The Inverter

Author: Faisal T. Abu-Nimeh
Last update: September 14, 2009

Abstract

In this tutorial we will build and simulate a CMOS inverter using Virtuoso Schematic Composer and Spectre. The inverter will be characterized using Transient, AC, and DC Analysis.

Contents

1 Library and Cell Creation

1.1 Creating Libraries

Launch *Cadence Virtuoso* (refer to the previous tutorial if needed). You should see two windows: The CIW with a title of "*Virtuoso®6.1.x*" and the library manager with a title of "*Library Manager*". Each time you want to start a **new project (not a cell)** you should create a new library **and** attach the created library to certain technology node. The technology we will be using is On Semi C5 (0.50 micron) previously known as AMI. Therefore, to create a new library:

- Go to the *Library Manager*'s menu **File⇒ New⇒ Library**
- In "Name" textbox type ECE412 then hit OK
- You will be asked to choose a Technology file for the new library. Select "Attach to an existing technology library" as seen in figure 1 then hit OK
- Select "NCSU_TechLib_ami06" as shown in figure 2 then OK.

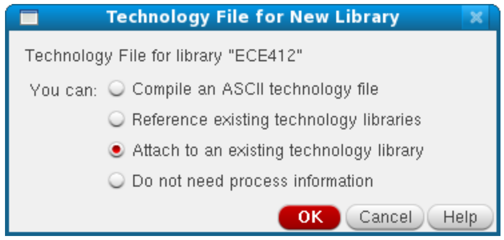


Figure 1: Technology File for New Library

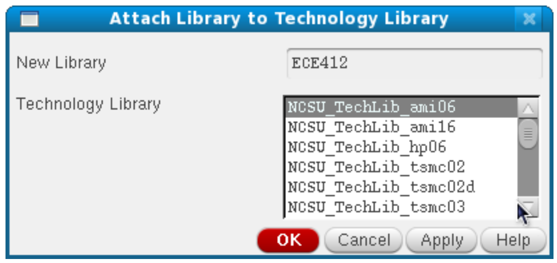


Figure 2: Attaching Technology Library

Now you should see the library you created in the list, any cell created inside this library will use the same technology node, therefore, If you fail to attach the correct technology to your library none of the simulations we will perform next will work. To know what kind of technology is attached to certain library just right click the library name (e.g. ECE412) then **Properties** the "Library Property Editor" should contain **techLibName = "NCSU_TechLib_ami06"** as shown in figure 3.

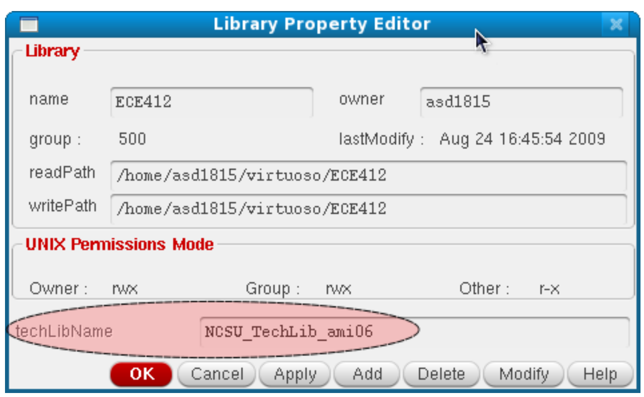


Figure 3: Library Property Editor

1.2 Creating Cells

We will use the same library we created in section 1.1 for **all** cells (e.g. Inverters, Op Amps, NAND gates, etc) we're going to simulate using On Semi 0.50 micron technology. For simplicity the CMOS inverter will be demonstrated in this tutorial. To create an inverter:

- Select and Highlight the library you created in section 1.1 e.g. (ECE412)
- Go to the *Library Manager*'s menu **File⇒ New⇒ Cell View**
- A Pop-up window appears titled "New File" as shown in figure 4. In the "Cell" textbox type inv .
- Make sure that the "View" and "Type" are set to **schematic**. Hit OK.

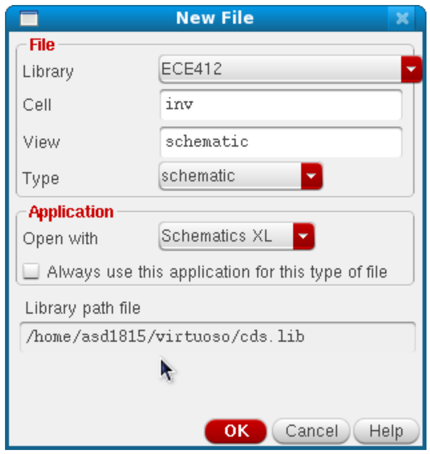


Figure 4: Creating a new Cell View

A new blank editor should appear in front of you. You are encouraged to familiarize your self with the interface, just move your mouse cursor over the menu and check out what they do. Also, it is a good habit to remember the keyboard shortcuts associated with this editor, just go to the menu on top and look for any letter beside it e.g. if you go to **Create⇒ Instant⇒ Cell View** You will notice the letter "I" beside it, hence, if you just hit "i" on the keyboard you can create an instance quickly. The more you use these shortcuts the faster your design process will be.

1.2.1 Adding Components

To create an inverter we need to instantiate four components: VDD, GND, NMOS Transistor, and PMOS Transistor. Once you create an instance two pop-up windows will appear "Component Browser" shown in figure 5 and "Add Instance". Using the Component Browser window select "NCSU_Analog_Parts" for "Library" then click on "N_Transistors" in the component list and click on "nmos4". Now move the mouse cursor over the schematic editor and place the transistor anywhere you like. Repeat the same process for PMOS Transistor and finally for ground and power click on "Supply_Nets" and choose "vdd" and "ground".

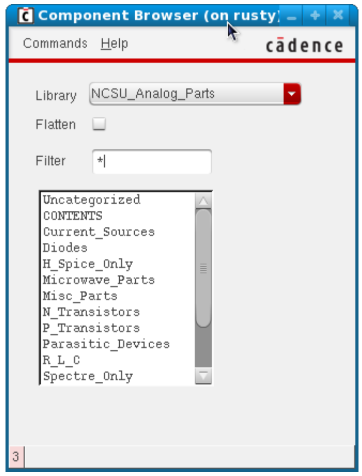


Figure 5: Component Browser

1.2.2 Wiring it up

Once we have all components on the schematic editor we have to connect them to form a valid inverter, so, go to **Create**⇒ **Wire**⇒ or simply hit "w" on the keyboard and click once on the Drain of PMOS then click again on the Drain of NMOS. You should see a blue wire connecting both ends. Repeat the same process for all ends PMOS Source and **Bulk** to VDD and NMOS Source and **Bulk** to GND. Finally we need an input pin and an output pin.

