Cadence Tutorial
Schematic Entry & Simulation
(Using Virtuoso Schematic and Spectre)

Department of Electronics & Communication Engineering
Indraprastha Institute of Information Technology, Delhi, India.

The following Cadence CAD tools will be used in this tutorial:

- **Virtuoso Schematic** for schematic capture.
- **Spectre** for simulation.

We will practice using CADENCE with a CMOS Inverter: creating (1) Schematic (2) Simulation

Computer Account Setup

Please see the Unix/Linux command before doing this new tutorial.

YOU SHOULD HAVE YOUR ENVIRONMENT SET UP FOR CADENCE AND ADDITIONAL TOOLS

Running the Cadence tools

Log in to your UNIX/LINUX account. Open the terminal window.

Now you should be able to run the Cadence tools. Never run Cadence from your root directory, it creates many extra files that will clutter your root. Instead please create a directory (e.g. cadence).

>> mkdir cadence

>>cd cadence

Now start Cadence by typing

>>csh

>> source cshrc

>>cd cadence_ms_labs_613

>>virtuoso

Please see the Fig. 1 for above command.

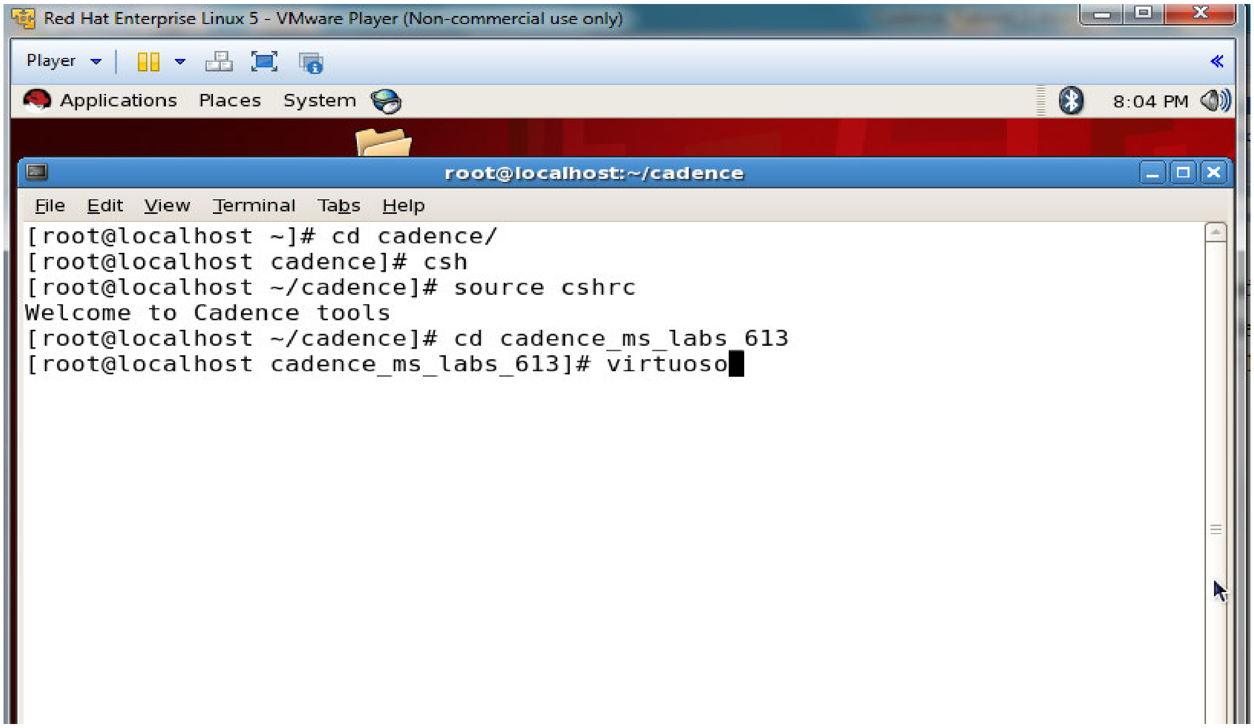

Fig. 1 Terminal window

The command will start Cadence and after a while you should get a window with the "Virtuoso@ 6.1.5**",** also called **Command Interpreter Window (CIW)** as below: Fig 2

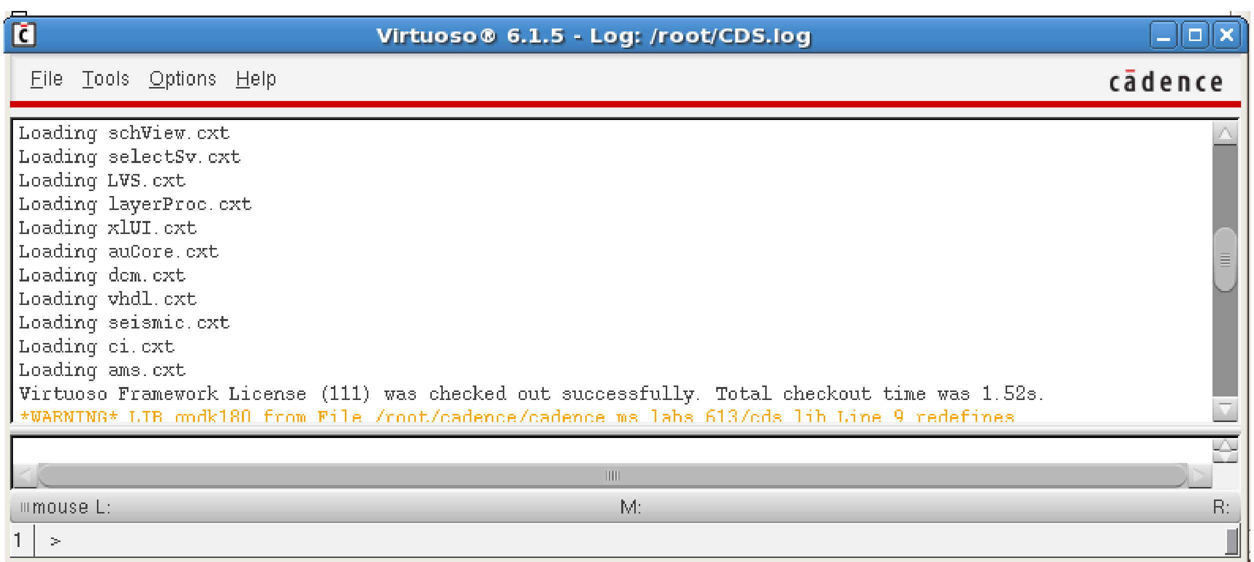

Fig. 2 Cadence virtuoso (CIW) window

For more information on the various Cadence tools I encourage you to read the corresponding user manuals. You can get to the manuals by pressing **Help -> Virtuoso Documentation** on any Cadence window (e.g. CIW)

Now we need to create a new library (to contain your circuits) so from the **Virtuoso (Fig 2) Command Interpreter Window (CIW)** go to **File -> New -> Library** from the File menu. You will see a "New Library" window (Fig 3). Fill in the name of the new library (e.g. CMOSInverter) in the dialog window (this will create the library in the directory where you started "Virtuoso", you could also choose to set a path if you wanted another directory). Click on "**Attach to existing tech library**" and click **OK**.

