Introduction

This tutorial will guide the readers through the basics of the Various Design features and Tools of the Cadence Design framework. It is assumed that the readers are already familiar with how to start the Cadence Environment. Once you successfully open Cadence, you can follow the following procedure.

Types of Libraries:

There are usually 3 types of libraries:

1. A set of common *Cadence libraries*- They come with the Cadence software and contain some very basic components like such as C, R, L, Voltage and Current sources.
2. Libraries that come up with a certain design kit and that are related to a certain technology. These are called as the *Technology libraries*. They can include different components that are specifically being designed /modeled according to their manufacturer's requirements. For example, transistors with a certain model attached
3. *User Designed libararies*, where users can store their designs. They can include components from both the cadence and technological libraries

Library Creation and Selection of Library

To start your work, the first thing you need to do is to open the Library Manager. Go to the Tools→ Library Manager. This will open the following screen.

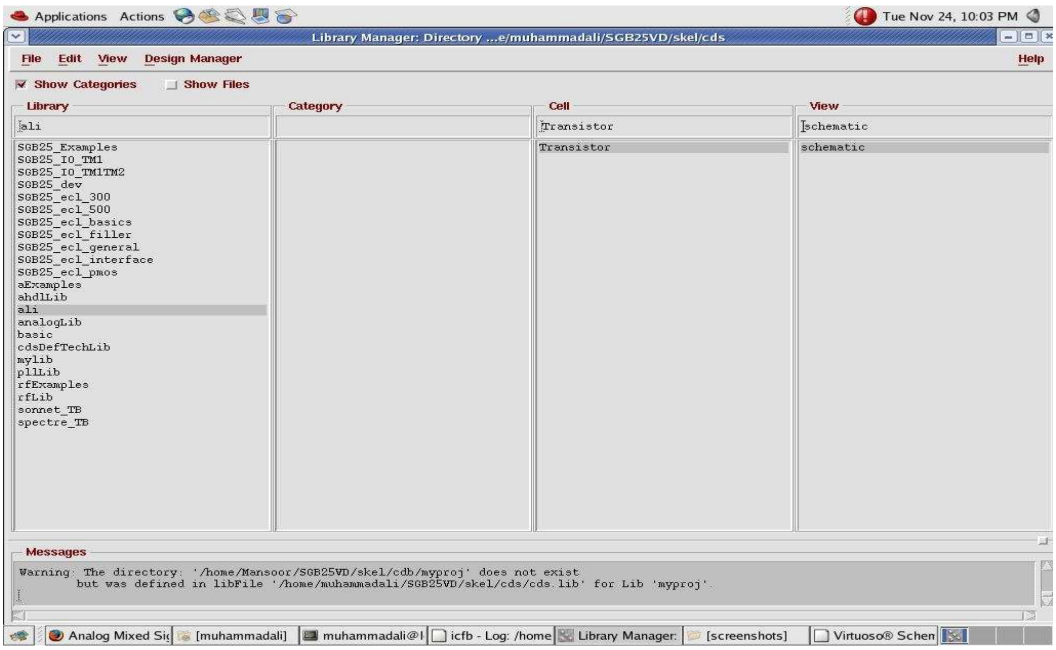


Fig.1: Library Manager

Note that Library Manger is a place which manages all types of libraries namely Cadence, Technology and user defined libraries. In the above window, for example, I have created my own library by the name 'ali' so this will be a user defined library. The above screen also shows you the Technology and Cadence libraries. As you can see, 'SGB25_ecl_basics' is a design kit library. In fact all the libraries starting with the name 'SGB25' are design kit libraries. 'Basic' and 'AnalogLib', on the other hand are Cadence libraries.

Analog Design with Cadence Design Framework

Now we are going to see how can we compose and simulate an Analog design on Cadence.

a) The first thing that you need to do is to create your own library. Open the Library manager from the CIW.

b) Select *File →New →Library*. A new window just like the one shown in the Fig.2 will appear. Enter a name for your library, say *My Library*.

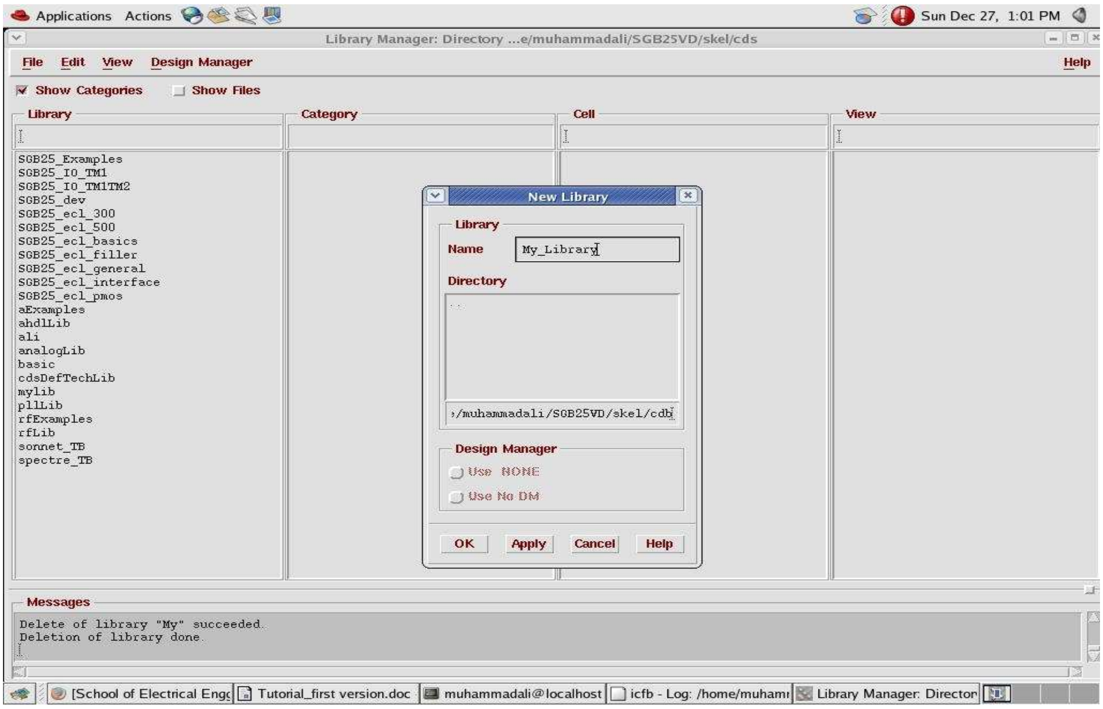


Fig.2: Creating a new library

c) If you want your library to be saved somewhere else other than the working directory, you will have to enter the path for it. Otherwise skip this path and the library will automatically be saved in your working directory.

d) Choose the *'Attach to an existing tech library'* option. As mentioned earlier by doing so your user defined library *My Library* will be attached to this existing technology file which will allow you to use its design kits and models for transistors.

