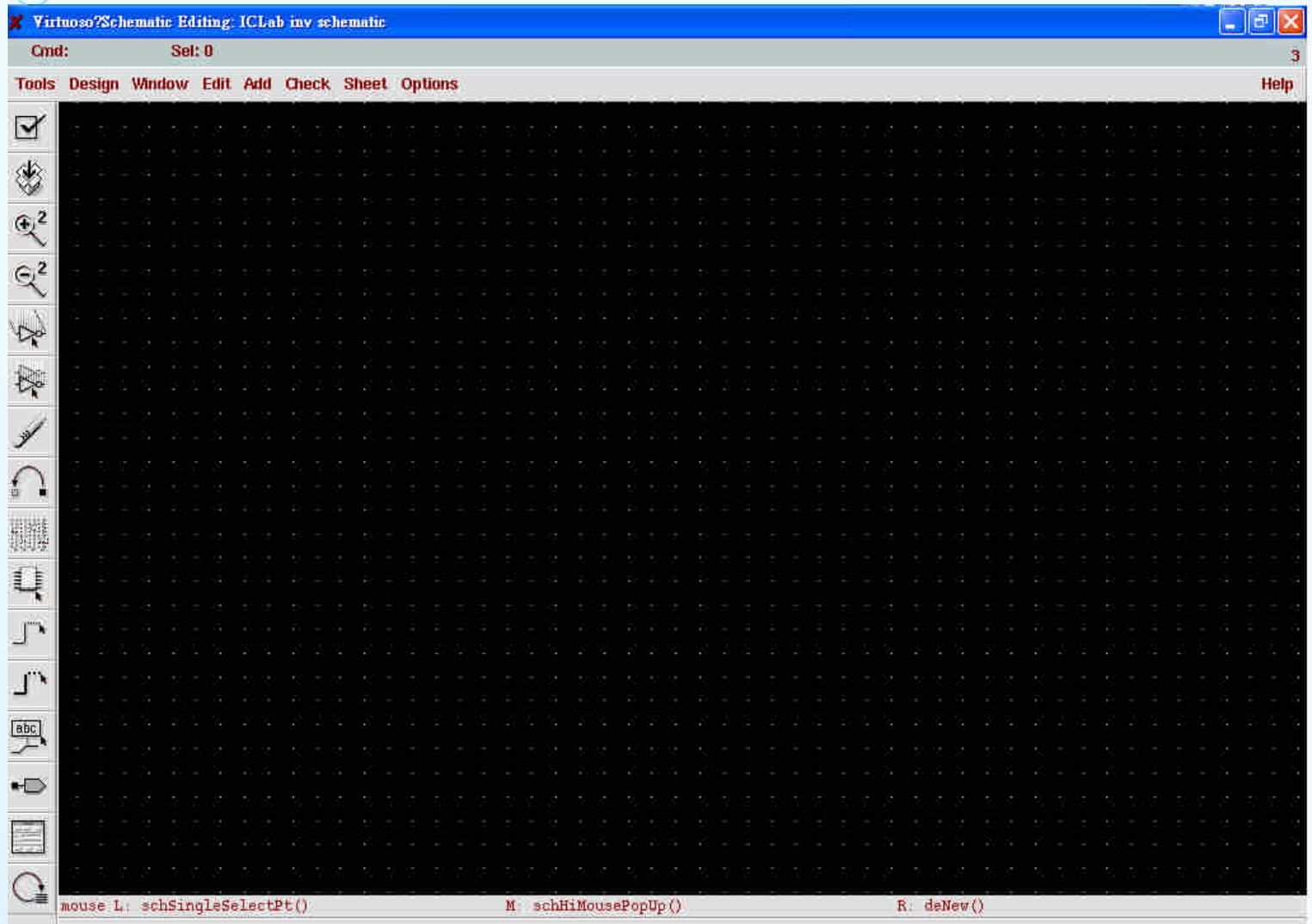Choose the Library at which the cell is created
Enter the desired name in the **Cell Name** field (nand2)
Choose **Composer-Schematic** in the Tool field
Press **OK**

