VLSI Lab Tutorial 2

Simulation Using Spectre

1.0 Introduction

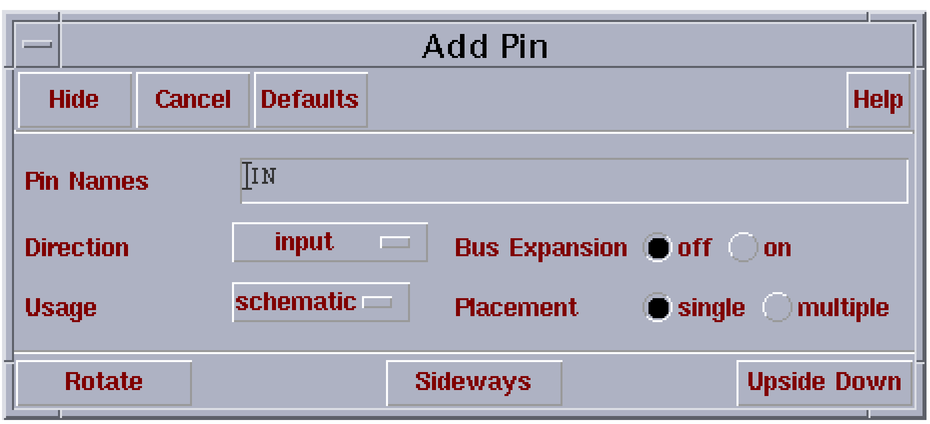
The purpose of the second lab tutorial is to help you in simulating your inverter design that you designed in the first lab tutorial. You will perform transient analysis and dc analysis for your inverter. We will use a simulator called spectre for our analysis.

Upon completion of this tutorial, you should be able to:

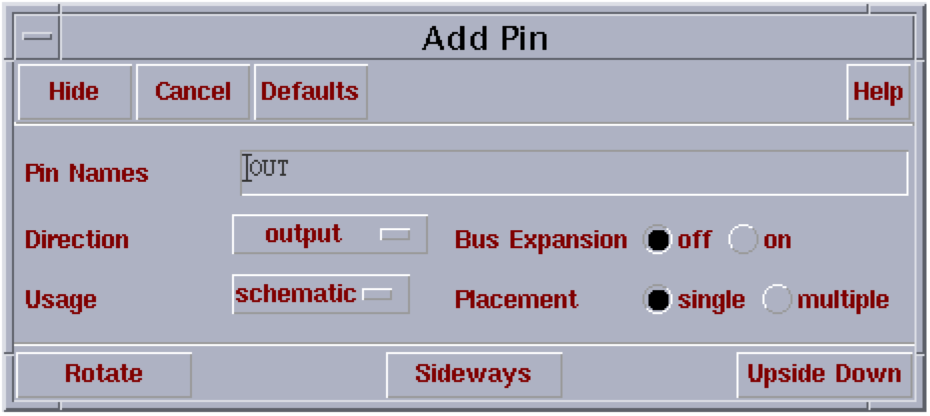
- Simulate your schematic using spectre.
- Perform transient analysis and dc analysis.

2.0 Schematic to be simulated

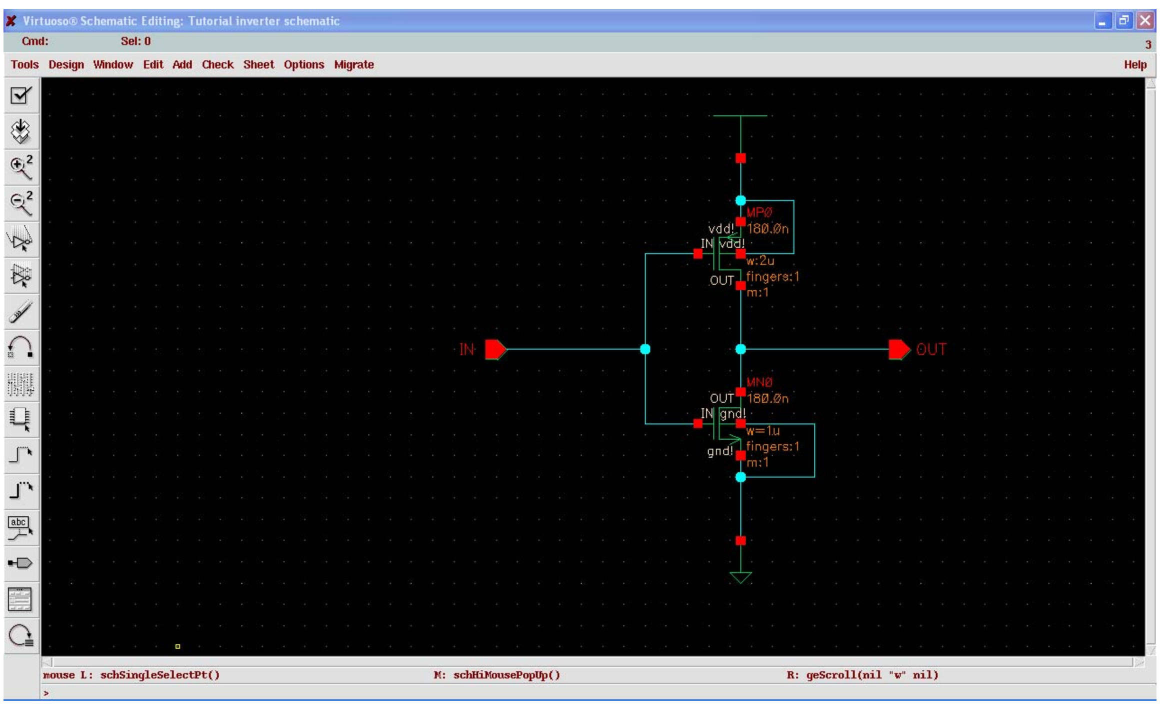
Use the schematic from Tutorial 1. The schematic consists of one nmos transistor and one pmos transistor (see Tutorial 1), a gnd and a vdd symbol and then an input port IN and and output port OUT. We are now going to add input, output ports (if not added before). You can place these either by pressing the **PIN** button on the left or by going to **Add** → **Pin**... Make sure you choose input for **IN**:



and output for **OUT**:



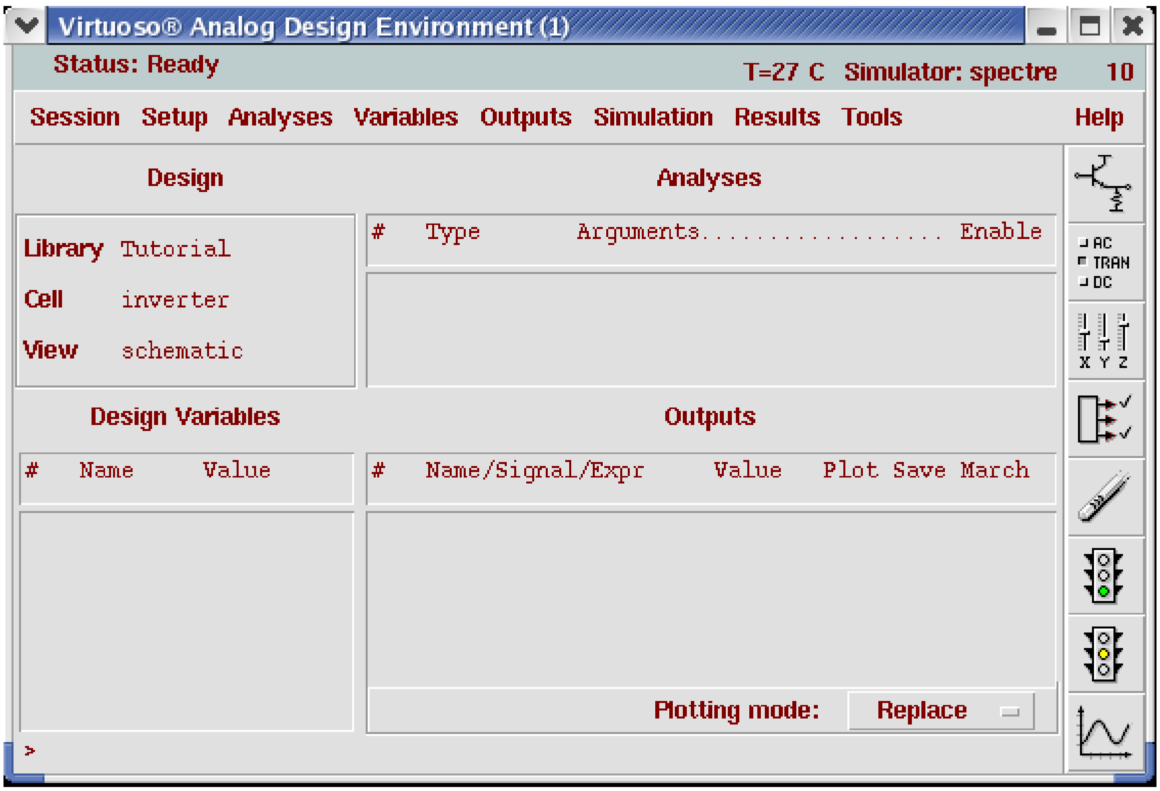
Your final schematic should look like this:



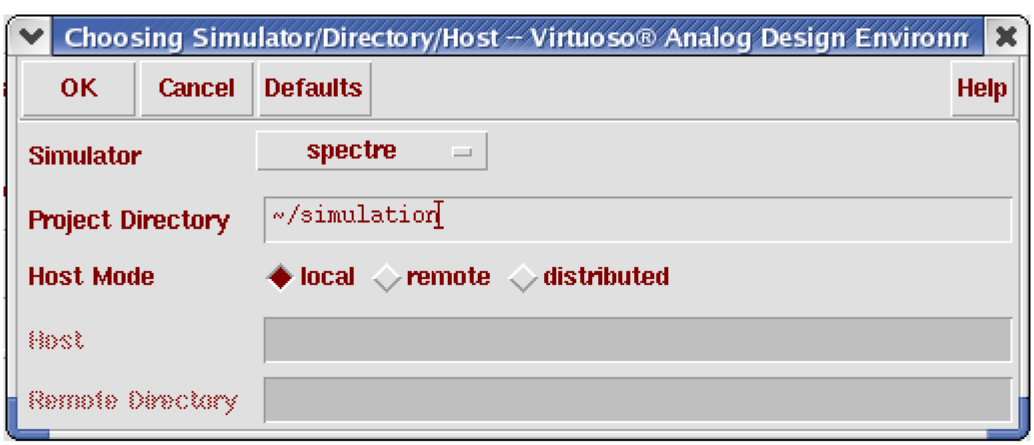
Check and save and make sure you don't get any errors or warnings. Assuming there are no errors we are now ready to start simulation!

