Introduction to MultiSim – Part 2

Prepared by:
Mohamad Eid
Summer 2007

The purpose of this tutorial is to demonstrate the use of MultiSim to simulate operational amplifier circuits (uA741 OpAmp), such as inverting and non-inverting amplifiers, filters, and oscillators. The circuit shown in Figure 1 is used as a simulation example in the current tutorial.

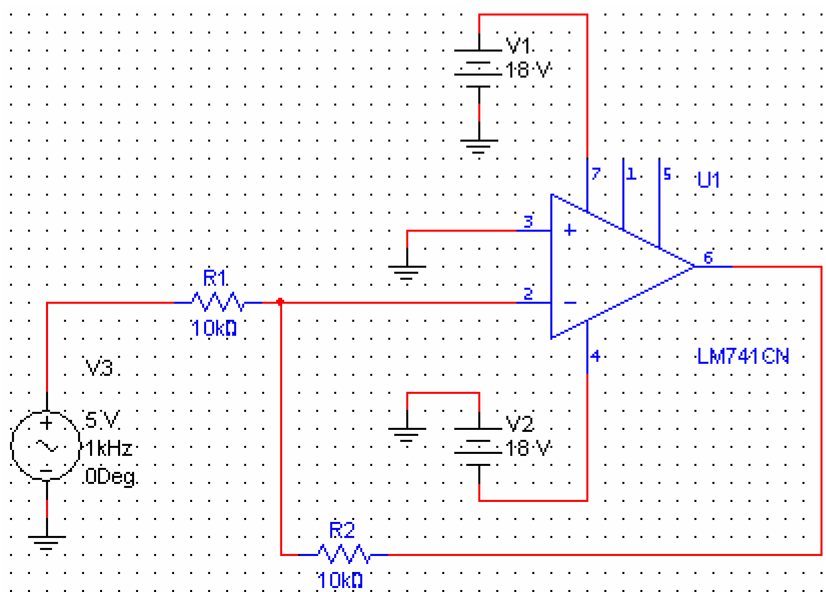
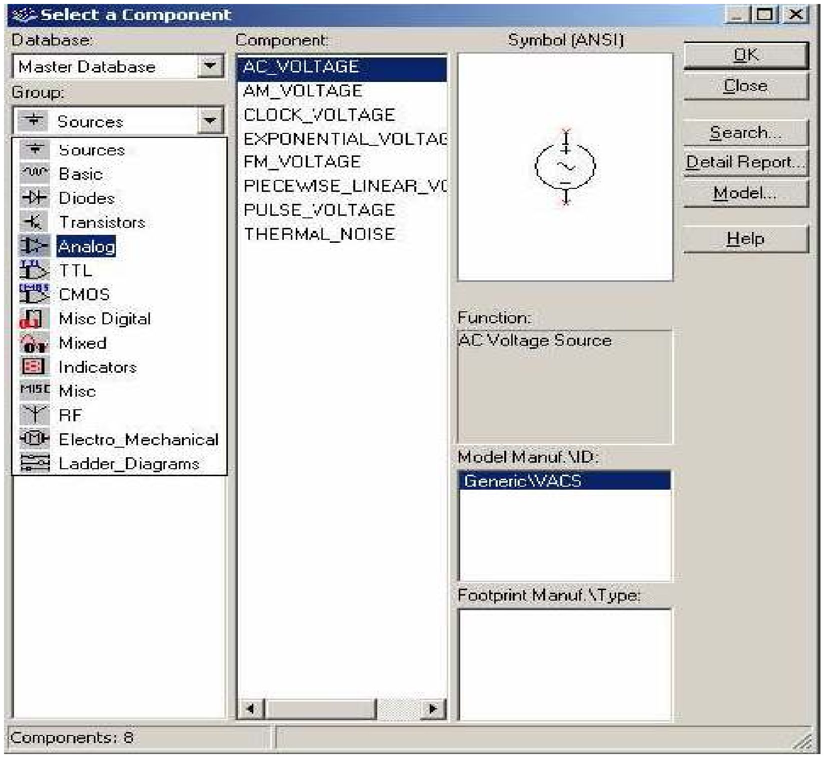


