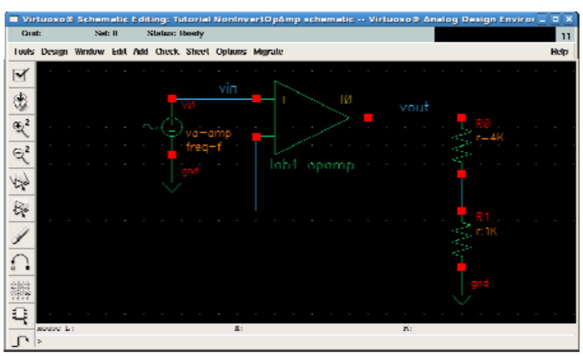
MichiganEngineering

University of Michigan
EECS 311: Electronic Circuits
Fall 2009
Cadence Tutorial

Non-Inverting Amplifier Example

Objective: Build a non-inverting amplifier using Cadence with a gain of 5 V/V.



$$Av = 1 + \frac{R_2}{R_1} \qquad R2 = 4k\Omega \quad R1 = 1k\Omega$$

MichiganEngineering

Setup Cadence Environment

You must do this before running Cadence, and you only need to do this once.

1. Login to a CAEN Linux machine in one of the CAEN labs. To connect remotely from a Windows computer or personal computer, login remotely to login.engin.umich.edu.
2. Authenticate using gettokens and your Kerberos password. Create a link from your home directory to your EECS 311 AFS space. Replace <uniqname> with your UMICH uniqname. If your directory does not exist, contact the course staff.

   >> gettokens
   >> cd ~
   >> ln -s /afs/umich.edu/class/eecs311/f09/students/<uniqname> eecs311

3. All your Cadence files should be saved on the afs file server. You can easily access this space from your home directory using the link you just created.

   >> cd ~/eecs311

4. Copy all of the the setup files to your afs space.

   >> cp /afs/umich.edu/class/eecs311/f09/cadence/setup_working_dir/* ~/eecs311
   >> cp /afs/umich.edu/class/eecs311/f09/cadence/setup_working_dir/.* ~/eecs311

5. For those who have used Cadence in a class before, check your home directory for the following 2 files: ~/.cdsinit and ~/.cdsenv. If they exist, rename them to ~/.cdsinit.tmp and ~/.cdsenv.tmp. Also remove any Cadence-related modifications you made to your ~/.cshrc file. The default .cshrc file can be found at /usr/caen/skel/std.cshrc.

Note: if you created these files for a class you are currently taking, you will have to undo these operations before launching Cadence for this other class.

MichiganEngineering

Launch Cadence and Create a Library

1. Login to a CAEN Linux machine in one of the CAEN labs. To connect remotely from a Windows computer or personal computer, login remotely to login.engin.umich.edu.

2. Launch Cadence from your afs directory. The Command Interface Window (CIW) will pop up.

   >> cd ~/eecs311
   >> gettokens
   >> icfb &

3. Next launch the library manager to create a new library by selecting Tools > Library Manager... from the CIW.

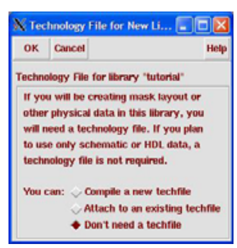
Launch Cadence and Create a Library

4. The **EECS311Lib** and **EECS311Examples** libraries should appear in the Library Manager. Select File > New > Library... from the Library Manager to create a new library.

Launch Cadence and Create a Library

5. Enter tutorial for the Library Name and click OK.

6. Select Don't need a techfile and click OK.



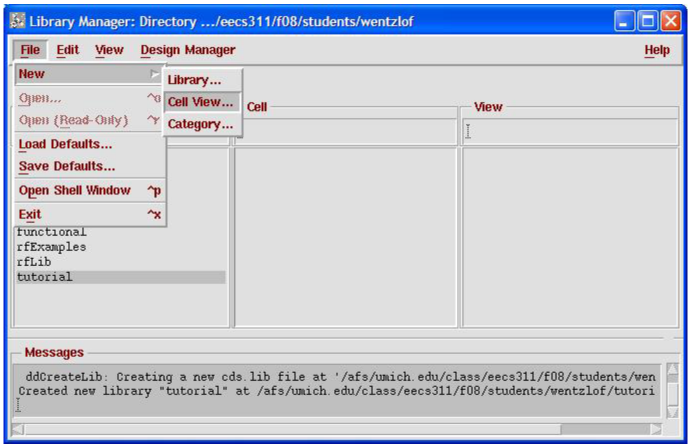
7. The library tutorial will now appear in the Library Manager

MichiganEngineering

MichiganEngineering

Create a New Schematic

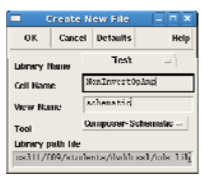
8. Click on the tutorial library in the Library Manager to select it, then go to File > New > Cell View...