3.0 SIMULATION

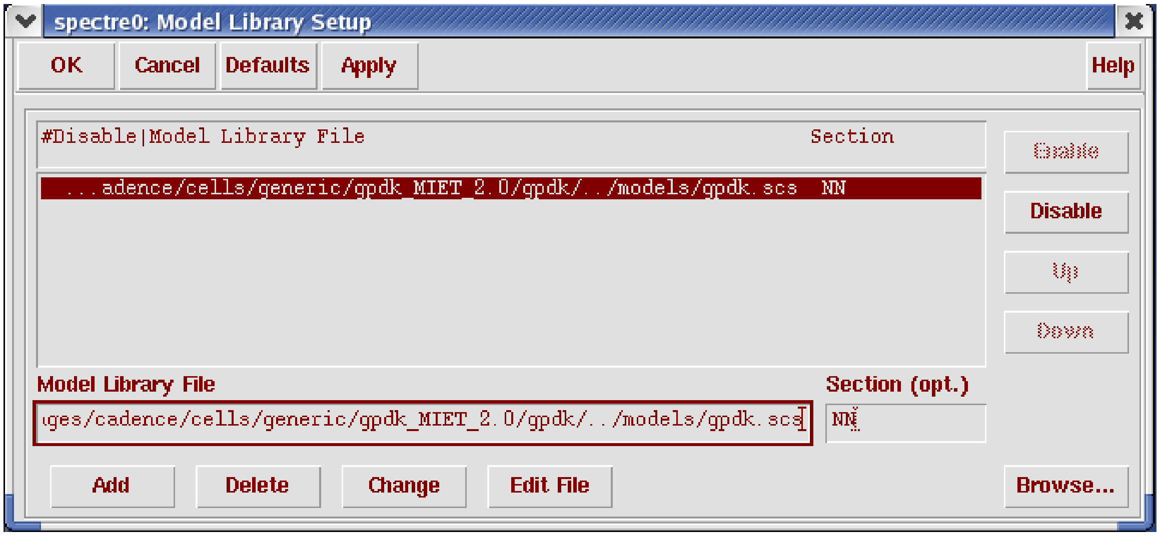
In the Virtuoso Schematic window go to **Tools → Analog Environment**. There is going to be another "What's New" pop-up window that you can read and close or minimize. The design should be set to the right *Library*, *Cell* and *View*.



First we need to choose the simulator, we will choose Spectre. Go to **Setup → Simulator/Directory/Host,** and choose **Spectre** in the pop-up window, then click **OK**:

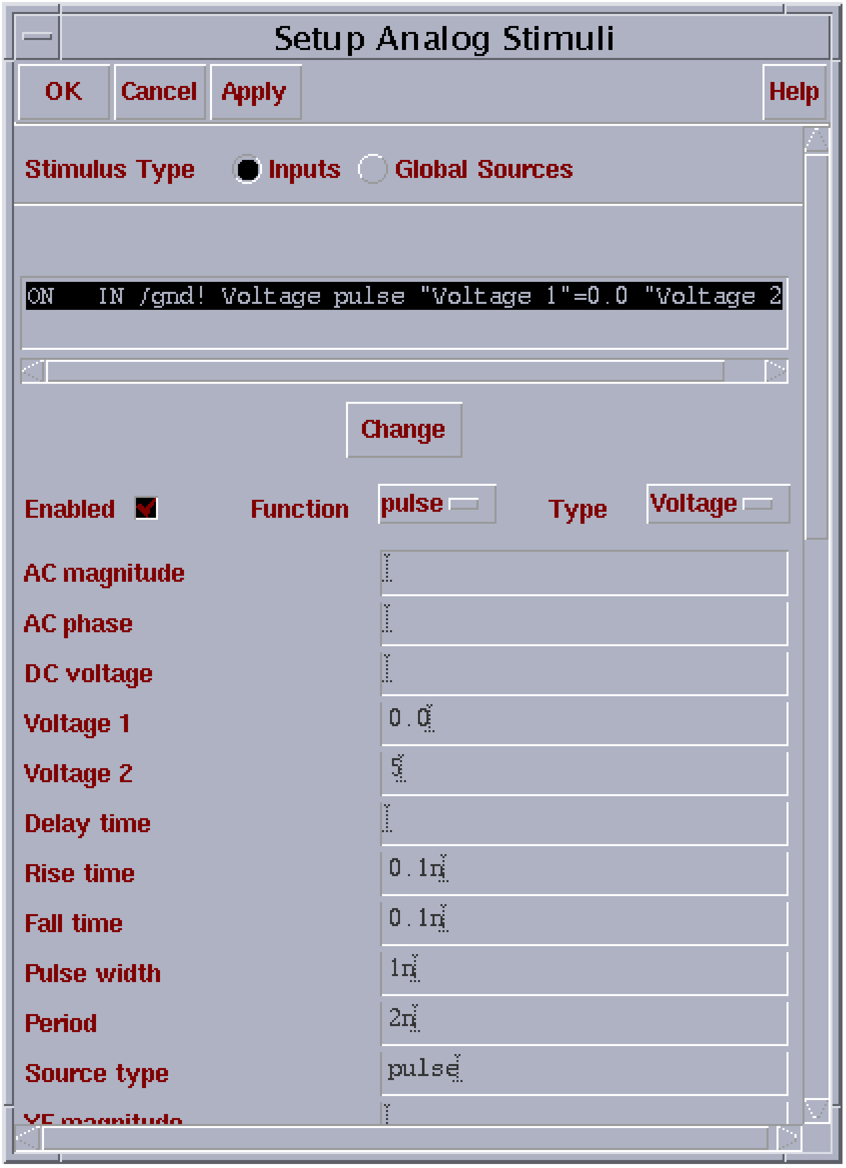


Go to **Setup** → **Model Libraries** and **choose** (you can type directly or use Browse) the **library** and then **click Add** (this is important, don't forget to do it), which adds the models for the NMOS and PMOS, then OK.

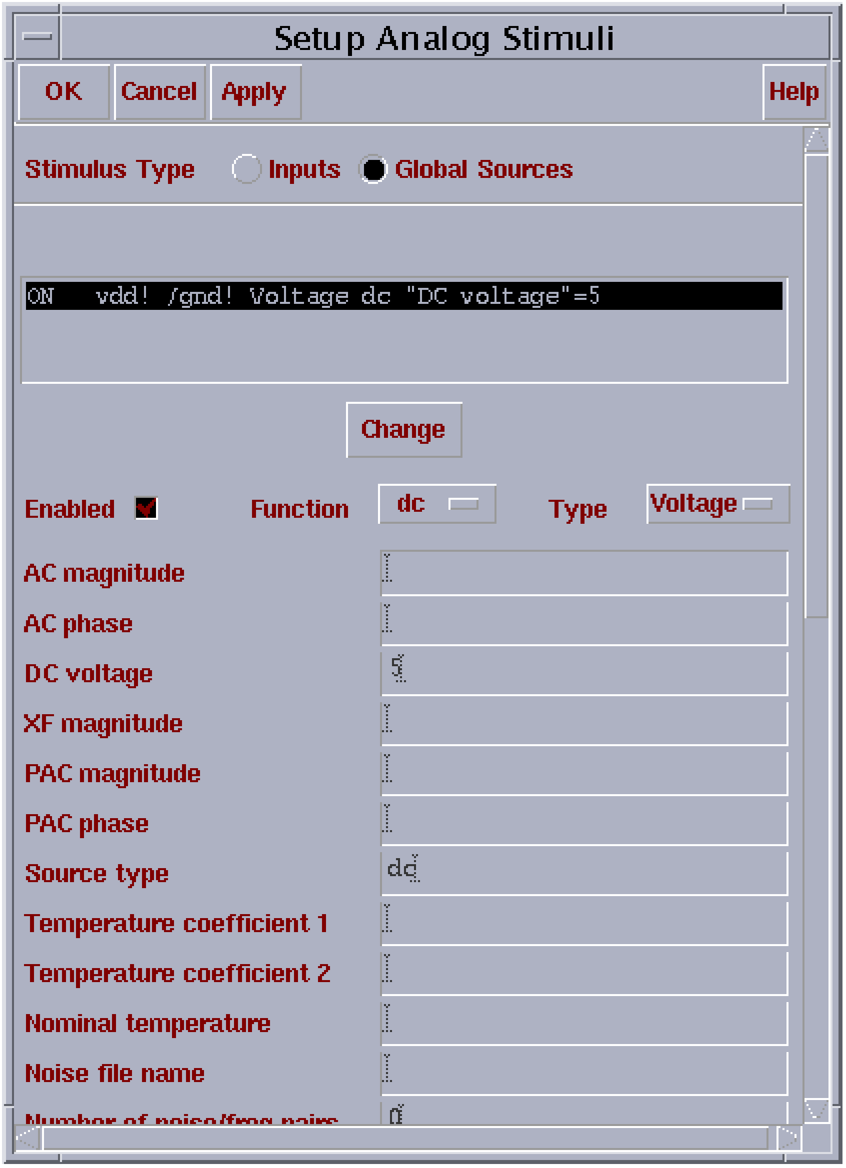


First we will perform a transient simulation to see that our inverter works correctly. In the schematic window go to **Tools** → **Analog Environment**. The design should be set to the right *Library, Cell* and *View.*

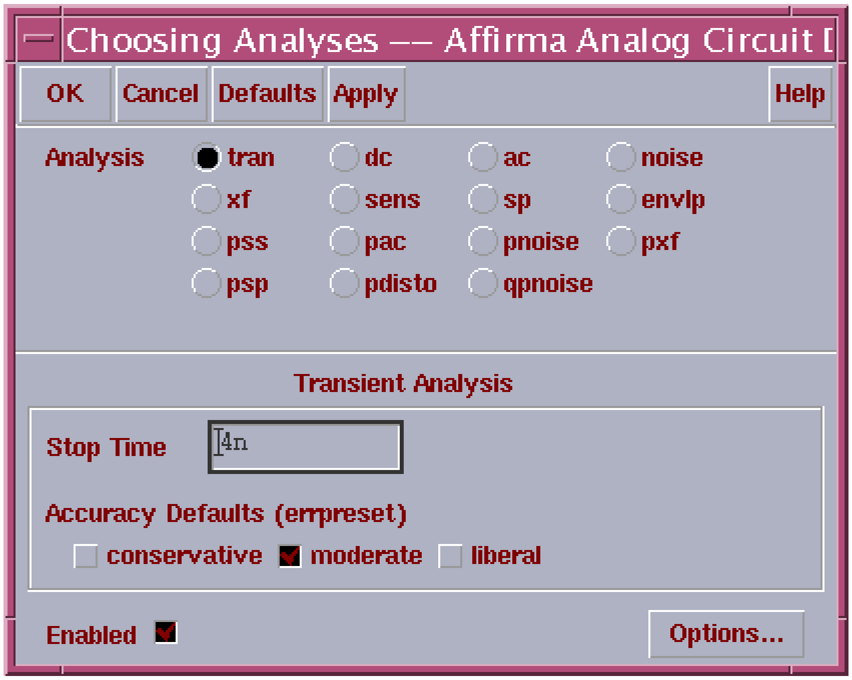
We first need to set up the right simulator (spectre), and then set the two model library files for the nmos and pmos (please revisit Tutorial 1 for the details). We also need to set up inputs and power supply since we don't have explicit voltage sources. Go to **Setup** → **Stimuli**. Initially you have the Inputs chosen; you should have only one (IN). Click on **Enabled, Function pulse, Type Voltage, Voltage1 0, Voltage2 5, Rise time 0.1n, Fall time 0.1n, Pulse width 1n, Period 2n,** Source type pulse, and click on Apply. The input should turn from **OFF** to **ON**. **CAUTION** Cadence is very fragile concerning this dialog box, make sure you enter the numbers exactly as above (i.e. no space between the digit and n, etc.). If you get syntax errors later in simulation they are most likely because of this.



Now we need to setup the **global sources** (*power supply*). Click on the **Global Sources**, you should have only one **(vdd!).** Click on **Enabled, Function dc, Type Voltage, DC voltage 5, Source type dc, and click on Apply**. The **vdd!** should turn from **OFF** to **ON**. Click **OK**.

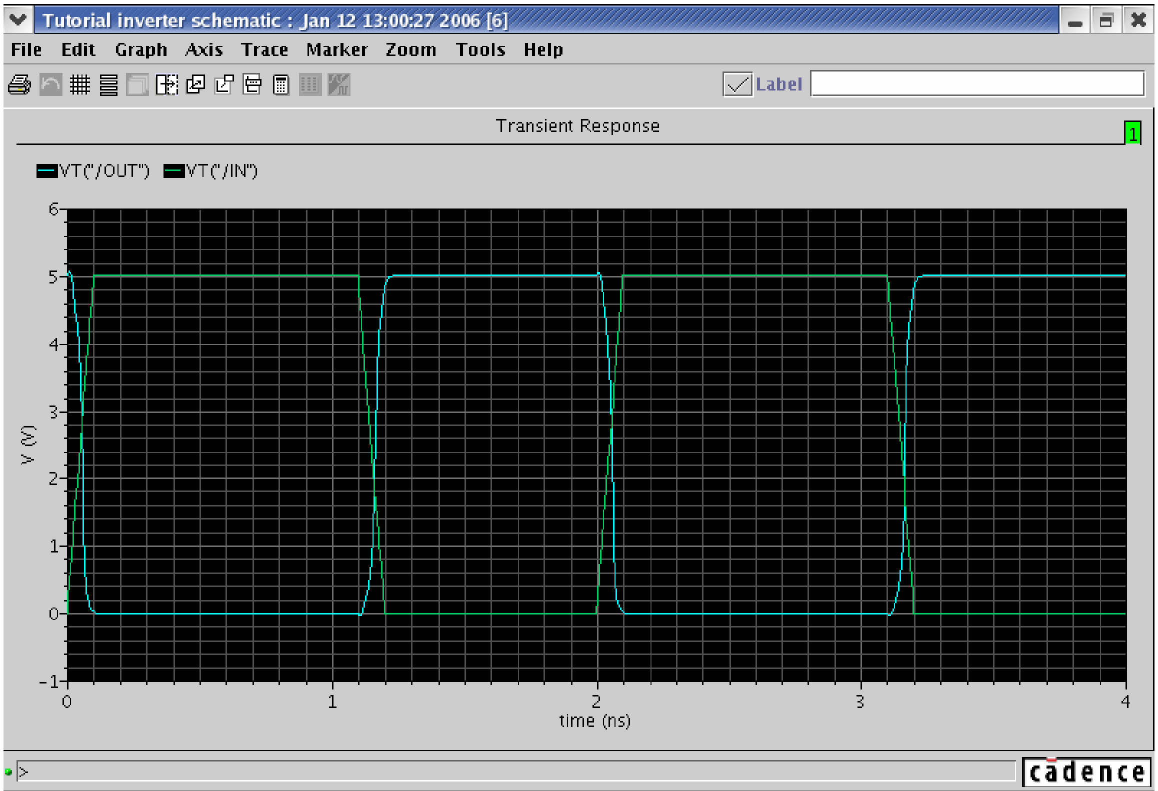


Now you need to choose the type of simulation, go to **Analyses** → **Choose**... In this case we will choose **tran** which is the default, **4n** as the **Stop time** (2 periods) and **moderate** as the **accuracy default**.