Figure 1. An inverting unity gain operational amplifier circuit

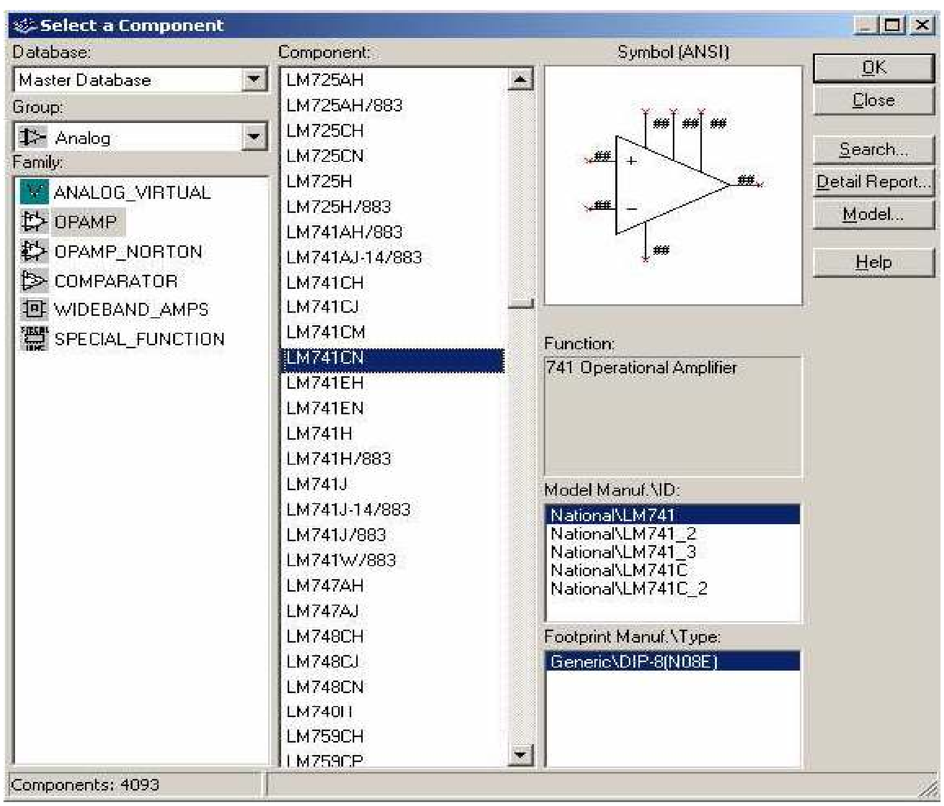
1) Start the MultiSim program as shown in the previous tutorial (For Windows users the default location can be found by clicking: Start ->All Programs -> Electronics Workbench -> DesignSuite Freeware Edition 9 -> MultiSim 9).

2) You will be creating a new schematic and simulation so choose File->SaveAs, navigate to or create a directory where you can save this schematic and simulation then fill in the filename as shown below. (I called this one tut1.) Click **OK** when you have navigated to the proper directory and entered a name for the project.

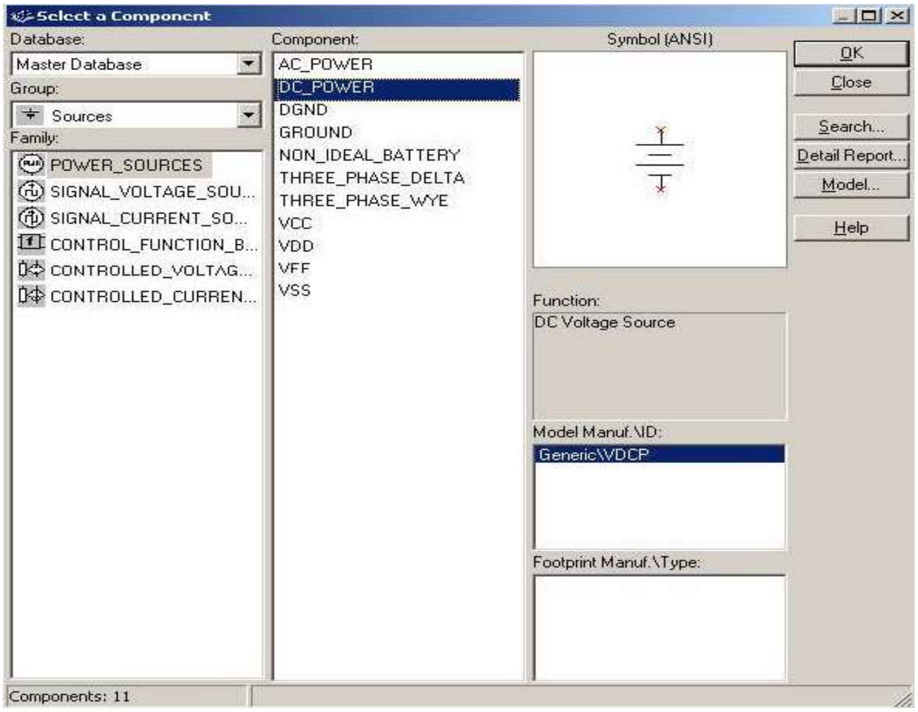
3) You should now have a blank schematic. Start placing components by selecting Place->Component from the menu bar. You will see another dialog box as shown. Start with the OpAmp so pick *Analog Components* in the drop down menu as shown:



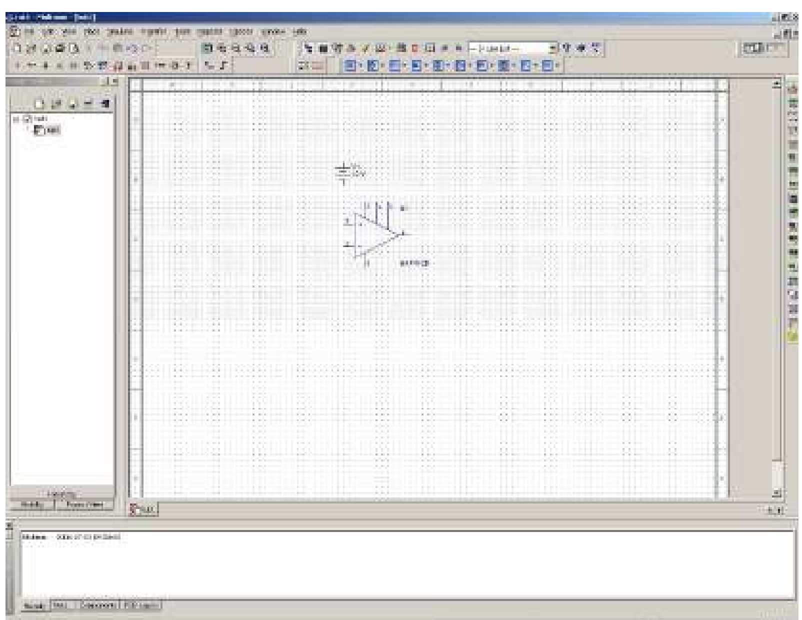
4) You should see a new dialog box. Select *OPAMP* then *LM741CN* as shown below then click **OK**.



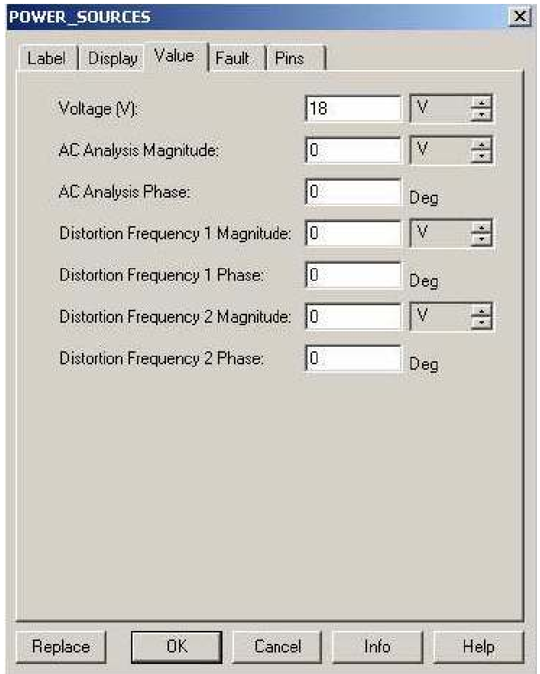
5) Place the OpAmp on the schematic. Next you must add a couple of power supply connections. You will add a DC source by again selecting Place->Component. Only this time pick *SOURCES* from the drop down menu. Then select *POWER_SOURCES* and *DC_POWER* as shown below then click **OK**.



