

Due to some issues that ADE GXL simulation environment has (probably because of inappropriate setup), we will run simulations in the ADE L design environment, which includes all the necessary tools that is needed for simulating the designs given in the class assignments.

*This brief tutorial is similar to section 2 in “Cadence Simulation Tutorial” posted in the course website, except here we will use the ADE L simulation environment.*

## 2.2 Transient Analysis

1. Start ADE L from Virtuoso Schematic Composer: **Launch → ADE L**

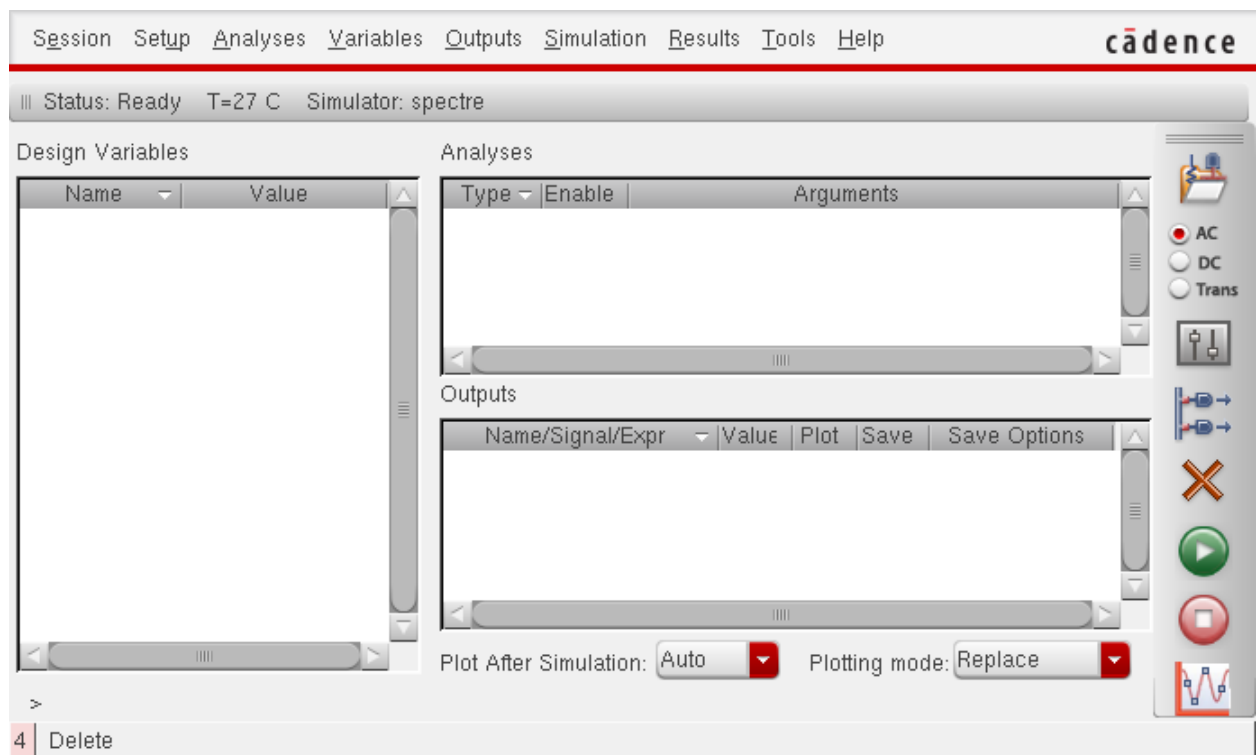


Figure 1: ADE Test Editor

2. The Analog Design Environment window should pop-up, where all parameters, models, paths, etc will be defined. The following settings should be set by default , however, you need to verify them to insure a valid simulation:

- **Setup → Simulator** verify that spectre is selected in the drop down menu.
- **Setup → Model Libraries** verify that you have

`/opt/soft/ncsu-cdk-1.6.0.beta/MSU/allModels.scs`

enabled under Global Model Files.

3. Choose a simulation type: go to **Analyses → Choose**. For Analysis select “tran” and for stop time type 20n that is twenty nanoseconds. Refer to figure 2.