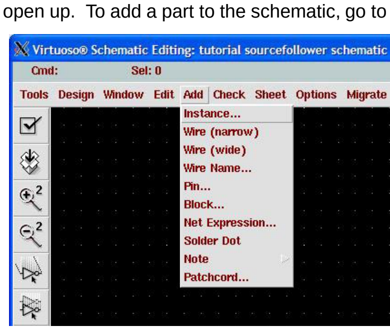
MichiganEngineering

Create a New Schematic

9. Enter NonInvertOpAmp for the Cell Name, and choose Composer-Schematic for the Tool. The View Name will automatically default to schematic. Click OK.



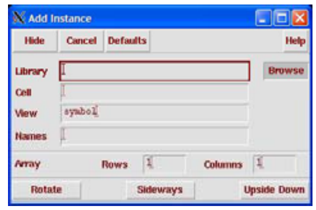
10. The new schematic will open up. To add a part to the schematic, go to Add > Instance... or by hitting the "i" key.



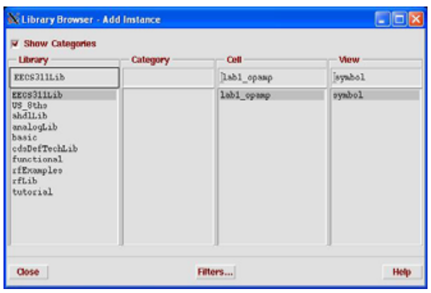
MichiganEngineering

Add Components

11. To find a part, click Browse from the Add Instance dialog box.



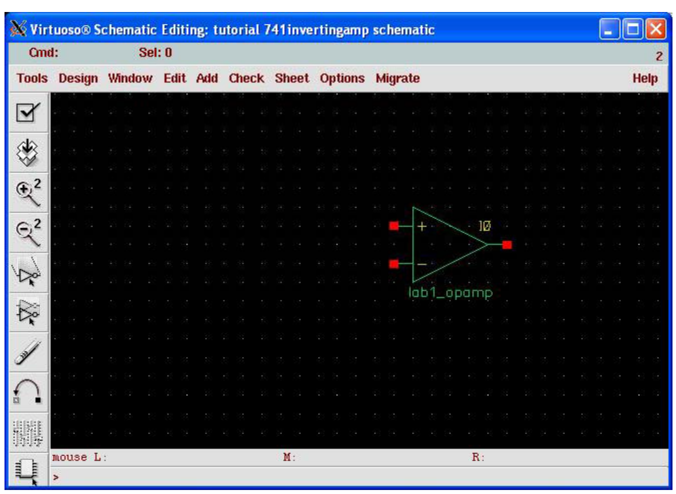
12. The Library Browser will pop up.
Select EECS311Lib, lab1_opamp, symbol. Click Close.



MichiganEngineering

Add Components

13. Place the opamp on the schematic. You may exit commands by pressing the Esc key.



MichiganEngineering

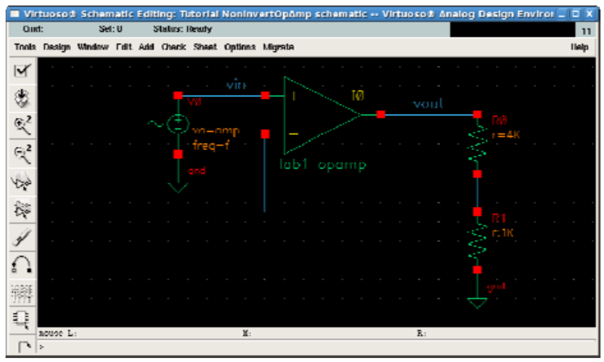
MichiganEngineering

Add Components

14. Using the Add > Instance... command, add a 1kOhm and 4kOhm resistor (analogLib > res), a sine wave input (analogLib > vsin), and ground symbols (analogLib > gnd). Place them as shown in the figure below. You may copy a component by going to Edit > Copy or by hitting the “c” key.

15. To edit the properties of a component, go to Edit > Properties or select the component and hit the “q” key to bring up the properties menu. To move components you may simply click and drag, hit the “m” key or hit Shift+”m” to break the components connections and move it.

