Lab 7: Op-Amp Models

Background:

Op amps are typically implemented on integrated circuits and are complex circuits which we can model in a few different ways. These models are centered around the circuit shown in Figure 1.
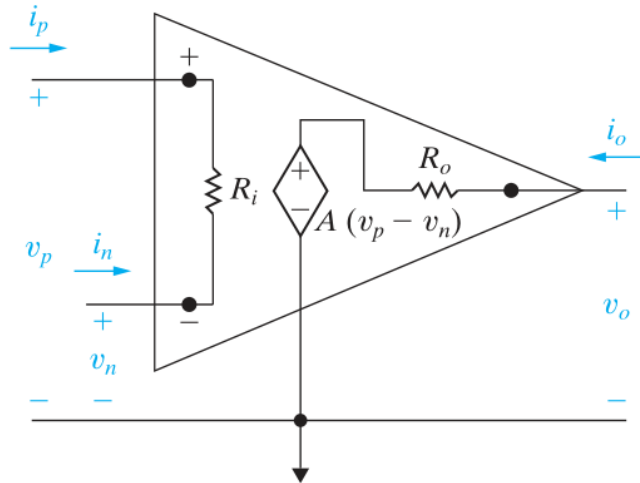

Figure 1: An Equivalent Circuit for an Operational Amplifier

The ideal op amp model is the circuit in Figure 1 with the following assumptions:

- $A = \infty$

- $R_i = \infty\,\Omega$

- $R_i = 0\,\Omega$

- The op amp's output voltage cannot exceed the boundaries of its input power (rail) voltages which not shown in Figure 1.

Objectives:

- Explore different op amp models

- Export LTspice data to Excel

- Use Excel to calculate simulated op amp properties.

- Use LTspice's **.param step X list** fucntion to vary a circuit parameter over a list of values.

Required Equipment:

- LTspice XVII

- MS Excel or equivalent spreadsheet program

1   Part A Procedure:

1. Design an inverting amplifier circuit and with a $5\,\text{k}\Omega$ resistor attached to the input voltage. Use the ideal op amp assumptions to solve for the feedback resistor (R_f) values to yield gains of -10, -20, and -25. This will, of course, result in 3 different values of R_f. Attach a load resistor R_L of $1\,\text{k}\Omega$ from the op amp's output pin to ground.

2. Set up a simulation for your circuit using using LTspice's **opamp** part. This part behaves like an ideal op amp. You can find it by clicking on Component button, go into the **[Opamps]** directory, and locating **opamp**. To use the part, you also add the command **.lib opamp.sub** by clicking the SPICE directive button (**.op**), enter the text **.lib opamp.sub**, and place the command on your schematic. Notice that this LTspice component does not have pins for rail voltages. LTspice allows this part's output voltage be any value (as in not constrained by any rail voltage values). In this sense, this is even more optimistic than what we consider to be the ideal op amp.

    (a) Use the reference designator (name) of **Rfa** for the feedback resistor, **R1a** for the resistor attached to the source, and **RLa** for the load resistor.

    (b) Give the net label of **Vin** for the input voltage, **Vna** for the op amp's inverting input, and **Voa** for the output.

    (c) Populate your component values and enter **{RF}** as the feedback resistor's value.

3. In the same LTspice schematic/file, duplicate the previous circuit. Delete the **opamp**, and replace it with **UniversalOpamp2**. This is component's behavior more closely imitates a real op amp's behavior. Give the same reference designators, net labels, and component values above except replace the suffix of "a" with a new suffix "b" where applicable. For example, use **R1b** instead of **R1a**. Use net labels to apply +/-25V to the op amp's rails. The same can be done for attaching your **Vin** source.

4. Repeat the previous step with the Figure 1 circuit in place of the op amp. See this video for instructions in implementing the dependent source, more specifically, a voltage-controlled voltage source, in LTspice: https://www.youtube.com/watch?v=E-O2E7iJAPo. Use a gain of **200k** for the dependent source. Replace the suffix of "b" with a new suffix "c" where applicable. Note that this op amp model does not have rail voltages. Use an R_o value of $75\,\Omega$ and an R_{in} value of $2\,\text{M}\Omega$.

5. Use a **Vin** of 1V for all 3 circuits. This way, output voltage will be the same as circuit gain.

6. Use the **.step param Rf list....** command to have LTspice simulate all 3 of your resistor values. This is the same command used in the previous lab, but with the word **list** added.

7. Run your simulation command to generate the **.raw** output file.

2   Part B Procedure:

1. Create a new Excel file.

2. Export all LTspice variables to a **.txt** file using these instructions: https://www.analog.com/en/technical-articles/ltspice-importing-exporting-pwl-data.html. Hold down the shift key while clicking to select all LTspice variables.

3. Open the resulting **.txt** file in a text editor, select all of its text (Ctrl+A) and paste it into the Excel file.

4. In the same Excel worksheet, organize all of the data into 3 tables, one for each of circuits A, B, and C. Organize your table to look like the one below.

Circuit A								
rf	V(vna)	V(voa)	I(R1a)	I(Rfa)	I(Rla)	Io	Opamp A	Ro

Circuit B										
rf	V(vnb)	V(vob)	I(R1b)	I(Rfb)	I(Rlb)	Ix(U1:V+)	Ix(U1:V-)	Ix(U1:In+)	Ix(U1:In-)	Io	Opamp A	Ro

Circuit C												
rf	V(vnc)	V(voc)	I(R1c)	I(Rfc)	I(Rlc)	V(vsc)	I(E1)	I(RLc)	I(Ri)	I(Ro)	Io	Opamp A	Ro

Figure 2: Template for Excel Table

Take note of the following things:

   (a) The blacked out cells will remain blank.

   (b) You can merge cells with Ctrl+1 –> Alignment tab –> check Merge cells.

   (c) The reference designators for the op amps were not specified, so you may have U1 and U2 transposed from this example.

   (d) The imported LTspice data that is absent from the table should be deleted.

   (e) The last three columns **Io**, **Opamp A**, and **Ro** will not come directly from LTspice. They will be calculations based on LTspice data.

   (f) The second-to-last column means the gain of the op amp. It does not refer necessarily to circuit A.

5. Use KCL to implement a formula to calculate **Io** by referencing the data from the columns to the left. **Io** is defined to be the current flowing *into* the op amp's output terminal. Be sure you are aware of the direction of current flow for you resistors. If you are not sure which way LTspice

has decided is the direction of positive current flow through a resistor, run your simulation, then hover your mouse over the resistor. You will see an arrow appear that tells you the direction of positive current flow.

6. Look at Figure 1 and write an equation that relates v_n, v_p, R_o, A, i_o, and v_n to each other. Keep in mind that v_o in pointing into the op amp. Set v_p equal to zero since our circuits all have this as true.

    (a) We will consider A and Ro to be our unknowns in this equation, and the rest of the variables (v_n, i_o, and v_o) will come from the LTspice data. If you use the data in the *first two rows*, we have two equations and two unknowns.

    (b) Use substitution along with data from Circuit A's first two rows to come up with a formula for A based on the LTspice outputted data. Enter this formula under **Opamp A**. It should reference **V(voa)** for v_o, **Io** for i_o, **V(vna)** for v_n. This should result in the number LTspice uses for its **opamp**, which will be a large number.

    (c) Use this result and the rest of the LTspice data to solve for **Ro**. Put this in a formula under the **Ro** header.

7. Copy and paste your formulas for **A** and **Ro** into the tables for Circuits B and C. Of course, we already knew these values for Circuit C ($75\,\Omega$), so you should see these values match what is on your schematic. This match is a good indicator your formulas are correct.

8. Note that many of the LTspice data columns were not used in this lab. They were left in for the possibility of doing more with this assignment in future terms.

3   Deliverable(s):

1. Submit your Excel file to this lab's D2L folder.