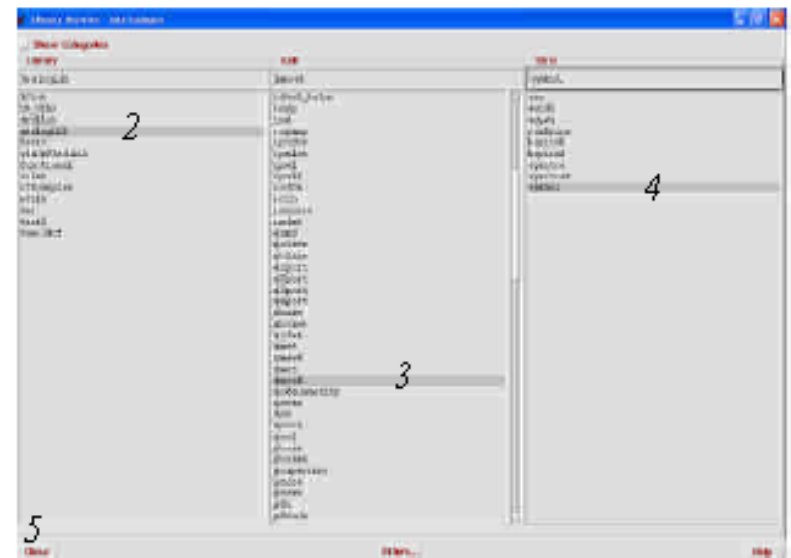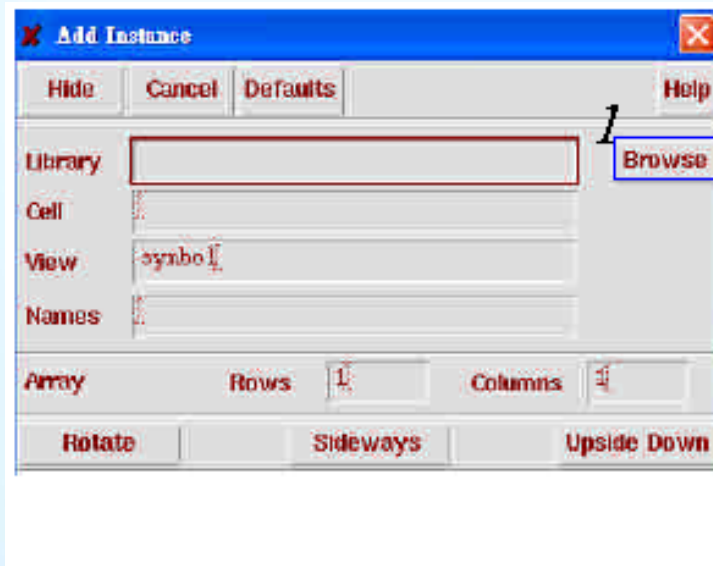# Schematic Editing Window

Inserting Symbols

Press key "**i**"or the instance button from the toolbar

Press "***Browse***"from the popped window

Pick the desired cell from library manager

Choose ***symbol*** view for schematic editing, and press ***close***

analogLib → nmos4 →symbol

Editing Properties of Symbol

After closing the library manager, the corresponding library and cell name appears in the popped window
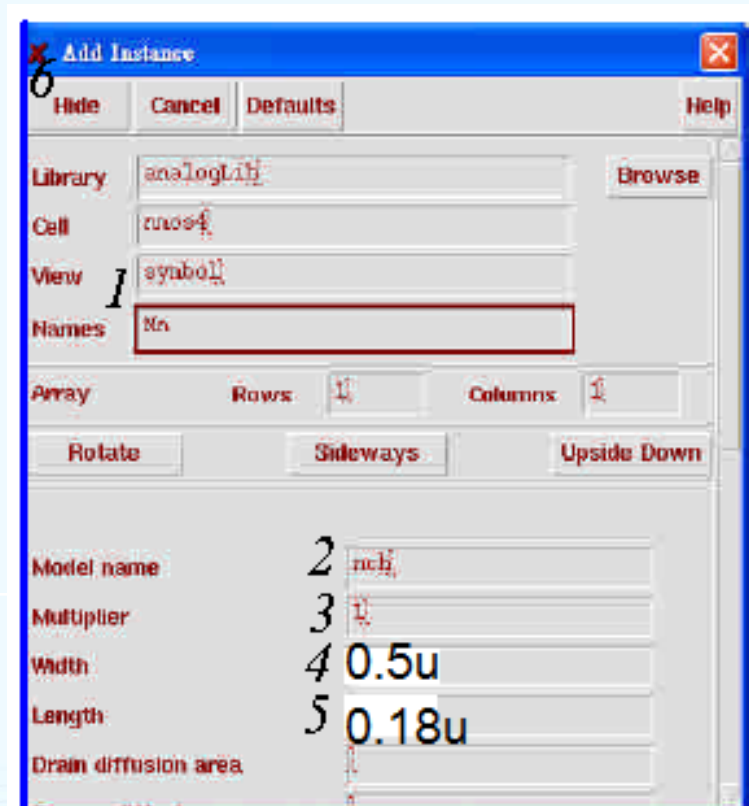Fill some properties for the symbol

