Set up an active filter as shown. Set the input as a sinusoidal wave with an amplitude of 1 V. Set $R_1 = 1 \, k\Omega$, $R_2 = 2 \, k\Omega$, and $C_2 = 10 \, nF$. Use ±15 V to power the OpAmp.

**Circuit Analysis:** Calculate (analytically) the gain and cut-off frequency of this active filter.

**PSpice Simulation:** Simulate the circuit with PSpice. Generate the frequency response of this amplifier for the frequency range of 10 Hz to 100 kHz. Attach the Bode plots (both amplitude and phase). Find the cut-off frequency and the gain of the filter. Do they match the analytical results?

**Lab Exercise:** a) Set the function generator to produce a sinusoidal wave with an amplitude of 1 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 (It is sufficient to scan the frequency range of 0.1$f_c$ to 10$f_c$). Generate the Bode plots (both amplitude and phase) 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 from a sin wave (due to the slew rate). Report the frequency that this happens, lower the input amplitude until you have a sinusoidal output and continue with your measurements.

b) Determine the gain and cut-off frequency of this filter from your experimental data. Do they match your simulations/calculation results? You may want to measure the value of $R_1$, $R_2$, and $C_2$ and repeat your analytical calculations of gain and cut-off frequency and compare with the experimental results.
c) Compare the properties of this filter with first-order RL or RC low-pass filters. Explain when one should use this filter as opposed to a RL or RC filter.

**Experiment 2: Integrator**

Set up a circuit similar to Experiment 1 with $R_1 = 10 \, \text{k}\Omega$, $C_2 = 0.01 \, \mu\text{F}$, and $R_2 = 100 \, \text{k}\Omega$.

*Circuit Analysis:* Write down formula for output voltage as a function of input voltage (ignore $R_2$). Compute the output voltage if input is a square wave signal with a period $T$ and amplitude $V_i$ (no DC offset).

*PSpice Simulation 1:* Use PSpice to simulate the response of this circuit (transient analysis for 10 ms) to a square wave input with a frequency of 1 kHz and amplitude of 1 V (with no DC offset). You need to use VPULSE voltage source for this analysis. Check your simulation results and show that the circuit indeed integrates the input signal and follows the analytical calculations. Note that it takes a few periods before the transients die away and you get steady-state signal.

*PSpice Simulation 2:* The ideal integrator discussed in the class has no $R_2$ in the circuit. Use PSpice to simulate the above circuit with $R_2$ removed. Explains what happens to the output signal. Deduce from these two simulations the function of $R_2$.

*Lab Exercise:*

a) Set up the circuit without $R_2$. Apply a square wave with an amplitude of 1 V (no DC offset) and frequency of 1 kHz. Look at the output waveform. Similar to PSpice simulations, the output is at one of OpAmp saturation voltages. Expand the setting of the scope channel that shows the output to 5 V/division. Next touch the two terminals of the capacitor by your fingers. Note that as long as your fingers are touching the capacitor terminals, the output does a nice integration but as soon as you remove your finger, output drifts rapidly to around ±15 V. Why does this happen?

b) Now add the $R_2$ resistor. Sketch your input and output traces and show that the integrator indeed integrates the input. Compare with your simulations.

c) The price we pay for adding $R_2$ is to make this circuit into a low-pass active filter (similar to Experiment 1). The integrator works only if the frequency of the input signal is much larger than the cut-off frequency of this active filter. Calculate the cut-off frequency. Now drop the input frequency and look at the output trace. At what frequency the integrator does not work very well. Compare that with the cut-off frequency of the filter. For what range of frequencies this integrator works well?

**Experiment 3: 2nd Order Wide-Band Band-Pass Active Filter:**

*Design and Analysis:*

Design a band-pass active filter with a gain of 10, lower and upper cut-off frequencies of
15 kHz and 150 kHz, respectively, a small output impedance, and an input resistance larger than 1 kΩ.