4. To plot the input/output response versus time ADE allows the user to select certain wires/nodes to display. Go to **Outputs → To Be Plotted → Select on Schematic**. The schematic composer is brought to front, you can now select the desired signals. NOTE:
  - **Selecting nodes e.g. red boxes will plot currents. When selected the node will be circled.**
  - **Selecting wires e.g. blue lines will plot voltages. When selected the line will appear dashed.**

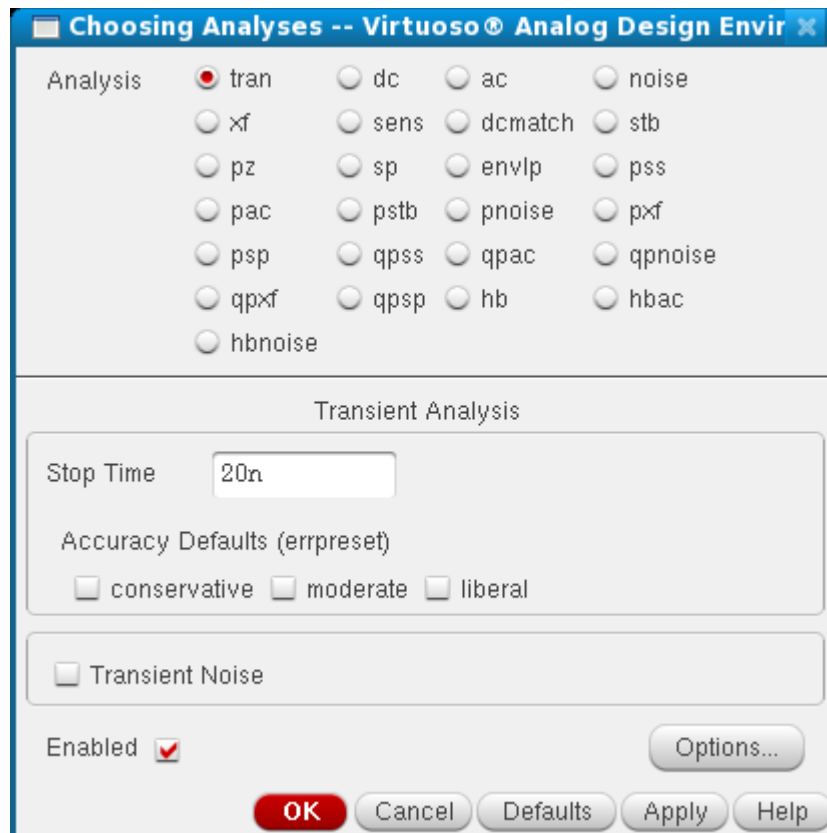


Figure 2: Transient Analysis

If you accidentally clicked on a node/wire but you don't want to plot it just click on it again to remove it. We are interested in plotting the input/output voltages versus time. Therefore, select the wire connected to the positive terminal of the pulse generator then select the wire connected to the output pin. The schematic should look like figure 3. When done selecting outputs press ESC on the keyboard to return to ADE test editor.

5. Your simulation environment is ready now and it should look like figure 4.
6. Start the simulation by clicking on the green play icon or go to **Simulation → Run**. The resulting plot should appear in front of you as shown in figure 5.

