

LAB 3 OrCAD

by Norman Fong (October 31, 2013)

This lab will utilize OrCAD Capture and PSPICE to simulate various aspects of the 741 Opamp. OrCAD allows you to draw and setup various circuits which can be exported as netlists. These netlists can then be simulated using PSPICE to examine various DC, AC and transient characteristics.

You can do the simulations at home with the Demo version of OrCAD. Feel free to play around and familiarize yourself with the environment as you will need to know this software well for Lab 4.

FOR YOUR REPORT: Include all simulated plots (screenshot, print to PDF, etc) with clear indication of the important values.

1) [**CLICK HERE**](#) to download the source files for this project and un-zip them to your computer (preferably to your network drive H:\). These files contain a schematic for the 741-Opamp.

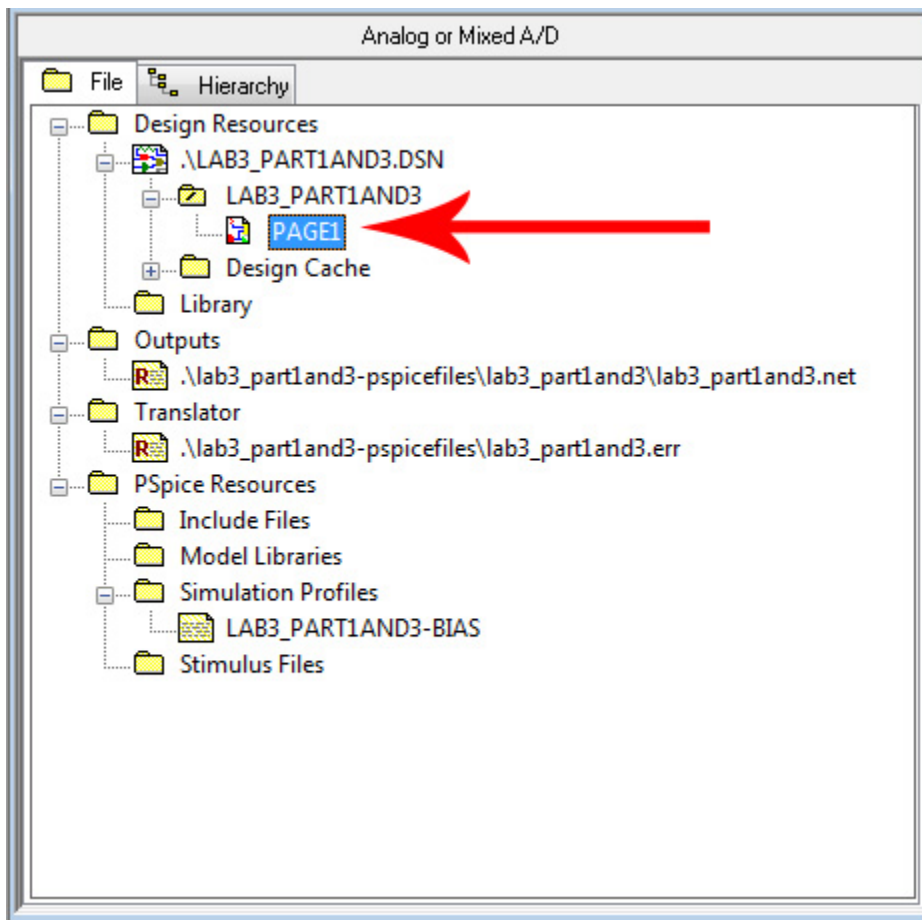
2) Open OrCAD Capture (Start> Cadence)

3) Now open the project (File> Open > Project) named "**LAB3_PART1AND3.OPJ**" within the source files you have downloaded.

4) The first window you see should be the project navigation window (pictured below). If the circuit schematic is not visible, double-click the schematic titled "PAGE1" under "Design Resources > \LAB3_PART1AND3.DSN > LAB3_PART1AND3"

5) You may be prompted with a window asking to use the Demo license, Click yes.

(Some students have reported that the window below does not appear when opening the project, rather they get a bunch of source code. If this is the case, make sure you are running the project off of your H: drive. This seems to fix the problem but the cause is currently unknown.)




Using OrCAD Schematics

- To modify the values for a particular component, double click on the component. For some components, like resistors and capacitors, a value will already be displayed on the schematic. You can double click on the value and modify it directly.
- Note that in PSPICE $m=M=10^{-3}$ and meg=Meg=MEG= 10^6 . **MI=Meg**.** Entering "M" for MHz is a common mistake made by new PSPICE users.