6) Put the battery (*DC power source*) on the schematic somewhat above the OpAmp as shown below.

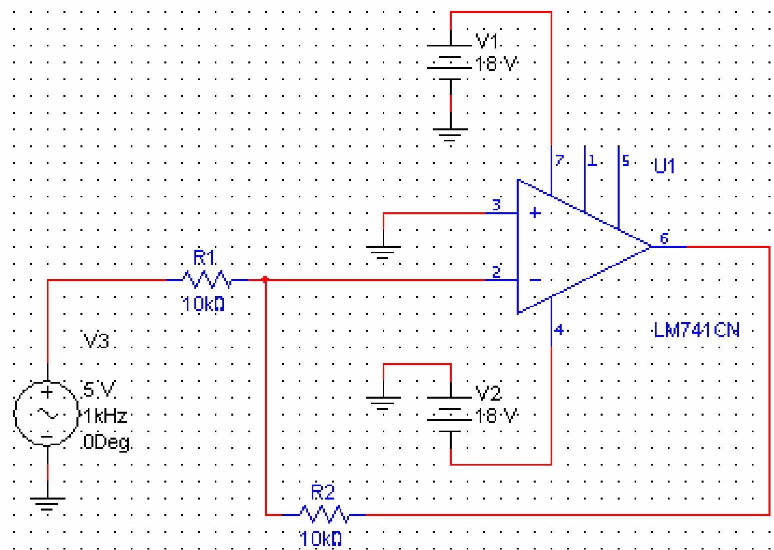


7) Double click on the “12V”. Change the voltage from 12 to 18 then click **OK**.

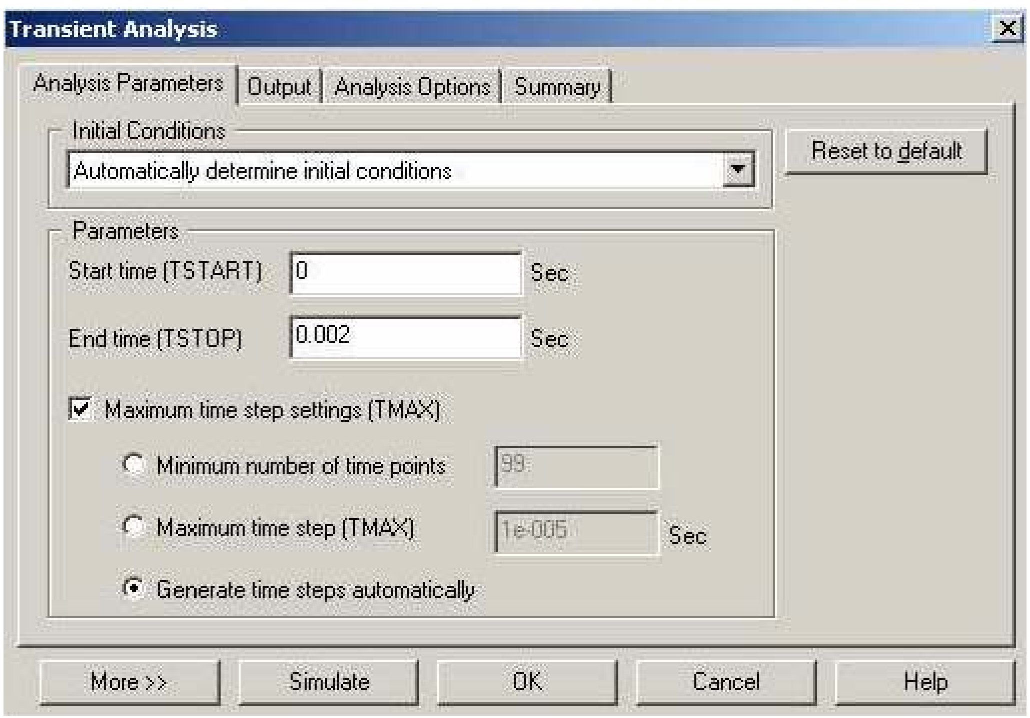


8) Place the rest of the components on the schematic as shown below. Use *Place->Wire* to add wires for connecting components together. Resistors are under the Basic components drop down menu. Note you must specify “k_” in the filter to locate the 10K resistors. Note *GROUND* symbols are at SOURCES->POWER_SOURCES. Your completed schematic should resemble the one below.