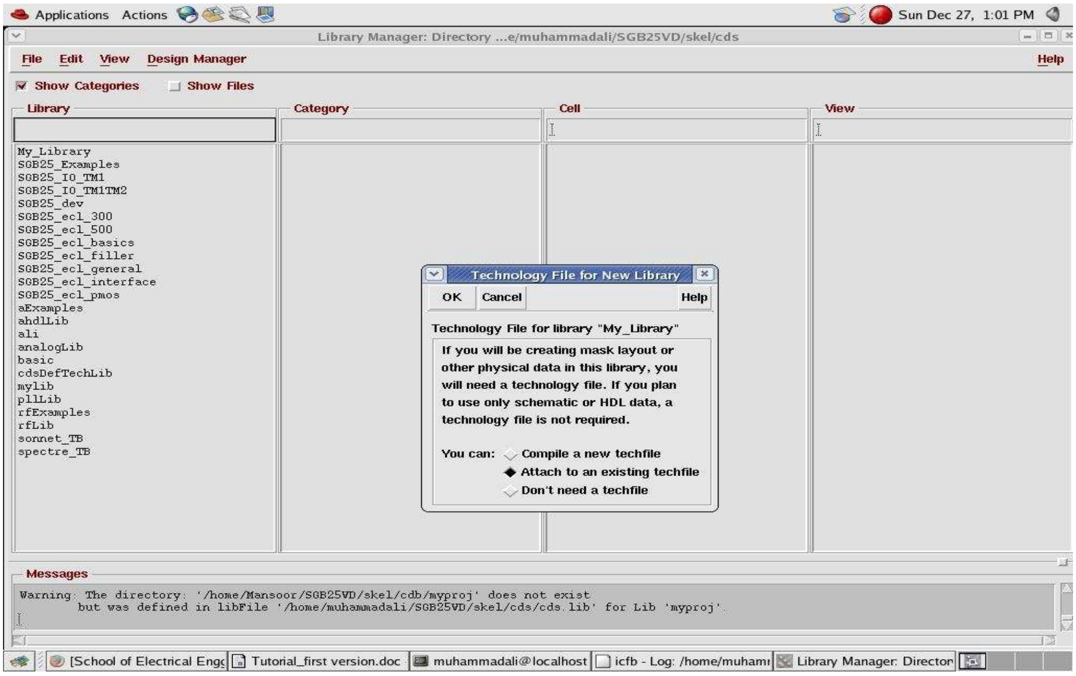


Fig.3: Attaching a new library

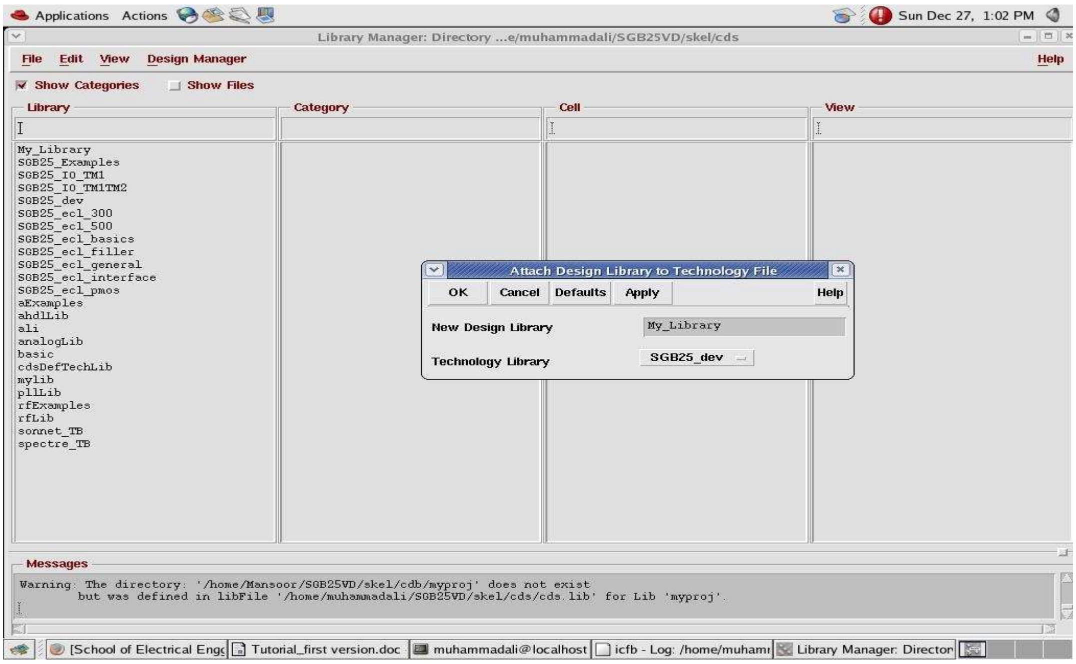


Fig.4: *Attaching a new library*

Schematic Entry with Composer

The most common and traditional method to describe the transistor level or gate level design in Cadence is via the *Composer Schematic Editor*. Schematic Editors provide simple, intuitive way to draw and place different components, wires etc. They actually define the electrical properties of the design like how much voltage or current will be generated at some specific node/point in the circuit. Schematics also include the power supply and ground connections as well as pins and interconnections. Some properties of different components in a schematic remain fixed throughout the design process like the power supply voltage, ground connections at different levels while some properties need iteration

a) Click on your library that you have just created
b) Go to *File*→ *New*→ *Cell view*.
c) Enter the name of the Cell say Transistor
d) Choose *Composer- Schematic* as the Tool. View name should be *Schematic*. We can other views too like Layout, Verilog etc by choosing the appropriate tool while defining the new cell
e) Click OK.