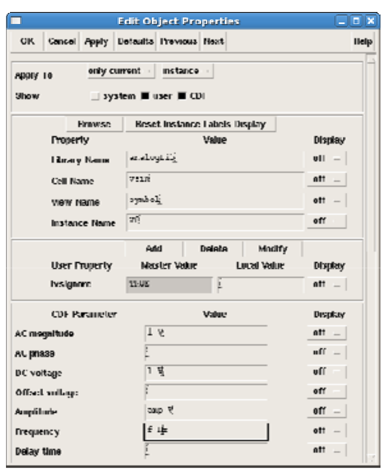
16. Use the **Add > Wire Name...** command to name the **vin** and **vout**, nets as shown below. You may auto-zoom the schematic by going to Window > Fit or pressing the “f” key.



MichiganEngineering

Edit Properties

17. Edit the properties of the sine wave input to have an AC magnitude of 1 V, a DC voltage of 1 V, an Amplitude with a variable value of “amp” and a variable Frequency of “f” as shown below. You need only enter/change these values. Units will be added on automatically.

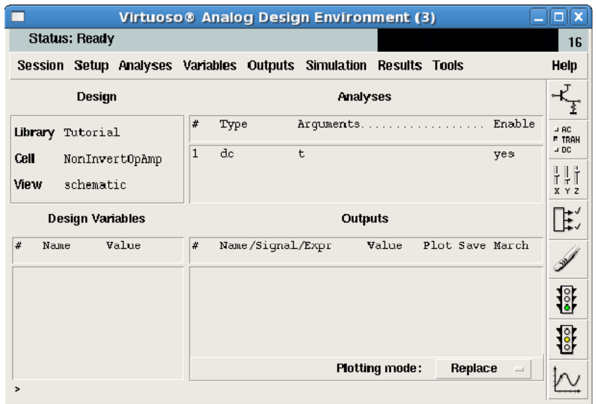


MichiganEngineering

MichiganEngineering

Analog Environment

18. Finally, click the **Check and Save** button in the upper left corner of the schematic window. Next open the Analog Environment by going to Tools > Analog Environment. The Analog Environment window will appear.

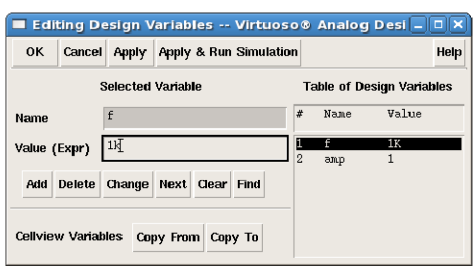


MichiganEngineering

Environment Variables

19. Load any variable names from the schematic by selecting **Variables > Copy From Cellview**.

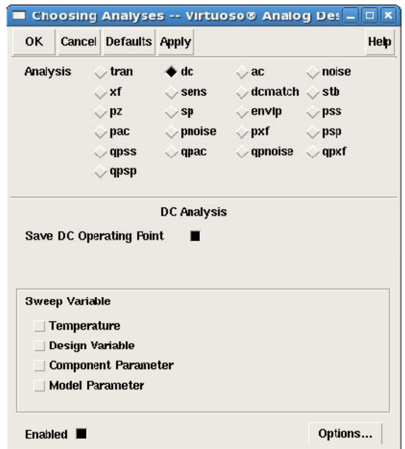
20. Select **Variables > Edit ...** to edit the variable values or double click on them in the environment window. Give the freq variable a value of 1k (Hz) and the amp variable a value of 1 (Vpk).



MichiganEngineering

Select Analyses

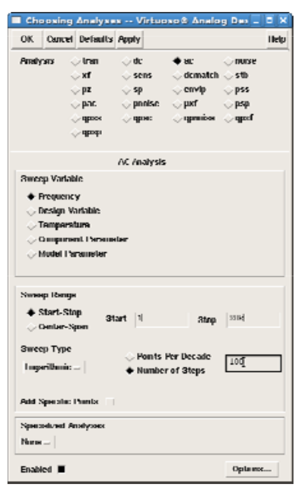
21. Go to Analyses > Choose... or click the Choose analysis button on the righthand side of the Analog Environment Window. Select dc as the Analysis type, select Save DC Operating Point, and ensure that enabled is selected at the bottom.



MichiganEngineering

Select Analyses

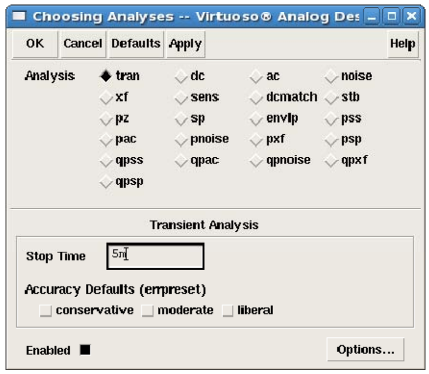
22. Select ac as the Analysis type. Select the sweep variable as Frequency and sweep over a range from 1Hz to 20kHz. Select the sweep type as logarithmic over 100 points or so. Ensure that enabled is selected at the bottom.