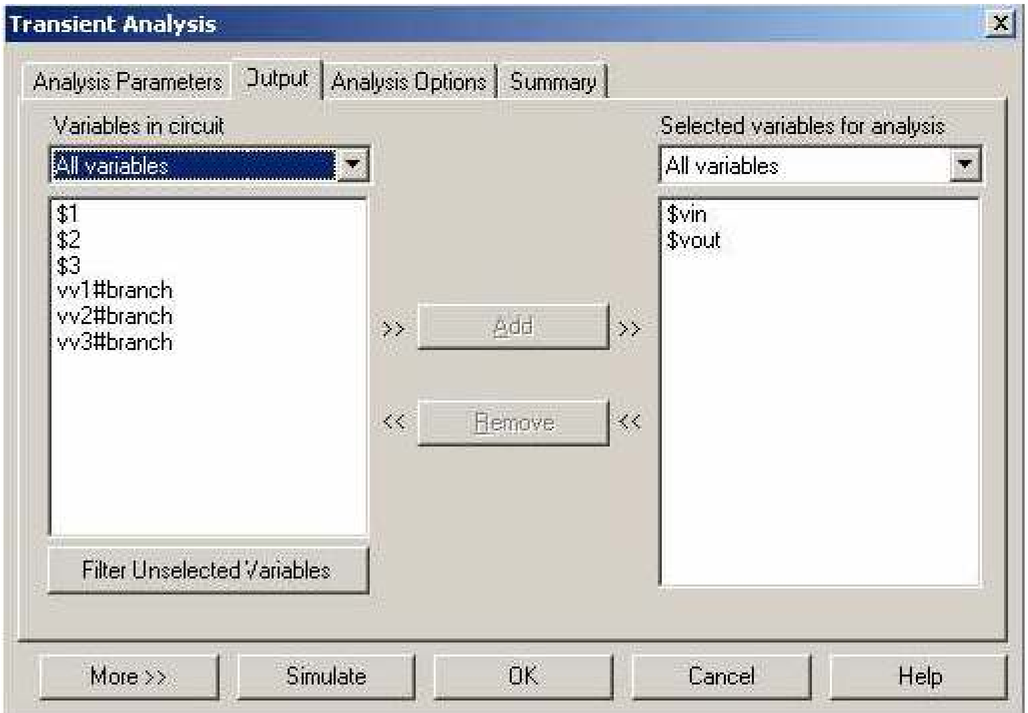
9) Be sure all components are connected as shown above. Now you will name the input and output signals to make them easier to locate in simulation. First highlight the wires above the 5V 1KHz source *V3*. Do this by placing the cursor on the red wire and left clicking once. Now right click once and select “properties” from the popup menu. Change the net name to Vin as shown on the next page then click **OK**. Use the same method to change the output net name to “Vout”.



