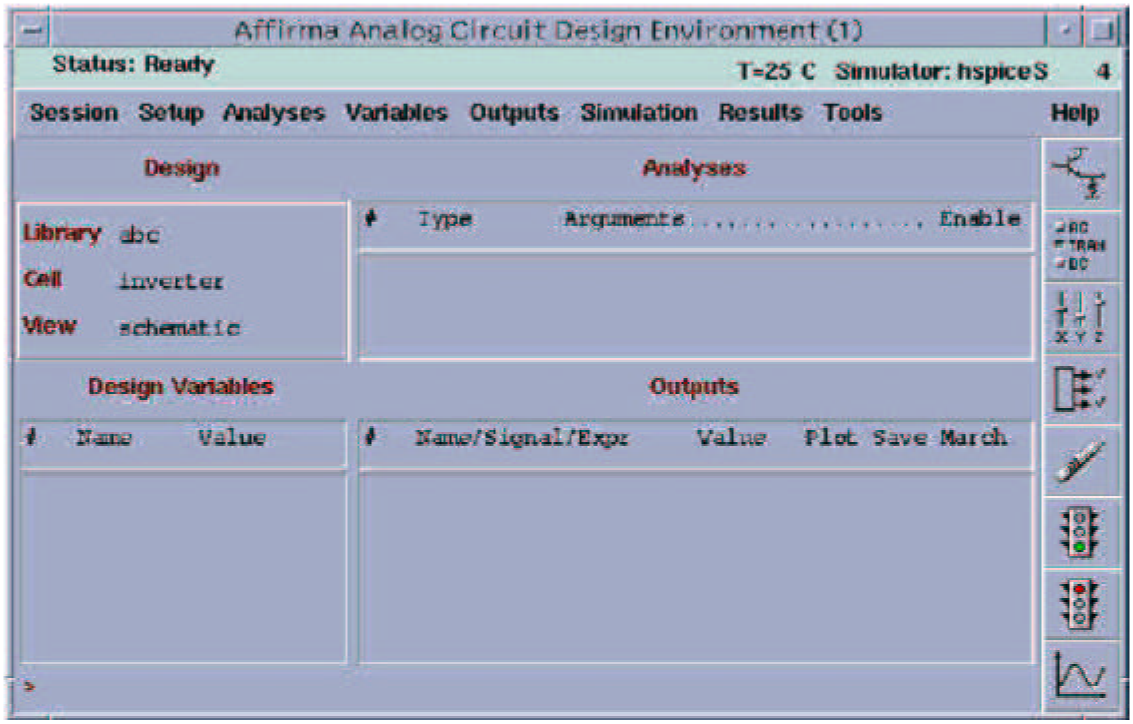
Circuit Simulation using Spectre

In order to do circuit simulation in Cadence, we need to setup ***Analog Artist*** first.

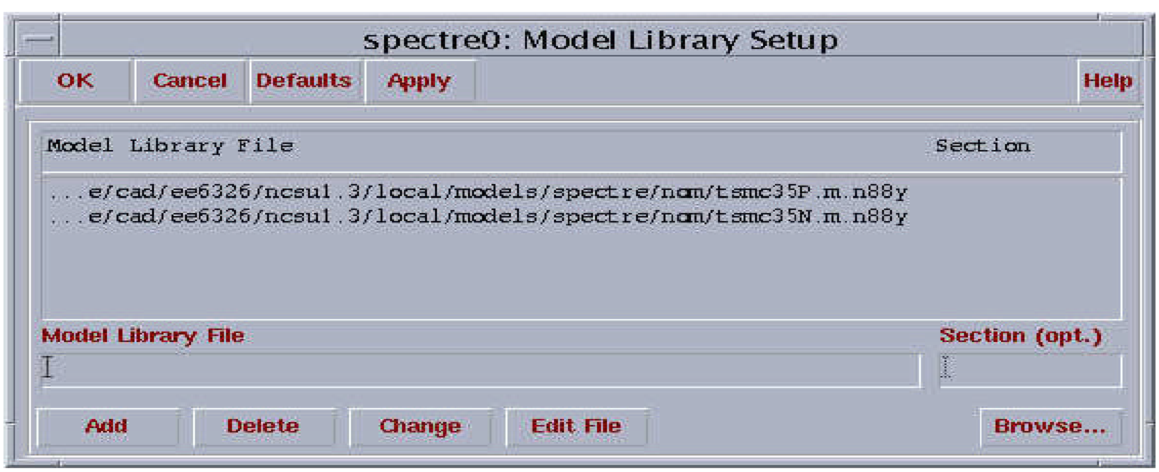
1. Go to **Tools** in Composer-Schematic menu and choose ***Analog Environment*** to open Analog Environment.