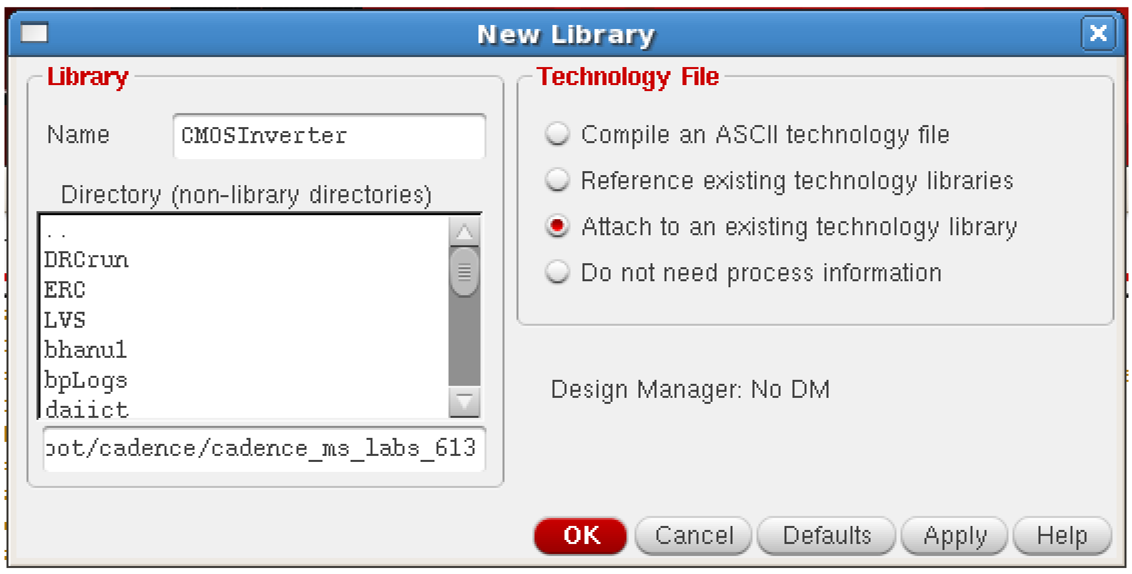

Fig. 3 New Library Window

The above steps can be also performed using the Library Manager. After you start Cadence and get the "Virtuoso CIW" window, go to **Tools->Library Manager** or press **F6** on keyboard**.** It will open the Library Manager window (Fig 4) as shown below. You can create the new library (CMOSInverter) from the Library Manager following the same steps as explained above. Now the "CMOSInverter" library should appear in the Library Manager window. It is easier to work with Library Manager. However, for this document we will work through Virtuoso- CIW window.

Let's start our first schematic now!

SCHEMATIC CAPTURE

In the **Virtuoso CIW** window go to **File -> New -> Cell View**. You will get a "Create New file" window (Fig 4). Fill in the information in the dialogue window as below and then press OK.

Library Name : CMOSInverter
Cell Name : myinverter (you can choose other name if you want)
View Name : Schematic
Tool : Composer-schematic

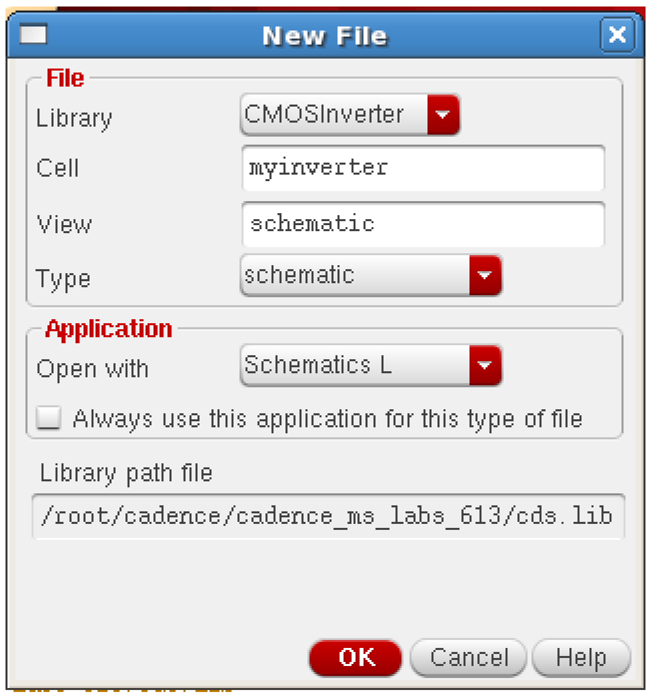

Fig 4 Create New File window

Wait for a while. "The schematic window will appear. You should get the "Virtuoso Schematic Editing" window as shown below (Fig 5). Spend some time analyzing the window. On the left side you have various shortcuts to common used commands such as: placing component instances (looks like an IC), drawing wires, placing ports, stretching, copying, zooming in and out, saving, etc. If you pass the mouse pointer on top of the buttons you get short pop-up help messages. You also have access to these commands (and others) from the menu. It is not possible here to describe all the functionality of Virtuoso Schematic so you are strongly encouraged to read the on-line user manuals.

You should notice that the top bar of the window will display the name of the library (CMOSInverter), cellview (myinverter) and schematic at the end.

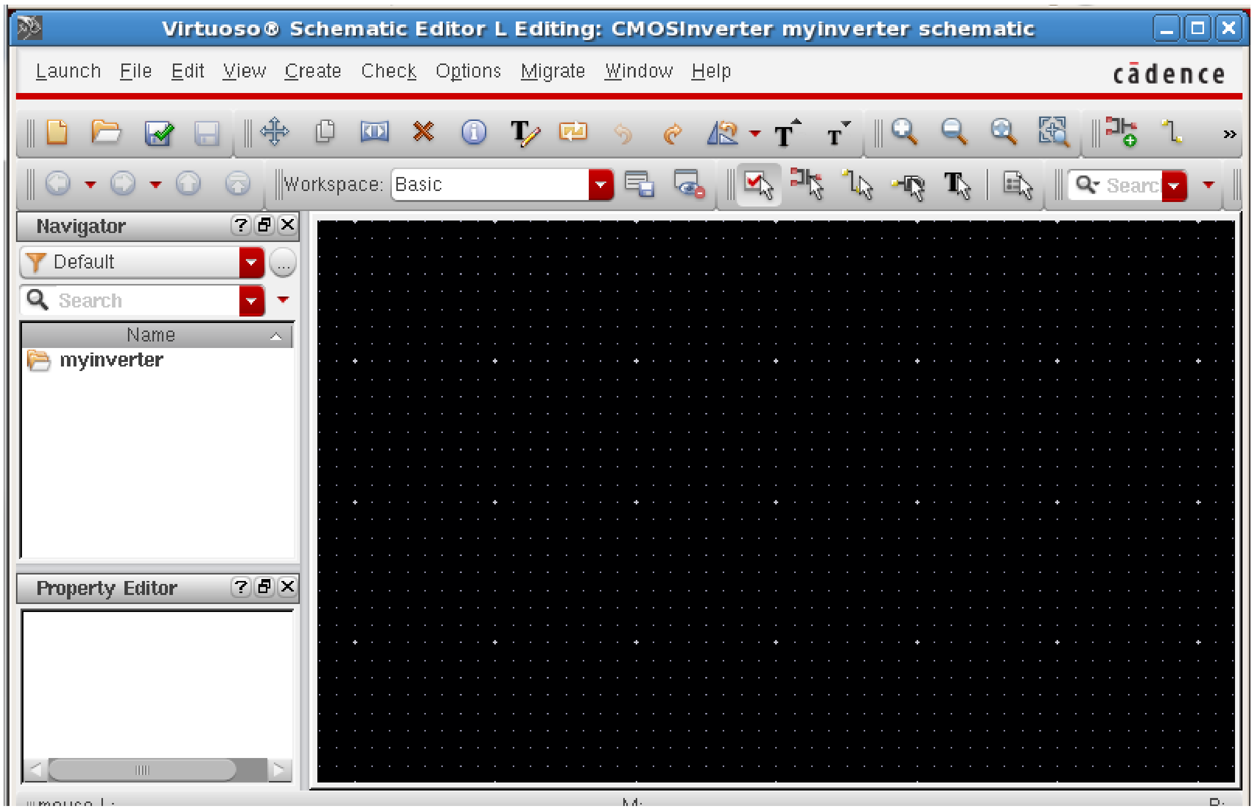

Fig 5 Virtuoso Schematic Editing window (Composer)

Let's start our first schematic to create the CMOS Inverter. Expand the Virtuoso Schematic Editing window if necessary. We will pace the NMOS and PMOS transistors on the schematic.