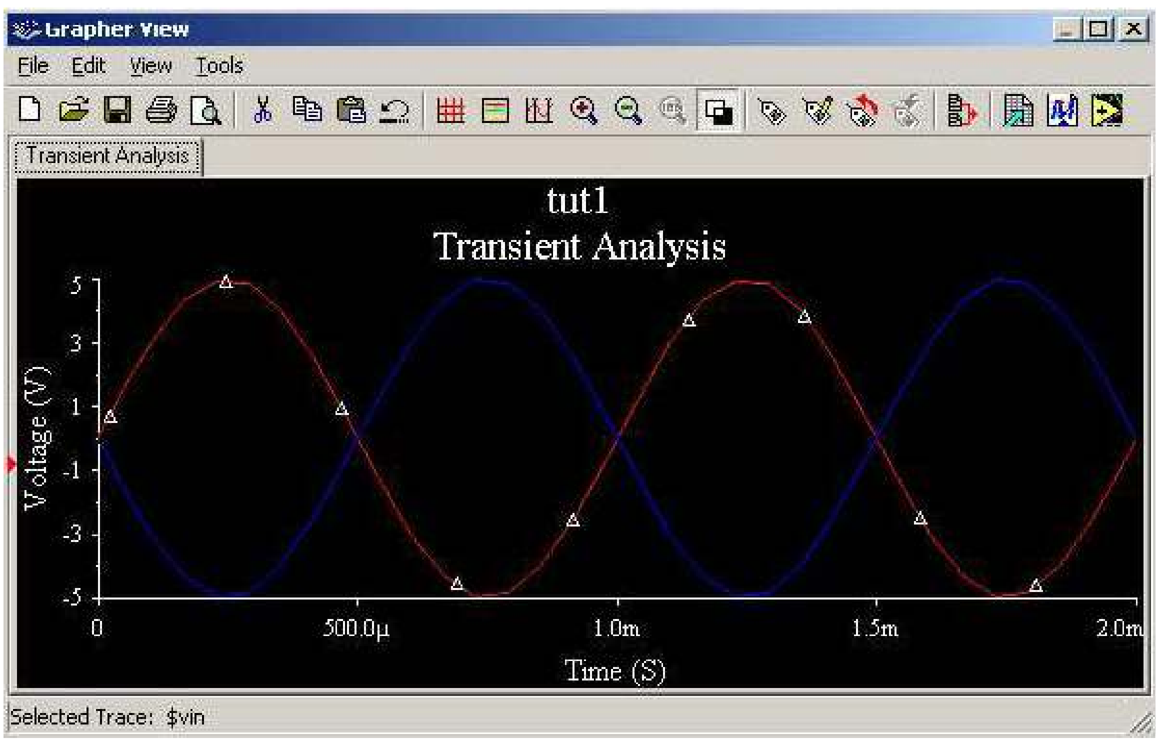
10) You will now run a simulation. First select Simulate->Analyses->Transient Analysis from the top menu bar. Change the End Stop Time (*TSTOP*) to 0.002 as shown below.



11) Next choose the *Output* page and select the two signals you want to see on the output. Your dialog box should now look like the one on the next page.

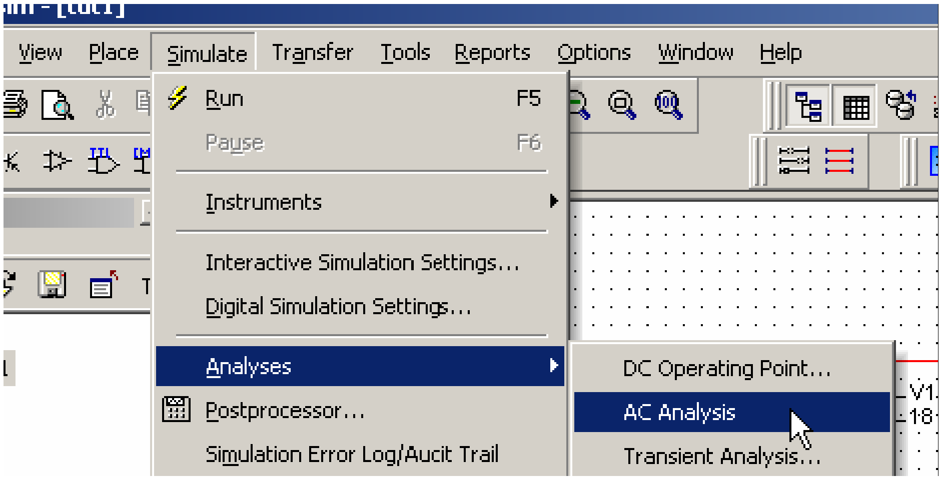


12) At this point you are ready to simulate. Left click on the **Simulate** button to run the simulation. You should get a result window that looks like the one below.