2. Go to ***Setup*** option in ***Analog Environment*** and choose the option **Model Libraries** and enter the following lines and click **Add**:

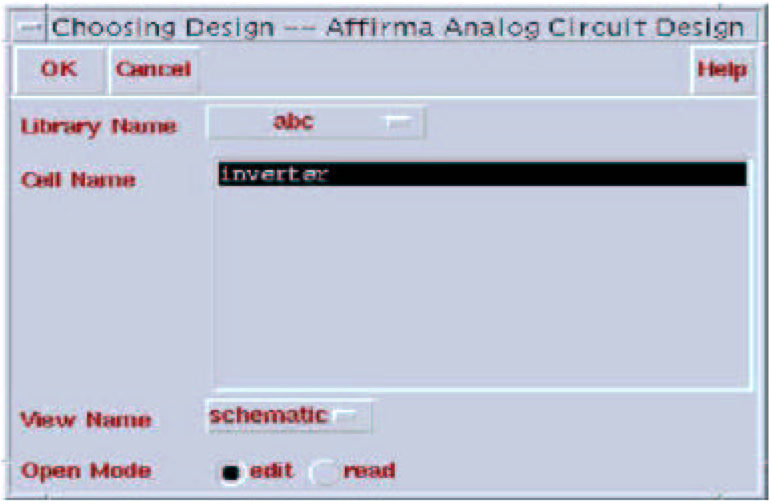
/home/cad/ee6326/ncsul1.3/local/models/spectre/nom/tsmc35P.m.n88y

/home/cad/ee6326/ncsul1.3/local/models/spectre/nom/tsmc35N.m.n88y

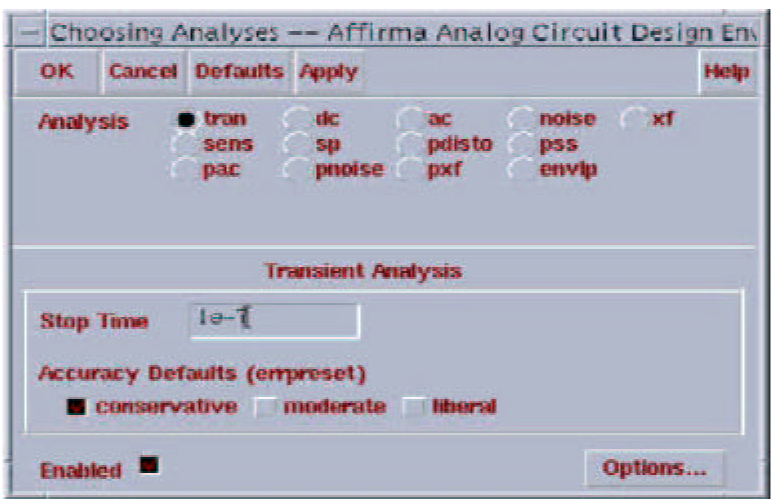


A. Transient Analysis

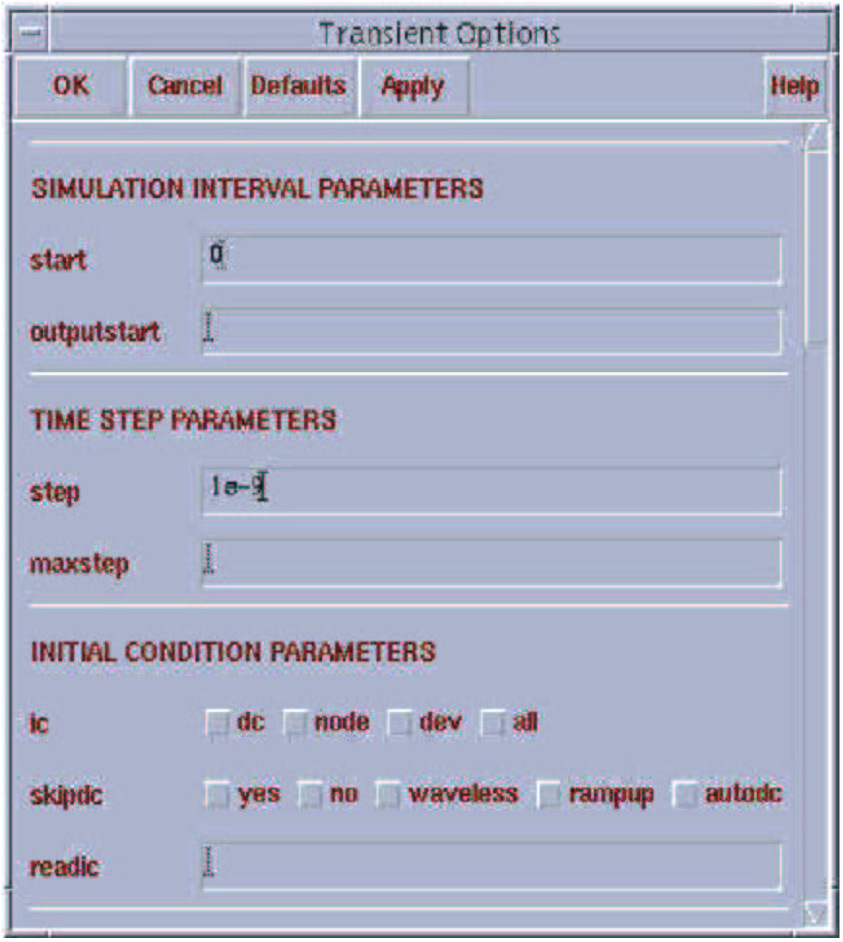
1. In the ***Analog Environment*** Window go to ***Setup*** and select ***Design*** and the Design window opens.



3. Once this is done choose the type of analysis you wish to perform. In this case now we are going to do a Transient Analysis i.e. plotting a time-voltage curve. Click on ***Analyses*** in the ***Analog Environment***.



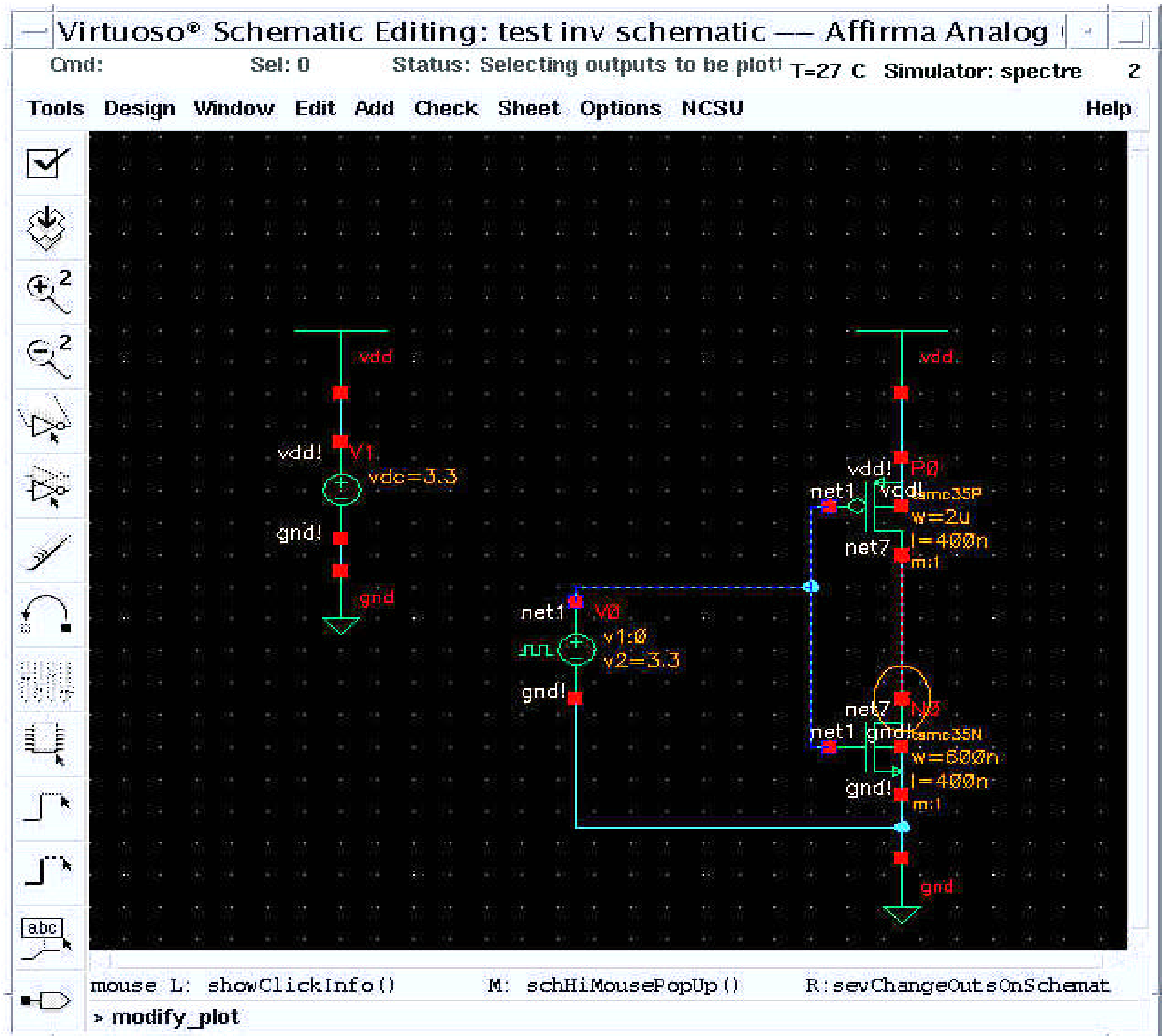
4. In the Choosing Analyses Window, select **tran** and enter the stop time as 1e-7. Choose **conservative** under ***Accuracy Defaults***. Make sure **Enabled** is ON. Then click on options.



5. Under this input the start time as "0" and the step time equal to 1e-9. This means that the transient analysis will start from time t=0 and go until t=100ns having a step time of 1ns.Click **OK** in the Transient Options window and finally click **OK** for the Choosing Analyses window. These values can be fixed according to your requirements.

6. In the Analog Environment Window go to ***Outputs*** and select ***Save All.*** To plot ***select signals to output*** , and select allpub and click OK. This will save the voltage values of all the nets. However this is not advisable for a big design since the simulation would consume a lot of time just deselect allpub and choose ***selected.*** Then select only those node voltages and branch currents which you require.

7. In order to observe the voltage at particular node, go to select ***Output, to be saved Select on schematic,*** and then click on the particular wire. The wire will be highlighted to indicate that it is selected. In order to plot the current, select the drain node.



8. Then to Analog Environment Simulation Window, select **Output, *to be plotted, Select on Schematic***, and then select the output node in your schematic. The voltage node will be highlighted and the current node will be denoted by a circle as in the earlier case above.

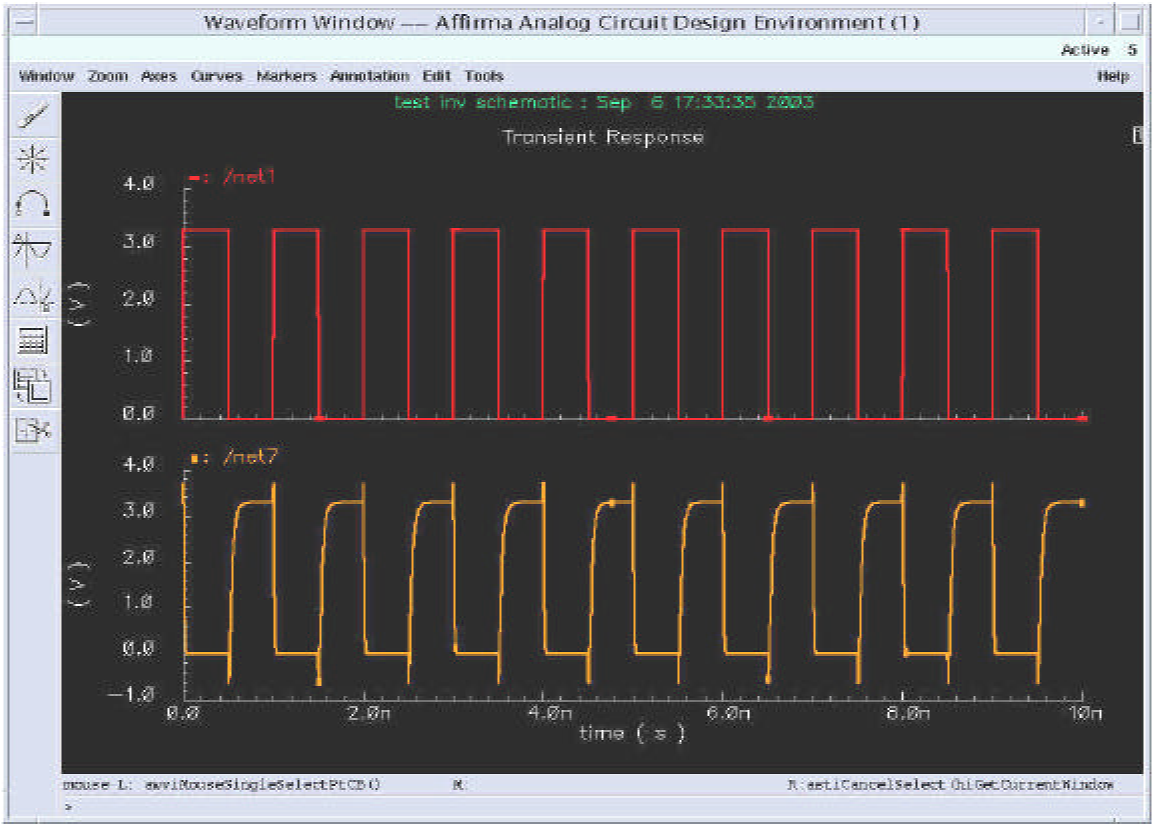
9. Once this is done, go to **Simulation** on the ***Analog Environment*** Window and select ***Netlist and Run.***

10. Wait for a few seconds and after the simulation is finished, a waveform window will automatically appear.

NOTE: In case if the designed circuit is huge and if the no. of outputs to be plotted is large, the simulation might take some time. Also sometimes it might appear that