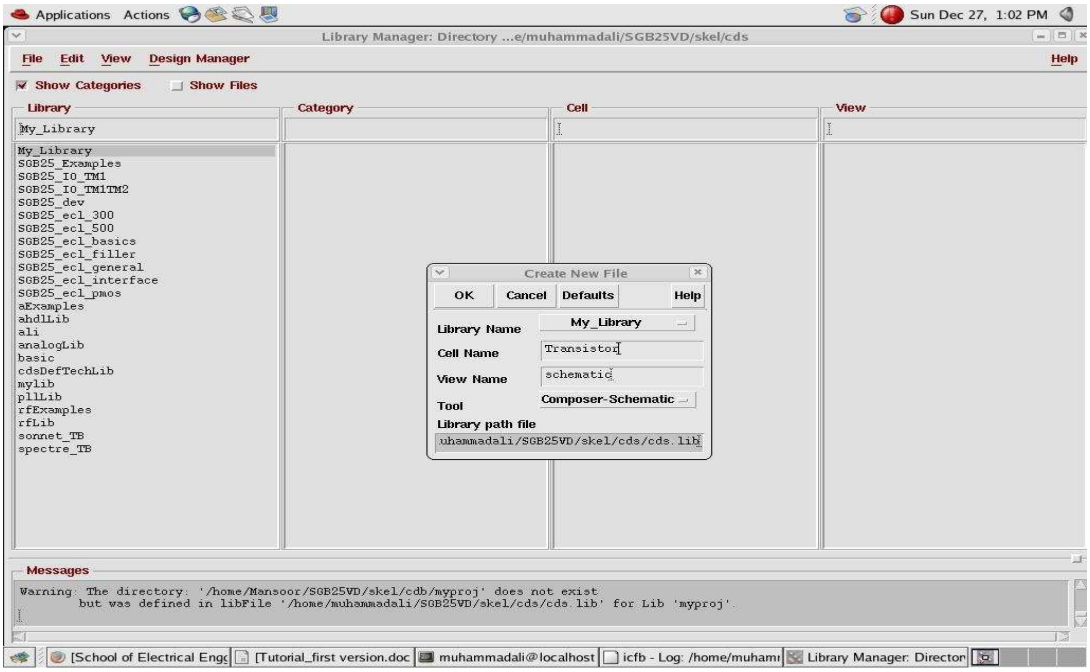


Fig.5: Creating a cell view

Double clicking on the cell view will open the schematic in the Virtuoso Design Environment. Here you can now insert different components according to your own requirements. You can perform multiple tasks by using the icons (functional keys) in the schematic window. Basically you can:

- *Create Components* by selecting the instance icon and browsing in the pop-up window through different libraries.
- *Wire Components* select the wire, click to the first component and drag it to the next one.
- *Set Instance properties* you can modify the properties of your instance any time you desire. These properties can be different parameters/values of the instances e.g. you can change the length and width of the transistors.

So let's say that we have to create a simple NMOS transistor schematic so that we can simulate it in order to observe various properties and graphical analysis. For that, you need to follow the following steps:

- Press *'i',* Browse to the AnalogLib and select 'vdc'. Then put it on the schematic.

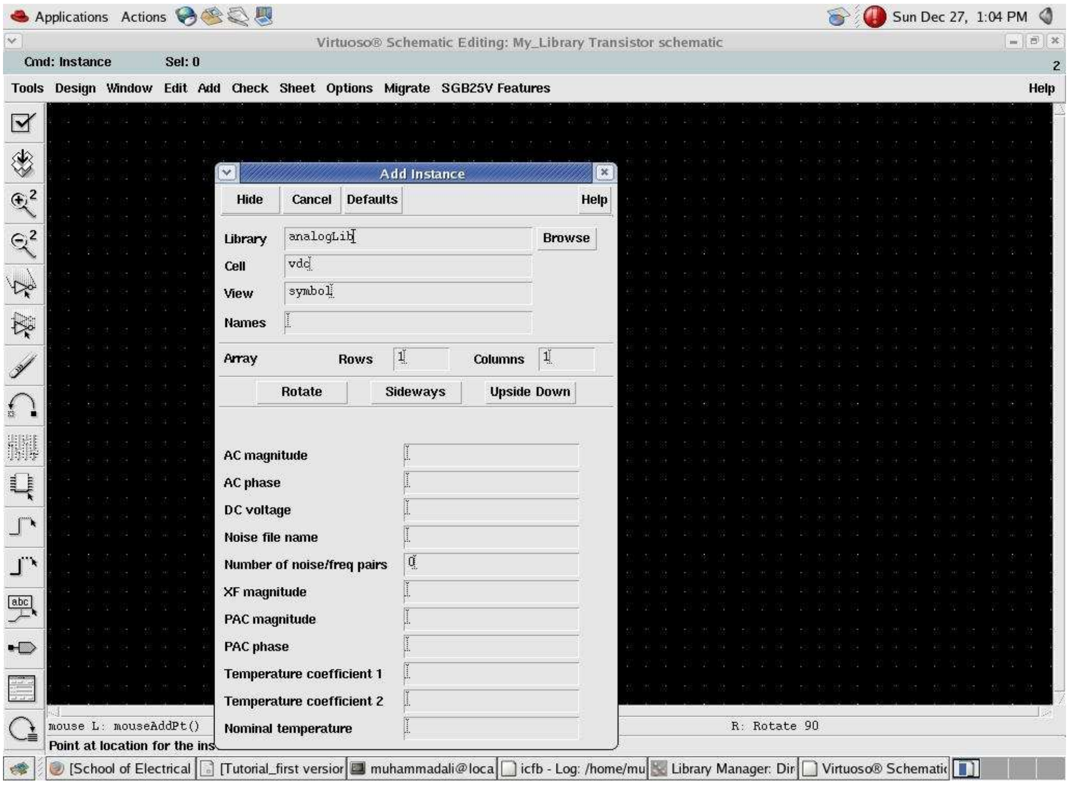


Fig.6: Adding an instance

- Follow the same procedure in order to complete your design which consists of 'gnd', vdc, NMOS transistor.
- Connect the components together by clicking *'w'* and joining the parts together.