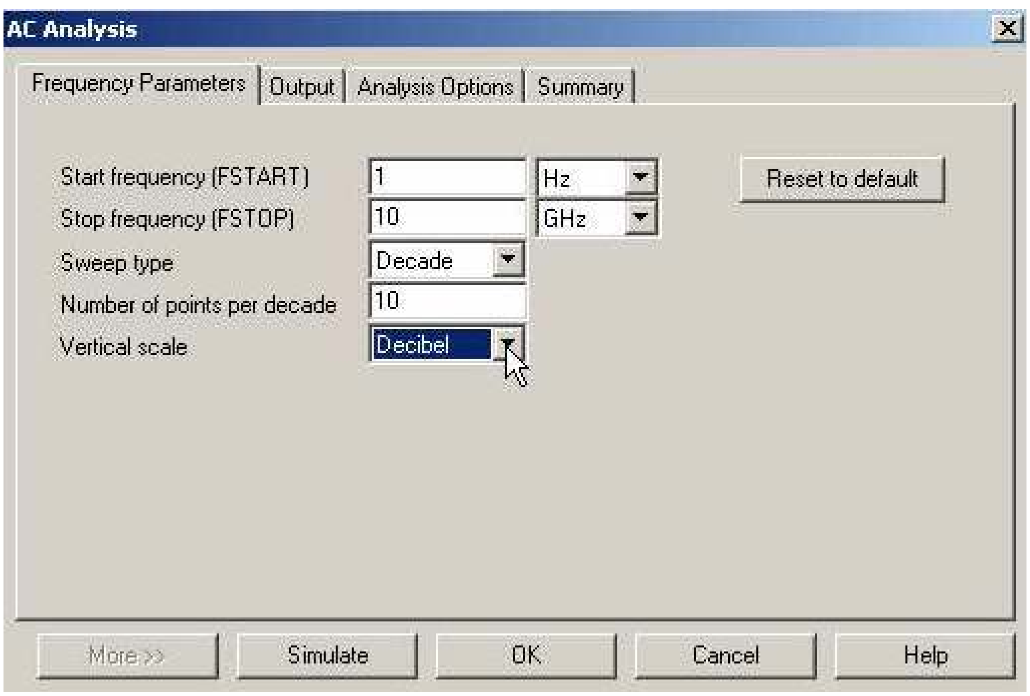


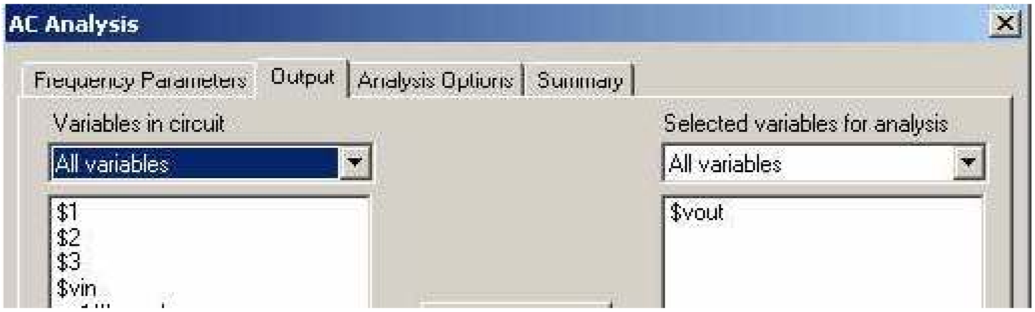
Now run an AC analysis to make a Bode plot of the response of your circuit. For an ideal Op Amp the gain would be always be 1. In the real world, capacitive and inductive effects at higher frequencies cause the gain (and phase) to shift. The Bode plot is a graph of gain and a graph of phase shift relative to input frequency.

13) Select Simulate->Analyses->AC Analysis as shown.