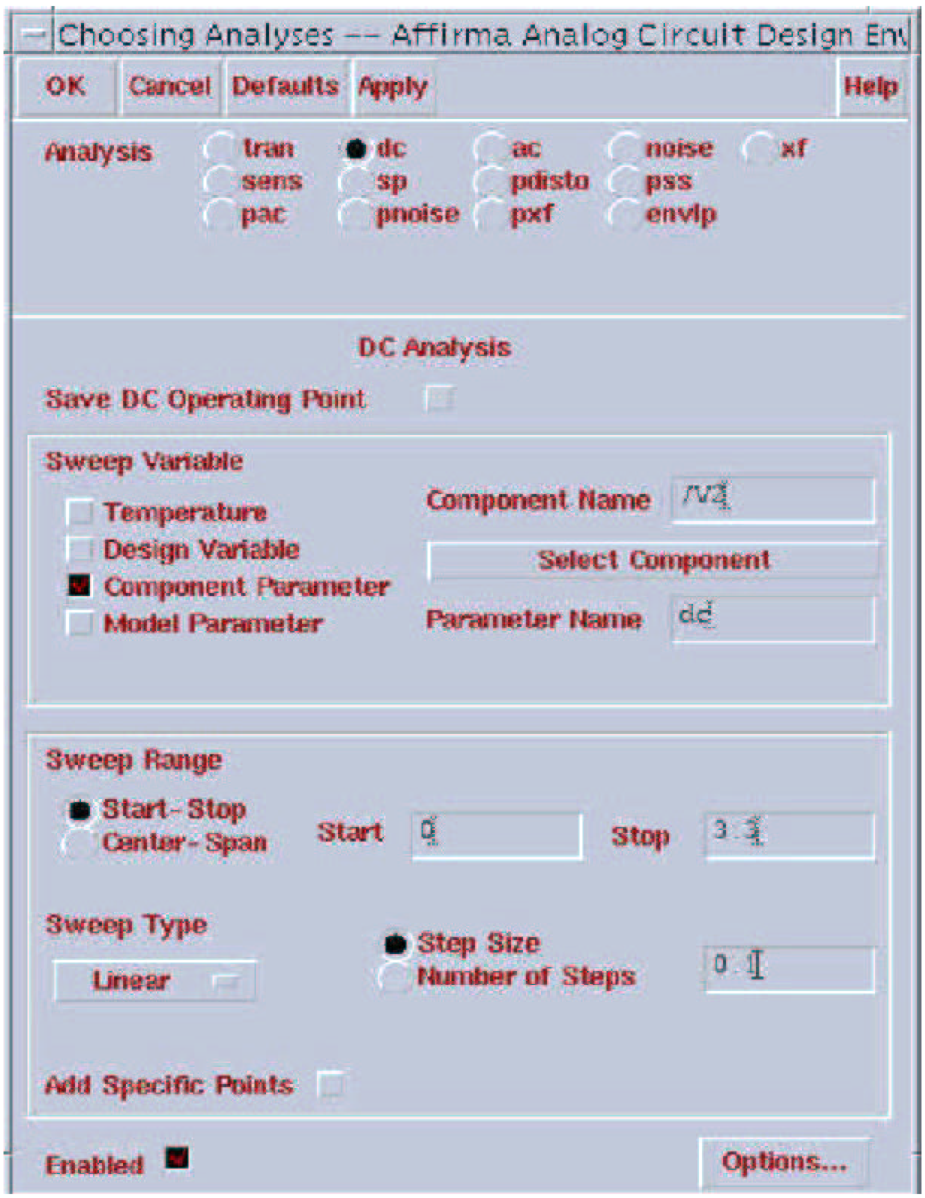
there is no output waveform appears .This means that there is a problem in your circuit you designed or you have made a fundamental mistake or short circuited some node. So always check twice before running your simulation.

11. Another way of opening the Waveform Window is going to the ***Results,*** and select ***Plot Outputs*** and choose ***Transient.***

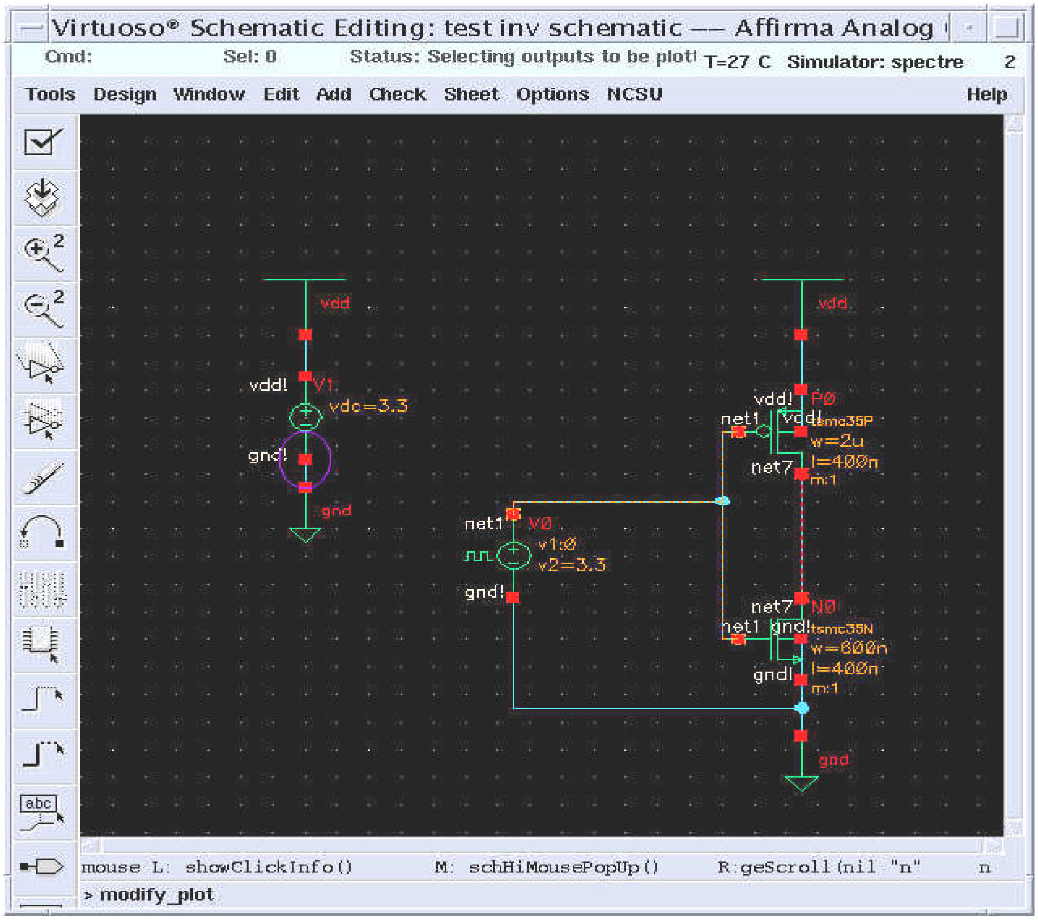


B. DC Analysis

1. In order carry out a DC Analysis, go to ***Choose*** in the ***Analog Environment*** Window and in the ***Choosing Analyses*** Window, select **dc**. DC analysis can be carried out for a number of parameters. Under the **Sweep Variable** select ***Component Parameter*** .Then click on ***Select Component*** and go to the schematic and click the component whose parameter you want to sweep. Select what sort of parameter you want to sweep. Then under the **Sweep Range** enter the initial and final values of the sweeping range. Then in **Sweep Type** select ***Linear*** and under step size enter a suitable step size or simulation. Then click OK.



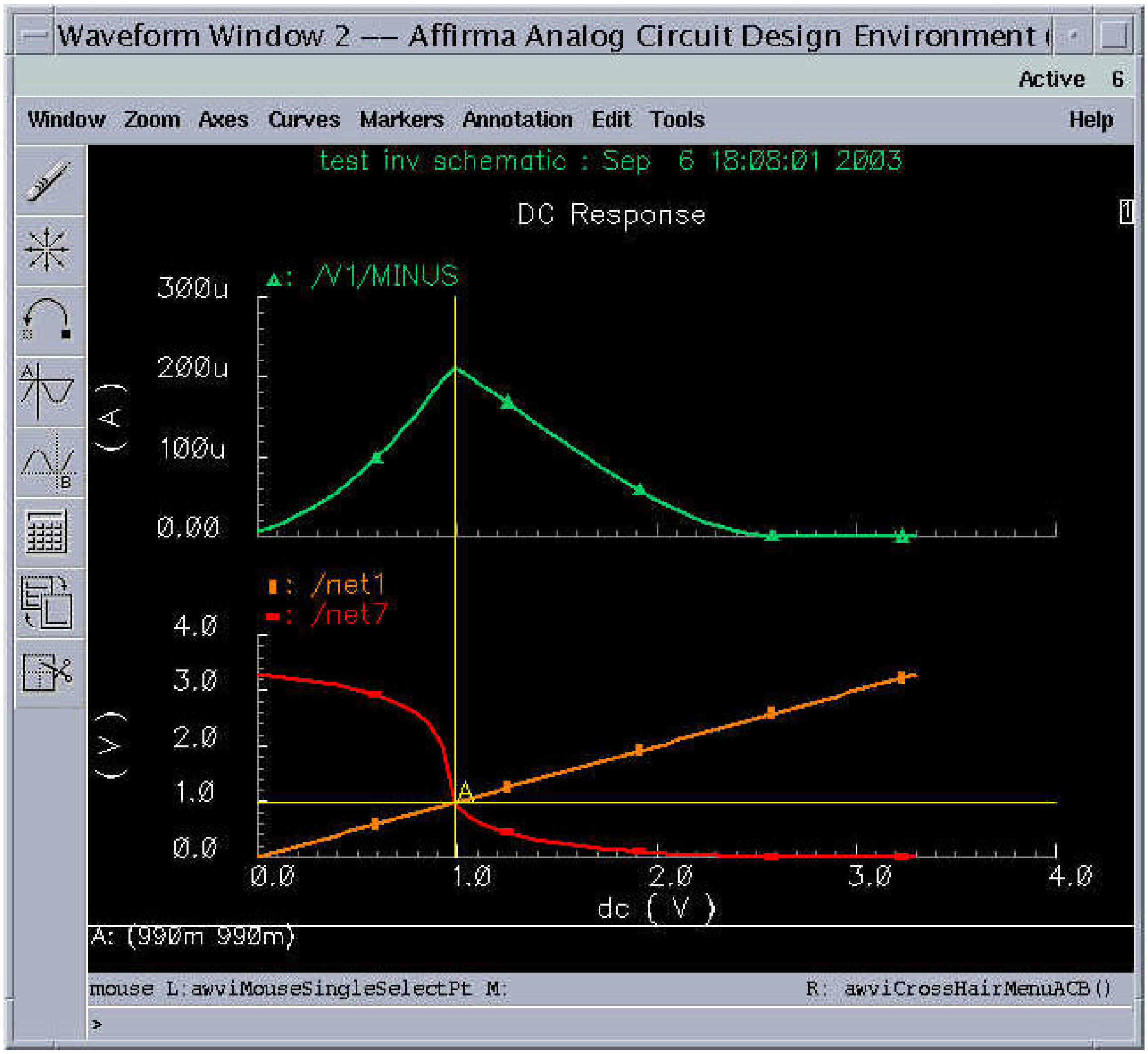
2. Once this is done, go to the ***Analog Environment window*** and follow the same procedure for plotting as you did for Transient Analysis.



3. Then go to **Simulation** in ***Analog Environment window*** and under Simulation choose ***Netlist and Run.***

4. A waveform will automatically appear showing the DC response of the circuit. We can separate all the graphs by clicking on ***Switch Axes mode*** on the Waveform window to the left. Also in order to combine two graphs just select one of them and place them over the other.

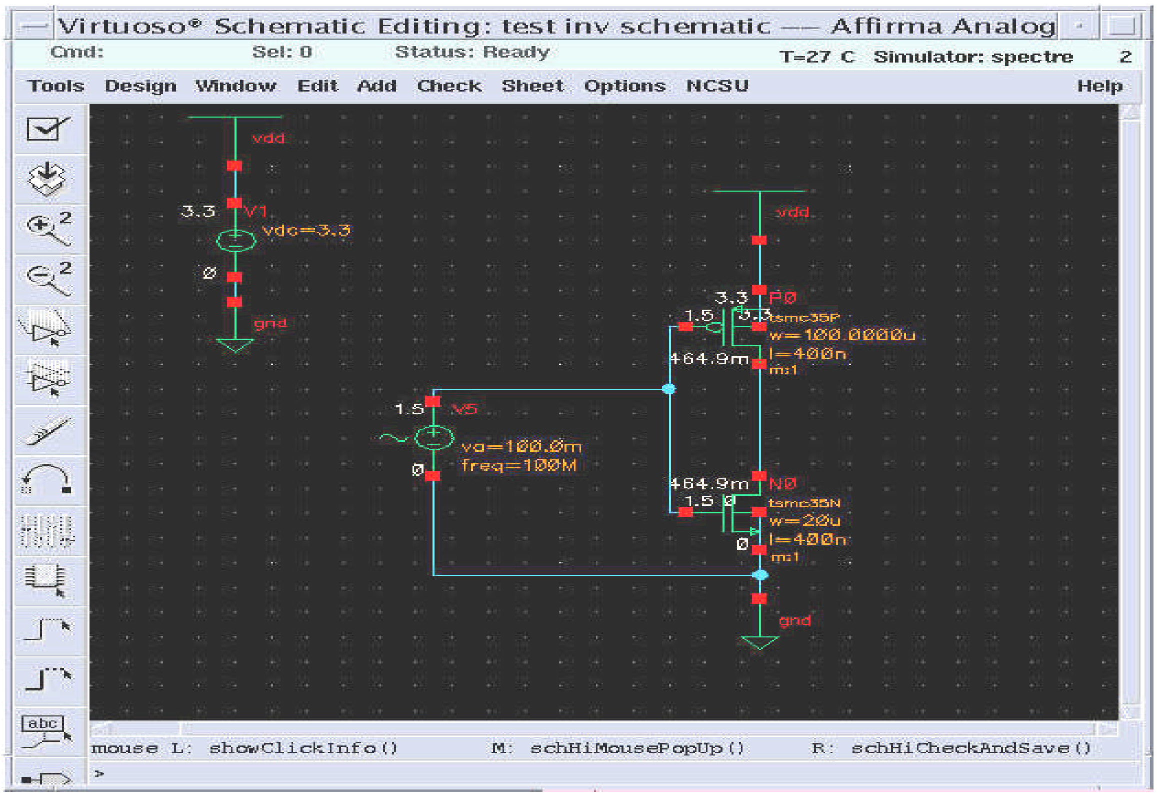
5. A plot of the DC characteristic curve of the inverter is shown and the drain current I_d is shown. On the Waveform window go to Markers and click on Marker "**A**". The marker can be useful to observe values at a particular point on the graph.


Waveform Window 2 — Affirma Analog Circuit Design Environment
Active 6
Window Zoom Axes Curves Markers Annotation Edit Tools Help
test inv schematic : Sep 6 18:08:01 2003
DC Response
/V1/MINUS
300u
200u
100u
0.00
(A)
/net1
/net7
4.0
3.0
2.0
1.0
0.0
(V)
0.0 1.0 2.0 3.0 4.0
dc (V)
A: (990m 990m)
mouse L:awviMouseSingleSelectPt M:
R: awviCrossHairMenuACB()

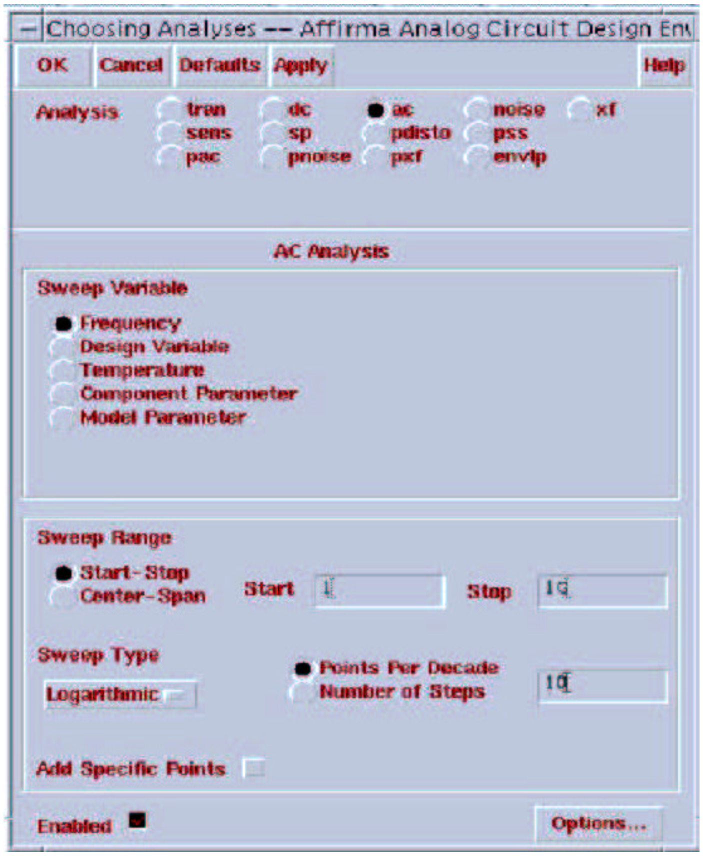
C. AC Analysis

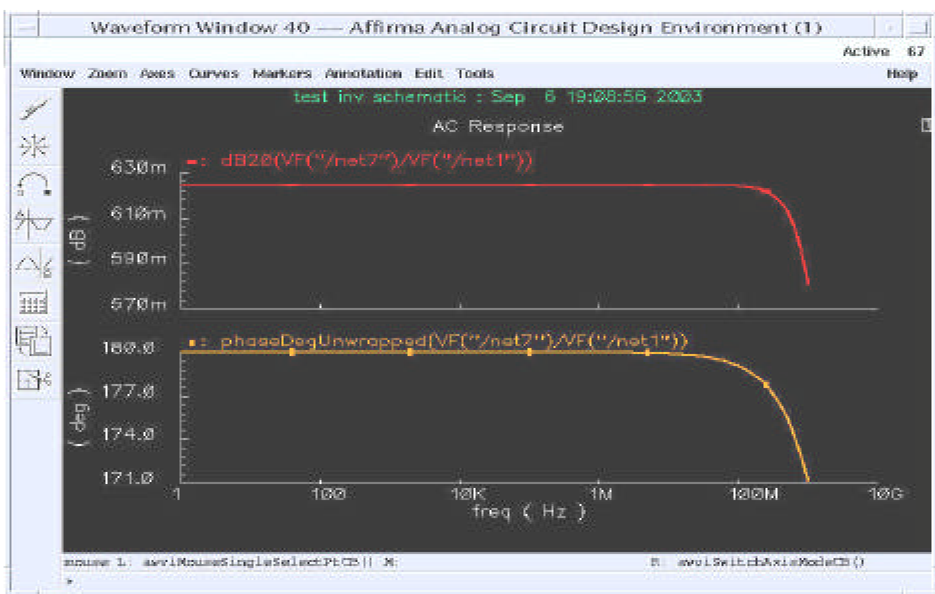
In order to conduct an AC Analyses, we will make use of an inverter circuit.

Draw it in the schematic window with the specifications given below. Some of the components we will be using will be a sinusoidal source (vsin). In the *Composer* window go to *Instance* and choose vsin.



1. Follow the same steps as above and invoke the ***Analog Environment window.***
2. Then go to ***Choose*** in the **Analog Environment window** and select **AC**.
3. In Choosing Analyses window, under **Sweep Variable** select ***Frequency.*** In the Sweep Range, enter in **Start** as ***1*** and in **Stop** as ***1G***. Then in **Sweep Type**, choose ***Logarithmic*** and select **Points per Decade** enter ***10***.
4. Click OK, choose the outputs you want to see and run the simulation.
5. The output is as shown in the waveform in the fig below.
6. Also perform a transient analysis and observe the differential inputs.





In order to learn Cadence much more, try out your own circuits and simulate them and observe their response. Also to learn about Cadence and Analog Environment, go to **Help** and view ***Openbook Main Menu***.