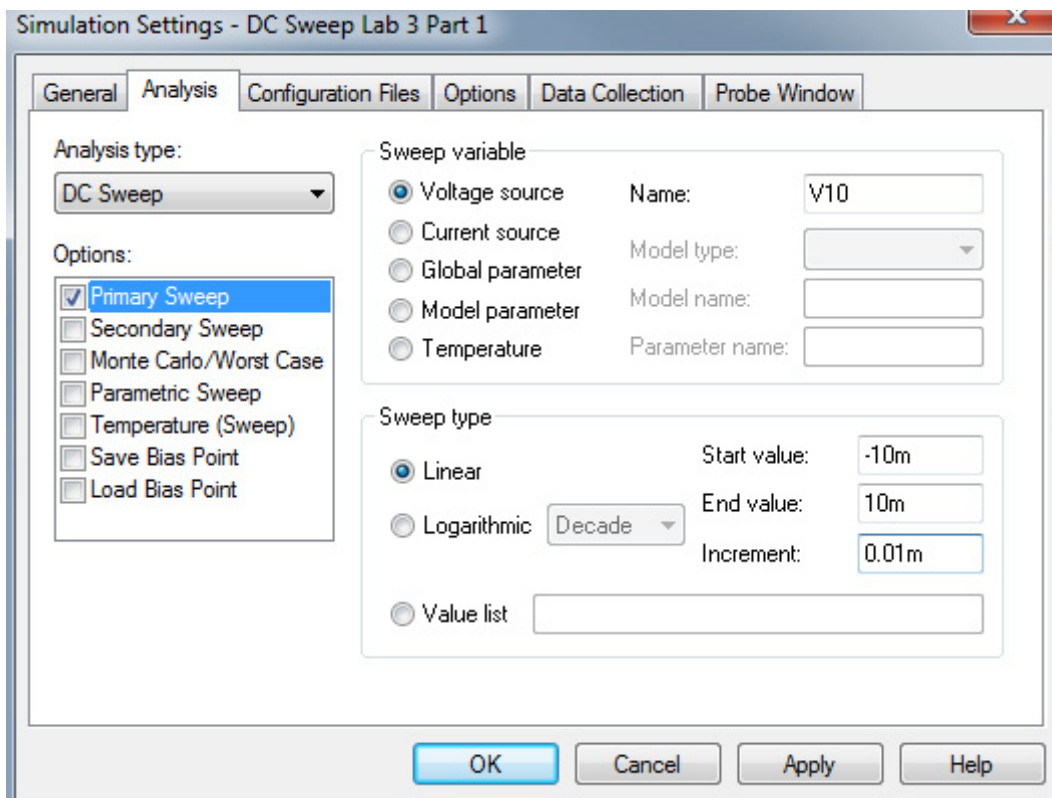
LAB 3 Part 1

1. DC Sweep


1) Create a new simulation by clicking the "New Simulation Profile" button (name it whatever you want). Setup your sim

by clicking the "Edit Simulation Profile"  button. The settings window may pop up underneath your current window.

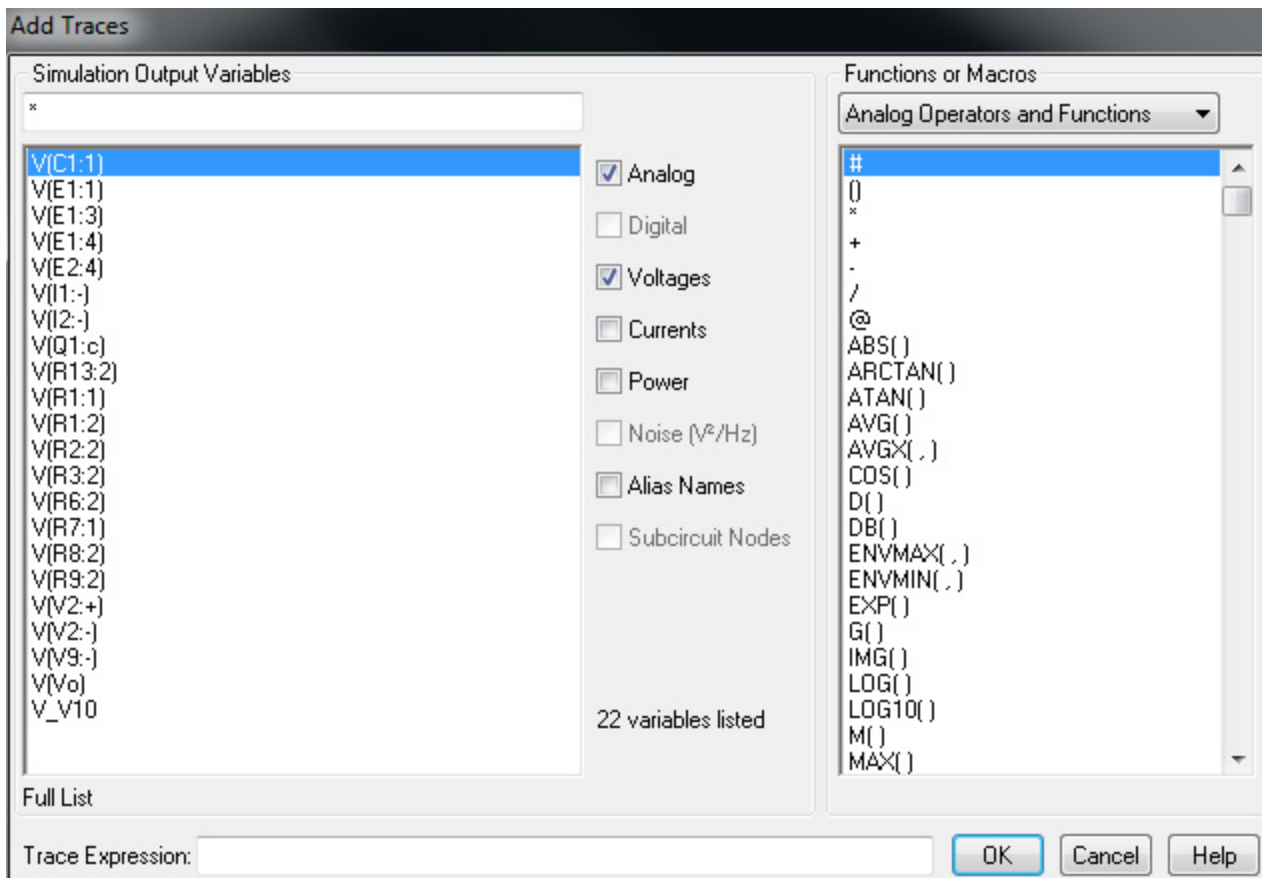
2) For part 1 you will now do a "DC sweep" of the differential voltage source (V10) from -10mV to 10mV. Enter the values shown in the figure using a "Linear" sweep type.



3) Now click the "Run Simulation" button. The simulation should complete without any errors and you should see a blank graph with the x-axis scale showing the voltage that was just swept (V_V10). Again, this may pop up underneath your current window.


4) Click the Add Trace button  from the toolbar. This will allow you to plot various parametrs as a function of V10.

5) The Add Traces window will open, as shown below. To shorten the list of output variables, uncheck "Currents" and "Alias Names" as shown.






6) Any simple mathematical expression can be typed in the "Trace Expression" box at the bottom of the window. To add an output variable, in this case $V(Vo)$, to the expression, just select it from the list or type it directly into the "Trace Expression" box. Then click "OK"

7) A plot of V_o vs. V_d (V_V10) should now be displayed.

8) Cursors will help you accurately measure points on the waveform. To turn on the cursors, press the "Toggle cursor" button  on the toolbar.

9) You have two cursors; the left mouse button controls "Cursor 1" and the right mouse button controls the "Cursor2". This is particularly useful for measuring gain slopes, etc. The X and Y values can be found in the table on the bottom-right of the window. You can use the "diff" values to easily determine a slope.

10) The zoom buttons will help you zoom in on a specific part of your waveform. The "Zoom Area" button  allows you to drag a box around the part of the waveform that you want to enlarge. The "Zoom Fit"  button allows you to zoom out to show the entire waveform in the window.

You can also place data marker by clicking the "Mark Label" button . You will probably want to do this to clearly indicate important data points.

You should now be able to complete Part 1.1 of the lab.

2. Transistor Parameters

1) You can obtain the transistor operating parameters (g_m , r_o , β , V_{CE} , etc.) by doing a Bias Point Analysis.

IMPORTANT!: Return to your schematic and set the differential voltage (V_{10}) equal to the "Input Offset Voltage" you observed in part 1.

2) Open your simulation settings to set "Analysis Type" to "Bias Point". Check the "Include detailed bias point" check box (the very first one) and simulate.

3) Once the simulation is complete, it should pop up a black screen in PSPICE. Click View > Output File. The transistor operating parameters are listed at the bottom of the output file. Note that there are two values listed for beta: BETADC and BETAAC. For comparing to theory, you want to use BETAAC.

4) If you return to your schematic you can plot the voltages and currents by clicking the "V" and "I" buttons on your PSPICE toolbar.

These numbers should be reasonably close to what you calculated in the prelab.

3. Common Mode Range

You should now have a general understanding of how the software works. For the next section you need to sweep the common mode voltage from 0-5V. Use the same procedure described above for "DC Sweep" but setup the simulation to sweep V5 instead of V10. Select an appropriate step size as well. If you use the same increment from before your simulation will take an excessively long time to compute.