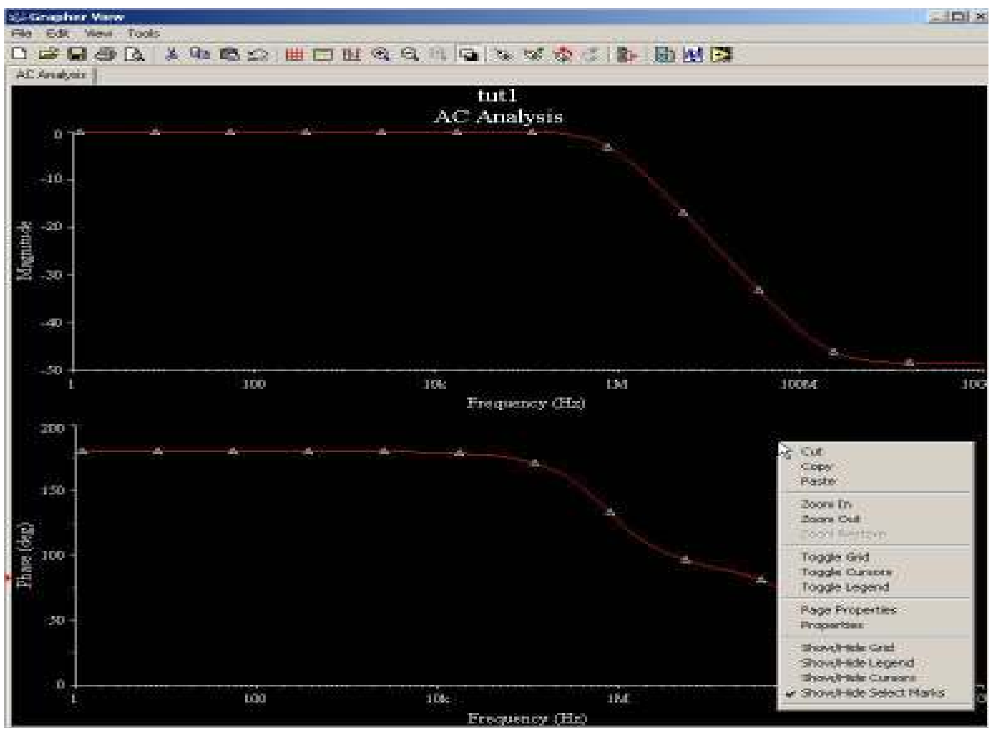


14) Make sure the frequency dialog box looks as shown below then click the *Output* tab. Make Vout the only output then click the **Simulate** button as before.





15) The simulation results should appear as below. Note the top graph is gain in decibels (0 db is the same as unity gain or 1). See how the gain rolls off starting around 1 MHz. The bottom graph is phase shift. Since this is an inverting configuration, expect the phase shift to be 180 degrees. But notice how it drops to around 90 degrees by 10MHz then rolls down to around 0 by 10GHz.

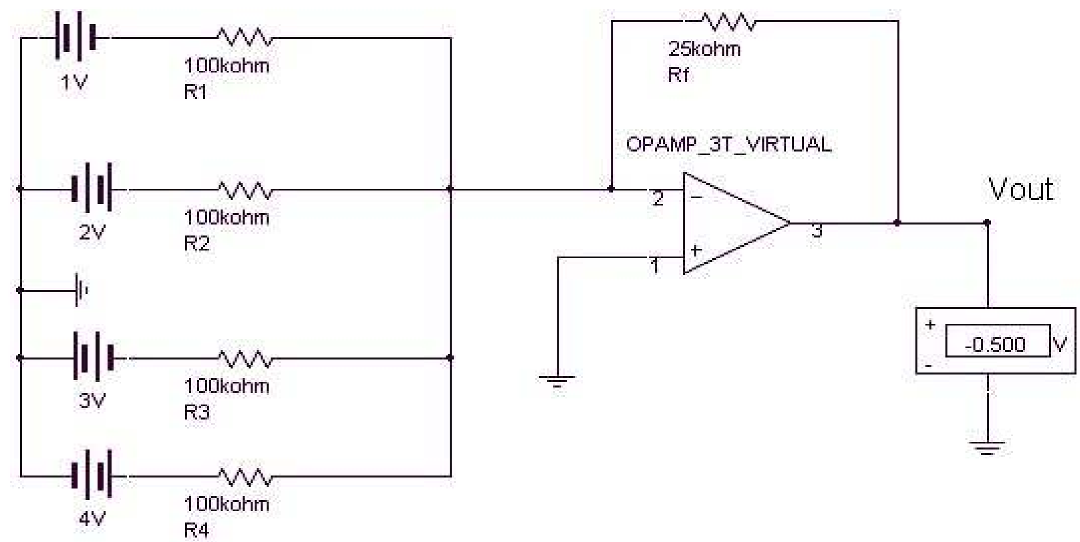


16) Put the cursor in the graph and right click to get the menu show. From there you can turn the grids on or off, add cursors, etc. You can also choose **Properties** and change the axes of the graphs. Use File->Print to print the Bode plot.

Examples

Simulate the following circuits:

a) Average Amplifier:



b) Bandpass Filter: