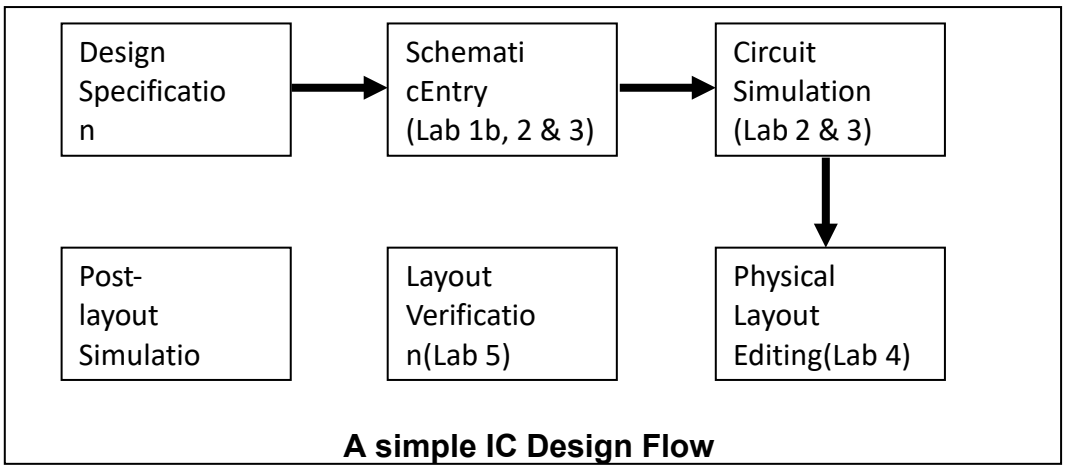
EE 323002
Lab 1 - Part b: Schematic Entry

Version 1.0 (Sept 2023)

Introduction

In this lab, you will learn to use the Cadence suit of IC design tools. Cadence tools are very popular in industry and can be used for schematic entry, circuit simulation, layout design, all the way up to a complete chip design.

In the coming labs, the basic of Cadence design framework and schematic capture of simple circuits (in transistor level) will be covered. You will learn how to build a chip through our lab exercises.



A simple IC Design Flow

I. Starting Cadence

Step 1. Enter to the Cadence working directory (working directory is the directory where you open Cadence and directory ~/ is your home directory). All your cadence-related files will be stored there from now onwards

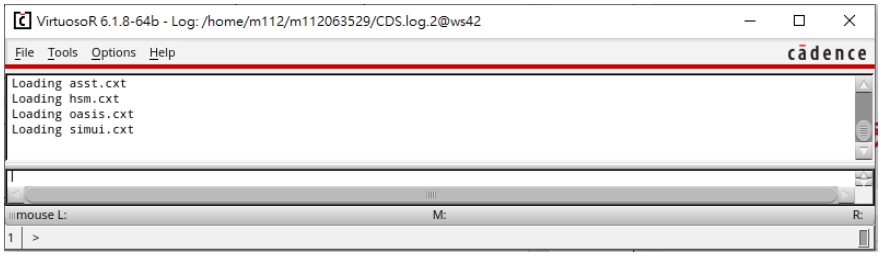
```
cd ~/323002cadence
```

1. Always start Cadence software under this working directory (`~/323002cadence`).
2. Add source /usr/cadtool/user_setup/03-spectre.csh in .tcshrc file.
3. Put technology file under (`~/323002cadence`)
4. And to activate the Cadence tool, under this working directory, **type in the following command lines ,**

```
virtuoso &
```

It will take a few seconds to execute. The main Cadence window, Command Interpreter Window (CIW) will appear as below.