LAB 3 Part 2

Transient

1) Return to OrCAD and close your current project (File > Close Project). Now open the project file for Part 2 of this lab "LAB3_PART2.OPJ"

2) If you examine the circuit you will see that a pulse generator has been added to the circuit. This setup will allow us to examine the Opamp's response to a step input.

3) **Change the value of capacitor C1 to the number you calculated in the prelab*.**

4) Setup the simulation to do a "Time Domain (Transient)" analysis.

5) Enter a "Run to time" of 75us. The period of the square wave generator is set to 50us, this simulation length will give you 1.5 periods to look at.

6) Perform the simulation and Add the input and output voltages to the plot window. Use your cursors to measure the negative and positive slew rates.

* For interest, you can try simulating the circuit leaving C1 at 1pF. This should give you a good idea as to what the purpose of this capacitor is.

LAB 3 Part 3

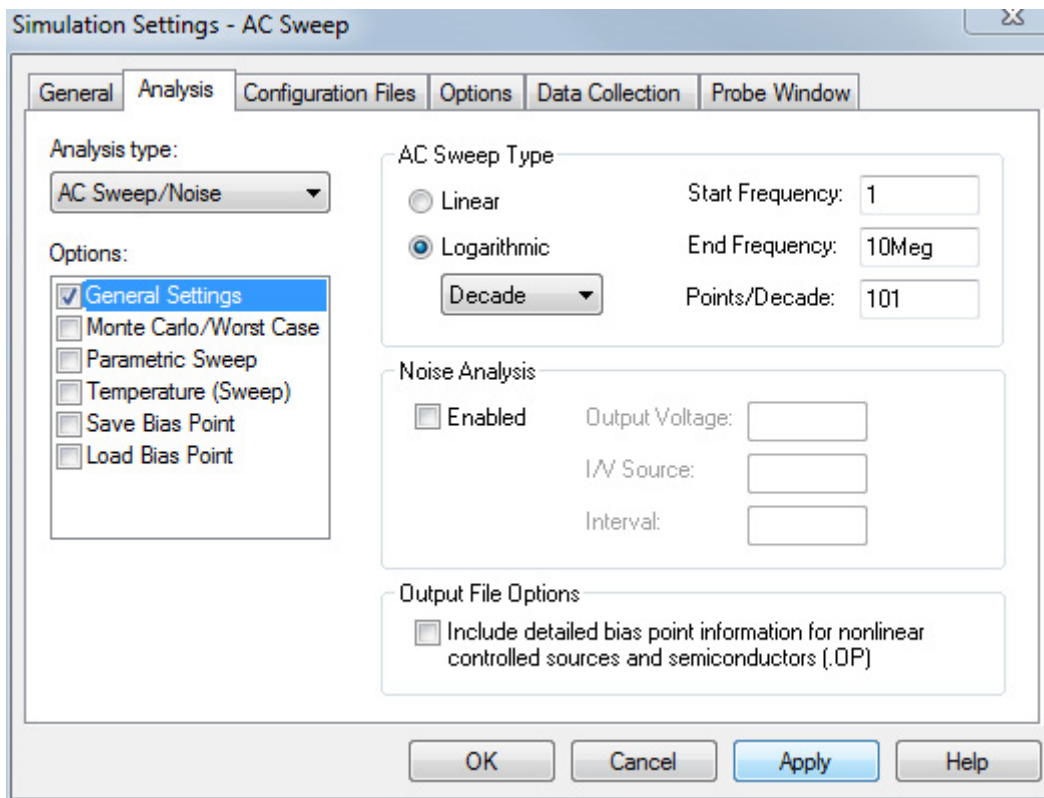
Frequency Response

1) Return to OrCAD and close your existing project. Part 3 of this lab will use the same project file as in Part 1. (LAB3_PART1AND3.OPJ)

2) An AC sweep will give you the small-signal frequency response of your circuit, with the circuit linearized around the DC operating point. Set the AC magnitude of differential signal source V9 (ACMAG) to unity (1V).

3) Ensure that you still have set V10, the differential input voltage, equal to the input offset voltage as in Part 1. **ALSO REMEMBER to change the value of capacitor C1 to the number you calculated in the prelab.**

4) Setup the analysis to do an "AC Sweep" from 1 to 10Meg. Select "Decade" for your scale and 101 Points/Decade.



5) Run your simulation and plot the output voltage in dB DB(V(Vo)). From here you should verify the unity gain frequency.

6) Add another plot to the window to simultaneously plot the phase of the output (Plot > Add Plot to Window)

7) To display the phase information, open the Add Traces window. The function for phase is "P()". So the expression for the phase of the output would be "P(V(Vo))". Enter this in the "Trace Expression" box and click "Ok". The phase and magnitude information should now be displayed in separate plots in the same window