1.2.3 I/Os

For this simple design we need two I/Os an input and an output: go to **Create**⇒ **Pin**⇒ or just "p" type the pin name as **IN** and select the Direction to be "input" then click on the schematic editor to place the pin icon anywhere you like. Now wire it up to the gates of the NMOS and PMOS transistors. Repeat for the output pin; the name is **OUT** and the Direction is "output" then wire it up to the Drains of the NMOS and PMOS transistors.

1.2.4 Inverter Schematic

After connecting all wires to the corresponding nodes your inverter should look like figure 6. To make your schematic fill the editor window hit "f" on the keyboard.

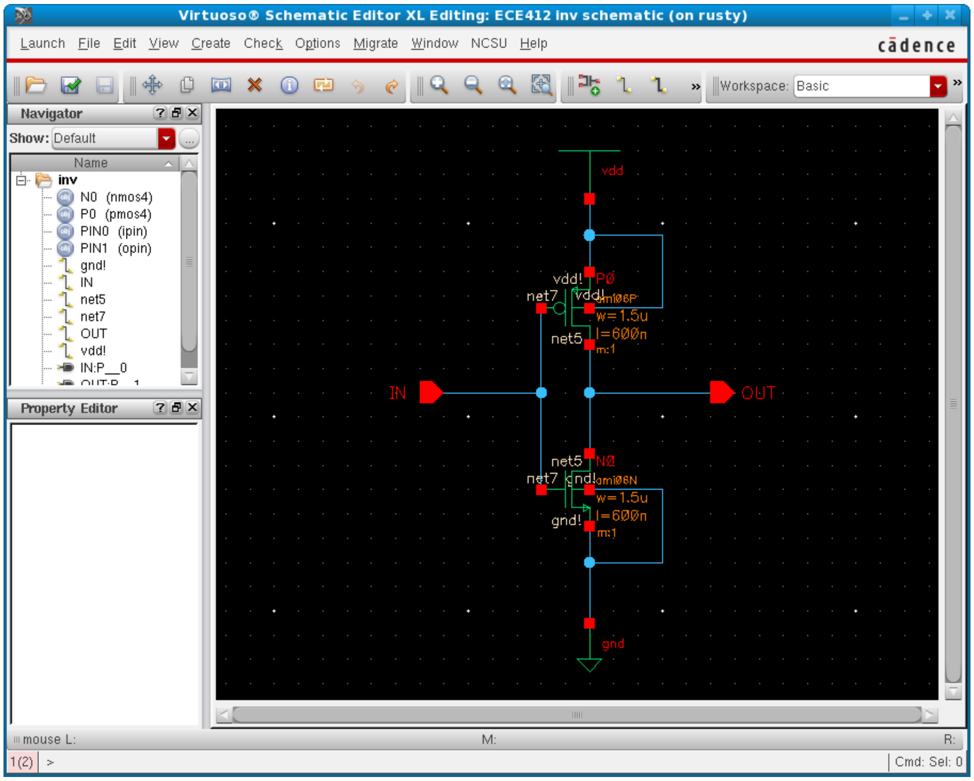
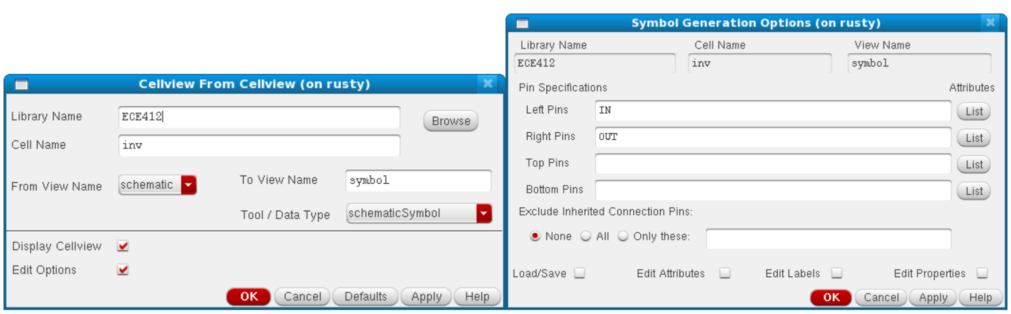


Figure 6: Inverter Transistor Level Schematic

1.3 Creating Symbols

It is sometimes desired to work with device-level schematic rather than transistor level, therefore, to create a symbol for the device (Inverter) created in the previous section: go to **Create**⇒ **Cellview**⇒ **From Cellview** a pop-up menu should appear as shown in figure 7a. Everything should be already filled out, just confirm that "Tool/Data Type" is **schematicSymbol** then click OK. Next you you can set the position of each pin as shown in figure 7b this becomes useful for multiple IOs.



(a) Creating a Symbol from Transistor level Schematic

(b) Generating Symbol IOs and Their Orientation

Figure 7: Creating An Inverter Symbol

A generic symbol should appear after acknowledging "Symbol Generation Options". Now you can edit the shape to look like an inverter by drawing a triangle and a circle at its tip. To draw a triangle: go to **Create**⇒ **Shape**⇒ **Line**, when drawing a triangle you have to close the shape in order to finish it. Drawing a circle is very similar however, you have to go to **Create**⇒ **Shape**⇒ **Circle** then click and release to form the center of the circle and move the mouse pointer outwards to increase the diameter. An example is shown in figure 8. Finally to save your symbol click on "Check and Save" of simply "F9" on your keyboard.