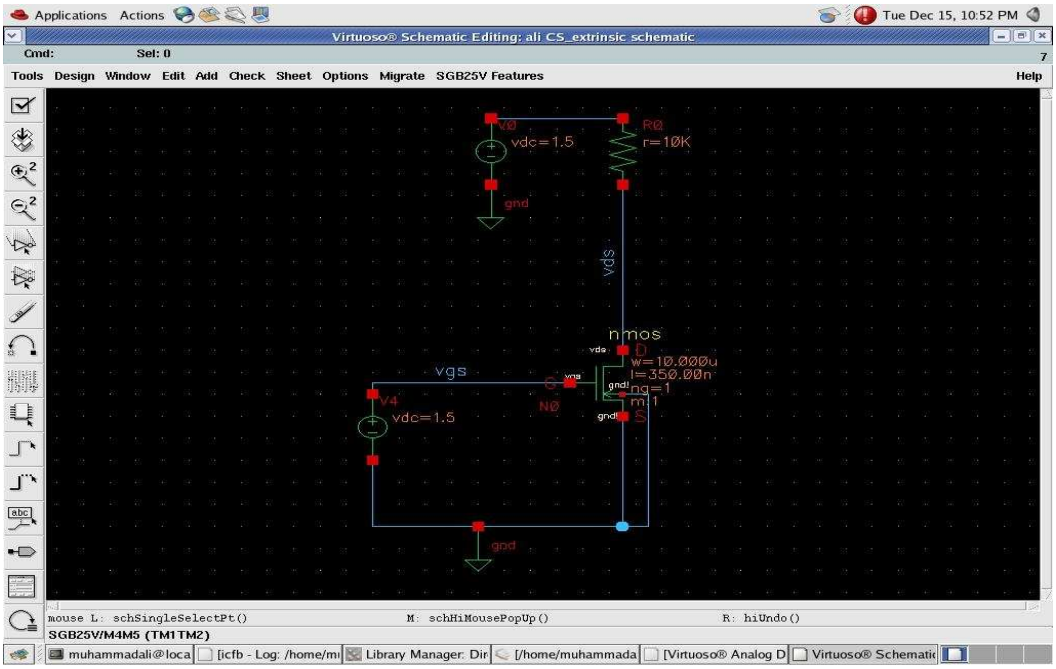


Fig.7: The circuit

- If you want to change the properties or values of the components, you can select that component by clicking on it and then pressing *'q'* will lead you to another window where you can modify the properties of that component. For example, if you want to change the Length of the NMOS transistor in the above figure, click on the transistor, then press *'q'*, you will get a new screen where different parameters of the transistor are already mentioned like the width, length, and finger.

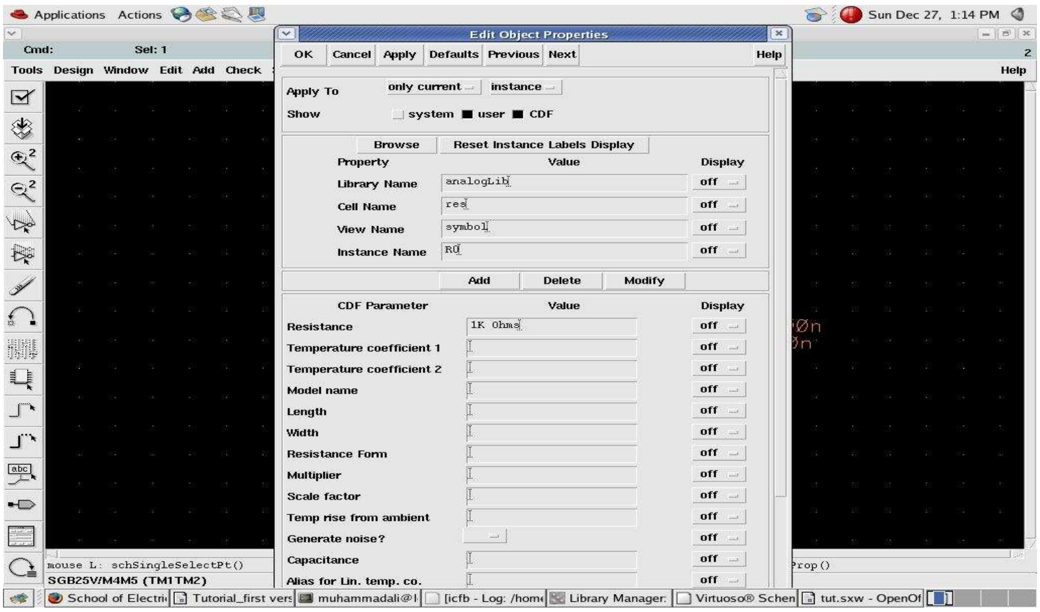


Fig.8: Object properties

➢ Now click on the length and change it accordingly, then click OK. After every change in the circuit schematic, don't forget to save it by clicking 'Design→Check and Save'. You can also label your wires. For that, press *'l'*, write down the name and click on that wire you want to label.

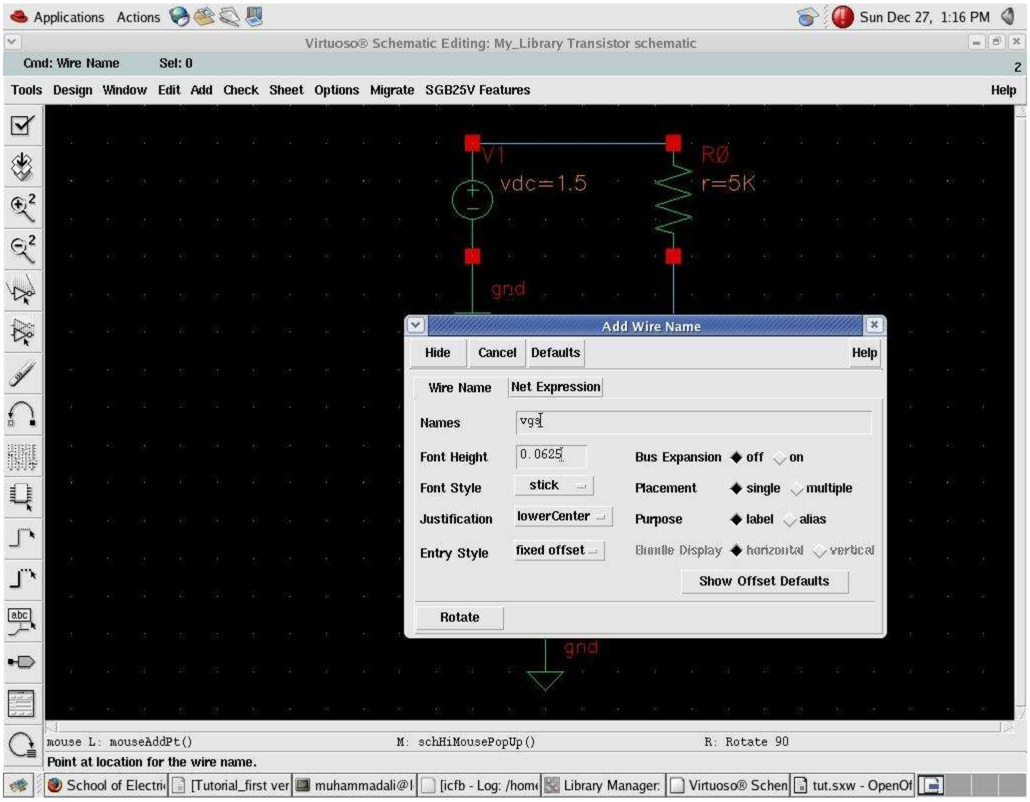


Fig.9: Labeling the wire

How to Simulate the Circuit

Supposedly, you have designed the same schematic that I have shown you in Fig. 7.Now you want to get it simulated so that you can observe the 'time domain' analysis and can also perform the 'ac' and 'dc' simulations. This can be done in the following way by using the *Analog Design Environment (ADE).*

DC Analysis

First we will perform the 'dc' simulation by using the following procedure.

1. In the Virtuoso Schematic Editor, go to the *'Tools →Analog Design Environment'*. The ADE screen will appear as is shown in the Fig.10.