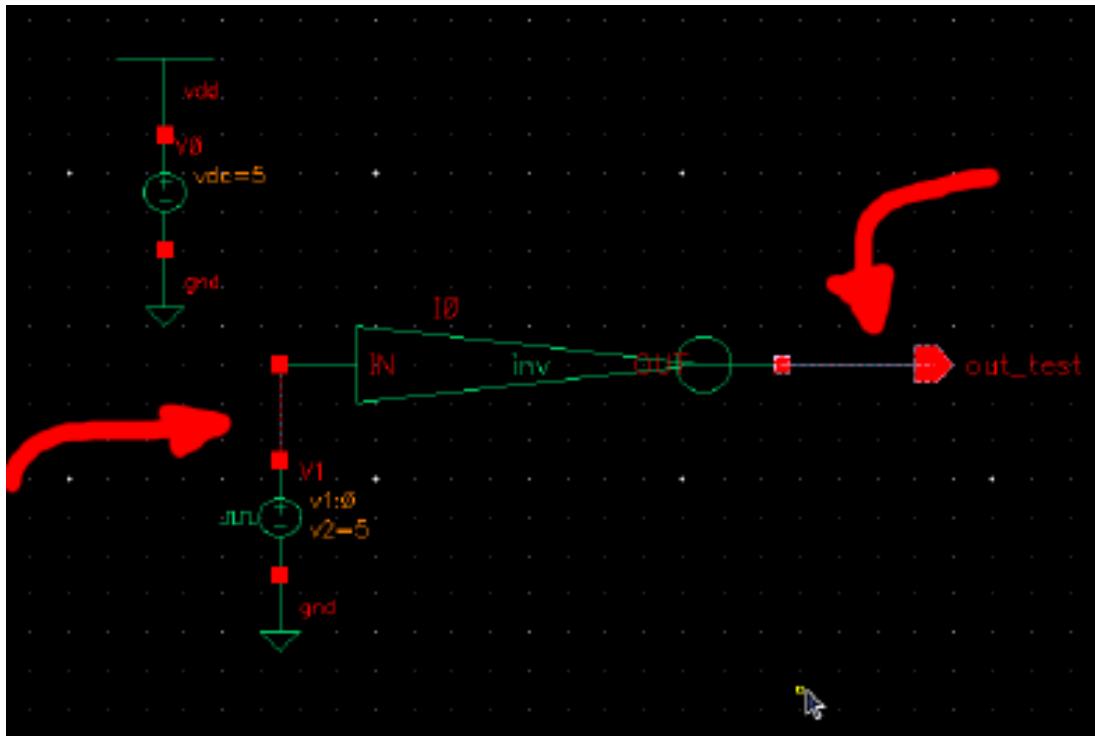


Figure 3: Select output to be plotted from schematic

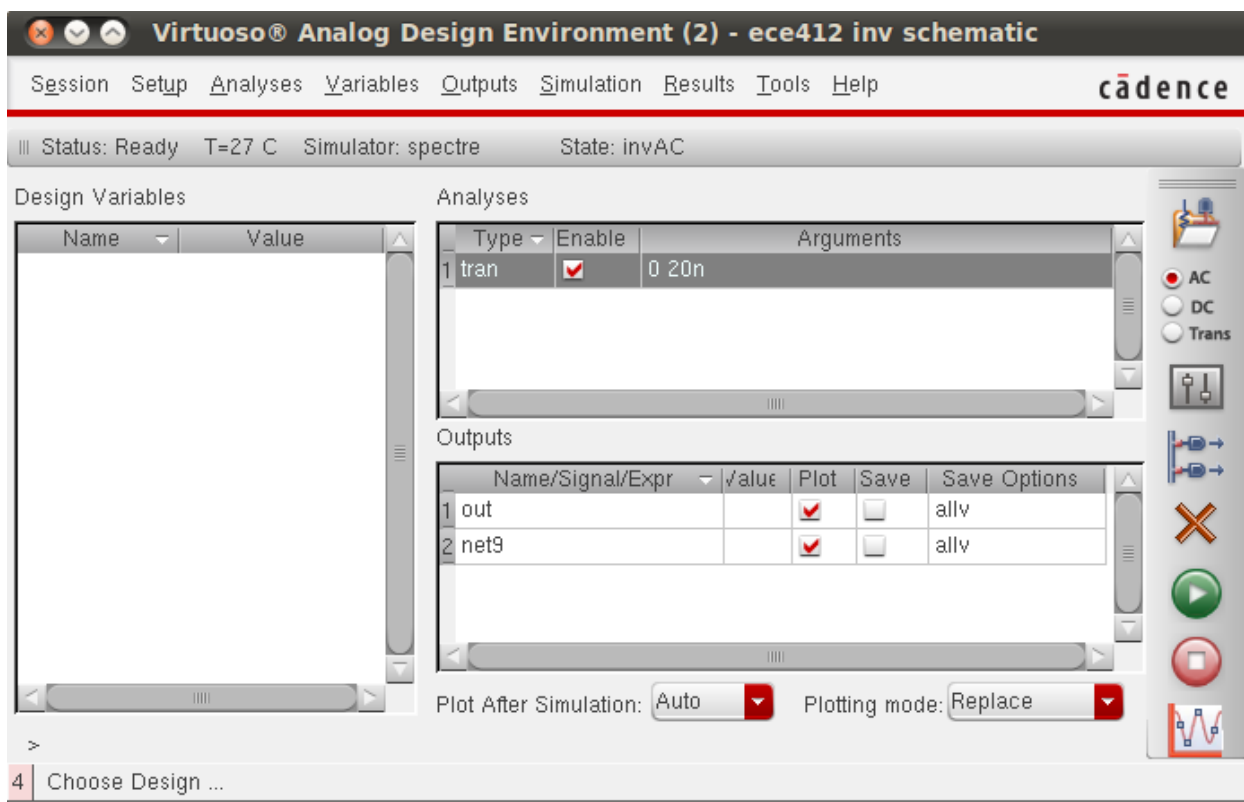


Figure 4: ADE L Test Editor with new settings

7. The ADE L (figure 4) environment state can be saved for future usage, to save go to **Session → Save State ..** and enter an appropriate file name in the **Save As** box.
8. To load a saved environment state, hit **Session → Load State ...** , then select the State Name needed to be loaded.