Name: Mn
Model Name: nch
Multiplier: 1
Width: 0.5u
Length:0.18u
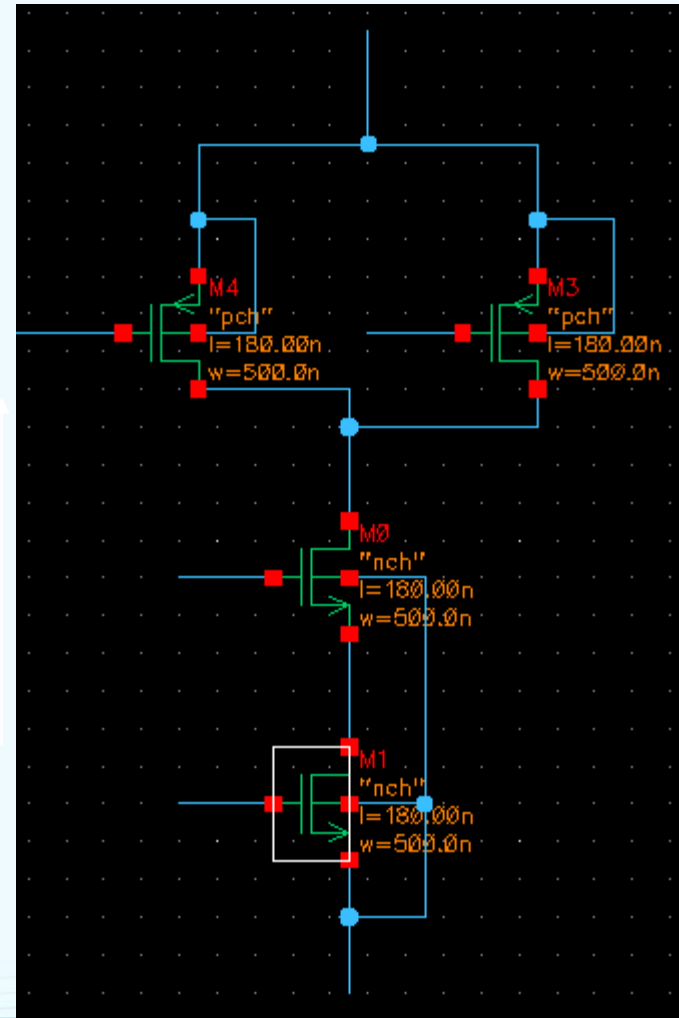
Press *Hide* Button

48um

Adding and Connecting

Insert nmos4, pmos4, symbols from the analogLib library

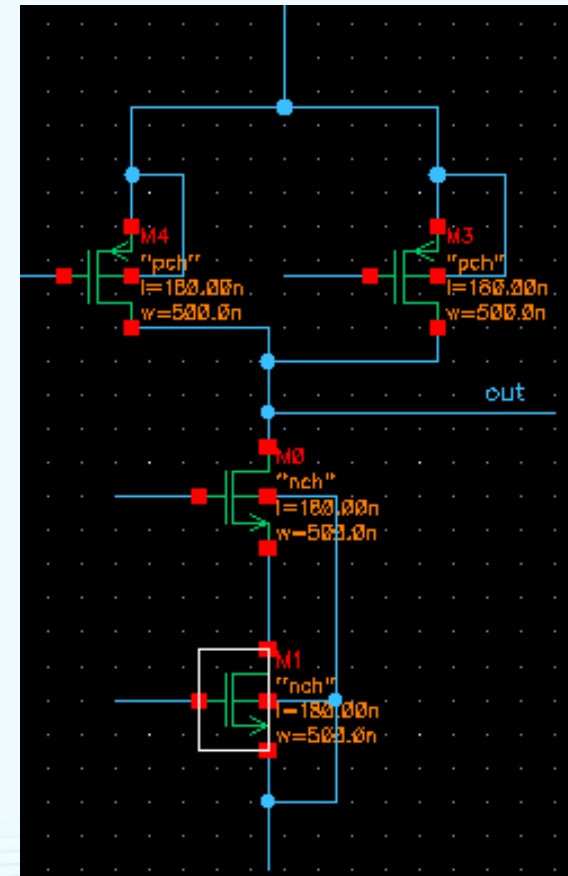nmos4: 0.5u/0/18u/1
pmos4: 0.5u/0.18u/1
vdc: DC voltage=3.3V
gnd
Press key "*w*"to place wire

Adding Net Names

Press key *"I"*(Label) and keyin the corresponding node name in the popped window
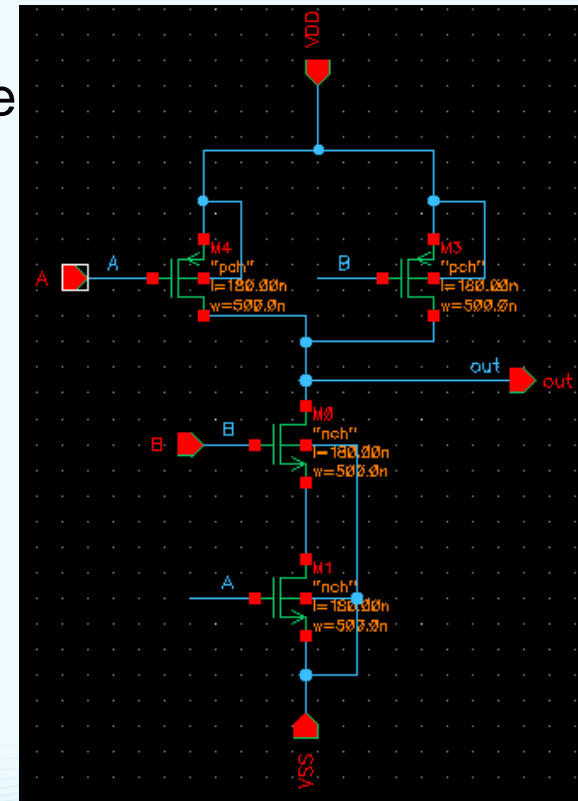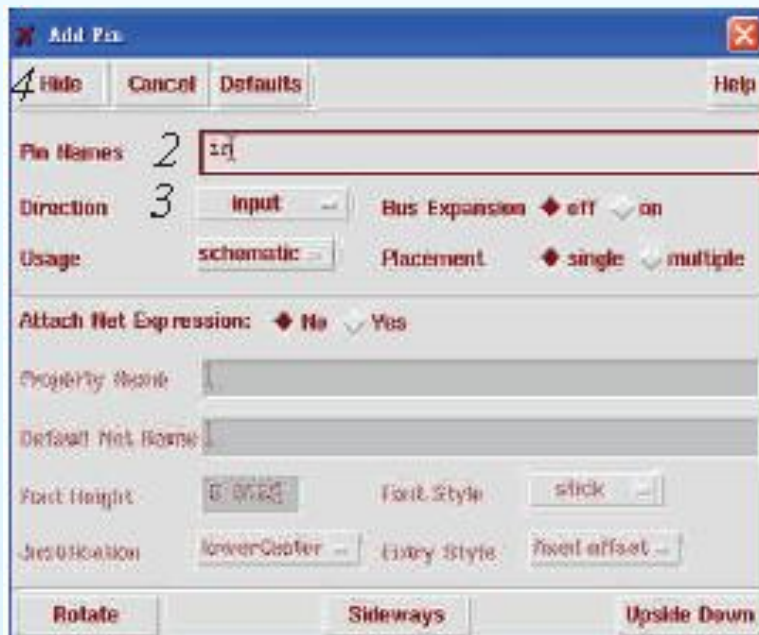
Press ***Hide*** and drag the "net name"to some wire/node