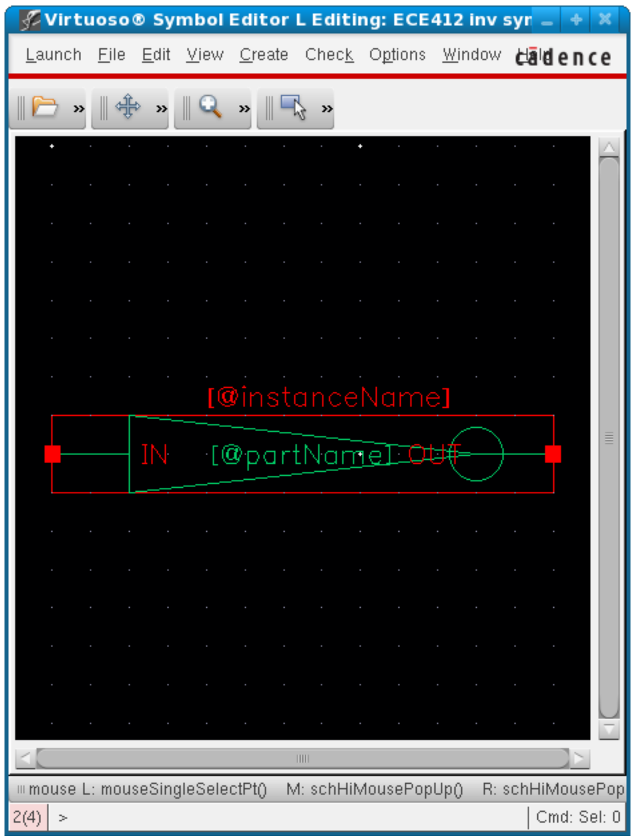


Figure 8: Inverter Symbol

2 Simulation

2.1 Test Circuit

There are several methods to test the performance and functionality of a given component. In this tutorial we won't use stimulus files, however, we will focus on using Virtuoso Schematic Editor and Analog Design Environment (ADE) to achieve this task. To build an inverter test circuit open the "Library Manager", make sure that the library e.g. ECE412 is highlighted then **File**⇒ **New**⇒ **Cell view**, create a schematic and call it *inv_test*. Now we will use the symbol we created in the previous section 1.3 as a DUT (Device Under Test):

1. To add the DUT: Create an instance by hitting "i" on the keyboard: For Library choose e.g.

ECE412 and then select inv then move the mouse cursor over the schematic editor and place the symbol anywhere you desire.

2. We need a power supply of 5Volts. Therefore, create an instance for **analogLib⇒ Sources⇒ Independent⇒ vdc** before placing **vdc** on the schematic editor look for a pop-up window titled "Add Instance", this should have appeared when you clicked on **vdc**, change the "DC voltage" field to 5 and hit OK. If you cannot find this pop-up window just place **vdc** anywhere on the schematic editor, select it, then **right click⇒ Properties** then change the DC voltage to 5 and hit OK.

3. To connect the DC voltage source to the inverter's global supply nets (vdd and gnd) simply create an instance for vdd and connect it to the positive pin of vdc and gnd to the negative end

4. The inverter's input and output pins should be connected depending on what you intend to characterize. For example, to test the functionality of the inverter we will connect a pulse generator as an input then plot/display the output of the inverter to verify it manually.

    - Add a new instance for **analogLib⇒ Sources⇒ Independent⇒ vpulse** under "Add Instance" pop-up window change Voltage1 to **0**, Voltage2 to **5**, and Period to **1n**. Leave the rest blank and hit OK.
    - Place the vpluse on the east of the inverter's input and connect the positive pin of vpluse with the input of the inverter.
    - The negative pin of vpulse should be connected to ground: We can either wire it to ground, create an instance for gnd, or copy gnd by hitting "c" on the keyboard then selecting the gnd component and placing the new copy on the negative pin of vpluse.
    - The last remaining pin is the inverter's output pin we will simply create a pin by hitting "p" and name it "out_test" then wire it to the inverter.

5. Verify that you have a power supply with a valid DC voltage (vdc), a valid input signal (vpulse), a valid output monitor (pin) e.g. shown in figure 9

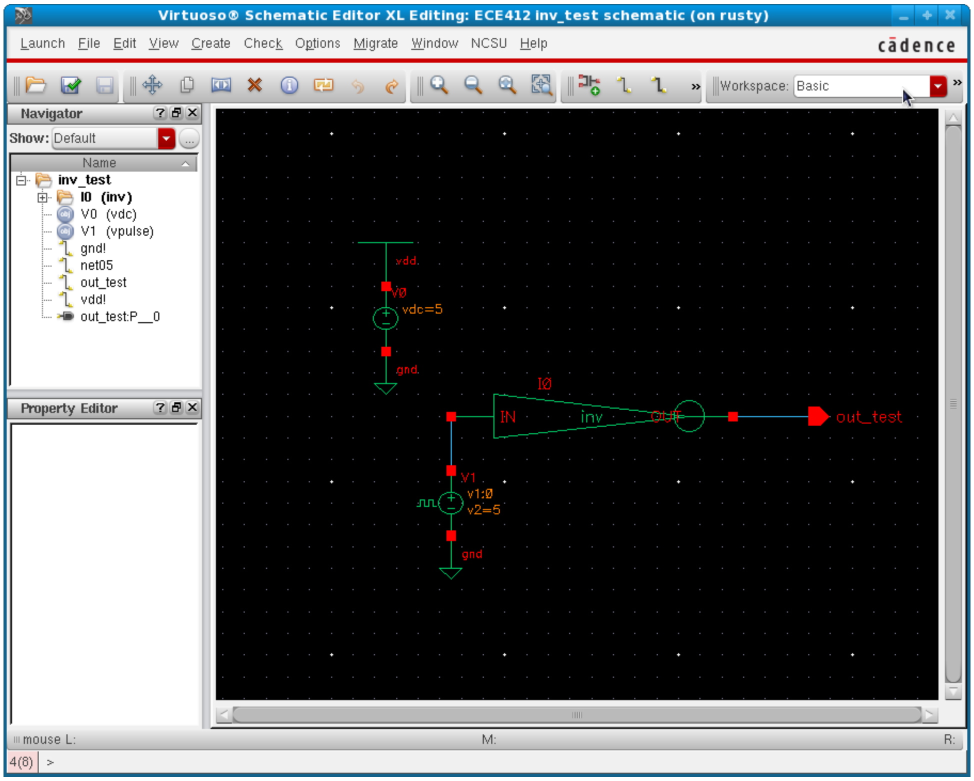


Figure 9: Inverter Test Circuit

You can create as many test circuits as needed. Usually, one would create a test circuit for different input/output schemes. Most of the time you can use the same circuit to do DC, AC, and transient response analysis.

2.2 Transient Analysis

One should use transient analysis to find any component/device's response versus time. First, create a test circuit as described in section 2.1. If you have already done so, go ahead and open it using Virtuoso Schematic Editor by double click on *inv_test* in the Library Manager. There are several ways to do transient analysis, however, we will use ADE GXL to store all simulation data in a unified location which can be used later with ease:

1. Start ADE GXL from Virtuoso Schematic Composer: **Launch**⇒ **ADE GXL**⇒ **Create New View**
2. Verify the name of the cell and choose open in new tab then hit OK.
3. A new tab titled Welcome to ADE GXL will appear, by default the left menu should contain "Data View" as depicted in figure 10

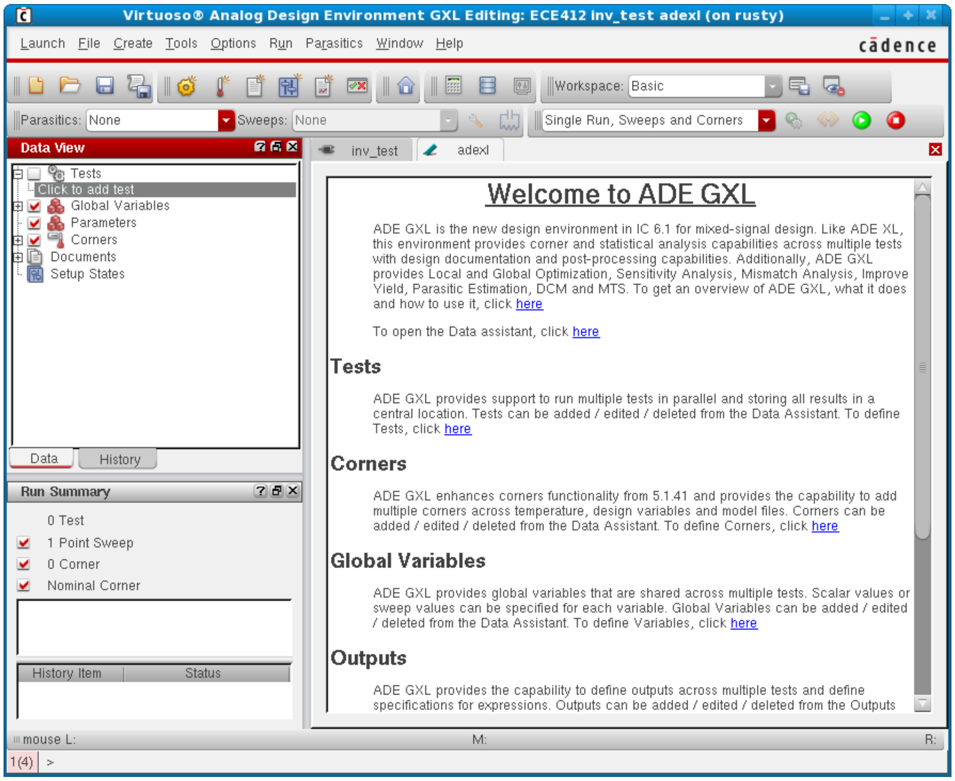


Figure 10: ADE Default Window

4. Under the "Data View" menu; expand **Tests** by pressing on the plus sign beside it then click on "click to add test" a pop-up window will ask select a design. Choose *inv_test* and hit OK.

5. Shown in figure 11 is the ADE test editor 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:

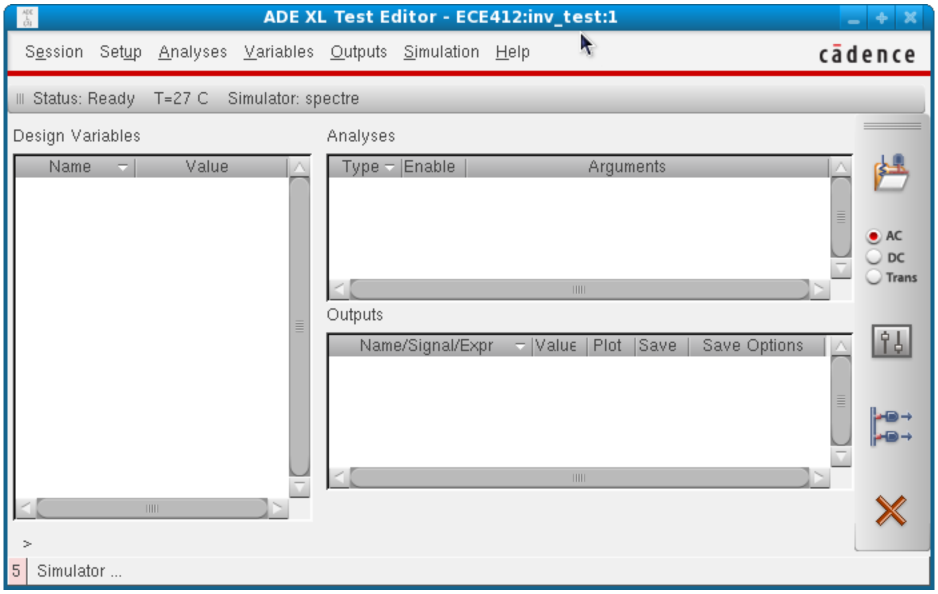


Figure 11: ADE Test Editor

- **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.

6. Choose a simulation type: go to **Analyses⇒ Choose**. For Analysis select "tran" and for stop time type 20n that is twenty nano-seconds. Refer to figure 12.

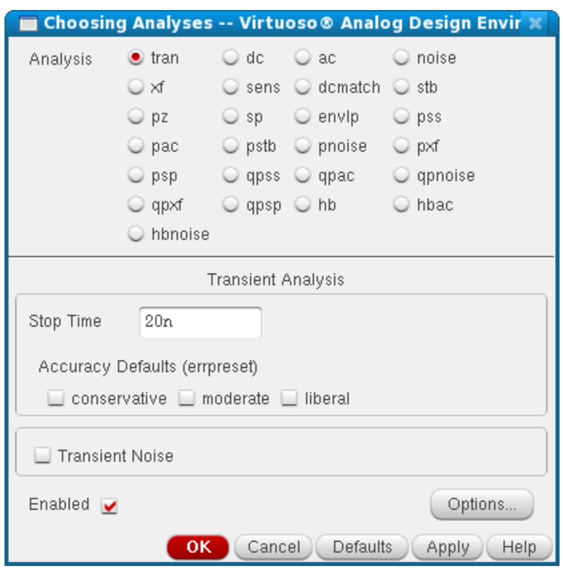


Figure 12: Transient Analysis

7. 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.**

   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 13. When done selecting outputs press ESC on the keyboard to return to ADE test editor.

