**ENSC 225 MICROELECTRONICS 1 Summer 2021**

## Experiment 1

# Ideal Op Amps

Due: June 7, 2021, 21:00

• Lab reports (pdf version) should be uploaded to the ENSC 225 on CANVAS.

### 1.0. Behavioral model for Op Amps

Circuit simulator software tool simulates operation of circuits built of electronic components, solving voltage and current equation in the electronic circuit. To analyze and design the circuits built of semiconductor devices simulators use electronic device models to calculate terminal currents and voltages of semiconductor devices. Circuit simulators such as Spice also permit simulation of electronic circuits using behavioral models.

Behavioral models are used to model the terminal characteristics of various circuit building blocks using all kinds of voltage and current sources and passive electronic components (such as resistors, capacitors, inductors). The behavioral models are black-box type models (macromodels) that do not provide any information about internal architecture of simulated circuits.

A simplest behavioral model of an ideal operational amplifier (high-gain voltage amplifier) can be built using a voltage-controlled voltage source (Fig.1, 2).

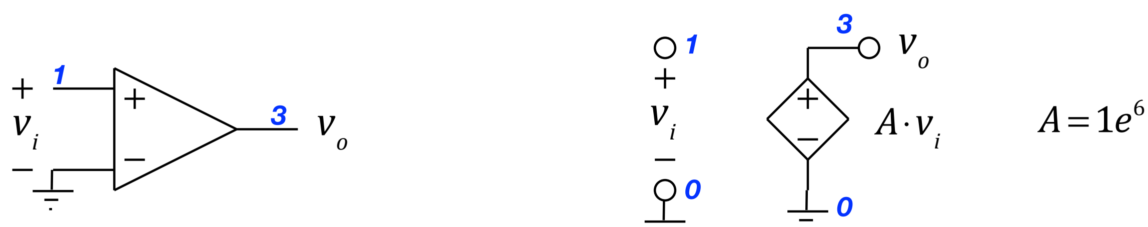


Fig.1. The ideal opamp symbol and its behavioral model

```
Eopamp 3 0 1 0 1e6
```

Fig.2. Spice input file (deck) for the ideal opamp from Fig.1, with voltage gain $A = 1 \cdot 10^6$ V/V.

The non-ideal opamp will have many extra elements to simulate all opamp parameters, such as finite input and output resistances, frequency characteristics with number of poles and zeros, limitations for input and output voltages, Common-Mode Rejection Ratio (CMRR), Slew Rate (SR), etc. Examples of macromodels of many opamps manufactured by Analog Devices you can find it the Analog Devices model library [https://www.analog.com/en/design-center/simulation-models/spice-models.html](https://www.analog.com/en/design-center/simulation-models/spice-models.html), and in other sources on Web. The entire library of opamp macromodels is built-in in the LTspice.

## 2. 0. Objective

In this experiment, we will examine properties and characteristics of operational amplifier circuits.

Your Lab Report must be no longer than 5 - 6 pages and should contain for each simulation experiment:

- Calculations requested for the targeted closed-loop gain
- Circuit schematic with labeled components and numbered nodes
- Input file (deck) for simulation – in case if you are doing simulation using the input file (deck); if you are using schematic entry in LTspice, input file is not needed
- Plots of simulated waveforms on white background (<u>Sec. 3.6 has important information</u>)
- Discussion of results

## 3.0 Procedure

### 3.1. LM107 opamp macromodel

In simulating the circuits in Sections 3.2 to 3.5 you will be using as an ideal opamp the macromodel of the OP37 opamp, available from the Analog Devices model library
https://www.analog.com/en/design-center/simulation-models/spice-models.html

- If you prefer to use LTspice graphical interface for schematic drawing and schematic capture (schematic entry), the OP37 opamp macromodel is in the LTspice model library. It is necessary that after drawing the circuit for simulation, you will produce the input file by right-clicking in a blank area within the schematic window and select **View -> SPICE Netlist** from the contextual menu (a new window will appear with the text of input deck). This text input file (deck) must be included in your Lab report.
- If you prefer to edit the input deck to LTspice using a text editor and proceed with the simulation as described in the LTspice Simulation Guide:
https://www.sfu.ca/~syrzycki/225/Docs/LTSpice_Simulation_Guide.pdf
you should download the OP37 macromodel from the Analog Devices model library:
https://www.analog.com/en/design-center/simulation-models/spice-models.html
and attach it to your simulation deck, for example using the **.INCLUDE** Spice command.

### 3.2 Noninverting operational amplifier

The circuit shown in Fig.3 will be used to simulate the Voltage Transfer Characteristic (VTC) of the noninverting operational amplifier. As the ideal op amp use the macro model of OP37 op amp with the power supplies of $V_{CC}$ = +15 V and $V_{EE}$ = -15 V.

a) Choose the values of $R_1$ and $R_2$ resistors to implement noninverting amplifiers with the closed-loop voltage gains of 1 V/V, 2 V/V, and 11 V/V. Varying the input voltage, $v_i$, plot the values of output vs. input voltages for all three amplifiers, using one system of coordinates. Make sure that the output voltage does not exceed the -10 V to +10 V range.

b) Provide explanation justifying the obtained results.

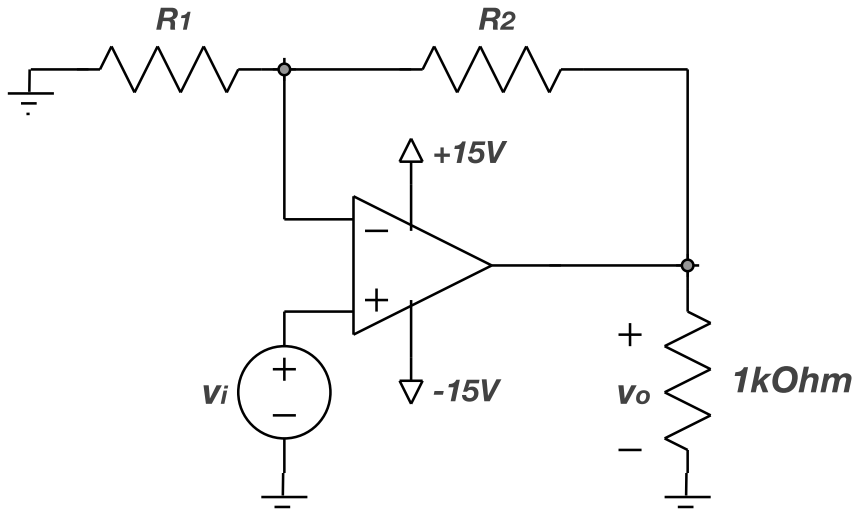


Figure 3. Noninverting operational amplifier.

### 3.3 Inverting operational amplifier

The circuit shown in Fig.4 will be used to simulate the operation of the inverting operational amplifier. As the ideal op amp use the macro model of OP37 op amp with the power supplies of $V_{CC}$ = +15 V and $V_{EE}$ = -15 V.

a) Choose the values of $R_1$ and $R_2$ resistors to implement the inverting operational amplifier shown in Fig.4 with the closed-loop voltage gain of $A_V$ = -5 V/V.
b) As the input signal use the sinusoidal signal of the frequency1 kHz, and the amplitude of 1 V.

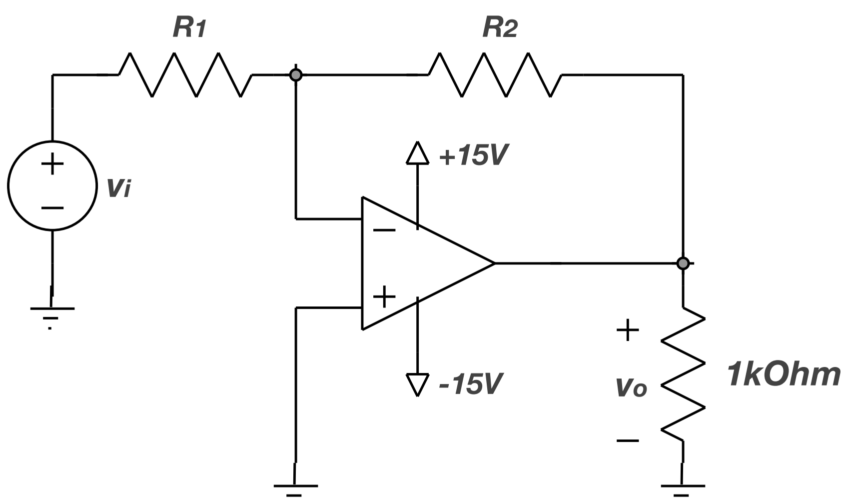


Figure 4. Inverting operational amplifier.

c) Plot 2-3 periods of input and output voltage waveforms.
d) Repeat the steps (b) and (c) for a triangular $v_i$ waveform of the frequency1 kHz, and the amplitude of 1 V.

e) Provide explanation justifying the obtained results.

### 3.4 Operational amplifier as a voltage summer

Using the ideal op amp macro model of OP37 op amp with the power supplies of $V_{CC} = +15$ V and $V_{EE} = -15$ V, construct the voltage summer that produces the output voltage $V_{out}$:

$$V_{out} = K_1 \cdot V_{in1} + K_2 \cdot V_{in2} - K_3 \cdot V_{in3}$$

Where $K_1 = 2.0$, $K_2 = 3.0$, and $K_3 = 0.5$.

a) Draw the schematic of this weighted summer.
b) Choose the values of resistors to implement the targeted $V_{out}$ function.
c) Apply a sinusoidal signal of the amplitude 1 V and frequency of 1 kHz to all three inputs. Plot the input and output waveforms of the weighted summer.
d) Provide explanation justifying the obtained results.

### 3.5 Operational amplifier as integrator

a) Construct the integrator featuring the dc gain of -10 V/V, as shown in Fig. 5.

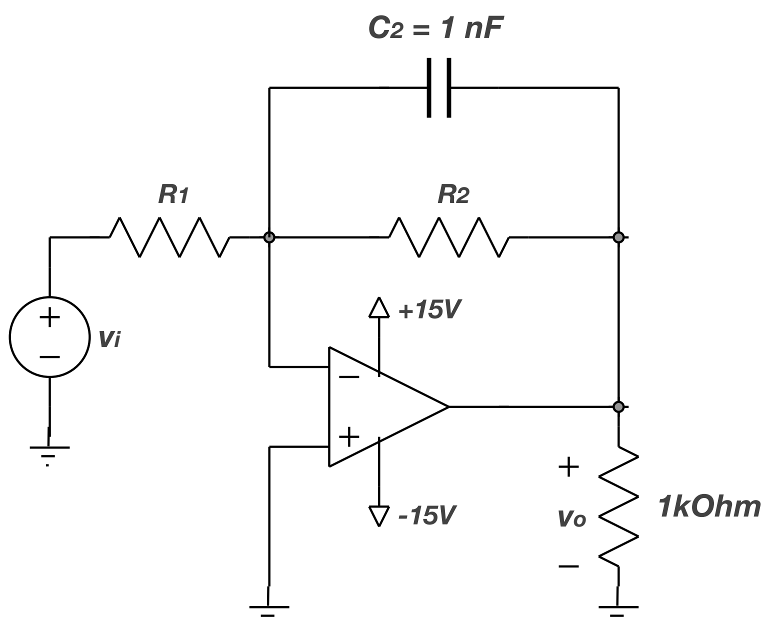


Figure 5. Opamp integrator.

b) Set the function generator to produce a sinusoidal $v_i$ voltage of the frequency $f = 10$kHz and the peak-to-peak output voltage $v_o$ of 10V. Sketch the $v_i$ and $v_o$ voltages vs. time (two to three time periods), and provide explanation justifying the obtained $v_o$ waveform as a function of $v_i$.
c) Repeat the step (b) using the square $V_{IN}$ waveform.

d) Repeat the step (b) using the triangular $V_{IN}$ waveform.

### 3. 6 Plots of simulation results for Lab reports

Plots of your simulation results must be clear, easily readable, and on a **white background**.

• If you decide to use plots as produced by LTspice, you must convert them to white background and thicken the waveforms to make plots readable. In order to change the background, click on **Tools** -> **Color Preferences** and set **Background** to **255 255 255**, and **Axis** to **0 0 0**. There is a U-tube video how to change the background color to white and how to thicken the waveforms in the plot: https://www.youtube.com/watch?v=P4uPejY9Cpk

• You may also import the output text files as columns of numbers and plot it in any graphical package (like Excel or Kaleidagraph). In Excel, make sure to use engineering notation and not its default setups that haven't been intended for engineering applications.

• A failure to produce readable plots on a white background will result in grade penalty.