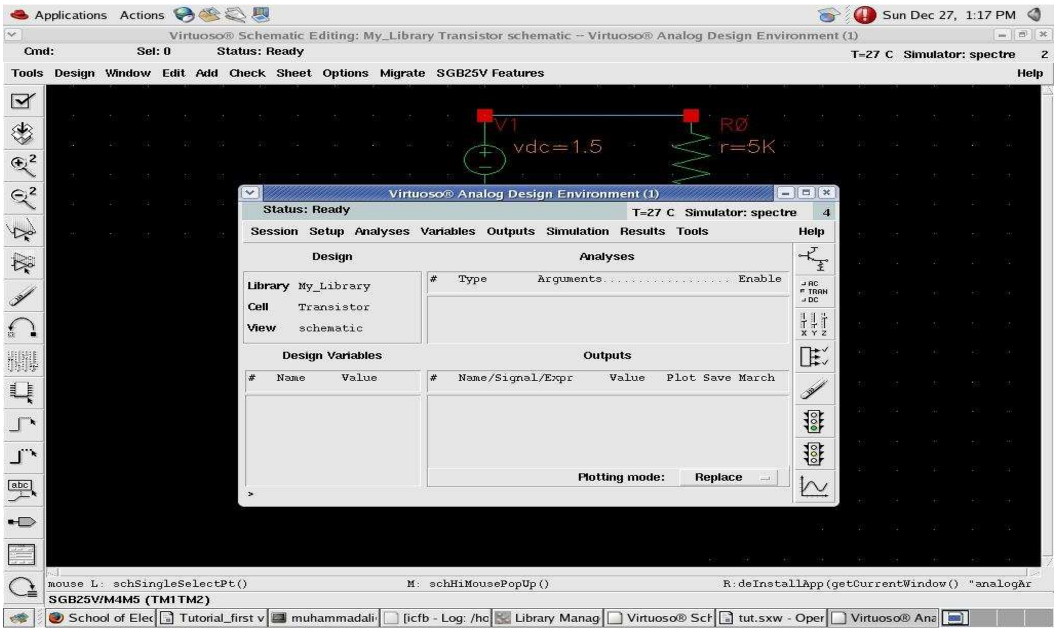


Fig.10: Analog Design Environment (ADE)

2. Now go to Analysis → *Choose*. A window will appear asking you the type of analysis that you need to perform. As you can see, there a numerous types of analysis you can perform using ADE. Some of the very common are 'dc', 'ac', 'transient', 'noise' etc. As we need to perform the dc analysis, so click on the 'dc'. Then click on 'save dc operating points. Then click on 'OK'.

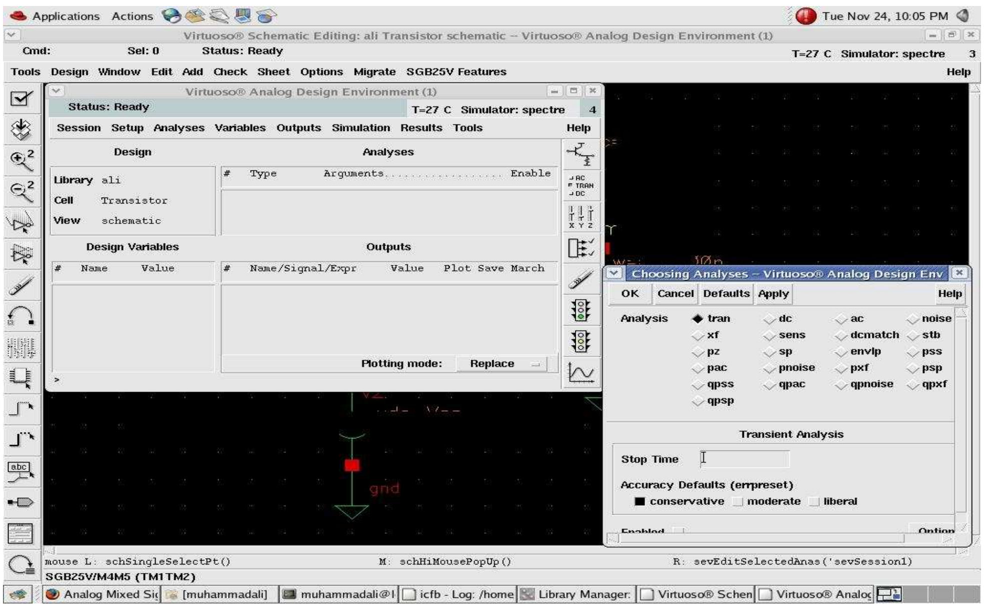


Fig.11: Analog Design Environment showing the types of analysis we can do

3. On the ADE window, you will now be having dc type in the analysis section with the argument’t’. Now click on simulation, and then click *'netlist and run'*. The simulation will take place in another window just like the one shown below in Fig.12.

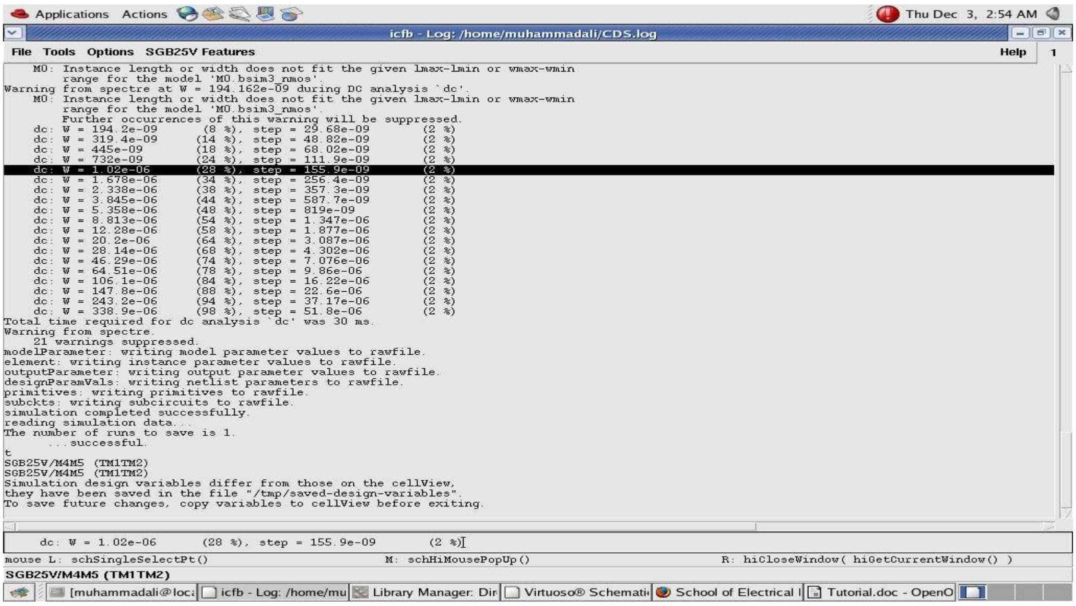


Fig.12: Simulation Window

4. You can change the properties of the components if you like. E.g. if you want to change the length or width of the transistor, select the transistor press ‘q’, then change the length and width in its properties.

5. Once the simulation is completed, you can figure out all the dc properties associated with your circuit. Go to 'Results→Print→DC Operating Points’. The Virtuoso schematic Editor window will then appear showing your circuit. Then click on any component to determine the DC operating point of that component. For example, by clicking on the NMOS transistor 'N0' in the circuit, a new window will appear containing all the dc parameters of the transistor 'N0'.

6. Click on 'Results→Annotate→DC Operating Points’. All the node voltages and currents will then appear on the circuit depicting the dc quantities in the Virtuoso Schematic Editor window.

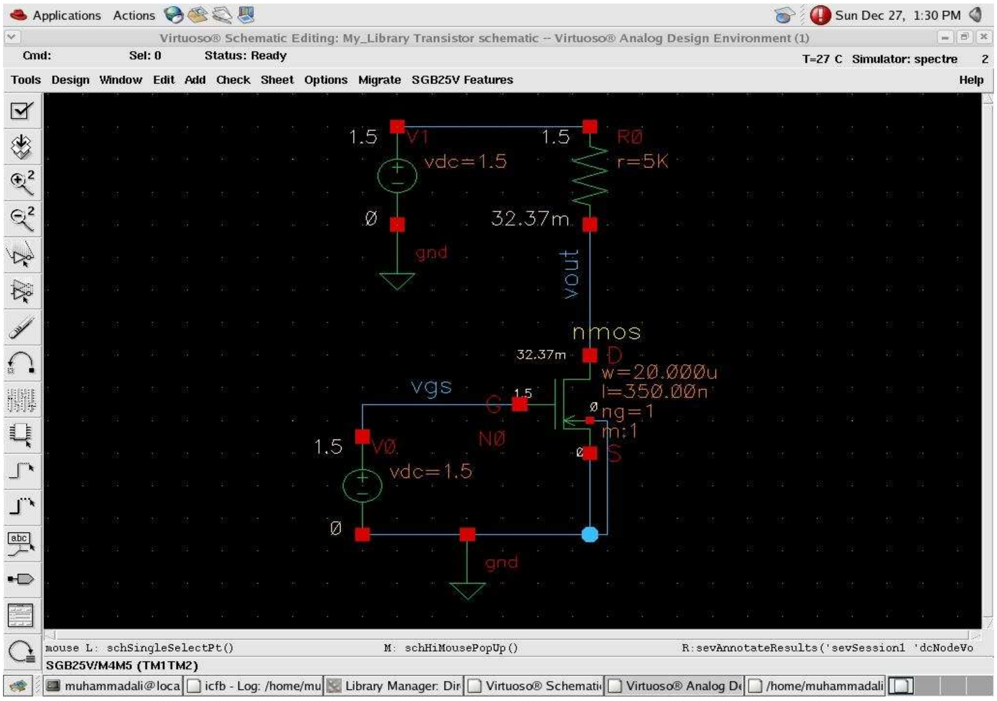


Fig.13: DC Point Annotated

Transient-Analysis

If we want to operate the 'Transient Analysis' of the circuit, we can follow the same procedure that we followed in doing the dc analysis. But before doing that, we need to slightly modify the circuit in the Virtuoso Schematic Editor as is written below.

1. Replace a constant dc source with a time domain sinusoidal input source having a specific amplitude and frequency as shown in Fig.14. For that press 'i', browse to the AnalogLib, select 'vsin' and put it there in the schematic window.
2. The rest of the circuit will remain the same.
3. Once you complete the circuit, go to Analysis→Choose→Transient. Set the parameters according to your requirements.
4. Save the Design
5. Open the Analog Design Environment (ADE) again.
6. Click on *Analysis*→*Choose*→*trans*
7. We need to specify the time for which the simulation is to be run. For example, if you want to do the transient simulation for 5usec then you will have to specify 5usec in the stop time column.
8. Click OK.

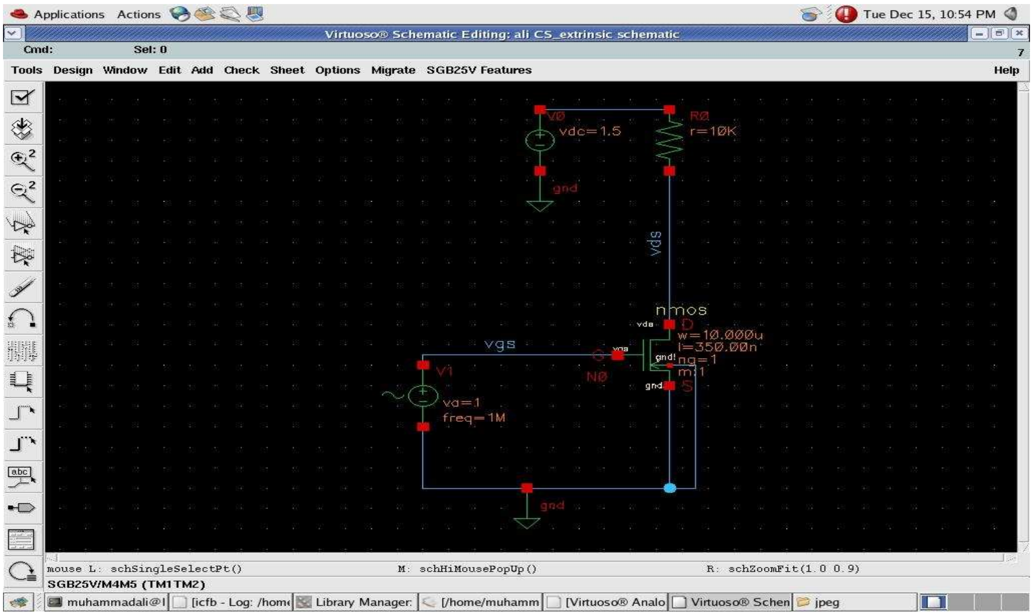


Fig.14: Circuit for doing the transient analysis

- In the ADE, you will see that the transient analysis will appear in the window.
- Next, go to the 'Outputs→to be Plotted→Select on Schematic Output'. It will take you to Virtuoso Schematic Editor where you have saved your schematics. You can then click any wire, label you want to plot. E.g. I want to see both the input and output transient waveforms. So I had labeled both the wires like 'vin' and 'vout' respectively as is shown in the figure and then I clicked on these 2 labels. You will see that in the ADE, these will appear in the output section.

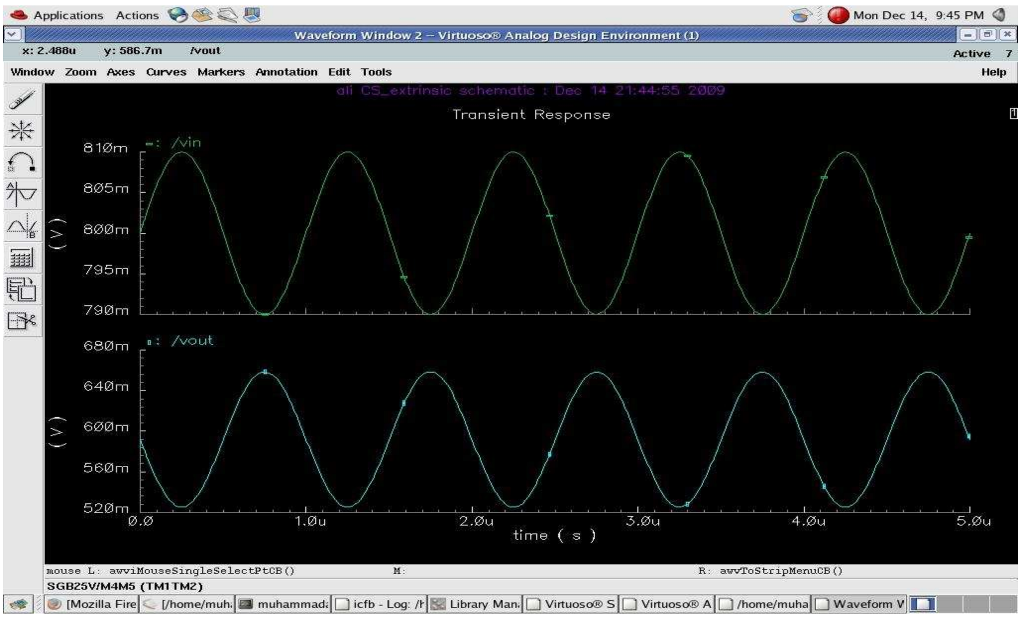


Fig.15: Time domain waveforms

- You can now run the simulation and both the waveforms will appear once the simulation will be completed.
- In the waveform window, go to 'Axes' and click on 'To Strip'. Both the 'vin' and 'vout' will appear on separate graphs as you can see in the Fig.15.
- With the help of Tools in the ADE, you can determine many important things. E.g. once the transient simulation is done, click on the 'Results' in the ADE, and click on '*Print → Transient Operating Points'*. This will print the operating points in terms of voltages, currents and power on the Virtuoso schematic Editor as is shown in fig.7.

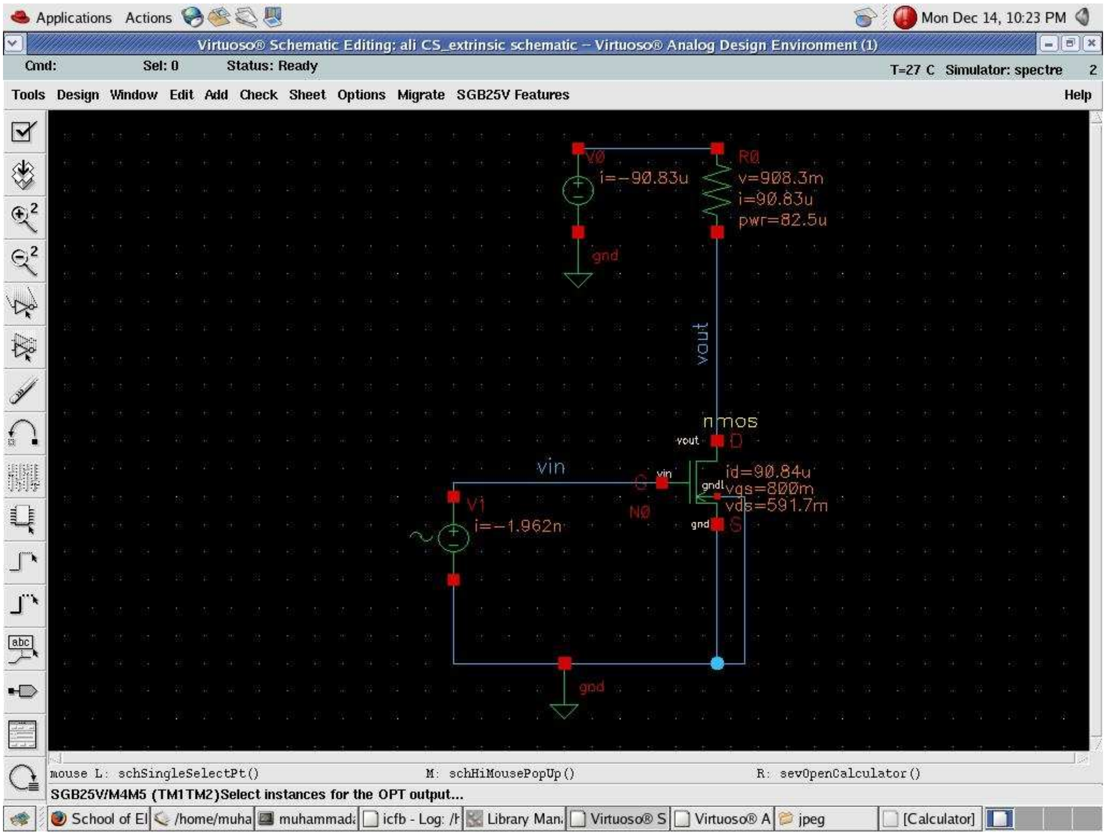


Fig.16: Transient Operating Points

AC Simulation

In order to figure out the frequency response of the circuit, we need to do the ac simulation. The procedure is same.

- Click 'Analysis→Choose→ac'.
- Select 'Frequency' as the sweep variable. Mention the sweep range. E.g. I have mentioned start at '1' and stop at '500MHz'.
- Click OK.