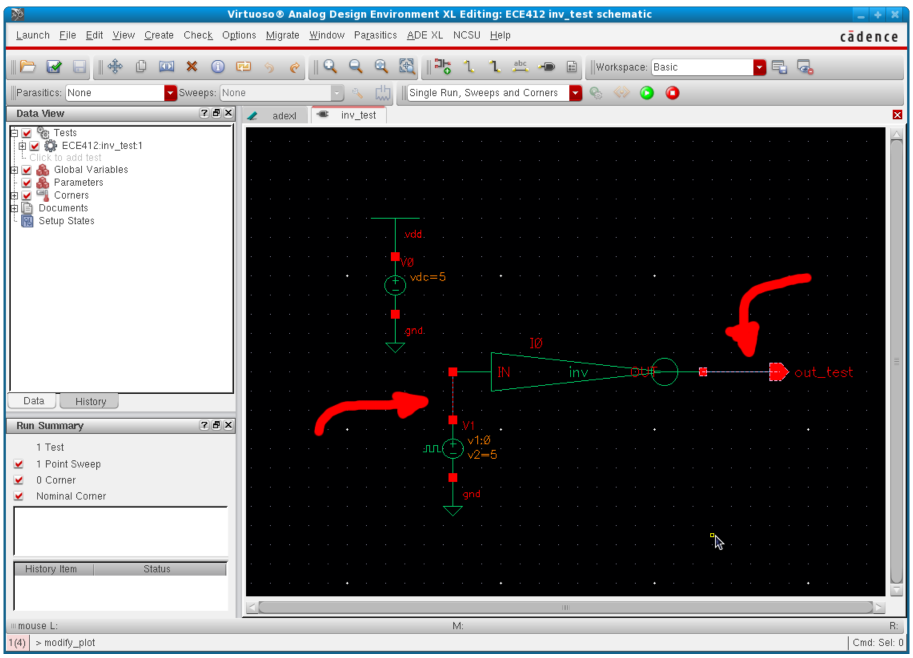


Figure 13: Select output to be plotted from schematic

8. Your simulation environment is ready now and it should look like figure 14.

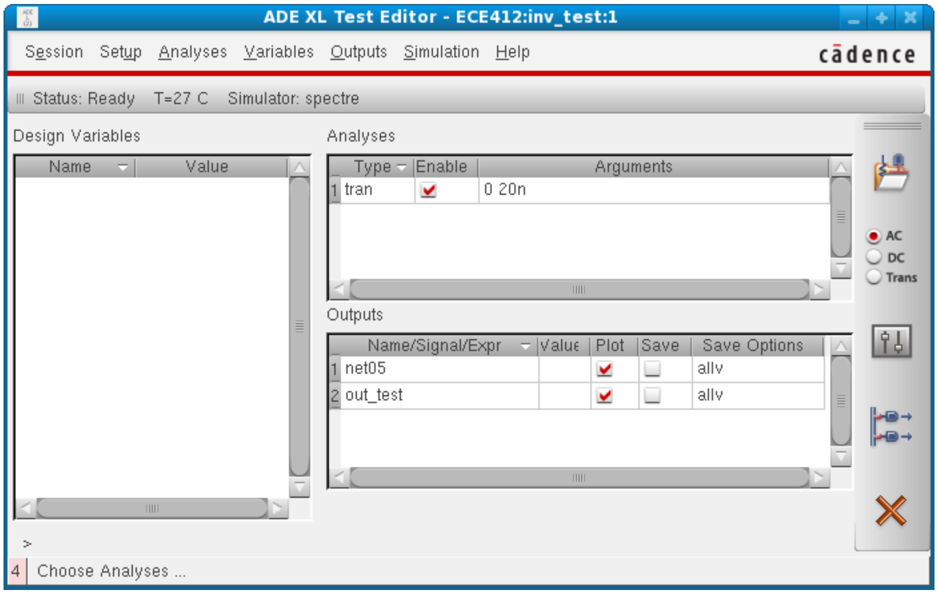


Figure 14: ADE XL Test Editor with new settings

When done you can close the ADE Test editor and return to AGXL which looks like figure 15. Note that all the options we set in the ADE test editor now appear on the left hand menu.

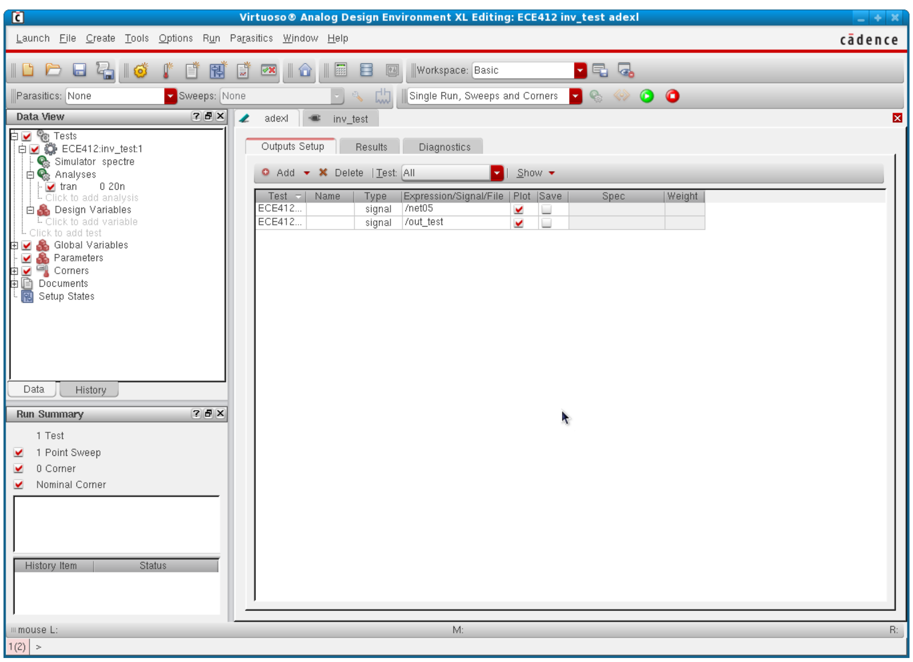


Figure 15: ADE GXL with new settings

9. Start the simulation by clicking on the green play icon or go to **Run⇒ Single Run, Sweeps, and Corners**. The result plots should appear in front of you as shown in figure 16. In case

you do not see any results, go to the results tab and click on "Plot All". To plot the two graphs on two individual axises; go to Axis and check Strips.