## 2.3 DC Analysis

1. To simulate the schematic design in DC mode, we will follow the same first two steps in section 2.2, and then :
2. Choose the analysis type by going to **Analysis → Choose** , from the window that will appear select “dc” and check “Save DC Operating Point”. Under Sweep Variable check “Component Parameter” then click on “Select Component”. The schematic composer should show up, click on the voltage pulse generator. Finally, a small pop-up window titled “Select Component Parameter” appears: Click on DC VDC “DC Voltage”. Both windows are shown in figure 6 below.
3. For Sweep range start at 0 volts and end at 5 volts. Make sure that the analysis is enabled then click on OK.

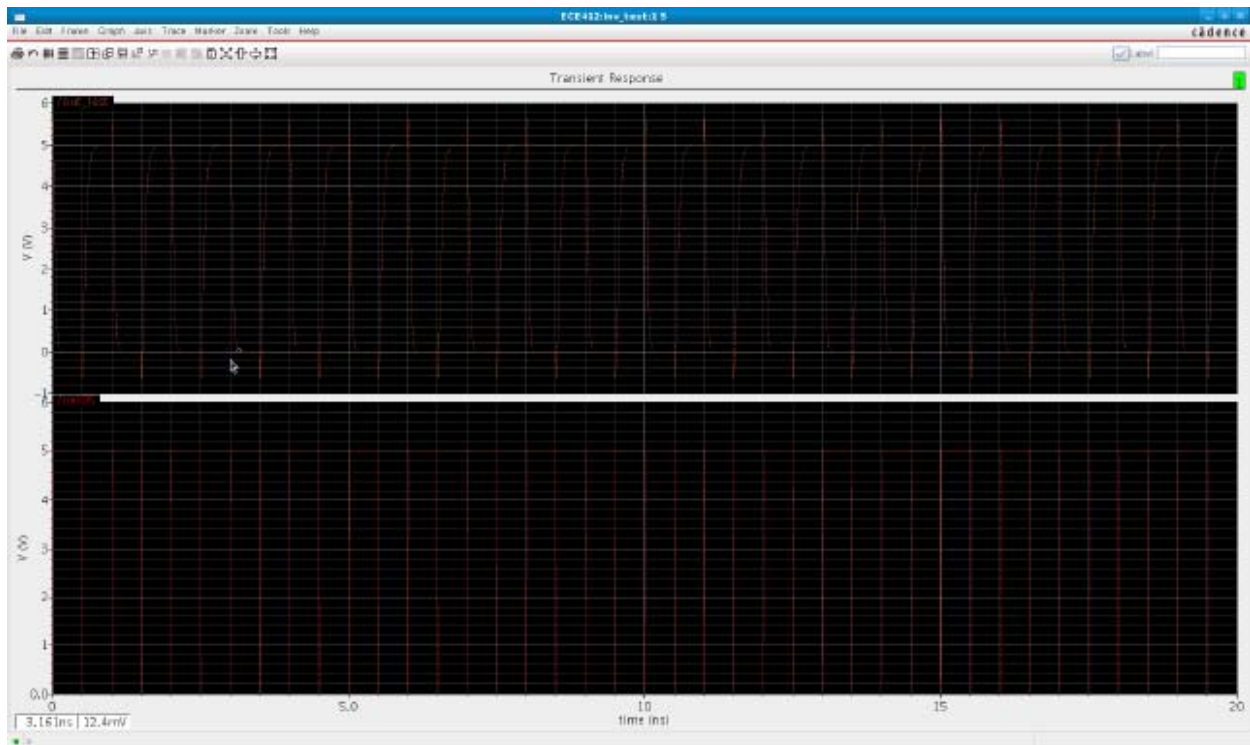


Figure 5: Transient response with two different axis's

4. Start the simulation by clicking on the green play button. The output should look like figure 7. We need to calculate the crossing point between the input and the output. This point will be used later in AC analysis.

5. The crossing point can be either estimated using the mouse by hovering over the crossing point or calculated using Virtuoso Calculator. In the graph window go to **Tools** → **Calculator**.
6. After launching the calculator look at the bottom half of the window near Special functions and look for "cross". Now click on "cross" you should see an integrated input box where the cursor is blinking in "Signal" text box.
7. We need to type a name of a signal or select a waveform from the graph window. To select a waveform from the graph window click on "Wave" in the upper half of the calculator (near Family) then go to the graph window and click on "test out" (the waveform itself or the label both would work). The "Signal" textbox should have "wave xx()" or "v(/out test ?result dc-dc)". Leave the threshold and edge number and type the same then hit OK.
8. The calculator's buffer text area should have the cross() function with the proper parameters. Click on evaluate buffer as depicted in figure 8 and write down the result, we will use it later in AC analysis. You can now close the calculator and the graph window.

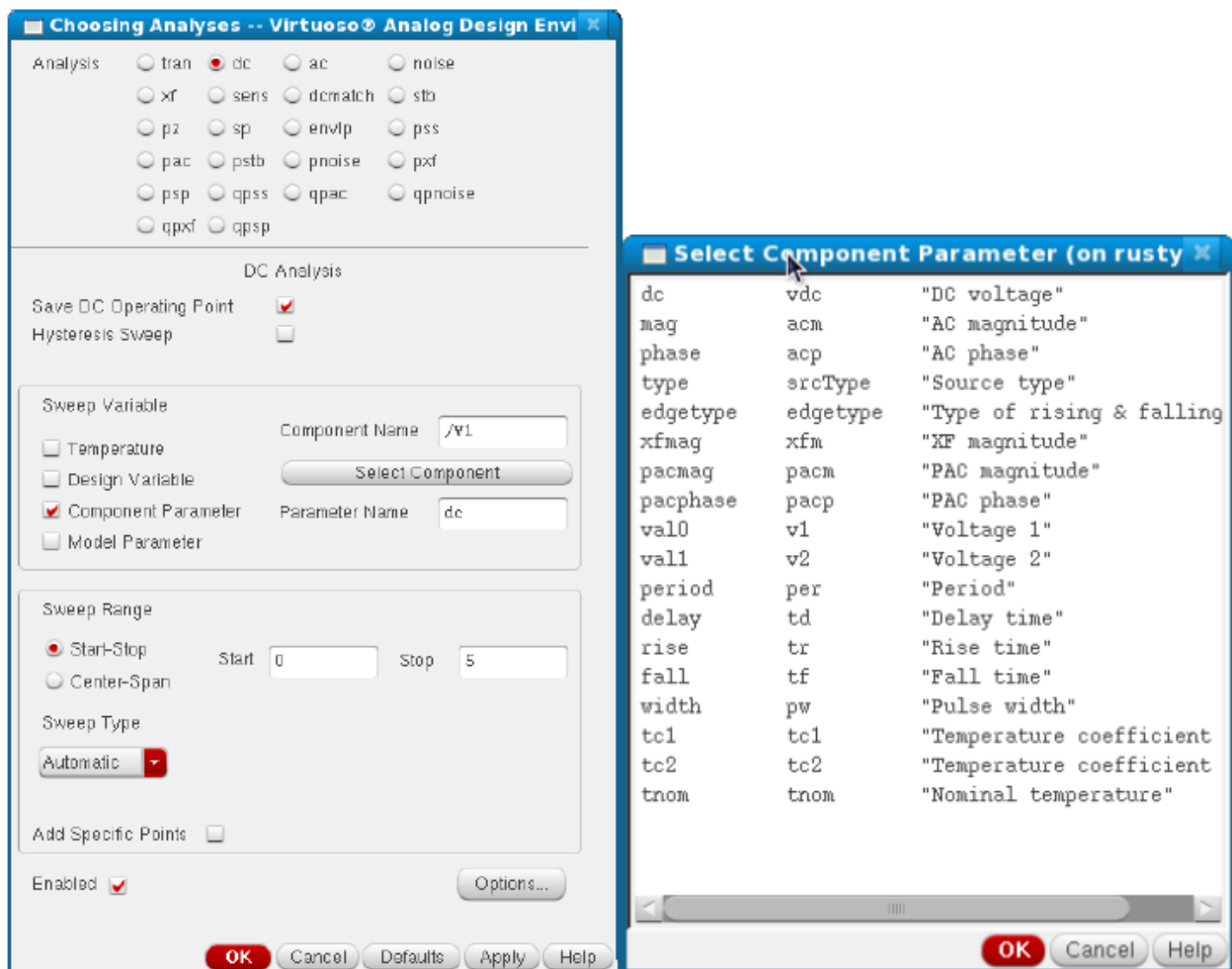


Figure 6: (a) Choosing Analysis window to select DC and Component (b) Selection of component parameters

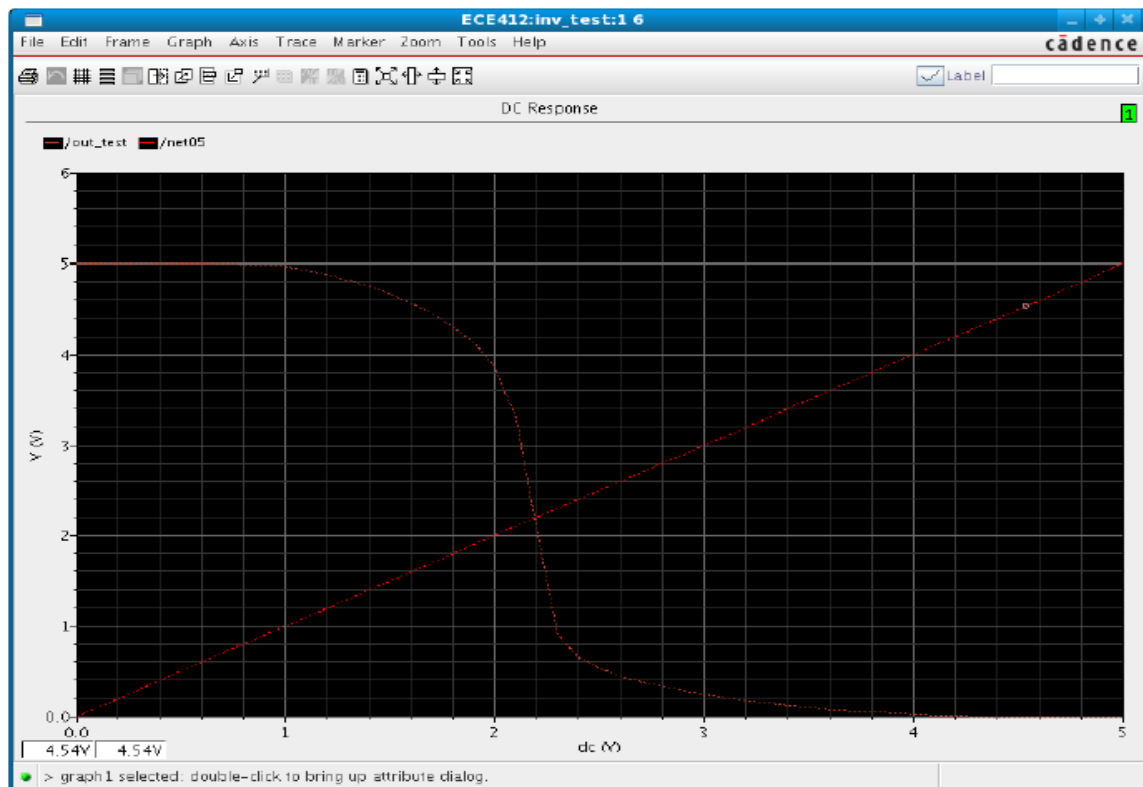


Figure 7: Inverter DC analysis

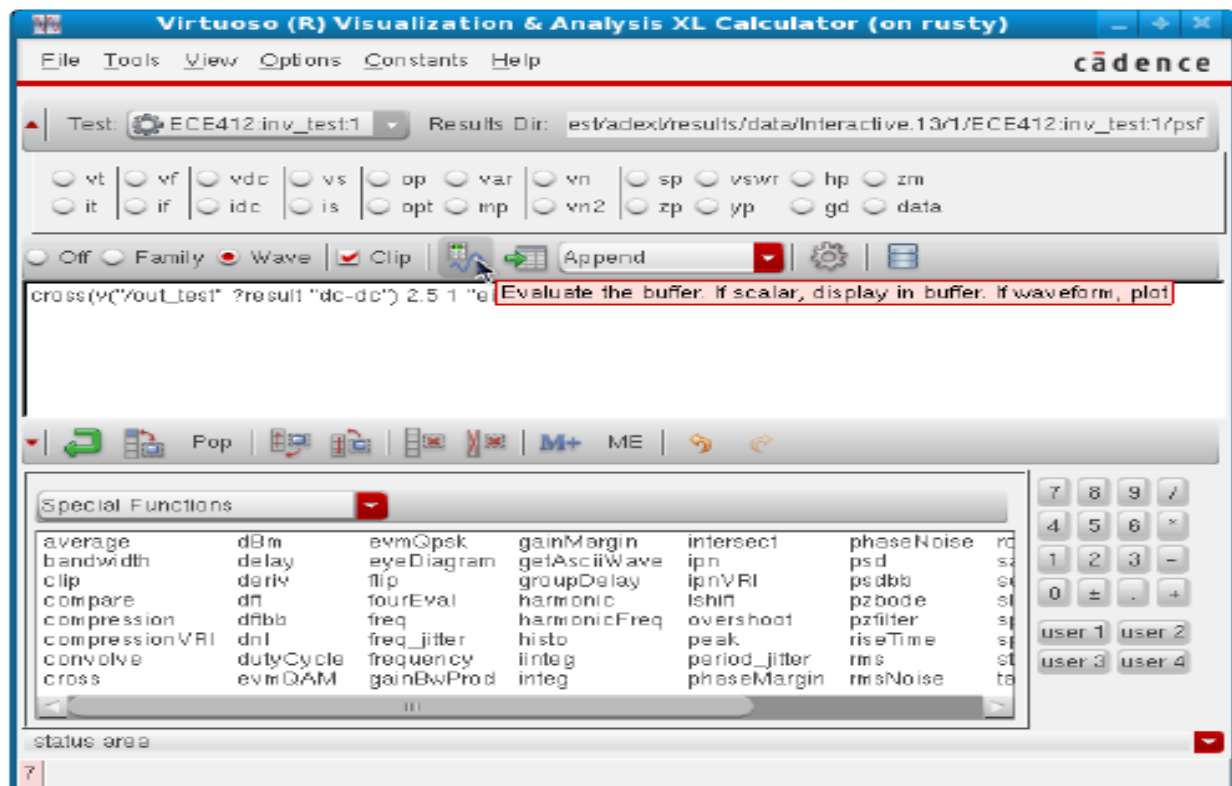


Figure 8: Virtuoso Calculator

## 2.4 AC Analysis

We will follow the same procedure executed in the previous two sections. The only difference here is that the test input has to be changed to an AC source.

1. Go to the schematic composer and click on the voltage pulse generator then hit “q” on the keyboard. The proper window should pop-up.
2. Change the Cell name from “vpulse” to “vsin” then click on apply. Now modify AC magnitude to 1, AC phase to 0, and DC offset to the crossing value we computed using the calculator in section 2.3. The proper window will look like figure 9.
3. Save the schematic, by clicking on the **Check and Save** icon.
4. On the left side menu add a new test. For Analysis select “ac”, Sweep Variable select “Frequency”, and Sweep range: start at 10 and stop at 1G with a sweeping type of Logarithmic and 10 points per decade. Enable it and hit OK. This is shown in figure 10.
5. Start the simulation by clicking on the play button. The simulation result should look like figure 11.