Placing Instances

Click on the "**Instance**" button (icon) on the left side (which looks somewhat like an IC, or go to Add -> Instance), this will pop-up an "Add Instance" window (Fig 6).

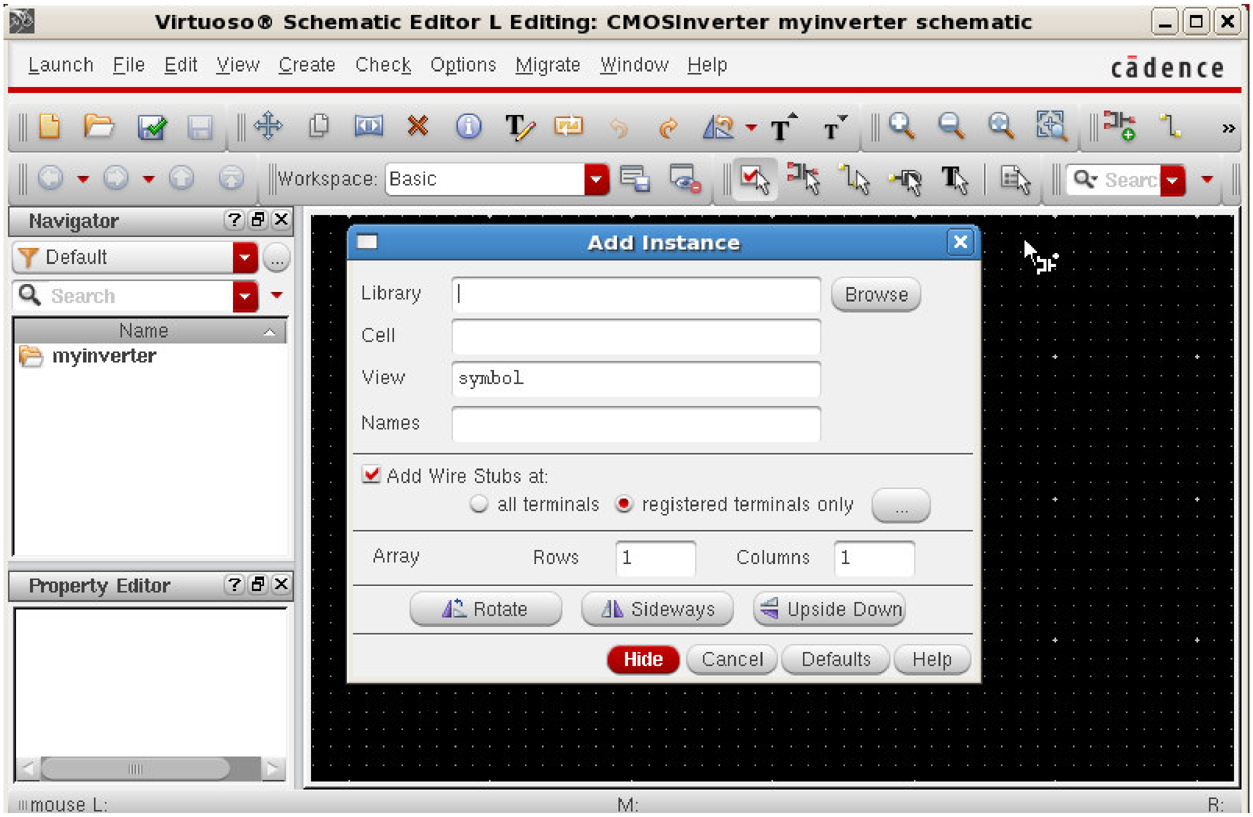

Fig 6 Add Instance Window

Now click on the Browse. Another window called "Library Browser – Add Instance" (Fig 6) will pop up. We will select PMOS transistor and will place it on the Virtuoso Schematic window. Follow the steps now. Select as follows in the Library Browser window (Fig 7).

Library =>gpdk180
Cell => pmos
View => symbol

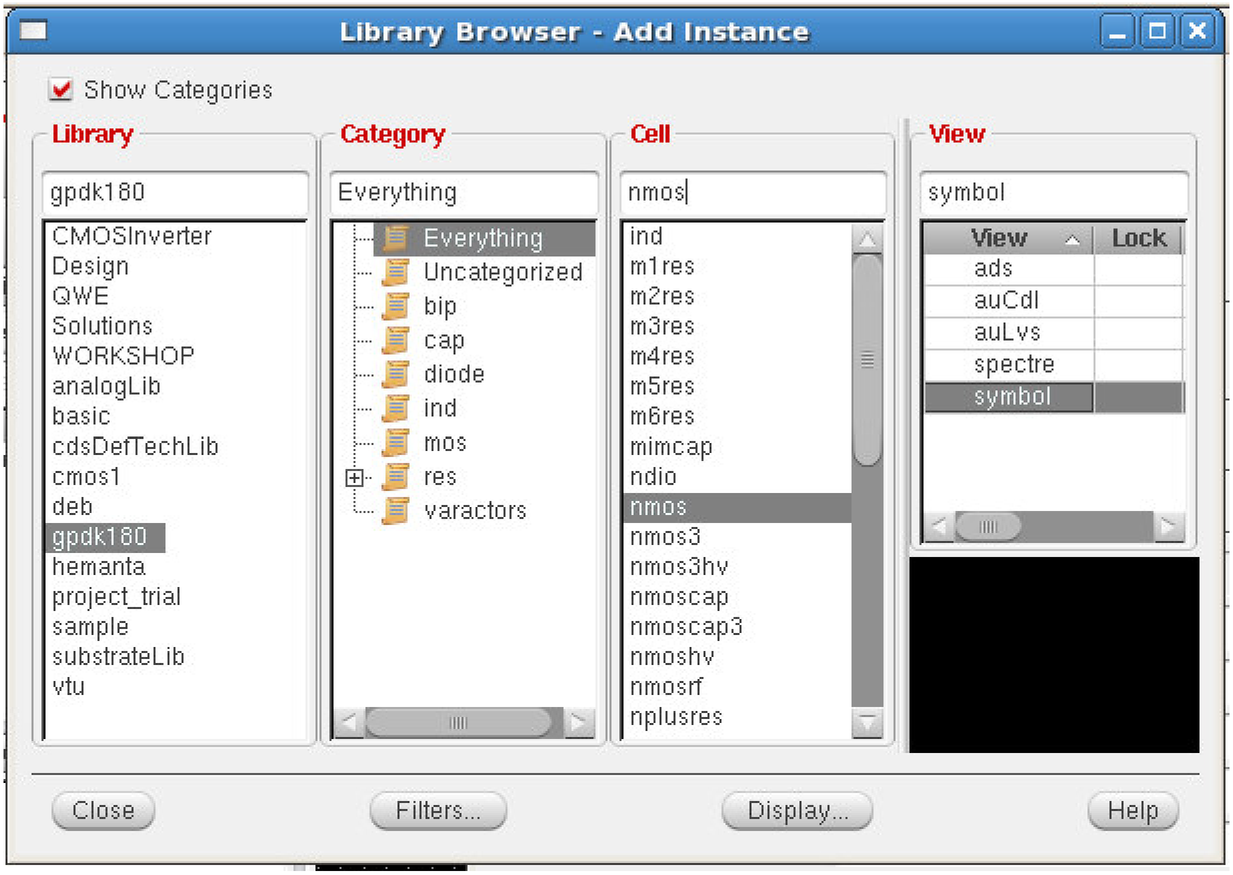

Fig. 7 Library Browser window

Change following properties of ***pmos*** in "Add instance" as given here.

Names => M1
Width =>800 nm
Length => 180 nm

Similarly, now we will place the NMOS transistor. Go back to the "Library Browser-Add Instance" by left clicking on this window. Select as follows:

Library =>gpdk180
Cell => nmos
View => symbol

Notice carefully. You need to select "**nmos**" under the cell for NMOS.

Now click on the "Add Instance" window. Change following properties of the *nmos* in the "Add instance" window.

Names => M2, Width => 360 nm, Length => 180 nm

Ok, you have NMOS and PMOS on your schematic. So far so good!!!!

To change the parameters of the instance, select the instance (by clicking on it with the mouse) and then use "properties" icon or press "q".

Now we also need to add wires, I/O pins and power supply.

First let's add wires (narrow) to connect transistor's terminals and form a schematic of the CMOS Inverter.

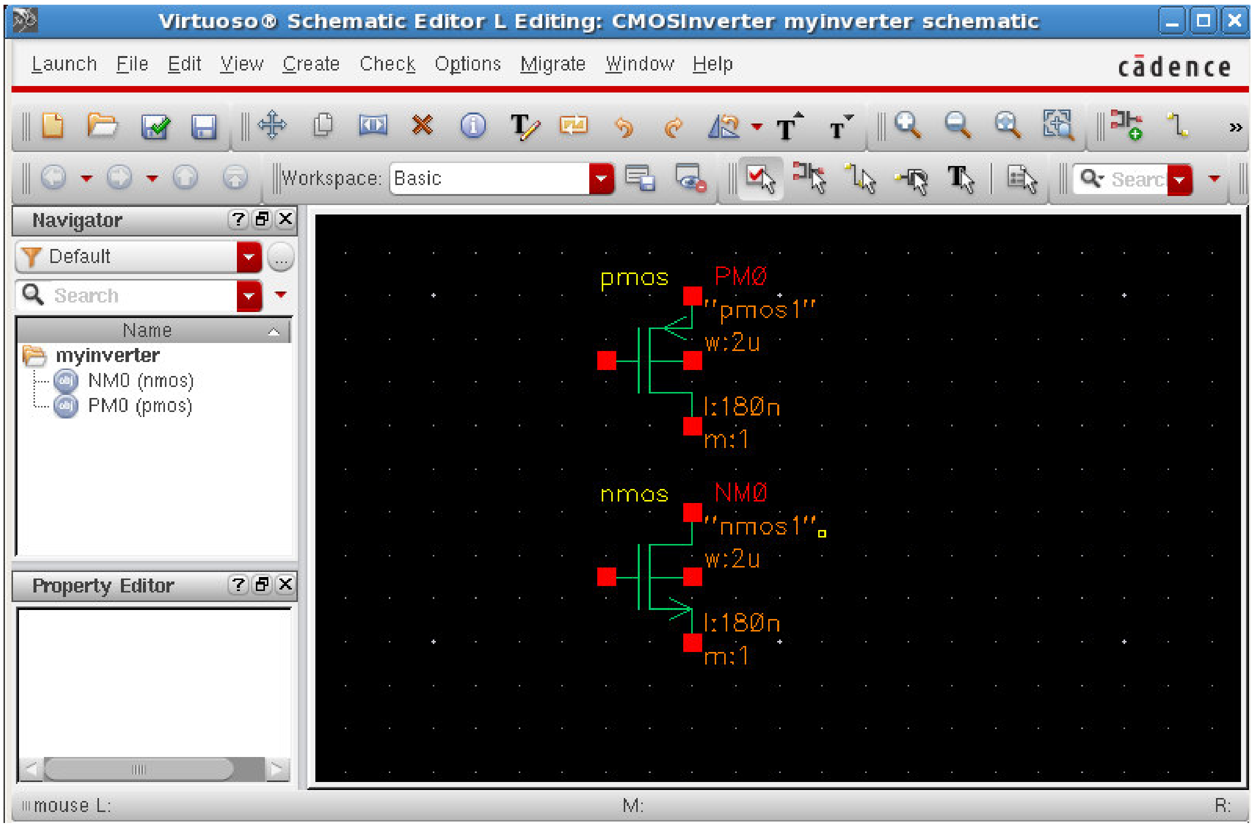

Fig 8 NMOS and PMOS on schematic

Connecting Wires

To connect the wires, click on the icon "Wire (narrow)" on the left side. You will see an "add Wire" window. You can choose the color whatever you want to. Now activate the Virtuoso Schematic Editing window by clicking on its title bar.

Move the mouse over or click on the **s** key on your keyboard. This snaps the wires to connect between the little diamond-shapes displaying by the nodes. You can click on the node (diamond-shape) with left mouse button, move the mouse over (you will see wire attached) and then double click at other end to connect wire between those points. Connect all the wires likewise. When you are done hit **"Esc"** on the keyboard. You can delete unwanted wires if connected accidentally. To delete the wire, select that wire by left mouse click and then hit Delete on the keyboard.

See Fig 9

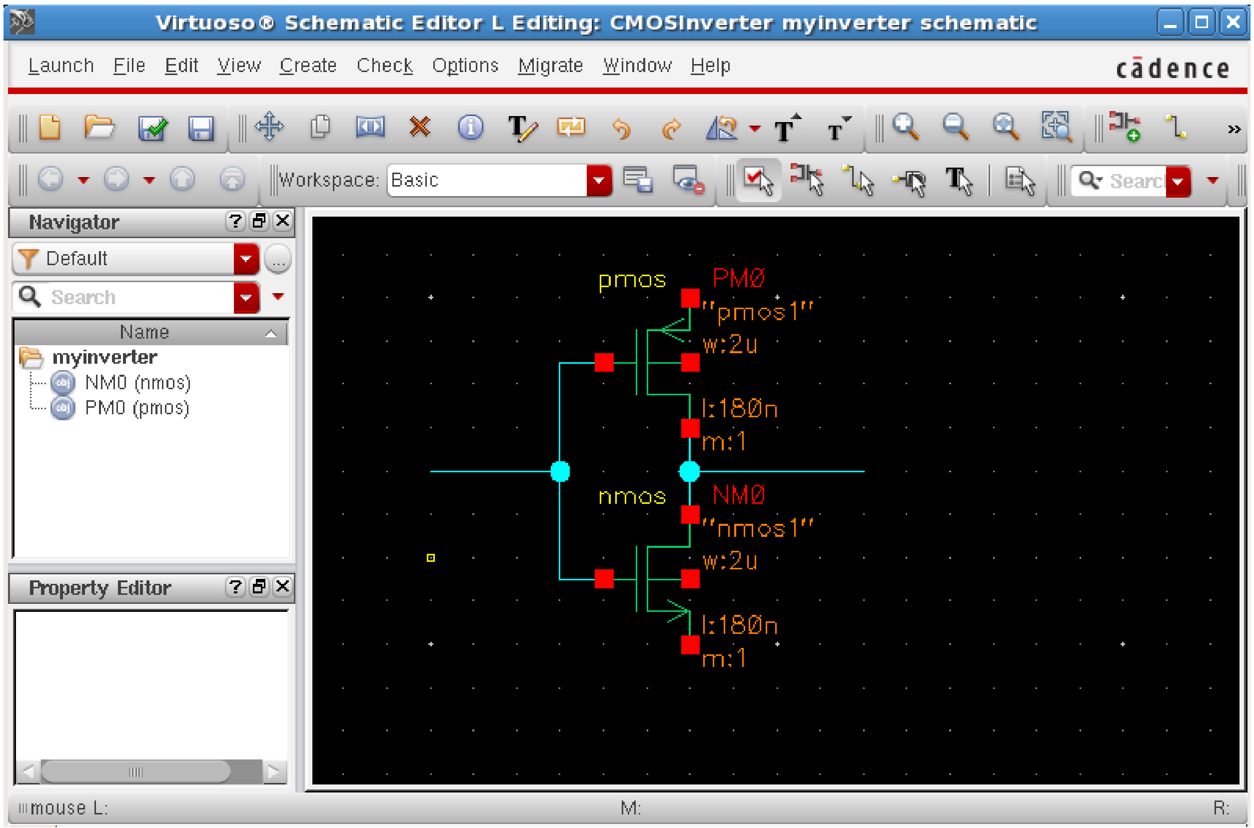

Fig 9 CMOS with no pins

Let's connect the I/O pins now.

Adding Pins

To add the *input* and *output* pins, click on the "Pin" icon at the lower left corner. The "Add Pin" form appears.
Under the Pin Name type *Vin*. Note that Direction in the form reads input, as shown below (Fig 10).

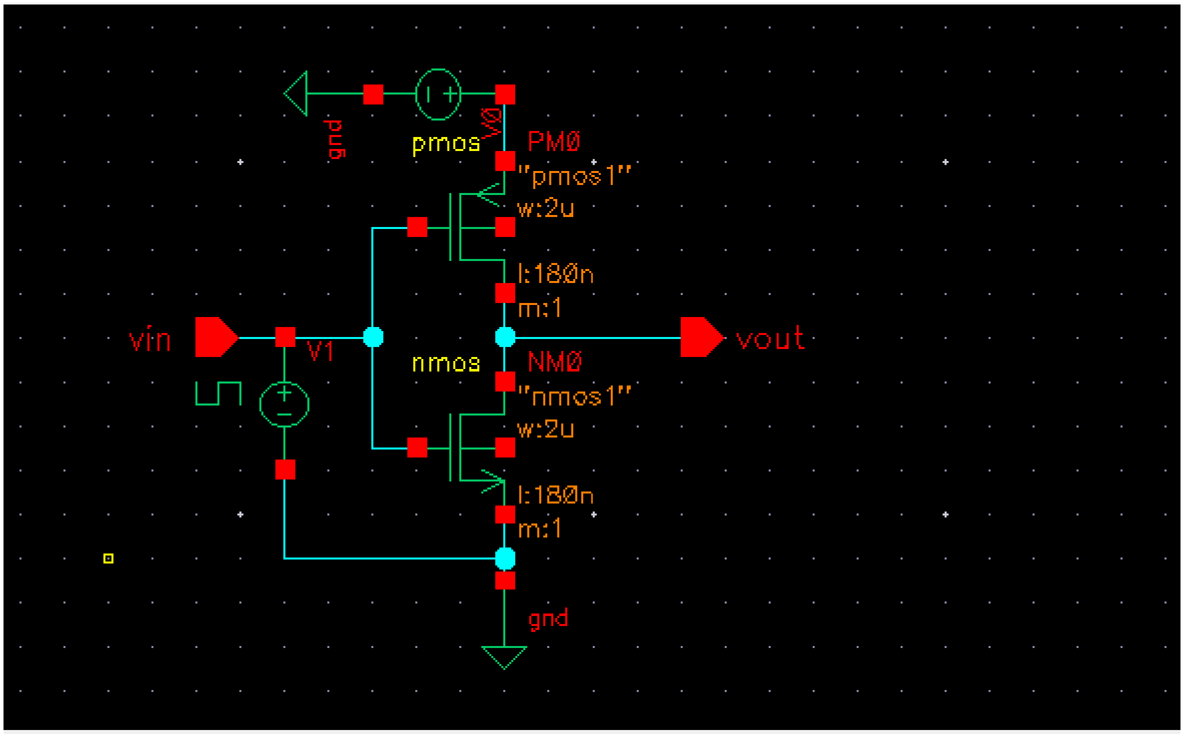

Fig 11 CMOS with pin, gnd and supply (Vdd)

The final schematic should look somewhat like this (Fig 17). Success?

It's a good idea to save your design from time to time in case the system crashes.

Everything worked fine so far!!!! Congratulation!!!

Check and Save your design

Now you need to Check and Save your design (either click the top left button or go to Design -> Check and Save). Make sure you look at the **"Virtuoso"** CIW window and there are no errors or warnings, if there are any you have to go back and fix them!

The Virtuoso window will give the message as shown below.

"Schematic" check completed with no errors.

"CMOS Inverter myinverter schematic" saved.

Let’s now perform the simulation on the inverter circuit to see the final results!!!

In the Virtuoso Schematic window go to Launch → ADE L
Then you will get the simulation window or ADE pop-up window

Please see Fig. 12.

Tools -> Analog Environment. You will get “Virtuoso Analog Design Environment (1)” window (Fig 12).

In the Virtuoso Analog Design Environment, go to “Setup -> Model Libraries……… ”.

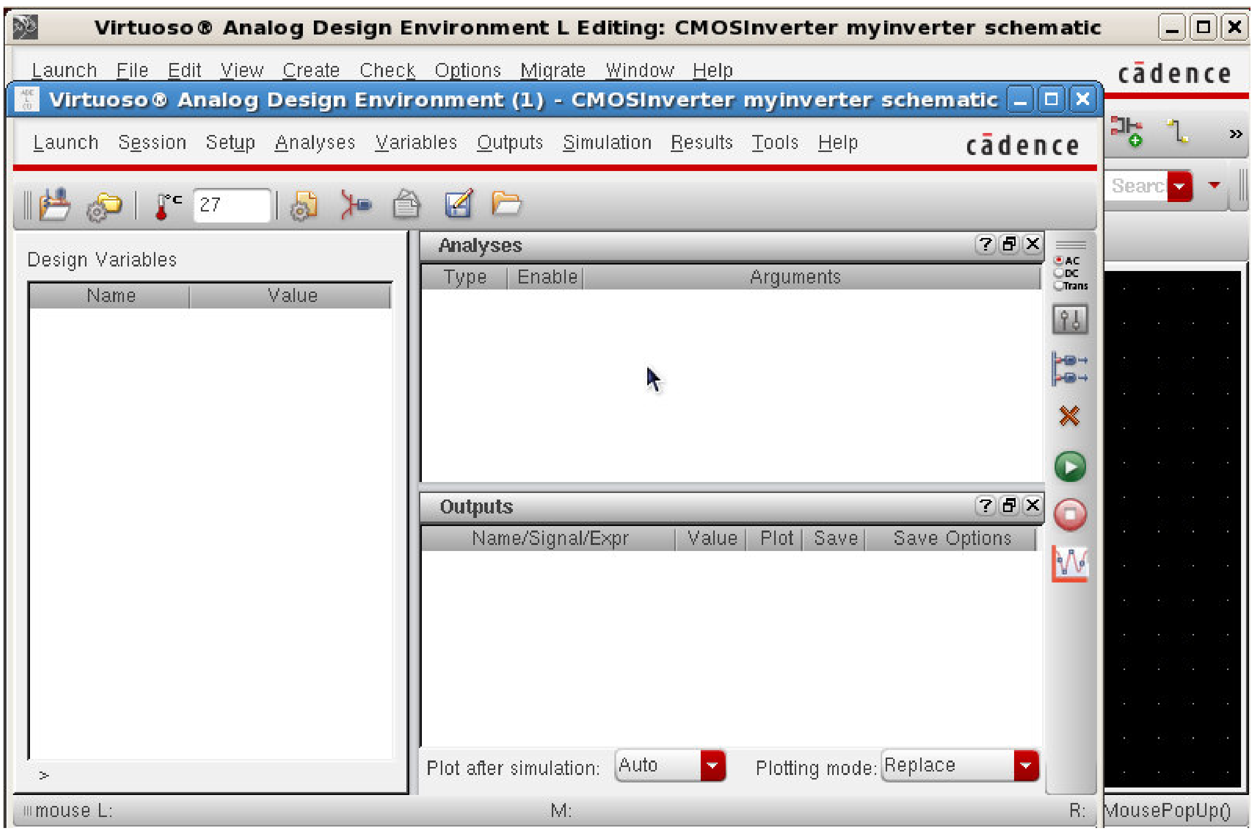

Fig. 12 ADE Window

Now you need to choose the type of simulation. From “Virtouso Analog Artist”(Fig 13) go to Analyses -> Choose... (Fig 13). In this case we will choose a transient analysis. Enter the stop time for transient analysis. Let’s type 200p for stop time. Click OK.

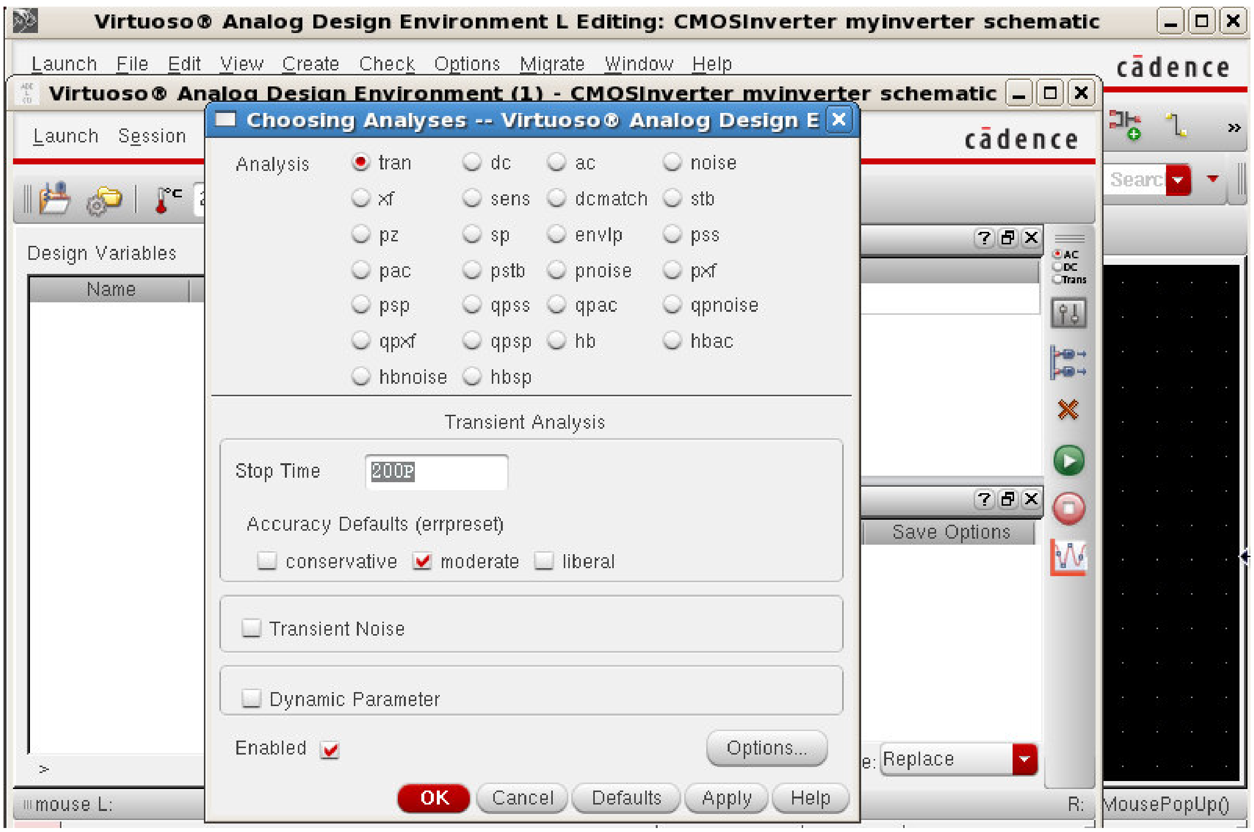

Fig 13 Choosing Analysis (Transient Analysis)

For DC analysis, please see the Fig. 14.

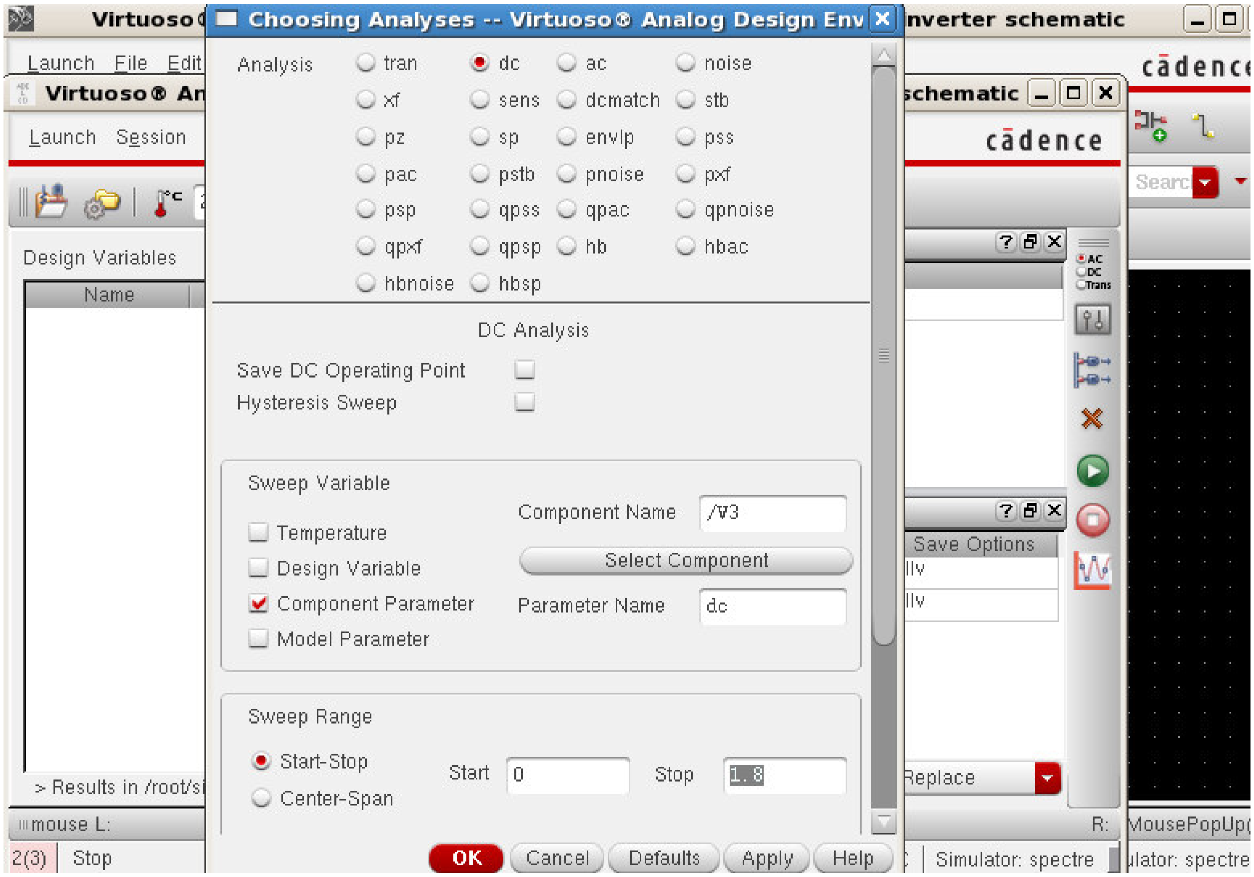

Fig 14 Choosing Analysis (DC Analysis)

Now in the "Virtouso Analog Artist" (Fig 15) go to "Outputs -> to be plotted -> select on schematic". That will bring your inverter cell view window in front. Select node voltages by clicking on the net. We will click on input and output nets (wires) to select input and output voltages. The currents can be selected by clicking on the terminals (red squares).

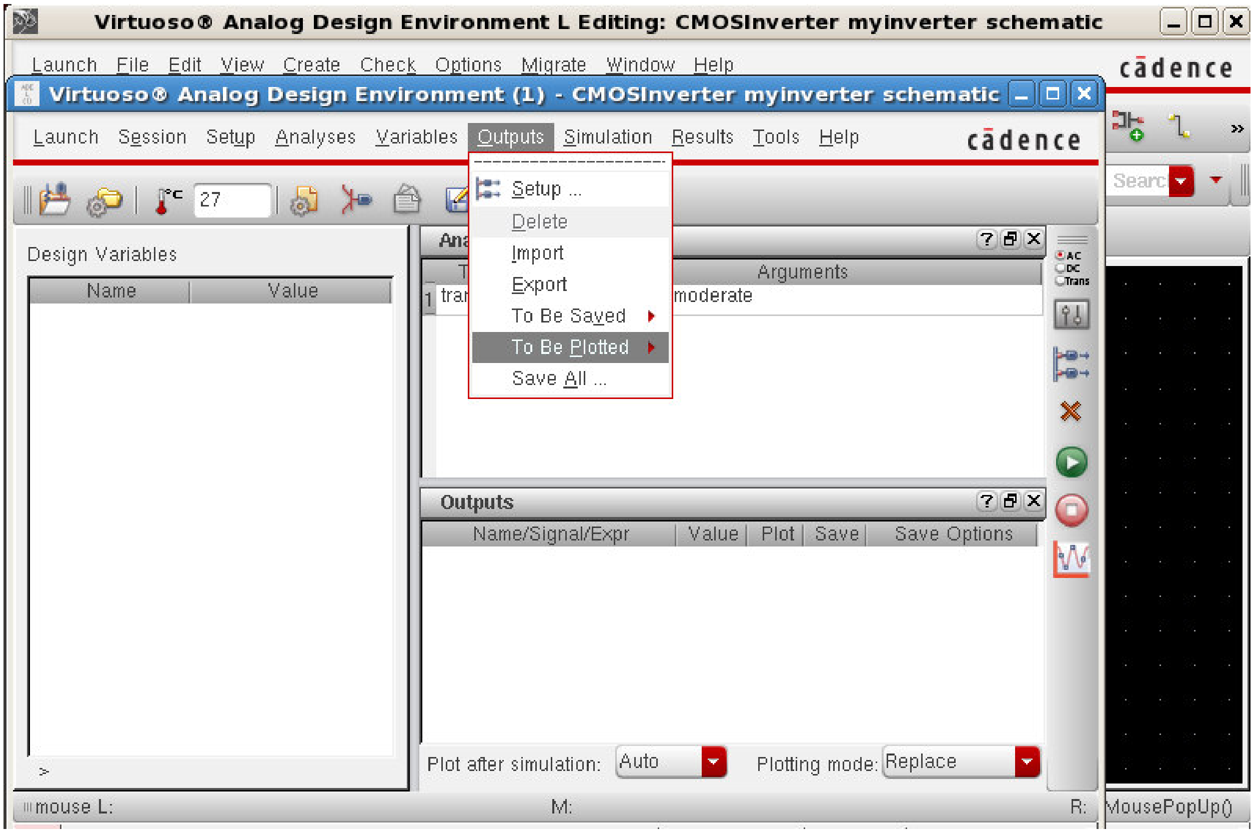

Fig. 15 ADE for Output plot

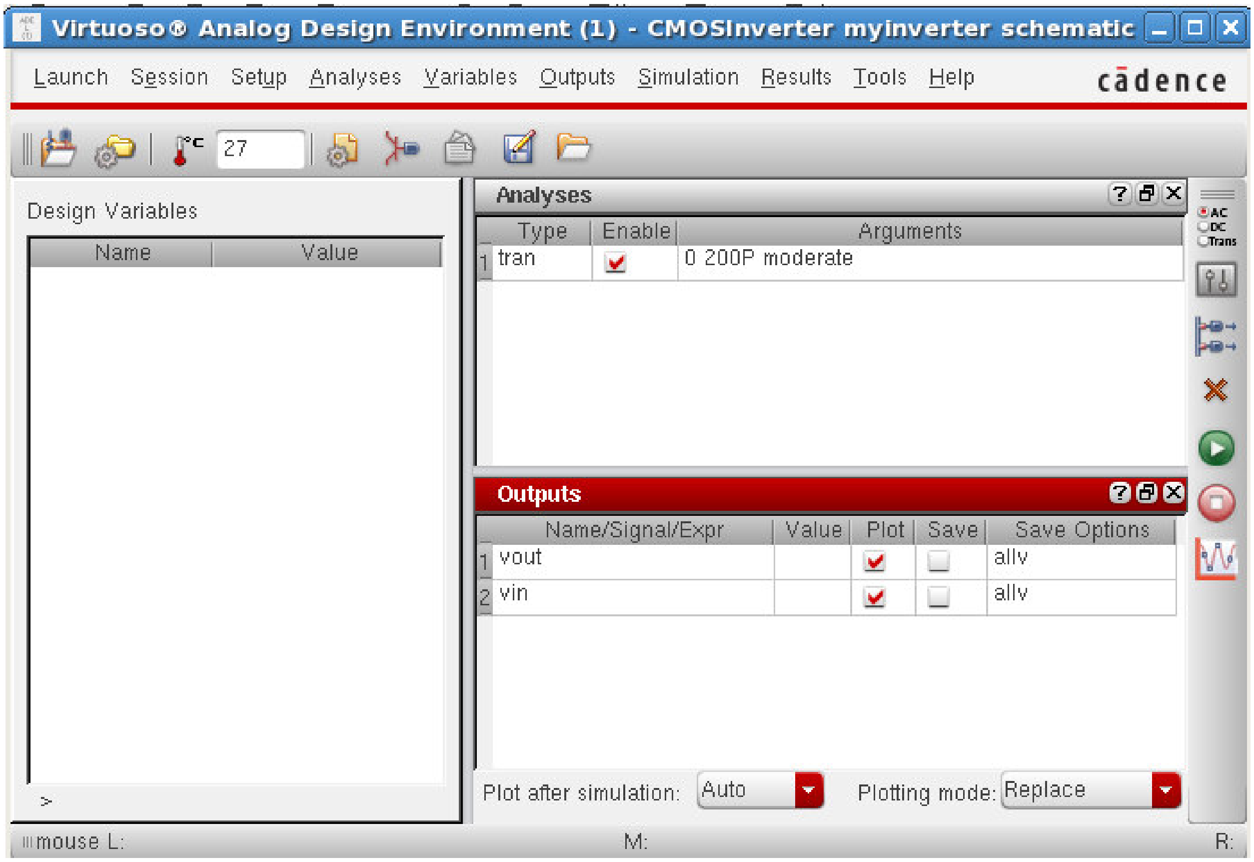

Fig. 16 ADE after setup

Now we can finally simulate! Click on the "Netlist and Run Simulation" button (looks like a green light) on the right or go to "Simulation -> Netlist and Run".

It will start simulation. You will need to wait for a while. You should check your "Virtuoso" window for messages while it is running the simulation.

In case you have errors you will need to go back and correct them. This can be tricky!

CAUTION Each time you change the schematic you have to do Check and Save!.

You should finally get the desired simulation results, input and output

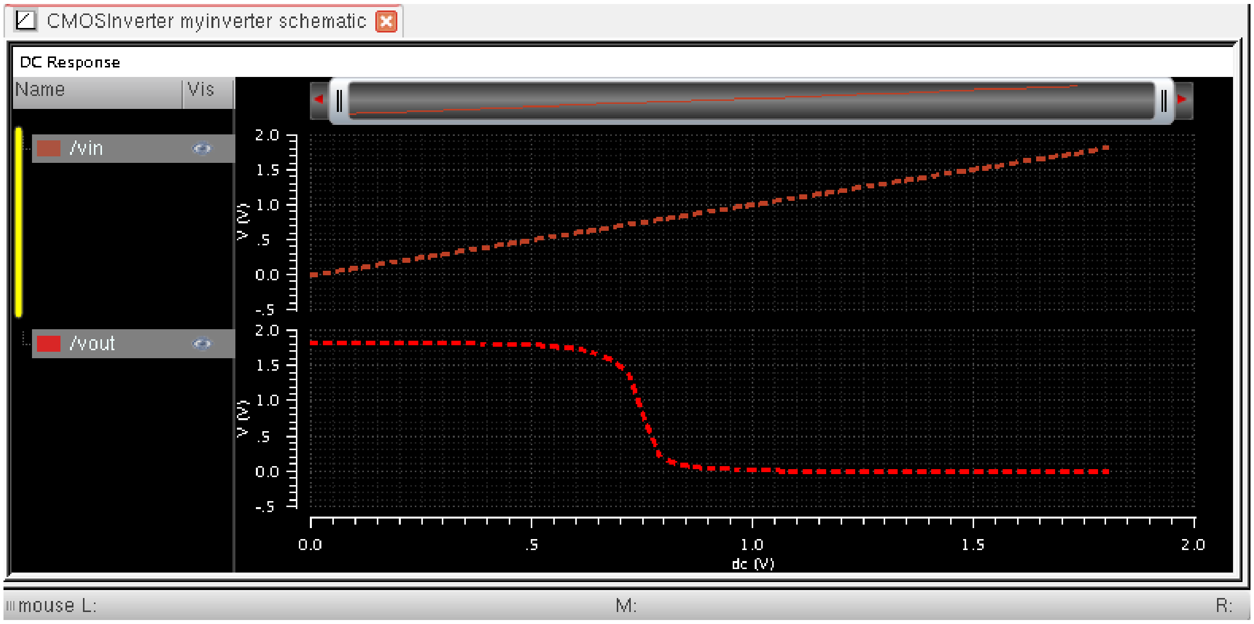

Fig. 17 Simulation Results (DC analysis)

You can zoom in your waveforms.

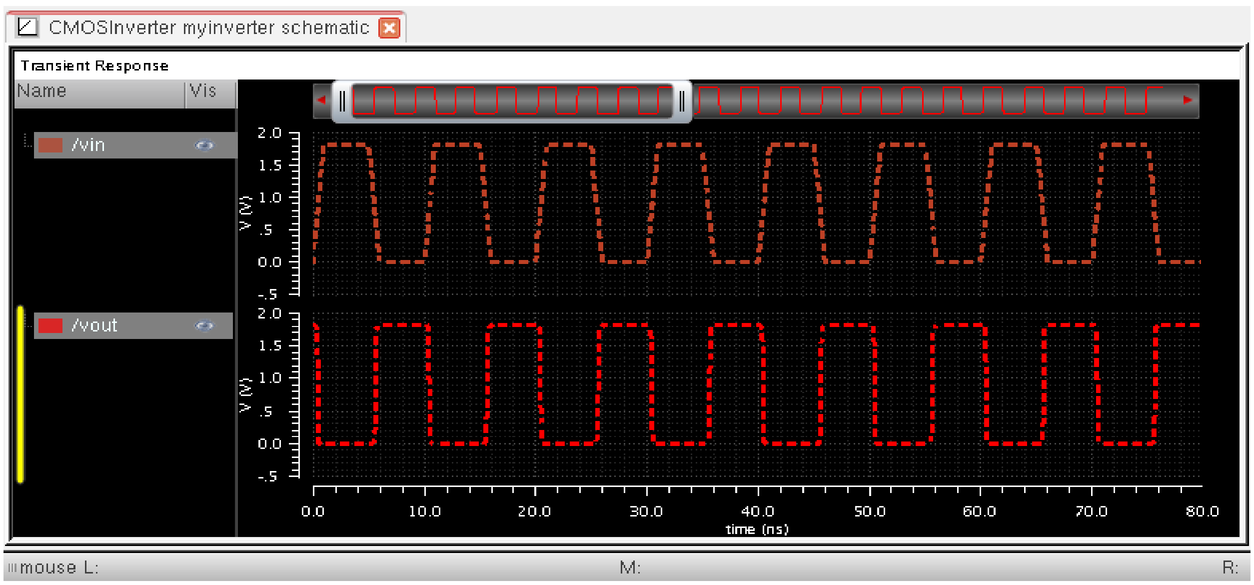

Fig. 18 Simulation Results (Transient Analysis)

1. Symbol generation
2. Delay Calculation (From Schematic)
3. Power Calculation (From Schematic)
4. Area Calculation (From Layout)

Lab Assignment: #2

Design and simulate a 2-input NAND gate. Calculate the area, power, current, and Delay.

Good Luck