Command Interpreter Window (CIW)



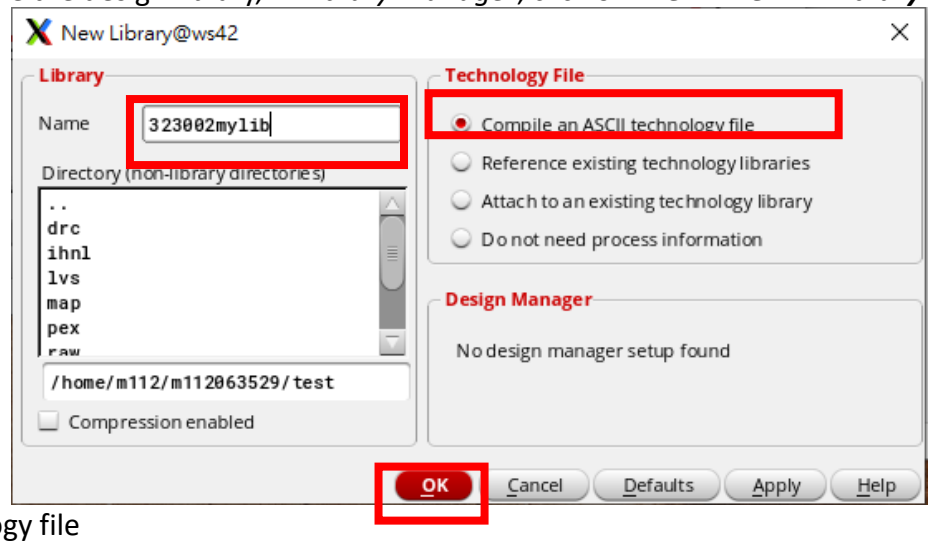
You should **frequently and carefully read** the messages shown in CIW because most of theinformation, warnings and error messages from the system are shown there.

5. To quit Cadence, click on the CIW's menu bar ***File*** → ***Exit***, then click on the "**Yes**" button in the pop-up menu to confirm.

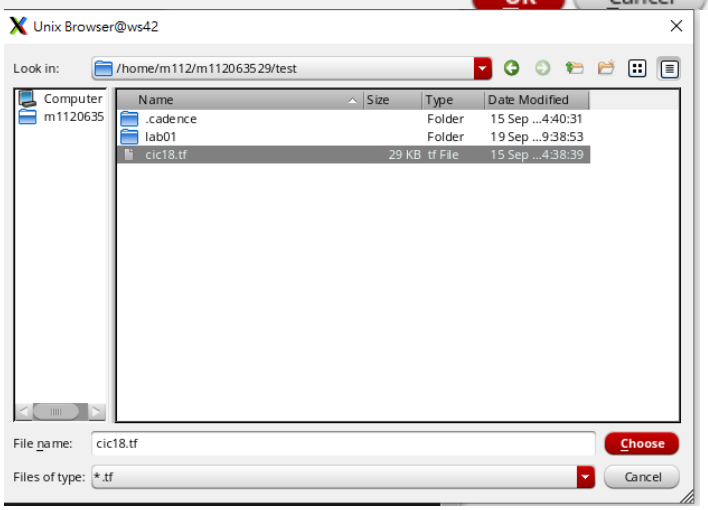
II. Creating Cadence Library and Specifying the technology library used

Now, you are going to create an empty Library. Later, you will have all your designs stored insidethis library: "323002mylib".

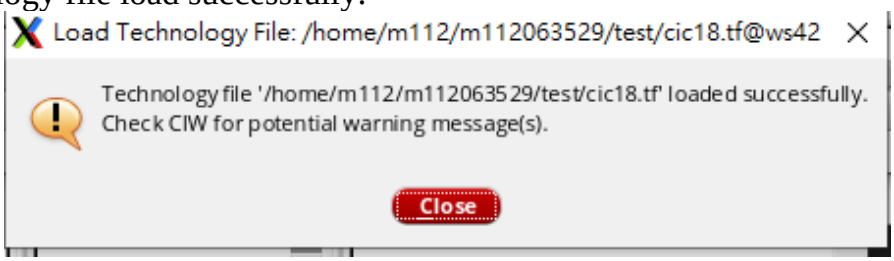
To create you're the design library, in *Library Manager*, click on ***File*** → ***New*** → ***Library***



Load technology file



Make sure technology file load successfully.