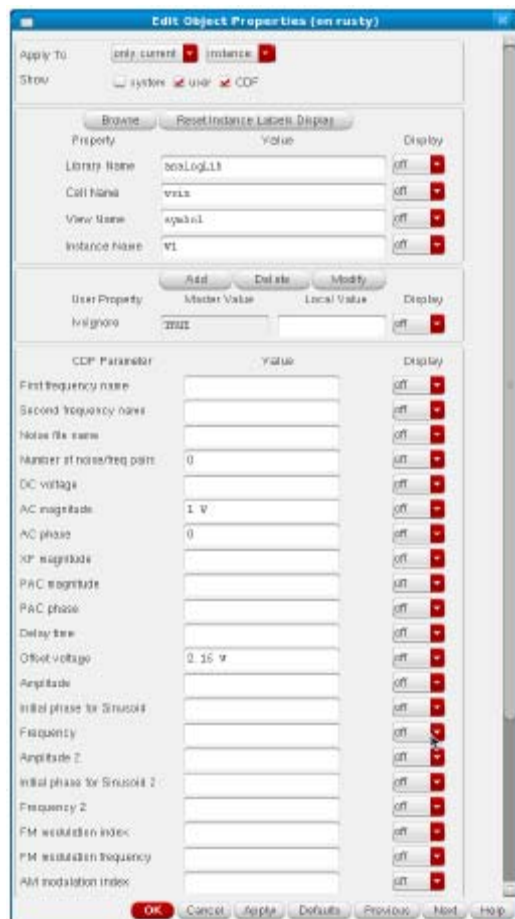


Figure 9 AC input source



Figure 10 AC analysis properties

6. To plot the Magnitude version of the curve in 20db or 10 db, right click on it and then select the desired plot from the upper-right drop-down menu in Virtuoso Visualization & Analysis Tool (figure 11 – the box next to “Label”). Similarly you can draw the phase by selecting “phase” from the same menu.

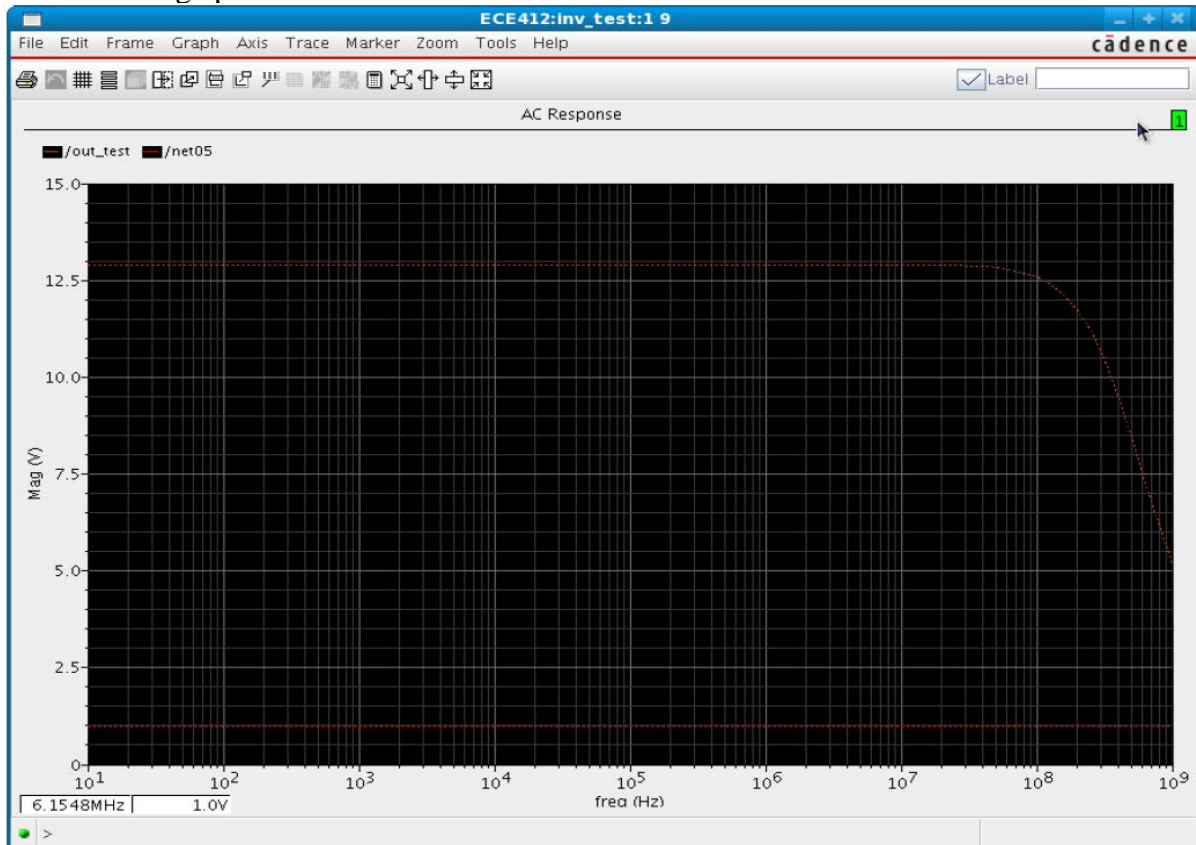


Figure 11 AC Analysis for an inverter