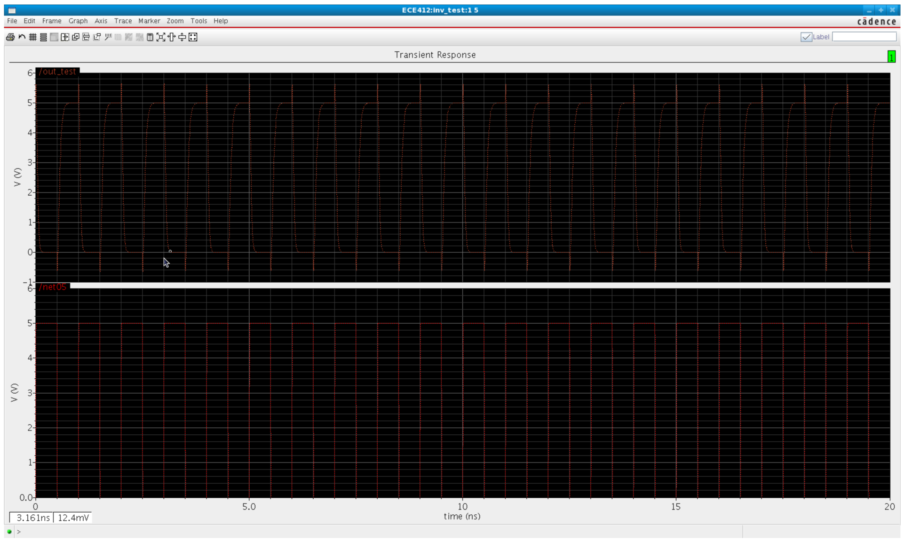


Figure 16: Transient response with two different axises

2.3 DC Analysis

We will follow the same procedure explained in section 2.2. We can utilize the same simulation environment created in the previous section, however, we need to add a new type of analysis:

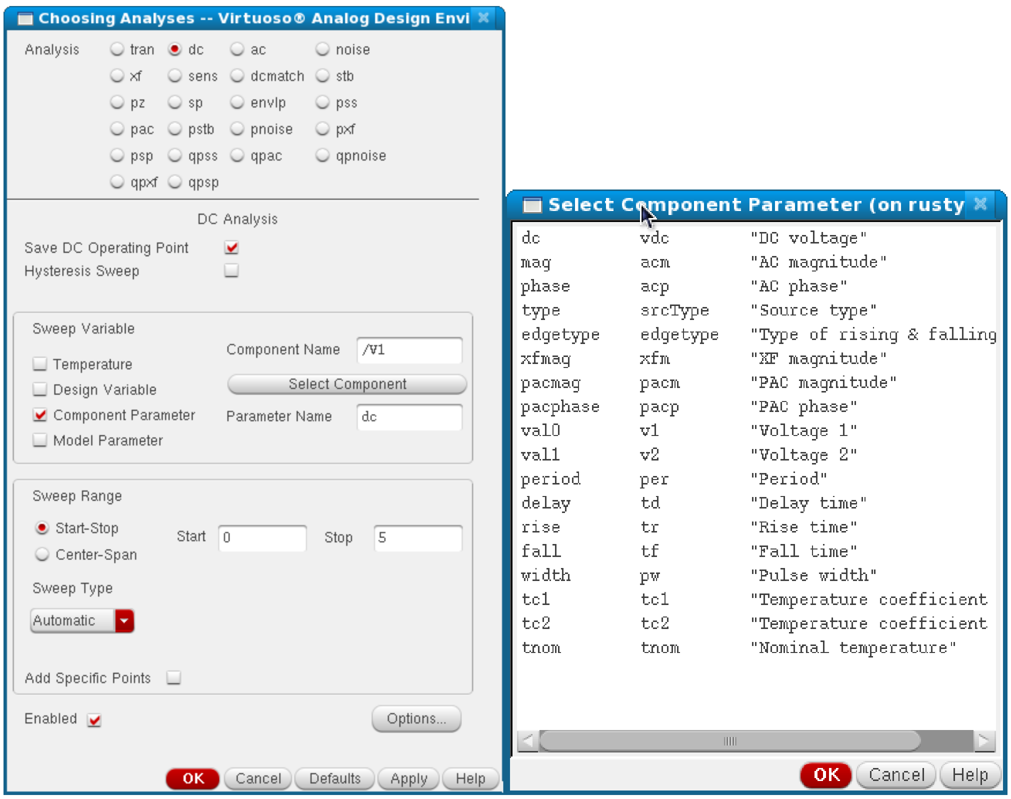
1. Disable transient analysis under the ADE GXL window, Data View, ECE412:inv_test:1 and remove the check box beside "tran" to disable the previous simulation.

2. Create a new analysis by clicking on "Click to add analysis" in gray. If you do not see it make sure you explain the test by clicking on the "+" sign.

3. Choosing Analysis window pops-up. Under analysis select "dc" and check "Save DC Operating Point". Under Sweep Variable check "Component Parameter" then click on "Select Component". The schematic composer should appear, 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 the figure 17 below.

4. For Sweep range start at 0 volts and end at 5 volts. Make sure that the analysis is enabled then click on OK.

5. Start the simulation by clicking on the green play button. The output should look like figure 18. We need to calculate the crossing point between the input and the output. This point will be used later in AC analysis.