➢

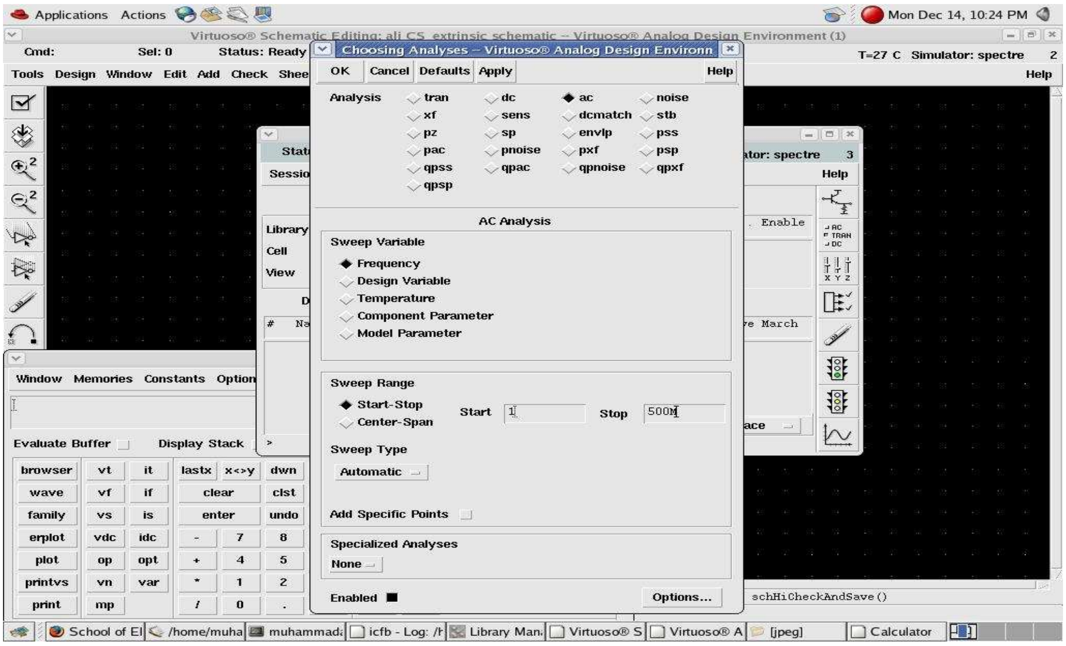


Fig.17: AC Simulation setup in ADE

So now if we want to display the gain of the circuit, go to 'Outputs→Setup'. In the new window, you can write down the expression *'db20 (abs (VF ("/vout")/VF ("/vin")))'*, name it as 'gain (db)'. Now add it and click on OK. Next run the ac simulation, you will see the following output. This is the frequency response of the circuit.

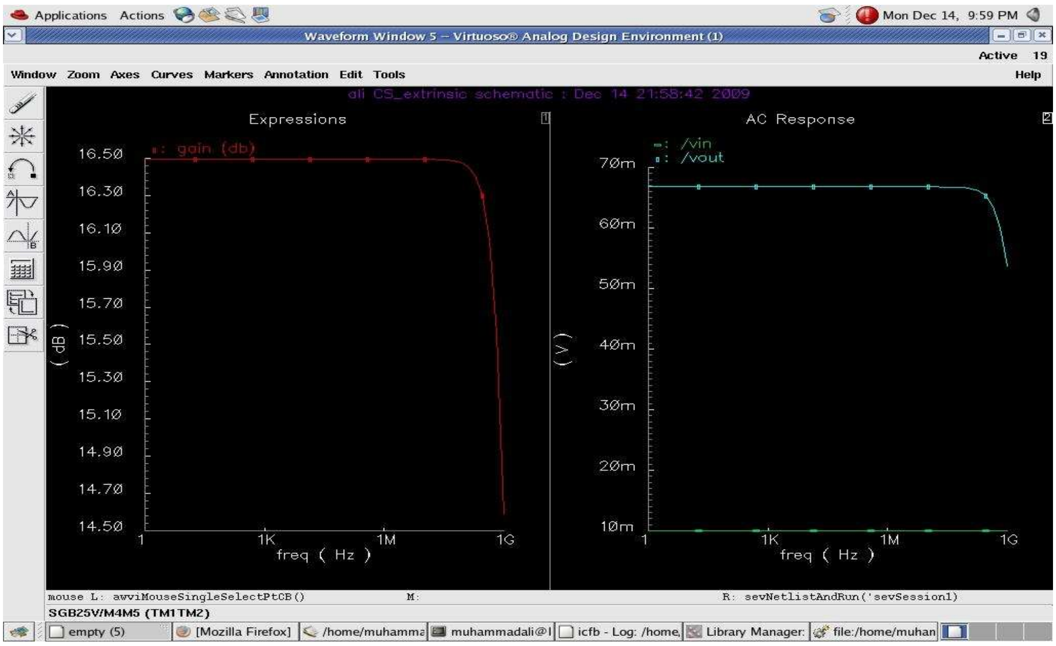
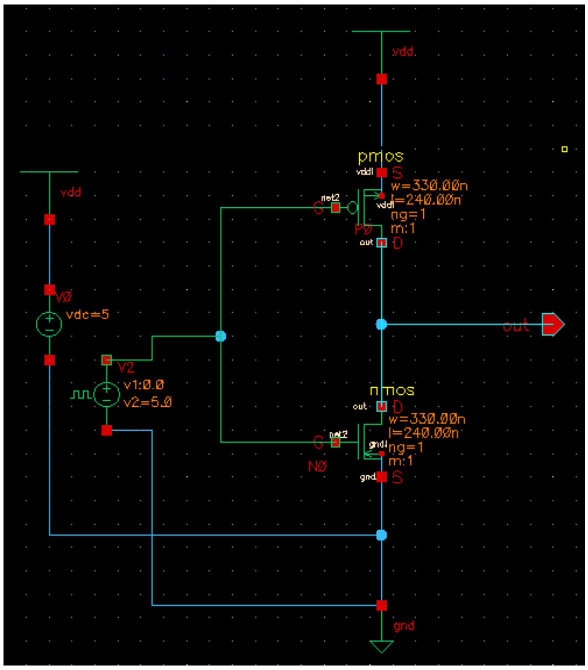


Fig. 18: Frequency Response and the input and output voltages

Inverter

- Press *'i'*, Browse to the SGB_Dev and select 'nmos' and a 'pmos'. Then put it on the schematic.
- Follow the same procedure in order to complete your design which consists of 'gnd', vdc.
- Add V_pulse source is added which is present in analoglib.
- Connect the components together by clicking *'w'* and joining the parts together.



Transient-Analysis

If we want to operate the 'Transient Analysis' of the circuit, we can follow the same procedure that we followed in doing the dc analysis. But before doing that, we need to slightly modify the circuit in the Virtuoso Schematic Editor as is written below.

1. Once you complete the circuit, go to Analysis→Choose→Transient. Set the parameters according to your requirements.
2. Save the Design
3. Open the Analog Design Environment (ADE) again.
4. Click on *Analysis*→*Choose*→*trans*
5. We need to specify the time for which the simulation is to be run. For example, if you want to do the transient simulation for 3n sec then you will have to specify 3n in the stop time column.
6. Click OK.

Next, go to the 'Outputs→to be Plotted→Select on Schematic Output'. It will take you to Virtuoso Schematic Editor where you have saved your schematics. You can then click any wire. So click on the wire having input voltage and the wire connected to the output respectively as and then press escape. You will see that in the ADE, these will appear in the output section.