MichiganEngineering

Select Analyses

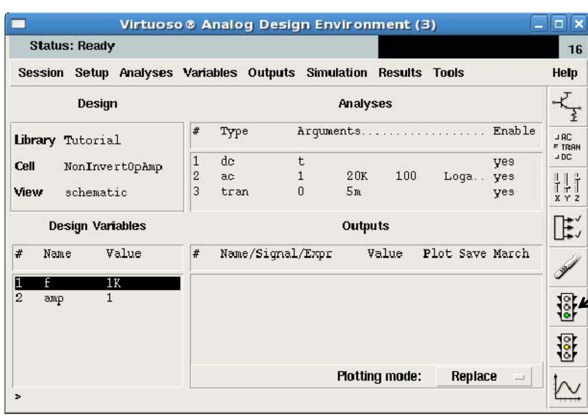
23. Select tran as the Analysis type. Set the stop time to be 5m (s) and ensure that enabled is selected at the bottom. Hit OK.



MichiganEngineering

Simulation

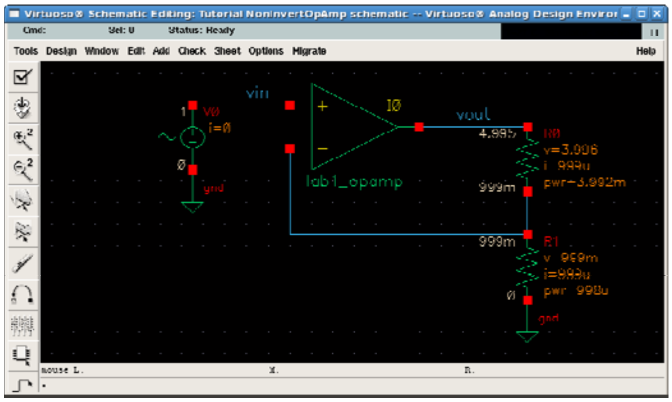
24. Select Simulation > Netlist and Run or the Netlist and Run icon on the righthand side of the Environment window to run a all 3 simulations.



Netlist and Run

Viewing Results

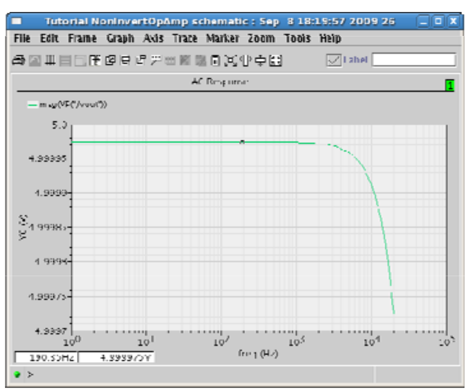
25. To see the DC voltage and current levels within the circuit, go to Results > Annotate > DC Node Voltages *and* Results > Annotate > DC Operating Points.



As expected, the op-amp shows roughly equal DC voltage at its terminals with no DC current going to either one. From the non-inverting architecture we also see that we obtain a gain of roughly Av = 5 V/V which is expected from the aforementioned equation.

Viewing Results

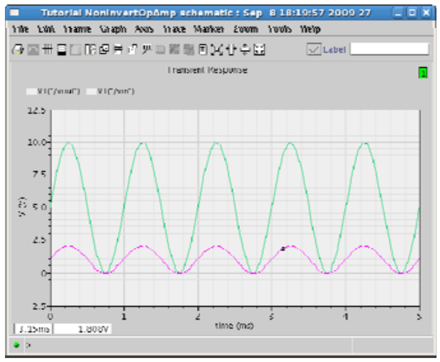
26. Select Results > Direct Plot > AC dB20 to view the small signal frequency response of the amplifier. Next select the “vout” wire in the schematic window and press “Esc”. The following window should appear.



Here again we see that the non-inverting amp is giving us a gain of about Av = 5. Since this is not an ideal op-amp we can see it starting to cut-off at higher frequencies. If you run a wider frequency simulation, you will see where the amplifier cuts off.

Viewing Results

27. Finally, select Results > Direct Plot > Transient Signal and select both “vout” and “vin” in the schematic window and press “Esc”. This allows us to view the input as a function of time. The following window should appear.



Once again we see that the output of the amplifier exhibits a 5 V/V gain.

Viewing Results

28. You may look at the input and output on separate graphs in the same window by selecting Axis > Strips as shown below.