(a) Choosing Analysis window to select DC and component

(b) Selection of component parameters

Figure 17: Pictures of animals

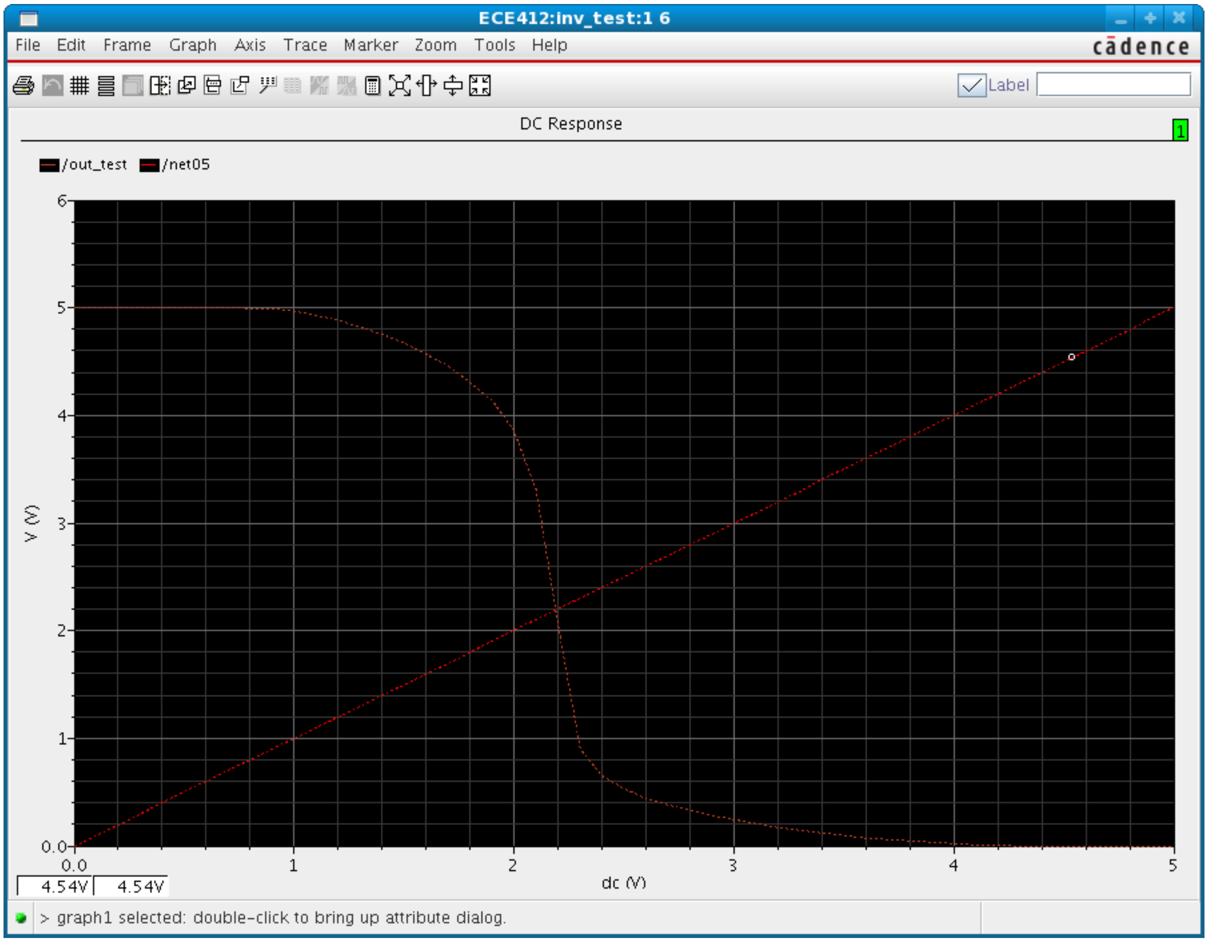


Figure 18: Inveter DC analysis

6. 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**.

7. 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.

8. 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.

9. The calculator's buffer text area should have the cross() function with the proper parameters. Click on evaluate buffer as depicted in figure 19 and write down the result, we will use it later in AC analysis. You can now close the calculator and the graph window.

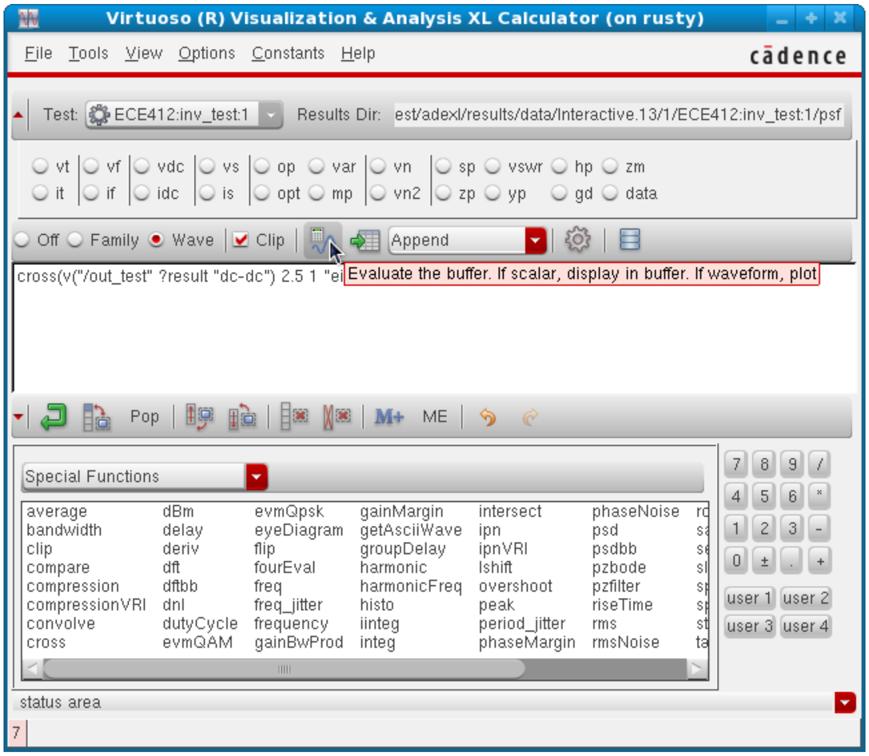


Figure 19: Virtuoso Calculator

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 change 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 20.

3. Save the schematic, by clicking on the Check and Save icon.

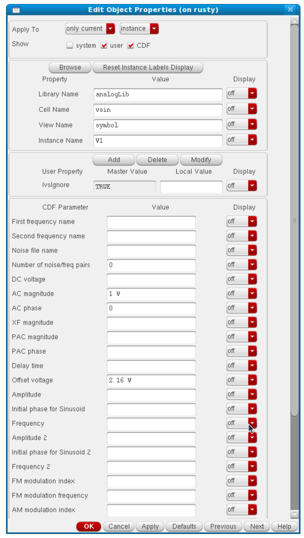


Figure 20: AC input source

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 21.

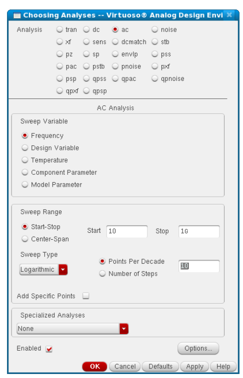


Figure 21: AC analysis properties

5. Under Data View: Disable all other analyses and make sure that "ac" is checked. Start the simulation. If the plots do not appear go to **adexl** tab then **Results** and click on "Plot All". The simulation result will look like figure 22.