Indicating I/O Ports

Press key "*p*"or the pin button from the toolbar

Specify the ***Pin Name*** and choose the ***Direction*** (input/output/bidirection)

If pin connect to some net, the pin name should be identical to net name to avoid confusion.

Press ***Hide*** and drag the pin to the right place

Saving the Schematic

Press key "*X*"or check and save button ☑ from the toolbar
To save the schematic and make sure schematic is error free
The main window reports the status of schematic
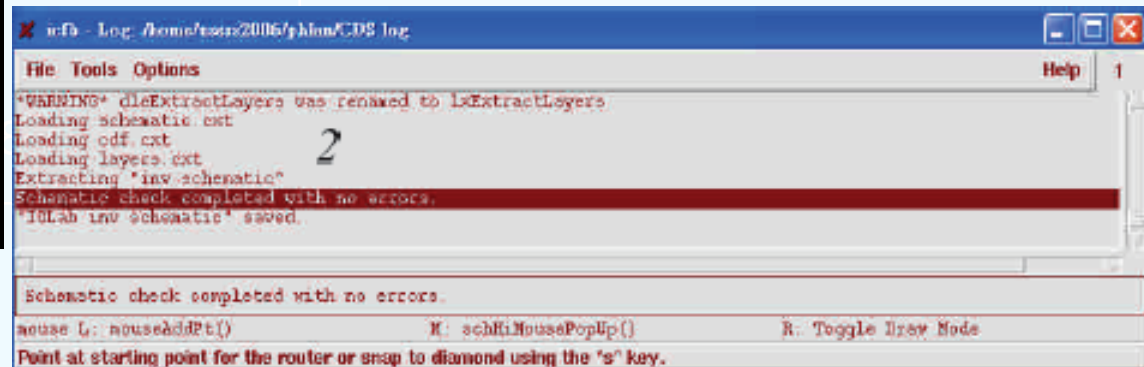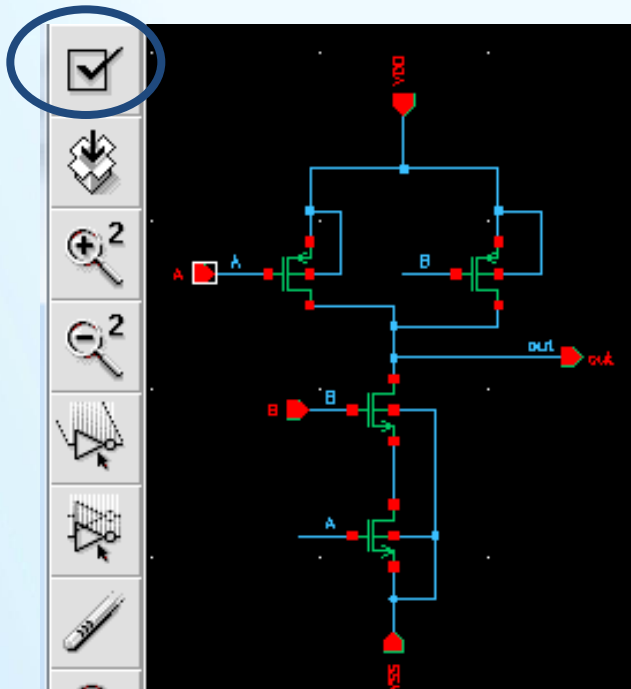
Open Existing Cell

From main window, choose *File → Open*
In the popped window, choose the *Library* and *Cell Name* you want
Choose the desired *View Name*, and Press *OK*

Creating Symbol
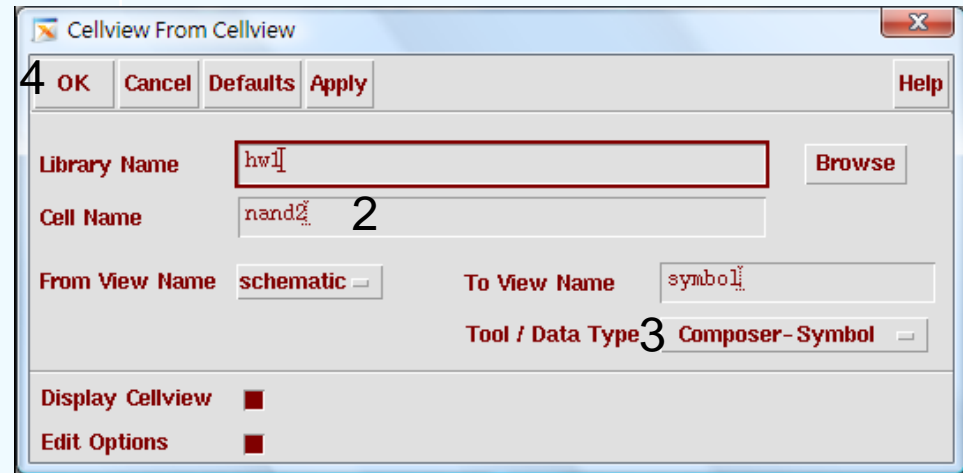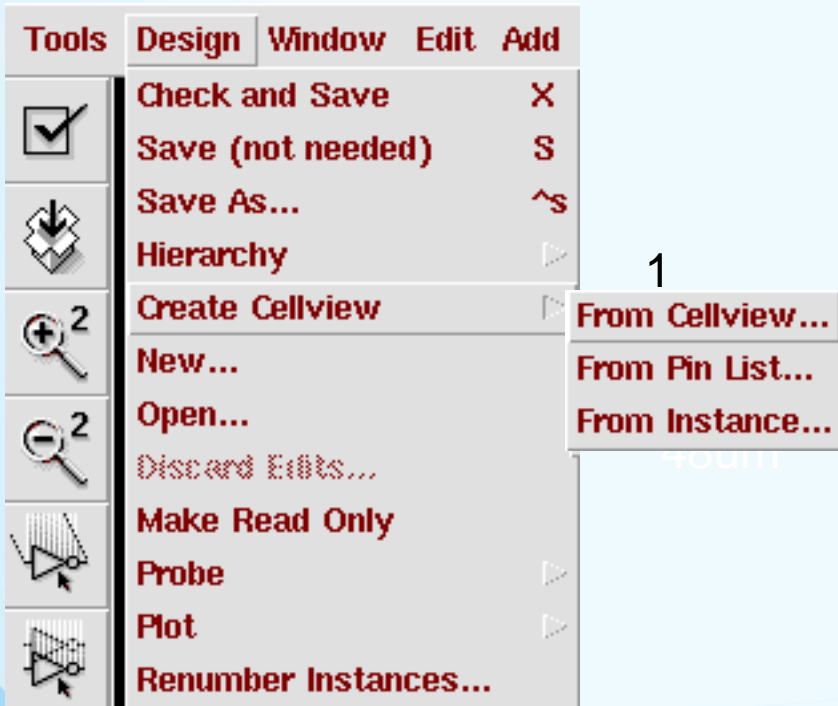
Open **nand2** schematic first

From schematic editing window, **Design → Create Cellview**

→ From Cellview…

Make sure that **Library** and **Cell Name** are correct.

Choose **Composer-Symbol** and Press **OK**

Defining I/O Ports

Another window pops up

The ***I/O Pins*** listed in the window accord with ***Pins*** that was inserted in Schematic Editing Window

You can decide the position of ***Pins*** on your own

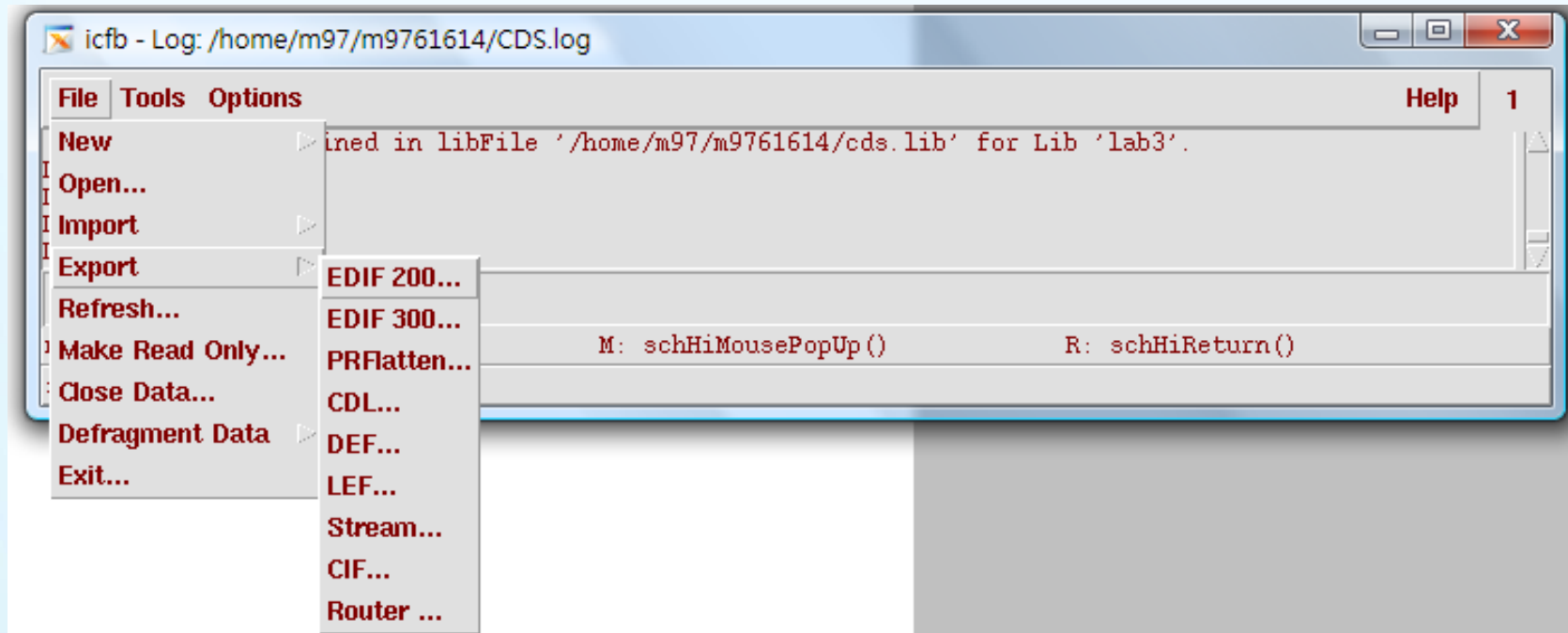Press OK if the I/O pins list is correct

Symbol Editing Window

Then, *Symbol Editing Window* pops up
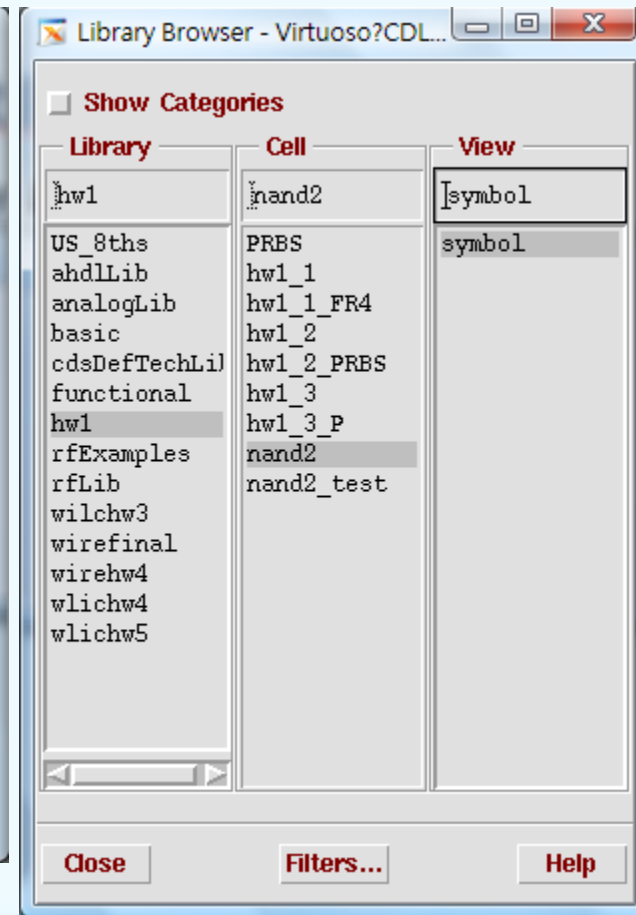Input (output) ports lie at left (right) as previous page's setup
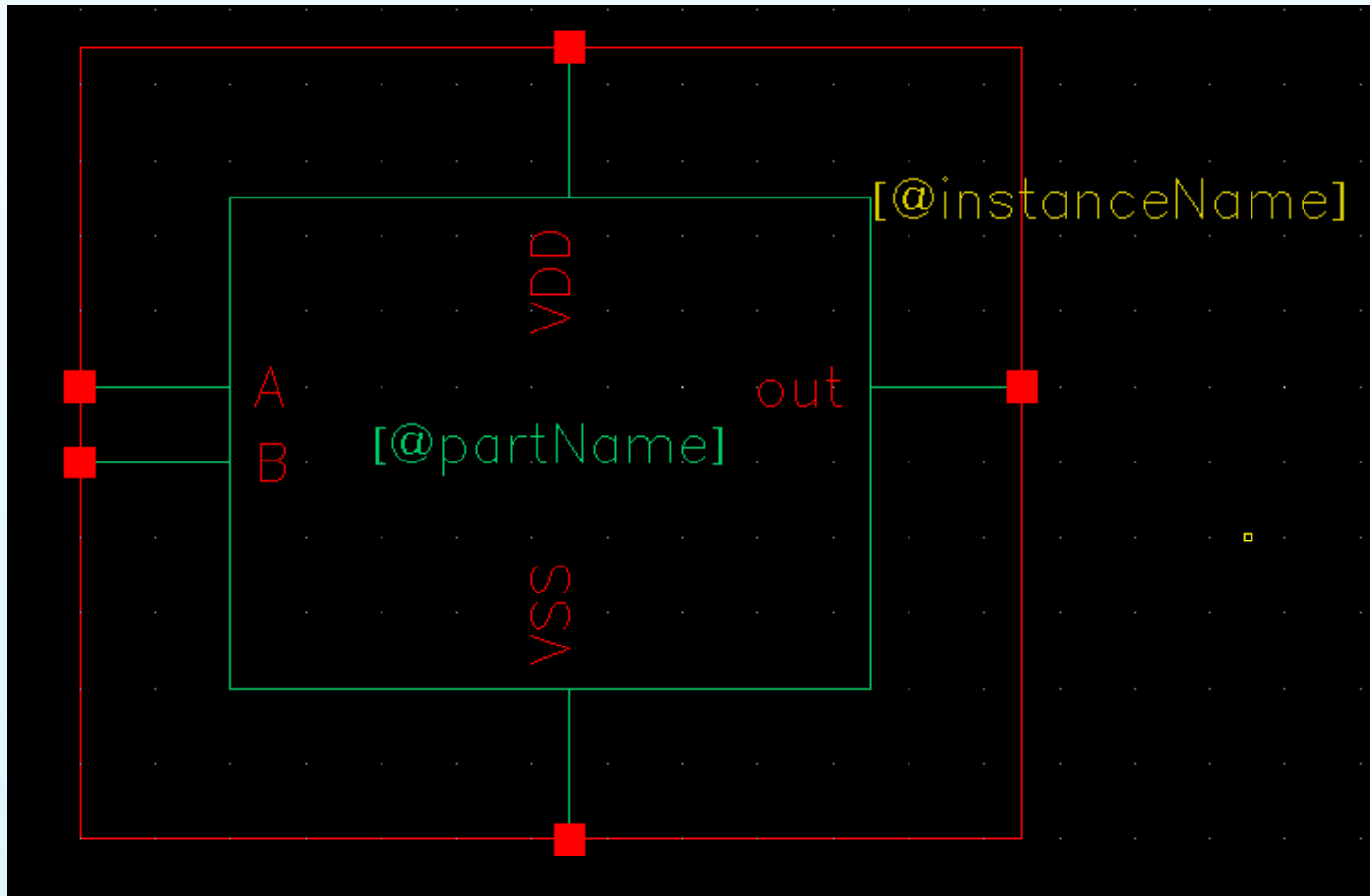
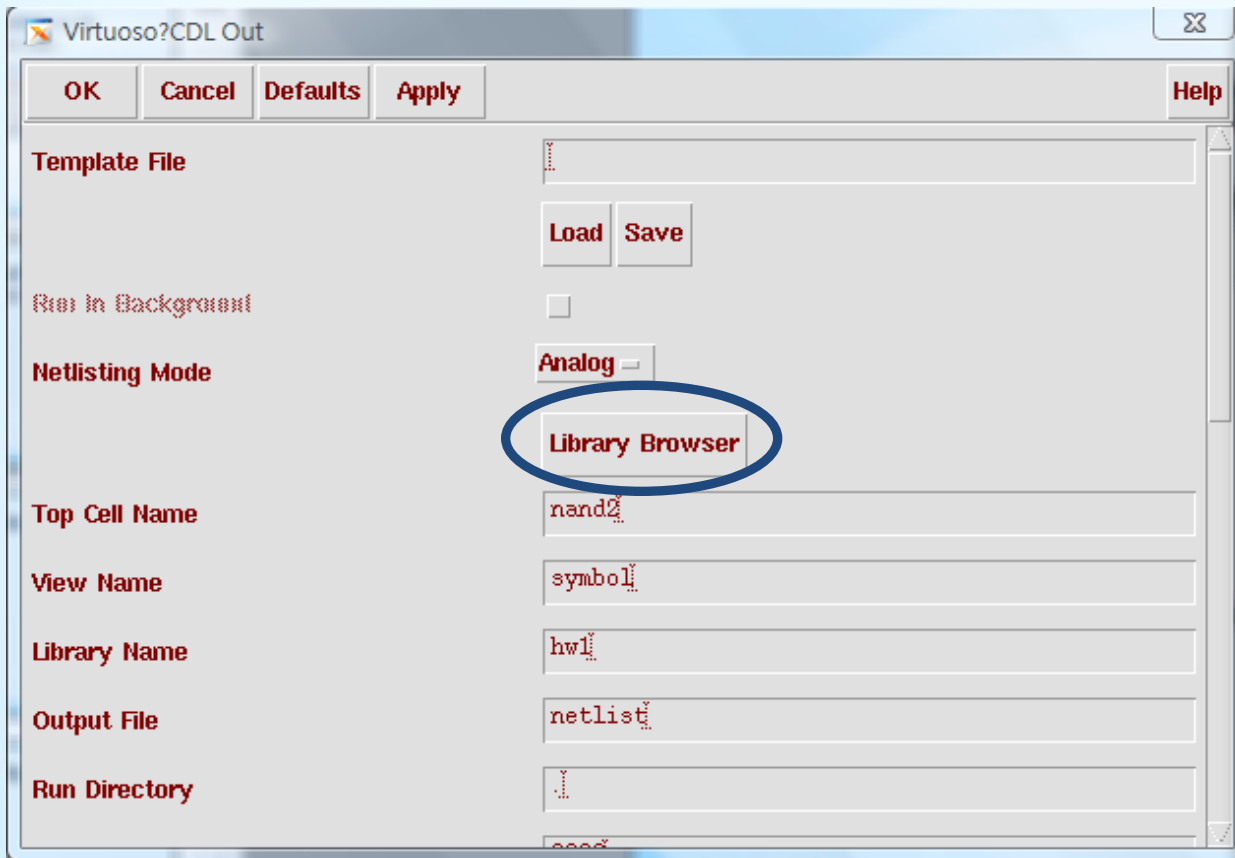# Export Netlists (1/3)
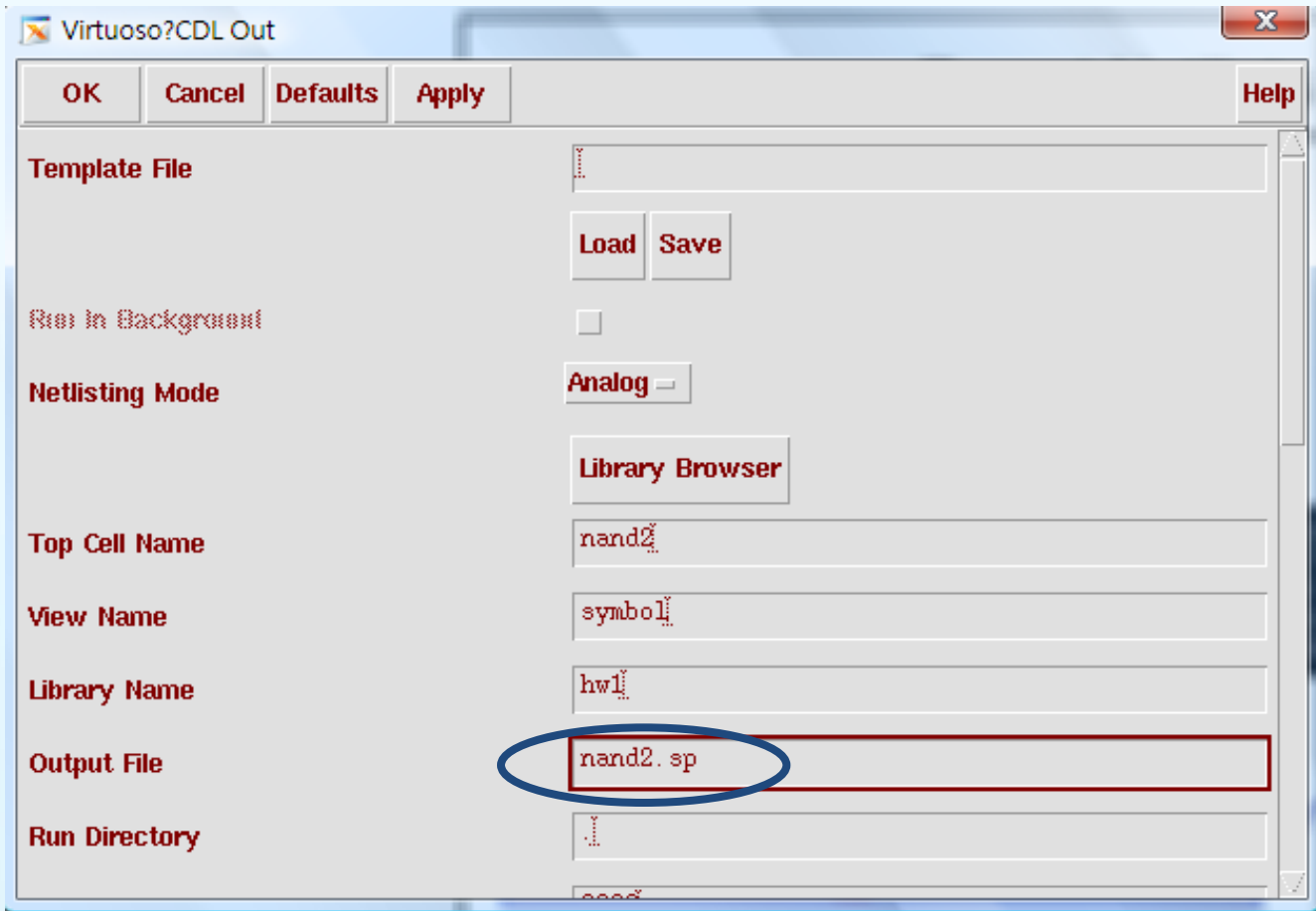
File → Export → CDL…

Export Netlists (2/3)

Choose *"Library Brower"*

Export Netlists (3/3)

Specify output netlist name (.sp)
View the generated netlist

Hotkeys

Instance	i	Copy	c
Narrow Wire	w	Move	M
Wide Wire	W	Delete	del
Pin	p	Rotate	r
Undo	u	Property	q
Redo	U	Zoom	z
Stretch	m	Check & Save	X
Descend Edit/Read	E/e	Ascend one level	Ctrl+e

Thank you!