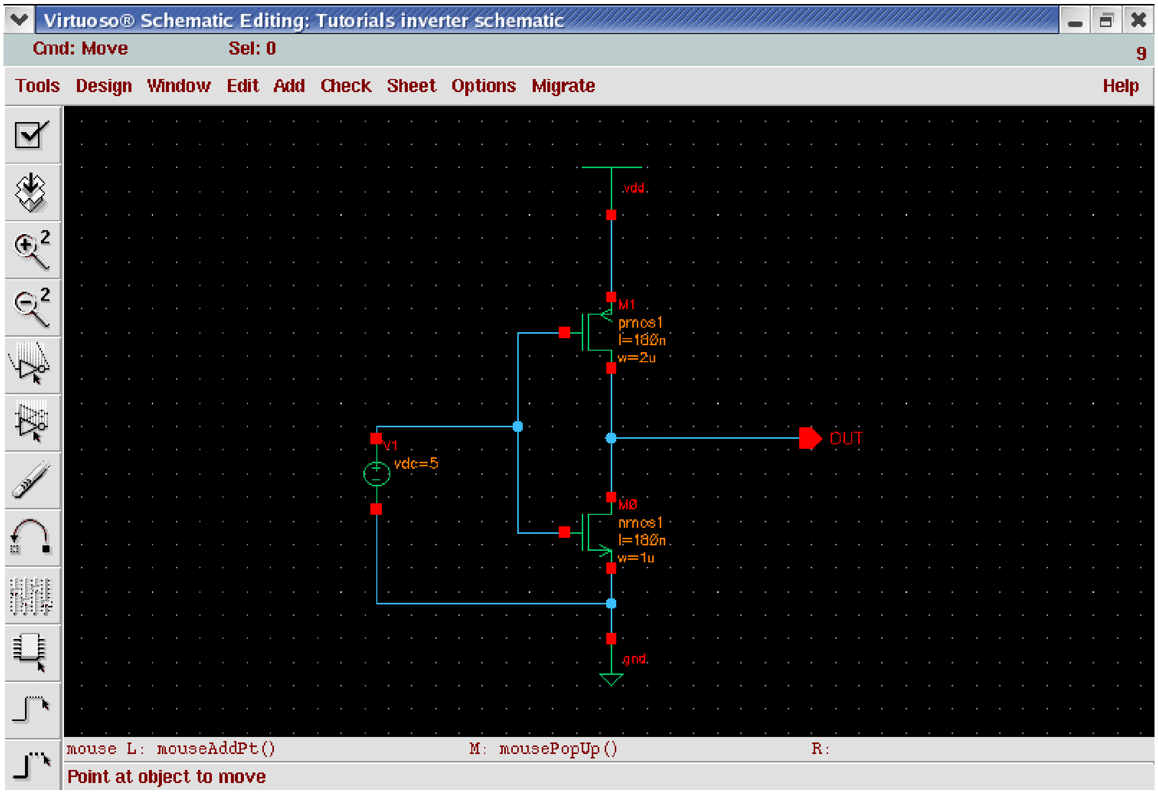


Now go to **Outputs** → **Save All** and click on **allpub** for signals to save (default) and **all for device currents.** Click **OK**. **CAUTION** In general, once you have a big schematic, you will want to only save a few signals for simulation, and this will make your simulation faster. For small circuits as we have now it doesn't make a big difference though.

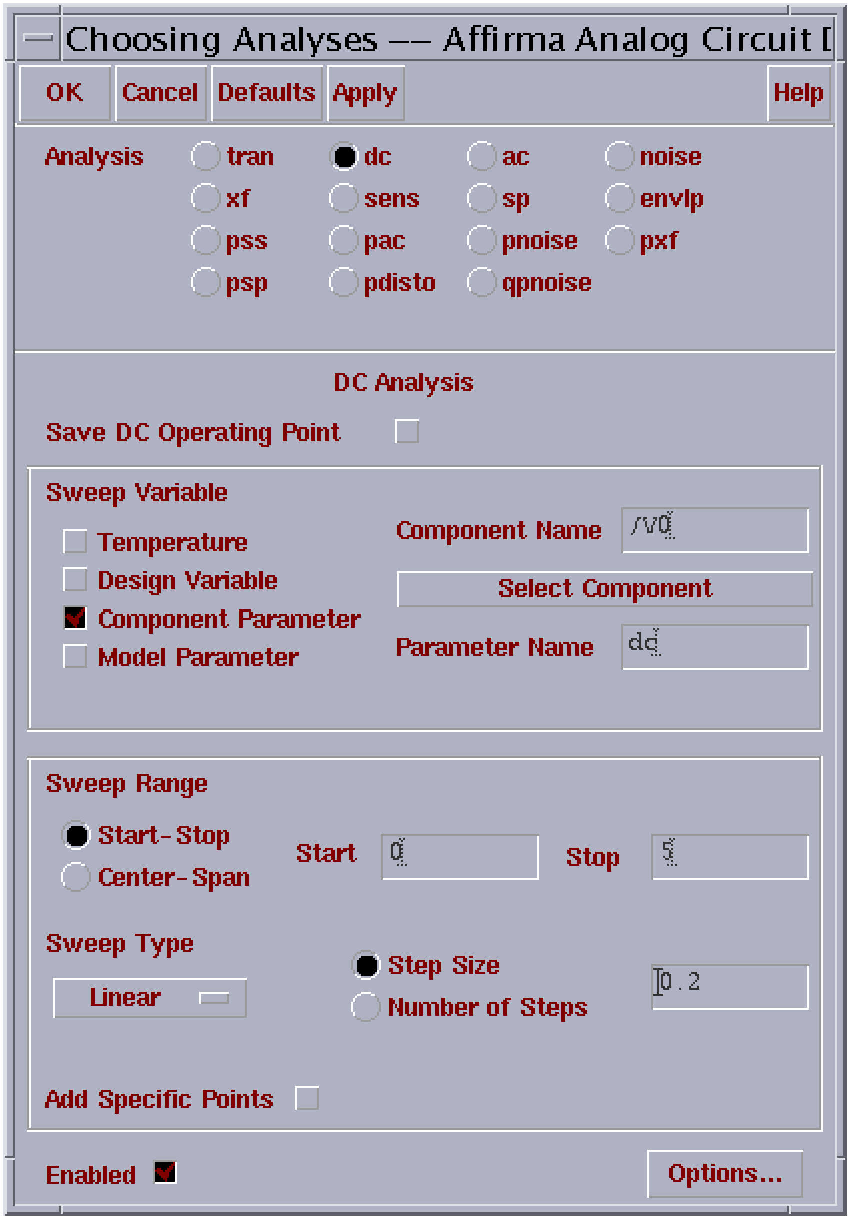
Now we can finally simulate! Click on the **Netlist and Run button** (looks like a green light) on the right or go to **Simulation** → **Netlist and Run**. Click **OK** on the Welcome to Spectre window which should start the simulation. In case you have errors you will need to go back and correct them. This can be tricky! You may need to do **Simulation** → **Netlist** → **Recreate** if you change the schematic. **CAUTION** Each time you change the schematic you have to do Check and Save!. Assuming there are no errors you can now admire the simulation results. Go to **Results** → **Direct Plot** →**Transient Signal** which will pop-up your schematic window. Now you have to click on the signals you want to see. Since this is a transient analysis we want to see the input and output voltages. In order to do this you have to click on the **input** and **output nets**, then the **ESC** key. You should finally get the desired simulation results, 2 periodic signals as expected from an inverter.



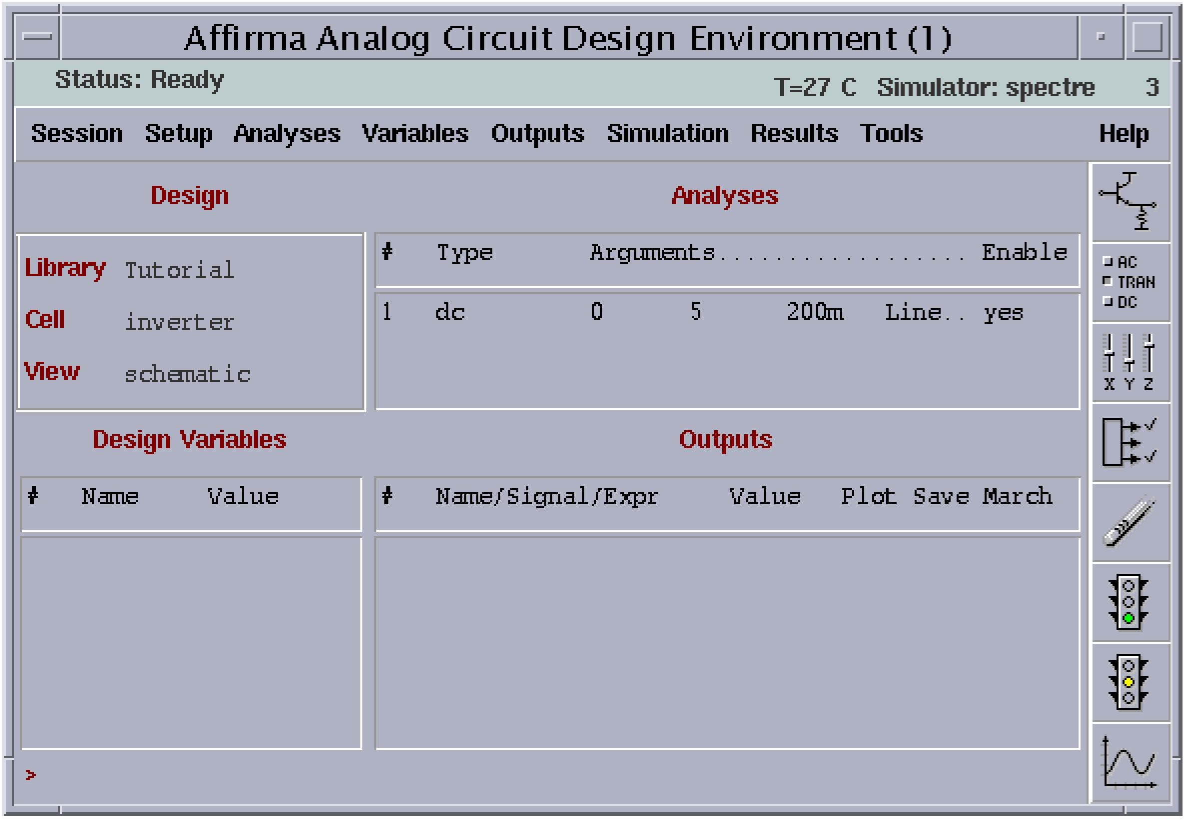
We are now almost done, before we finish let's try to also plot the VTC for the inverter. For this we first need to replace the **IN** pin by another **vdc** power supply from the **analogLib** library as in Tutorial 1. Change its **DC voltage** property to **5**



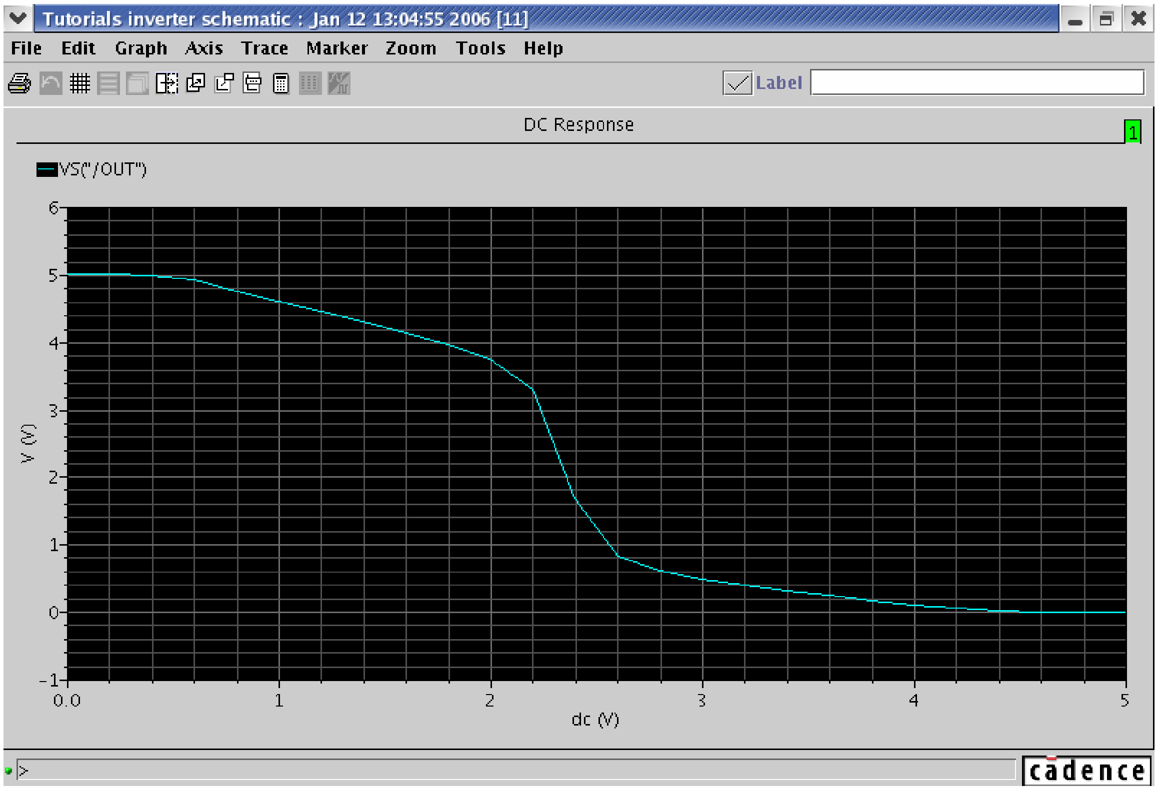
Check and Save (make sure you get no errors). Now go to **Analyses -> Choose, dc** and **Component Parameter**, Select Component, then the voltage source in the schematic, and then **choose 0 as Start, 5 as Stop and 0.2 as step**.



Now click on the old tran analysis and then go to **Analyses** → **Delete** so that you are left with only one Analysis.



Finally do **Netlist and Run**. If you don't have any errors you can now go to **Results** → **Direct Plot** → **DC** and click on the **output net** and the **ESC** key to get an inverter VTC.



It is a good idea to save your state before exiting the simulator in case you want to redo some of the simulations you can start by loading a saved state. Congratulations, this is the end of Tutorial 2.