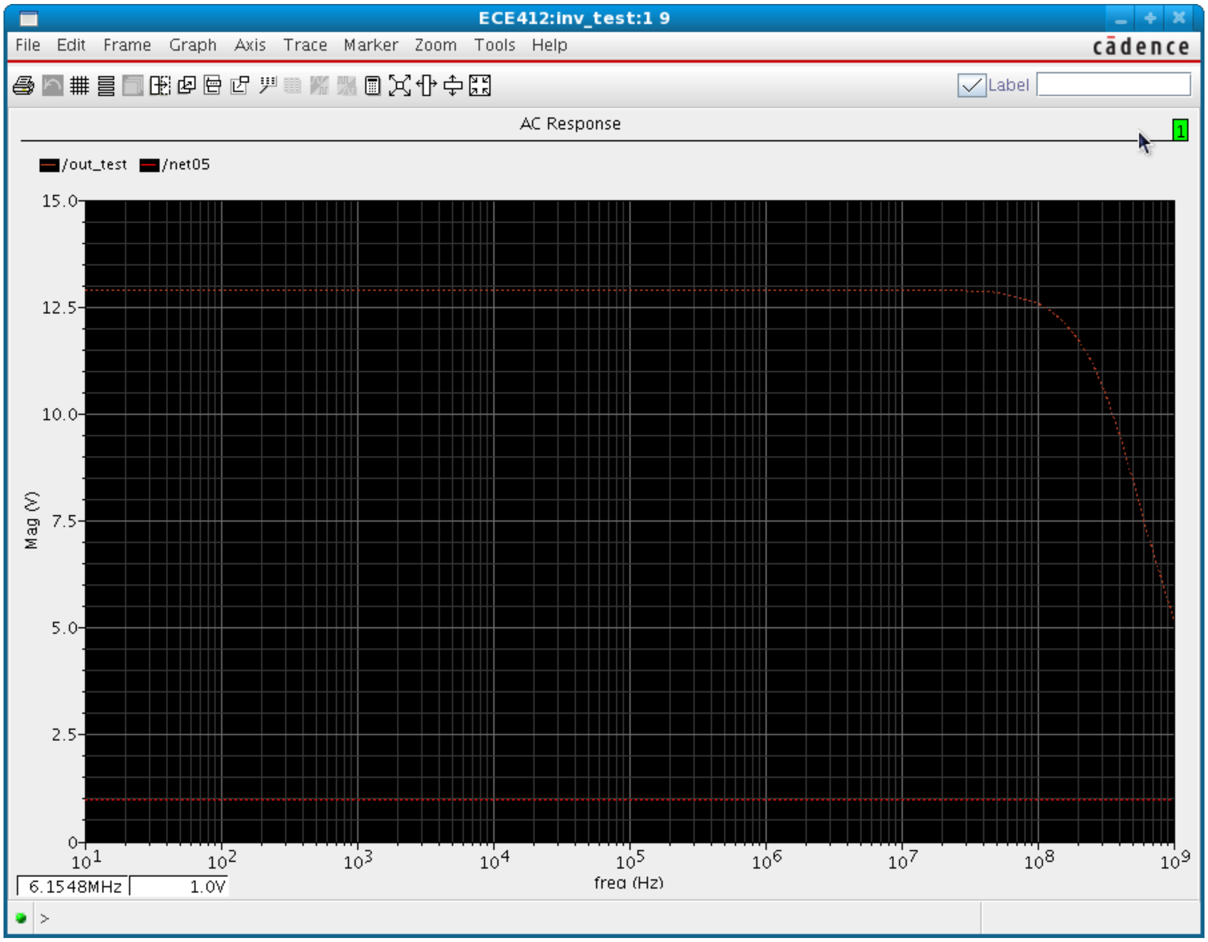


Figure 22: AC analysis for an inverter

6. To plot the 20dB Magnitude and Phase. Go to **adexl** tab then **Results** and right click on "/out_test" **Direct Plot**⇒ **AC Magnitude Phase**. The graph is depicted in figure 23.

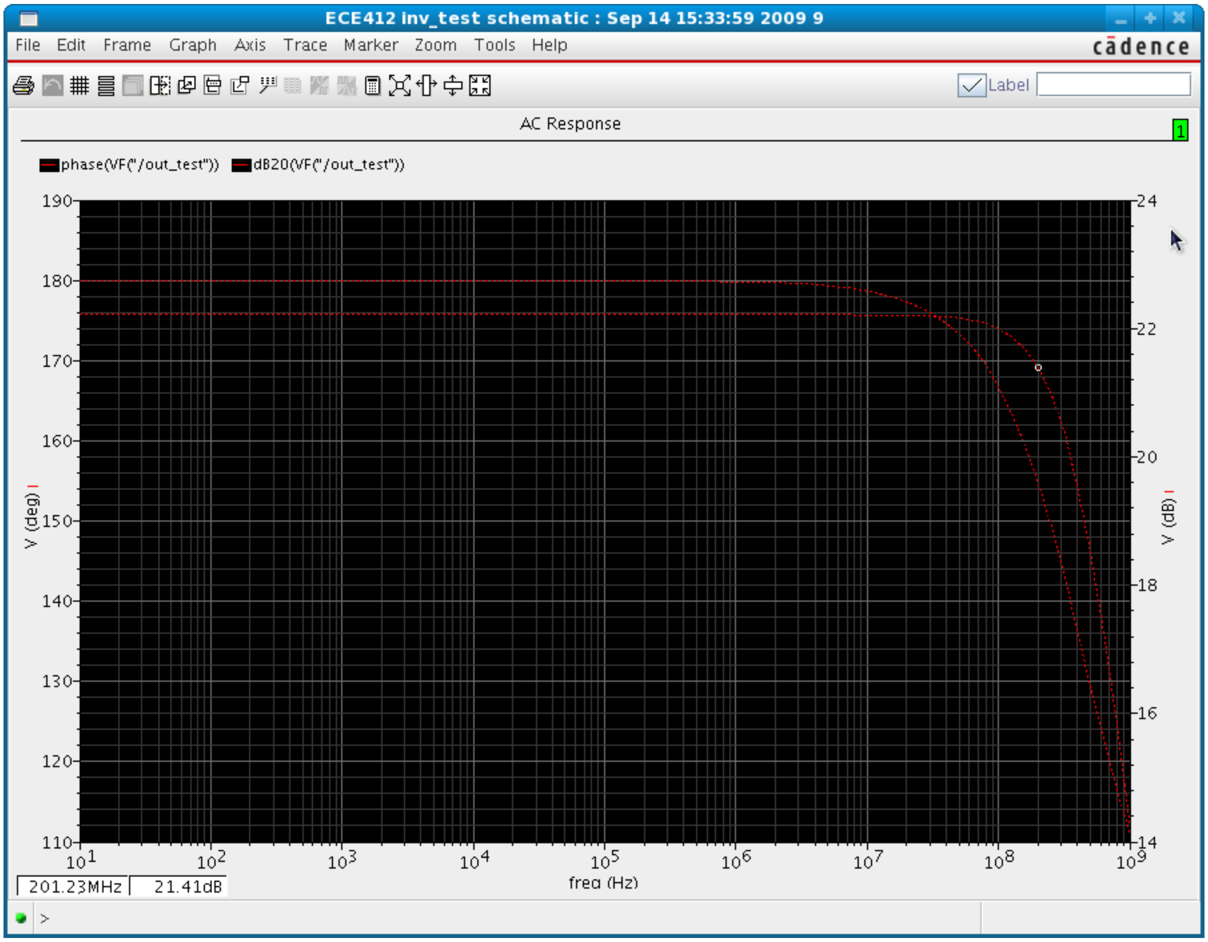


Figure 23: 20dB Magnitude and Phase