**Experiment 1: Inverting Amplifier**

*PSpice Simulation 1:* Set up an inverting amplifier circuit as shown with $R_1 = 1 \, \text{k}\Omega$ and $R_2 = 10 \, \text{k}\Omega$. Use the device library in PSpice for uA741. Set the input signal amplitude to 0.1 V.

a) Use PSpice to generate the frequency response of this amplifier for the frequency range of 10 Hz to 1 MHz. Attach the Bode plots (both amplitude and phase). For plotting the data, choose decade or octave scaling with sufficient number of points, *e.g.*, decade with 21 points/decade. What is the low-frequency gain of this amplifier. Does it match analytical formulas?

b) The cutoff frequency for this circuit is defined similar to that of filter circuits, *i.e.*, difference between the maximum gain and gain at cut-off frequency is 3 dB. Find the cut-off frequency from your simulations. What is bandwidth of this amplifier? What is the phase shift in the output at the cut-off frequency?

c) Plot the input resistance ($= V_i/I_i$) of this circuit as a function of frequency. Does it match analytical formulas?

*PSpice Simulation 2:* Set up an inverting amplifier circuit with $R_1 = 1 \, \text{k}\Omega$. Set the input signal amplitude to 0.1 V. Use family of curve options of PSpice to obtain Bode plots for this amplifier for $R_2 = 1, 5, 10,$ and $100 \, \text{k}\Omega$ in the frequency range of 10 Hz to 1 MHz (this option will plot all three plots in one page for comparison). Mark the cut-off frequency for each case. Note that as the gain is reduced (by using smaller $R_2$), the bandwidth is increased. Does it match $Af = constant$ formula?
**PSpice Simulation 3:** Use PSpice to simulate an inverting amplifier with $R_1 = 1$ kΩ and $R_2 = 10$ kΩ. Set the input to be a square wave with a frequency of 10 kHz and amplitude of 0.5 V (You need to use VPULSE function, see Class Web site). Run a transient analysis for about 5 periods. Make sure that $V_o$ has reached its “steady-state” waveform. If not, run longer transient analysis and plot only the last 5 periods. Plot both $V_i$ and $V_o$. Examine $V_i$ to ensure that it is correct. Note that the output signal is not a square wave due to the slew rate of the OpAmp. Use your simulation to calculate the slew rate of the OpAmp.

**Lab Exercise:** Set up the circuit with $R_1 = 1$ kΩ and $R_2 = 10$ kΩ. The pin arrangement of the IC is available in the Lab. Note that you have to supply power to the OpAmp (+15 V to $V^+$ pin and -15 V to $V^-$ pin). The ground pin of the IC should also be connected to your circuit ground (or common).

a) Set the function generator to produce a sinusoidal wave with an amplitude of 0.5 V and connect it to the input. Attach Scope channel A to the input and Scope channel B to the output. Vary the frequency and at various points, measure the output voltage and the phase shift between input and output. Generate the Bode plots for this amplifier circuit and compare with your PSpice simulations. As you increase the frequency look at the shape of the output signal. Stop your measurements if the shape departs significantly from a sin wave and becomes triangular (why does this happen?). In this case, reduce the amplitude of input signal until the output signal is again sinusoidal and continue your measurements.

b) Set the frequency of the input signal to be 1 kHz. Measure the input resistance of the amplifier circuit by using the technique discussed in the class (see lecture notes, page 29).

c) Set the frequency of the input signal to be 1 kHz. Measure (or try to measure) the output resistance of the amplifier circuit by using the technique discussed in the class. (see lecture notes, page 30).

d) Set the frequency of the input signal to be 1 kHz. Start with an input amplitude of 0.5 V and slowly increase the amplitude of input. Watch the output signal (which is growing proportionally). Observe that at some point, the top and bottom of the output signal starts to flatten out. This is because OpAmp is leaving its linear region and becoming saturated. Measure the saturation voltage and compare it to the bias voltage of the OpAmp ($\pm 15$ V). Note this voltage, we will use it in experiment 2.

e) Set the frequency of the input signal to be 1 kHz. Attach a 200 Ω resistor to the output. Start with an input amplitude of 0.5 V and slowly increase the amplitude of input. Watch the output signal (which is growing proportionally). Observe that at some point, the top and bottom of the output signal starts to flatten out. Note that the output voltage below the amplifier saturation voltage. This is because of OpAmp’s maximum current limit. From this experiment deduce the maximum output current limit of the inverting amplifier configuration as well the OpAmp chip itself. Remove $R_L$ from the circuit after you are finished with this part.
f) Set the function generator to produce a square wave with an amplitude of 0.5 V and a frequency of 1 kHz and connect it to the input. Sketch the input and output signals. If you invert the output, it should exactly match the input. Explain why? Increase the function generator frequency and look at the output signal. Note that the output signal starts to depart from a square wave. Sketch the output signal at 10 kHz. Use the output signal to calculate the slew rate of your OpAmp. Note that your measured slew rate may be different from your PSpice simulation. Explain the reason.

Experiment 2: Summing Amplifier

*Design and Circuit Analysis:* Suppose that we have designed and built a function generator which generate sinusoidal, triangular, and square waves for a range of frequencies. We now like to add one last capability to this function generator, ability to have a variable DC offset similar to function generators we use in the Lab. The summing amplifier discussed in the class can be used for this purpose (See Figure). Assume that $R_F = 20 \, \text{k}\Omega$, the maximum amplitude of the input signal is 5 V, and the OpAmp is powered by ±15 V supplies (and, therefore, its output voltage cannot be higher than ±15 V). Explain how you would choose $R_1$, $R_2$, $V_1$, and $V_2$.

*Lab Exercise:* Build the circuit you have designed in the Lab. Attach the function generator to the input (set the frequency to 1 kHz). Does it behave as you have designed it? Explore its limitations (by changing the function generator frequency and by changing the potentiometer): What is the maximum DC offset you can add to the sinusoidal wave with a 5 V amplitude? Record and explain your observations.
Experiment 3: Current Source (Do this experiment for an extra credit of 30 points)

Negative feedback can be provided by any elements in addition to a resistor. The circuit below, a current source, has the BE junction of a transistor in its feedback loop. This circuit acts as a current source. If you change the load resistance (potentiometer), the load current \( i_C = i_L \) remains constant, i.e., the current in the load is independent of load resistance (or the voltage across the load).

Circuit Analysis: Assume that the transistor is in its active-linear region \( i_L = i_C \approx i_E \) and there is 0.7 V voltage drop across the BE junction (note \( V_{BE} = -0.7 \) V because this is pnp transistor). Explain why \( i_L \) is independent of the value of the load (potentiometer).

PSpice Simulation: Simulate this circuit with PSpice using uA741 OpAmp and circuit model for 2N3906 transistor (this circuit works with any general-purpose pnp transistor). Generate plots of \( i_L \) as a function of \( R_L \) for \( R_L \) ranging from 0 to 5 k\( \Omega \). Attach the plot to your report. For what range of \( R_L \) the circuit behaves as a current source. At what value of \( R_L \), the circuit does not behave like a current source anymore? Explain the cause of this behavior. (Hint: Look at \( V_{CE} \) of the transistor).

Lab Exercise: Build the circuit. Vary the potentiometer resistance and measure its resistance and the current in the load. (Question: How would you measure the resistance of the potentiometer while it is still in the circuit?) For what range of \( R_L \) the circuit behaves as a current source. What is the value of \( i_L \)? As you increase \( R_L \), find when the circuit does not behave like a current source anymore. Compare your experimental results with PSpice simulation and analytical circuit analysis.