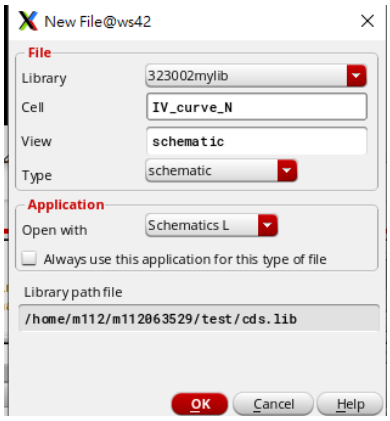
Well done! You have now set up the required design environment for your323002myLib design library. You can start designing your own circuits now! ***Note that whenever you create a new design library, you must specify which technology library to be used.***

III. Active MOSFET current-voltage characteristics

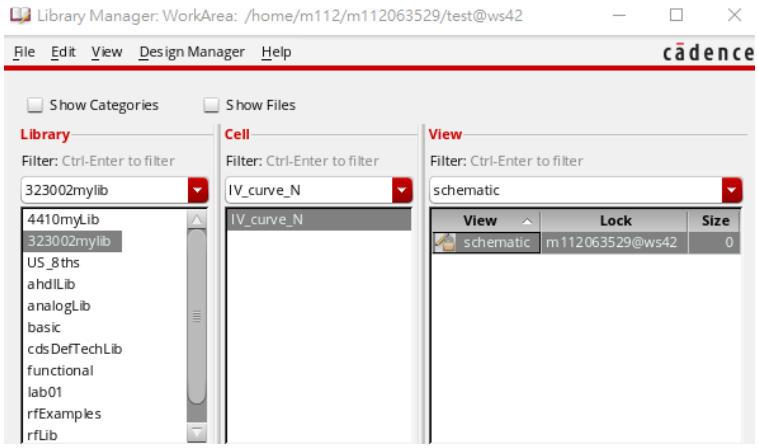
You are going to inspect the I-V characteristics of MOSFETs in this experiment.

1. File → New→ Cellview

Input *Cell Name* as ***IV_curves_N***.



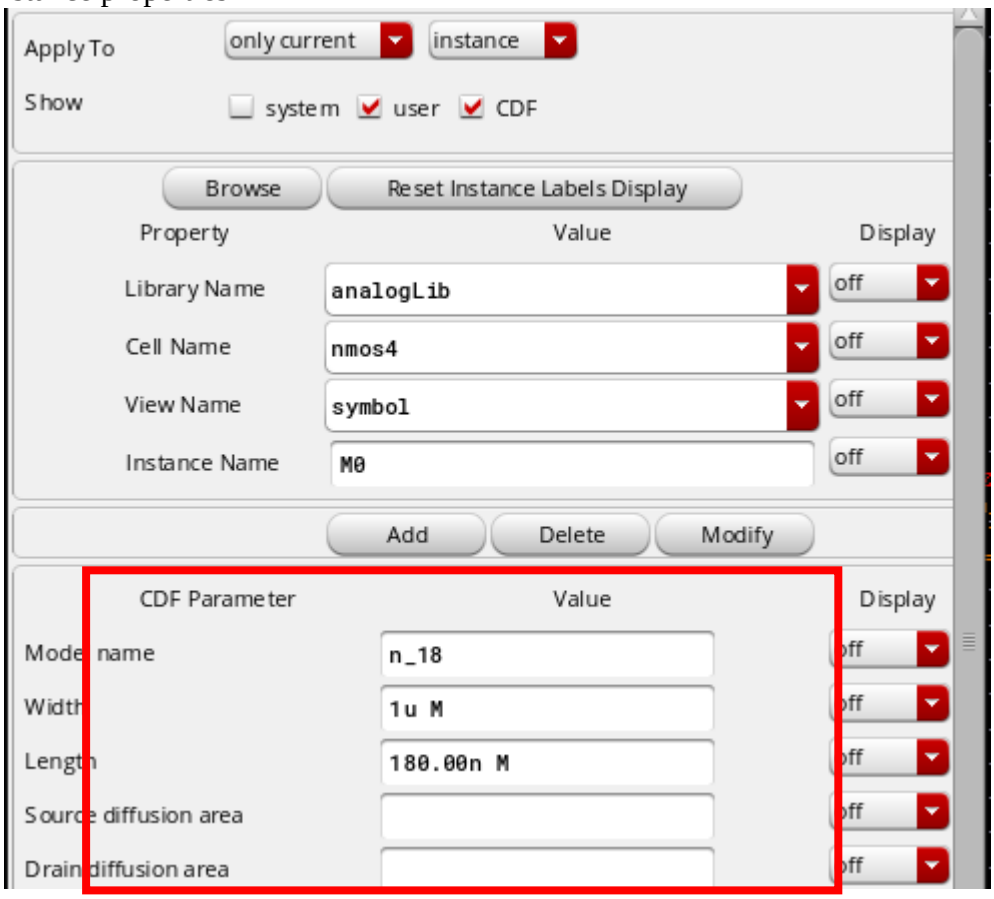
Click OK. A blank schematic window will appear. Note that the newly created IV_curves_N schematic cell view appears in the *Library Manager* as shown below:



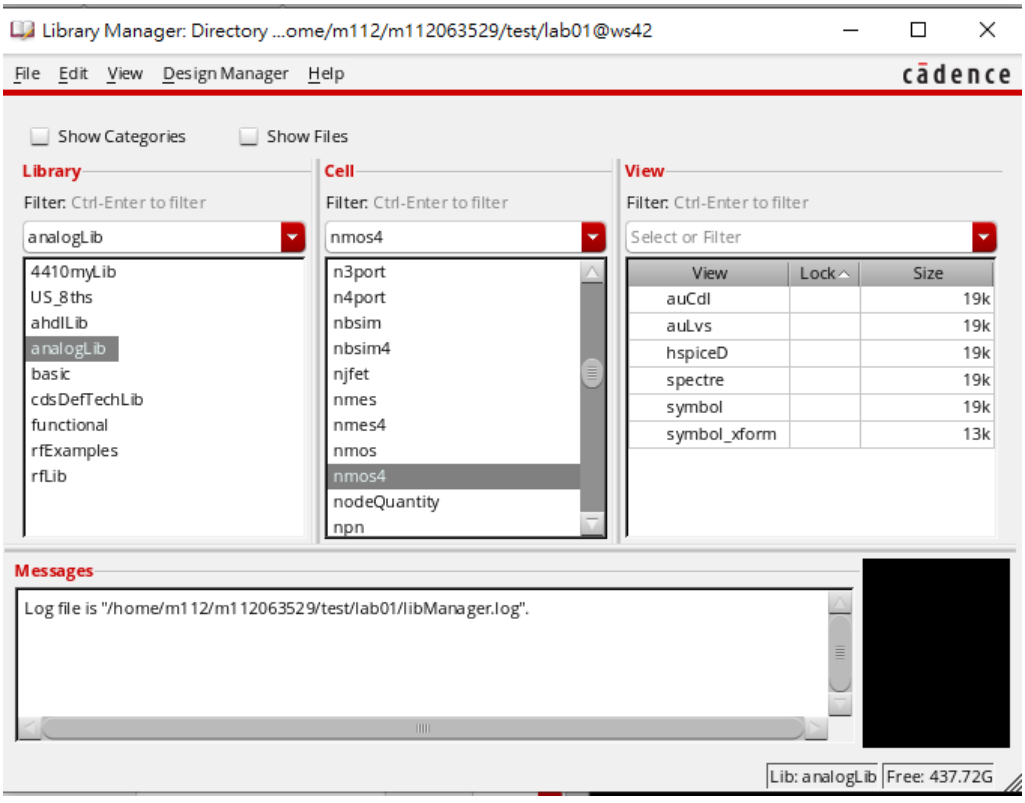
2. Create schematic in *Schematic Editor*.

- Click ***Create*** in the menu bar and choose ***Instance*** (or just press i) as shown below, or justclick this button  in the tool bar.
- Input Library as "*analogLib*" and Cell name as "*nmos4*" and input "symbol" in View field. You can flip and rotate the instance using buttons on this window. You can change model name and physical parameters by modifying the contents of the input box.

Press Q to see instance properties



Or you can click Browse button to choose them in *Library Browser*, as shown below:



- Now you can close the Browser window (selected library, cell, and view will be automatically filled into the *Add Instance* window) and click ***Hide*** button in the *Add Instance window* so that you can place an NMOS transistor in the *Schematic Editing* window. Left Click to place one selected instance, press Esc to finish placing (you can pressEsc a few times more to insure there no function active).

- Adding Wires to the Schematic

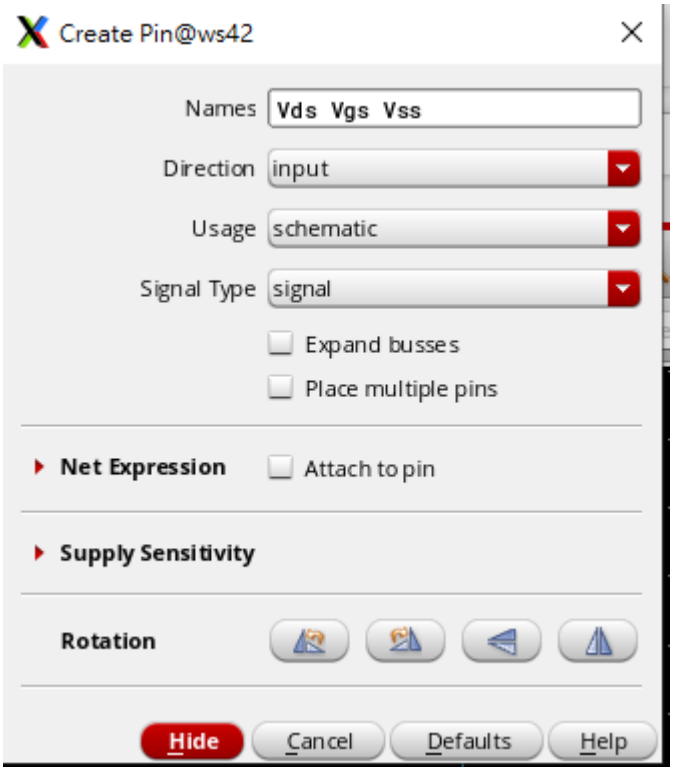
. Select the fixed menu ***Wire (narrow)*** button  . You can also press “w” to select wire.

- Then move the mouse cursor back into the schematic window. Note that two cursors (one in diamond shape and the other in square shape) move with your mouse pointer. These cursors can help you to connect the wires accurately from point to point.
- Point at a starting point (overlap the square cursor with the diamond one) and click the left mouse button to start drawing the wire.
- You can also use the keyboard key ‘***s’*** to snap the square cursor into the diamond one.
- Connect all the components with wires (refer to the first figure in the next page)
- To end the wire, double click the left mouse button. The finished schematic should look likethe following figure.

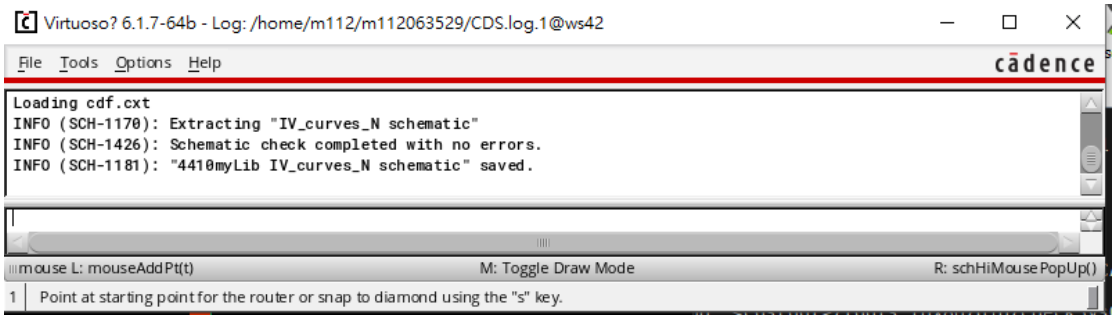
  Note: When this ***Wire*** command is activated, press "**F3**" on the keyboard to bring up an optionform. This form can also help you to draw wires, try it out yourself.

- Add Pin to input and output

  Press “p”to add pin

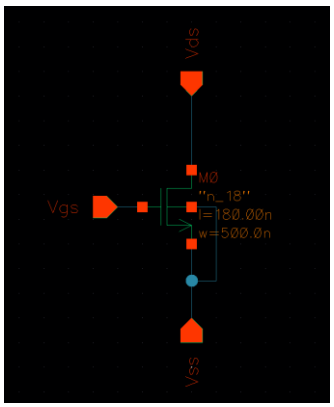


- Don't forget to check and save every time you finish editing a component: click on ***File*** →***Check and save*** (or this button ). If any errors and warnings occur, go to ***check***→ ***find marker***,and correct the errors and warnings. If everything works well, the CIW window would look like this:

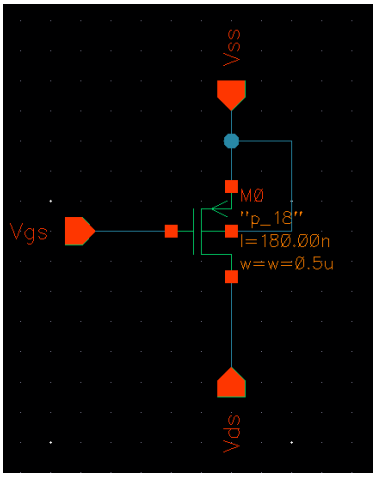


Goal is to make schematic of NMOS and PMOS

Nmos(Name:IV_curves_N): W=0.5u L=0.18u M=1



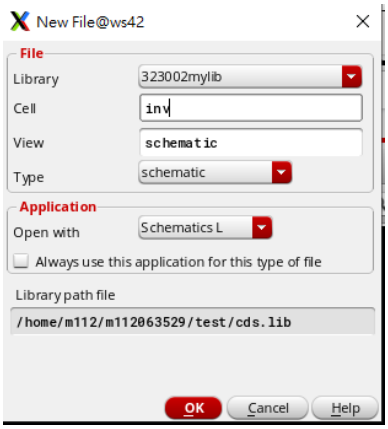
Pmos(Name:IV_curves_P): W=0.5u L=0.18u M=1



IV. Creating a new cellview – the schematic view of an inverter

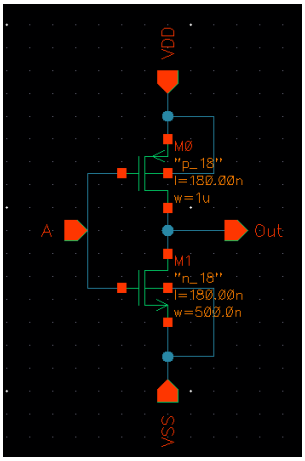
You are going to create an inverter design in schematic view, it is called "inv". The corresponding schematic tool used is called Composer.

1. In Library Manager, click on **File → New→ Cellview→ inv→ ok**



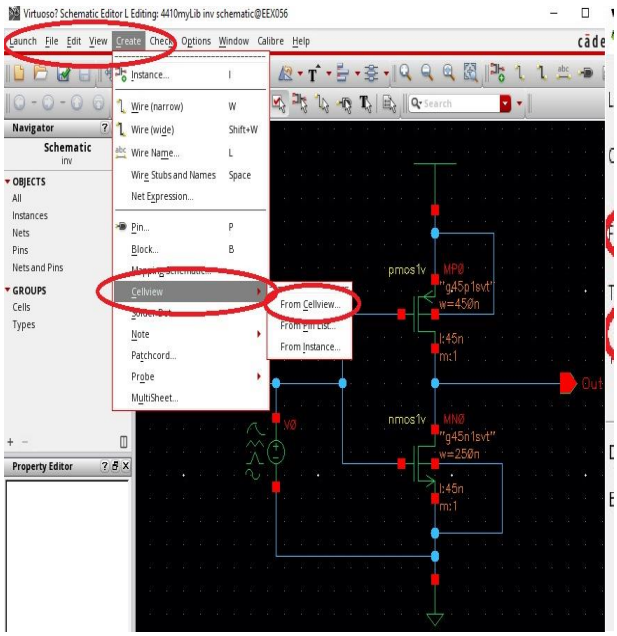
2. Create **inverter schematics**

   You need to modify transistor size and also make sure there is no errors.

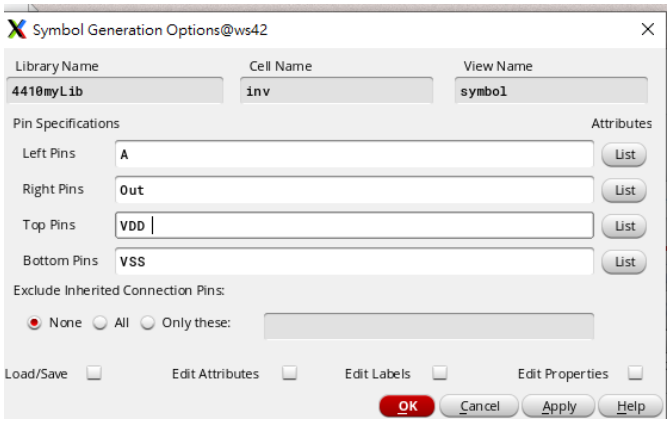


3. Creating a Symbol view

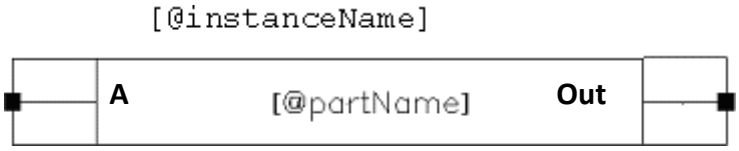
- In schematic editing window, click on ***Design → Create Cellview→ From Cellview.***This form appears. If there is no error, press ***OK***.



- Then the Symbol Generation Options form appears. Double check the pin names. If there is noerror, press ***OK***.



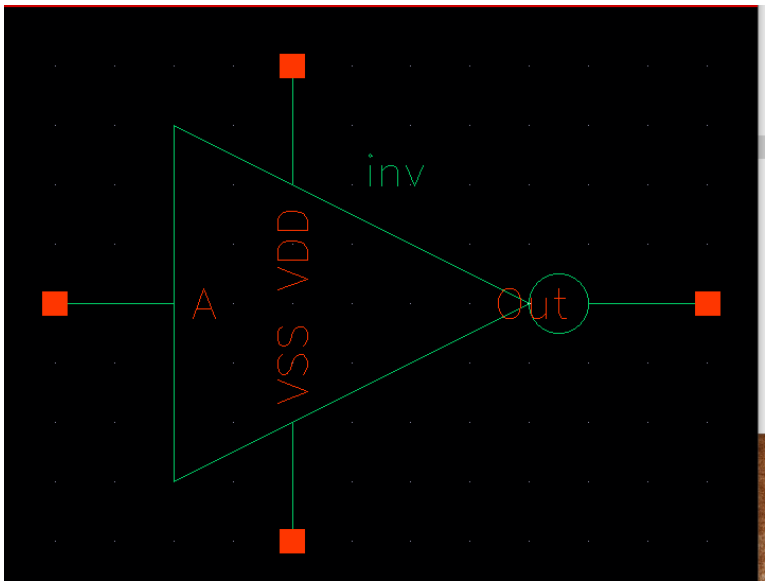
- The symbol view is then created automatically and will look as follows. Always remember to'***check and save***' the cellview so that the system



can cross-check the pin names and pin typesamong the symbol view and the schematic view.

Note: You can add lines/circles on the symbol view to make it more recognizable. To add lines/circles,at the pull down menu of the symbol view editing window, click on ***Add → Shape → Line*** , or***Add → Shape → Circle*** to

draw. You can create a symbol like this.



The End of Lab1 (Part b)