Simulate the circuit you have designed with PSpice. Generate the frequency response of this active filter. Attach the Bode plots (both amplitude and phase). Find the gain and cut-off frequencies. Do they match the design values?

**Lab Exercise:**

a) Set the function generator to produce a sinusoidal wave with an amplitude of 0.1 V and connect it to the input. Generate the Bode plot (only amplitude) 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 from a sine wave (due to the slew rate). Lower the input amplitude until you have a sine output and continue with your measurements. You may also see at very low frequencies that the shape of the signal is different than a sine wave. Do not include these points in your measurements table. (See experiment 4f for explanation of the problem.)

b) Determine the gain and cut-off frequencies of this filter from your experimental data. Do they match your design values? You may want to measure the value of $R_1$, $C_1$, $R_2$, and $C_2$ and repeat your analytical calculations of gain and cut-off frequency and compare with the experimental results.

c) Compare the properties of this filter with the 2nd order filter you designed in Lab 2.

**Experiment 4: Differentiator**

Set up a circuit similar to circuit of Experiment 3 with $R_1 = 1 \text{ kΩ}$, $C_1 = 0.01 \mu\text{F}$, $R_2 = 10 \text{ kΩ}$, and $C_2 = 100 \text{ pF}$.

*Circuit Analysis:* Write down formula for output voltage as a function of input voltage (ignore $R_1$ and $C_2$). Compute the output voltage if input is a triangular wave signal with a period $T$ and amplitude $V_i$ (no DC offset).
**PSpice Simulation:** Use PSpice to simulate the response of this circuit (transient analysis for 10 ms) to a triangular wave input with a frequency of 1 kHz and amplitude of 1 V (with no DC offset). You need to use VPULSE voltage source for this analysis. Check your simulation results and show that the circuit indeed differentiate the input signal. Note that it takes a few periods before the transients die away and you get steady-state signal. (alternatively, you can instruct PSpice to ignore the initial transients)

**Lab Exercise:**

a) Set up the circuit. Apply a triangular wave with an amplitude of 0.5 V (no DC offset) and frequency of 1 kHz. Look at the output waveform. Sketch your input and output traces and show that the differentiator indeed differentiates the input.

b) The ideal differentiator does not have $R_1$ and $C_2$. These elements have been added to stabilize the circuit and get rid of the noise. To see the impact of the $R_1$, remove that from your circuit. Sketch the output signal. Note the oscillations in the output due to the instability of the circuit. You can increase the frequency of the input and see that the oscillations grow (or take longer to die away). From this experiment and by comparing the differentiator circuit (without $C_2$) with the active filters discussed in the class, explain why addition of $R_1$ stabilizes the circuit.

c) Set the input frequency back to 1 kHz and now with $R_1$ in place, remove $C_2$. You should not see any impact. Start increasing the frequency. At some frequency, you MAY see that the differentiator will not work well. If this happens, sketch the output. Explain what $C_2$ does (you can add $C_2$ to the circuit to see its effect). If you did not see any effect from addition of $C_2$, explain why.

d) Learning the necessity of $R_1$ and nice addition of $C_2$, we note that the resulting differentiator is turned into a band-pass filter (similar to experiment 3). Starting at 100 Hz, slowly increase the frequency of the input signal and observe the output. For what range of frequencies this differentiator works properly? Compare this frequency range with the upper and lower cut-off frequencies of the band-pass filter.

e) Setting the frequency back to 1 kHz, apply a sin wave with an amplitude of 0.5 V to the differentiator. What do you expect the output to be? Sketch your output trace and compare with your expectations. Function generators do not generate a sin wave as it is difficult to make a stable, constant amplitude sin wave over a wide range of frequencies. Rather the function generator uses a circuit with several diodes to round the corners of a triangular wave and make it look like a sin wave! (See Hayes & Horowitz, p 203). The Lesson: One of the best ways to ensure that a signal is a sin wave is to differentiate it!