## Lab Unit 2: Signal Amplification Using Op-Amps

Many common transducers of analytical signals produce signals that are very small, either as voltages or as currents. For example, a typical photomultiplier tube in a spectrometer will produce a current on the order of picoamps to nanoamps. One of the most important steps in many analytical measurements is to amplify the signal. The difficulty in accomplishing this is that one needs to amplify the signals without also amplifying the noise.

Operational amplifiers, or "op-amps", are the most practical way of amplifying signals for the vast majority of all analytical measurements. In this lab you will look at some of the basic op-amp circuits and investigate their performance. You will also look at some properties that can limit their performance, such as frequency response (or speed) and noise.

I. Inverting Amplifier
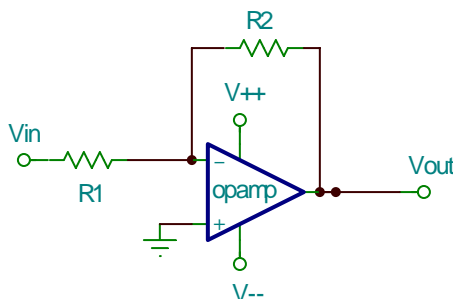
The inverting amplifier is one of the most common amplifier circuits and is shown above. For an ideal op-amp with infinity gain, we can usually make two approximations that greatly simplify the design of op amp circuits. The first approximation is that the op amp does whatever it can do to make the voltages at the inverting (v_-) and non-inverting (v_+) inputs the same. That is, the op amp tries to make $v_+ = v_-$. The second approximation is that the inputs of the op amp draw no current.

Using Kirchoff's current law and the assumption that no current flows into the op amp gives:

$$\frac{v_- - v_{in}}{R1} = \frac{v_{out} - v_-}{R2}.$$

Recognizing that $v_- = v_+ = 0$ (from the first assumption above) gives the gain equation for an inverting amplifier:

$$v_{out} = -v_{in} \frac{R2}{R1}.$$

In the following exercises, you will look at both the frequency response and the noise of the inverting amplifier.

A. Bandwidth and Gain

1. Measure the Bandwidth

Construct the inverting amplifier circuit (shown on the previous page) using an OP27 operational amplifier. Use R1 = 100 ohms and R2 = 100,000 ohms (100 kilohms, or more commonly, "100 k"). Set the function generator to produce a small sine wave of approximately 10 millivolts peak-to-peak at a frequency of 100 Hz.

Look at both the input and output signals on the oscilloscope. What are the peak-to-peak input and output voltages? Is there a phase shift?

Keeping the input voltage fixed, look at the signal output as a function of frequency of the input signal. You should see the output signal start to decrease in amplitude at high frequencies. Try to identify the "3 dB point", where the output voltage decreases by $1/\sqrt{2}$ and the phase shift changes by another 45 degrees. Record the amplitude and phase at enough different frequencies that you will later be able to make a plot of gain vs. frequency and phase vs. frequency. Keep in mind that you do not need many data points in regions where no change is occurring. More data points are needed in regions of rapid change.

Note that the "bandwidth" of the op amp circuit is the range of frequencies that the circuit amplifies. The low frequency end of the "band" is essentially 0 Hz, since the circuit has no trouble amplifying DC signals. The high frequency end of the "band" is the cut-off frequency. At frequencies higher than the cut-off, the gain begins to drop off. So by measuring the cut-off frequency, you are essentially measuring the bandwidth.

2. Change the Gain and Measure the Bandwidth

Change the gain of the circuit by replacing resistor R1 with a larger resistor of 1,000 ohms (1 kilohms, or simply "1 k"). Again, look at the magnitude at low frequencies. Determine the bandwidth of this circuit. In this case, you do not need to make complete measurements for a Bode plot. Instead, you can quickly find the cut-off frequency (the 3 dB point) by looking at the output for a voltage drop of $1/\sqrt{2}$ and a 45 degree additional phase shift. Then check the output voltage and phase at one or two frequencies above and below the cut-off to confirm that your measurement makes sense. At what frequency does the roll-off in response occur?

3. Change the Gain Once More
Replace resistor R1 with an even larger resistor of 10,000 ohms (10 kilohms, or simply "10k"). Again, look at the magnitude at low frequencies and then determine the bandwidth. At what frequency does the roll-off in response occur?

4. And Again
Replace resistor R1 with an even larger resistor of 100,000 ohms (100 kilohms, or simply "100k"). Note that this is a unity gain amplifier. Find the frequency where the roll-off in response occurs*.* You may see some interesting behavior *called gain peaking* at the frequencies just before the gain starts to drop. If you do, be sure to record this.

In steps 1 through 3 above you hopefully saw that the "bandwidth" (the range of frequencies capable of being amplified) depends on the gain of the amplifier. So, you can't make a single amplifier that is both high gain and high speed, for example. To do that you'll need to cascade multiple amplifiers together to make a multi-stage amplifier.

B. Noise

1. Intrinsic Noise and Noise Pick-up
Go Back to first amplifier configuration in Part A (the highest-gain configuration, with R1 = 100 ohms and R2 = 100,000 ohms). Instead of connecting to the function generator, connect the input wire (on the left side of R1) directly to the non-inverting input *using the shortest wire possible.* At this point, you have the input defined as zero volts. For a perfect amplifier, the output would be exactly zero. Now you are going to look at the *noise*.

Connect the output of the circuit to the oscilloscope and set the scope to trigger on "line". There are often 2 sources of noise: one is the intrinsic noise of the electronics, which is usually distributed over a wide range of frequencies. The second source of noise is "pickup", which often comes from the magnetic fields associated with anything that uses 110 VAC as its energy source. By triggering on "line", anything that is synchronized with the 110 VAC power line will be stationary on your scope. Anything that is fluctuating is at some other frequency. Chances are you will see both stationary 60 Hz noise and plus fluctuating "fuzz" noise. The stationary waves are at 60 Hz, 120 Hz, 180 Hz, and other multiplies of the 60 Hz power line. The "fuzz" noise comes from the other environmental sources and the intrinsic noise of the circuit.

Measured signal   =   60,120,180..Hz noise   +   "fuzz"= broadband noise
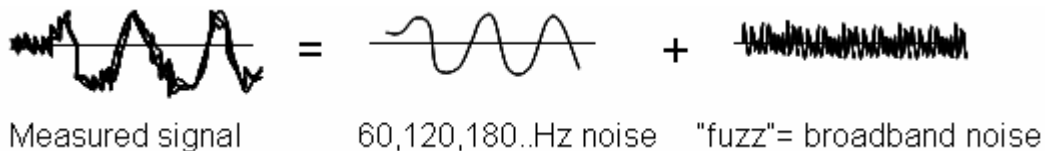
Measure the amplitude of the 60-120-180 Hz stationary waves and of the "fuzz". The fuzz is easiest to measure by looking at one point on the time axis and measuring the total vertical width at the particular time position. It shouldn't matter what time point you pick, but it is probably easiest to do at the top of one of the 60-120-180 Hz waves.

2. Changing Noise Pick-up
Now instead of using a short wire, use a longer wire, approximately 24 inches long, made into a loop.
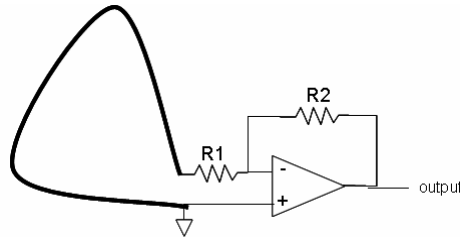

Look at the output noise now. How much larger is it? Move the wire around. As you move it closer to the top of your breadboard you will probably see it increase significantly – you are now picking up the magnetic field associated with the 110 V power supply that is embedded in the top of the breadboard.

Now take the same wire and twist it so that it loops around itself as shown. What you are doing is making something that is commonly called "Unshielded Twisted Pair", or simply, "UTP". How large is the noise now?
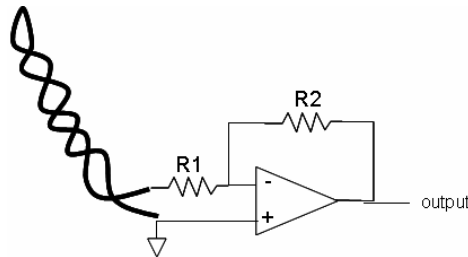
II. Summing Junction and Filtering

The inverting op-amp configuration is extremely versatile. Multiple voltage sources can be summed together at the inverting input, and the processes of amplification and filtering can be done at the same time. A summing amplifier is shown below.
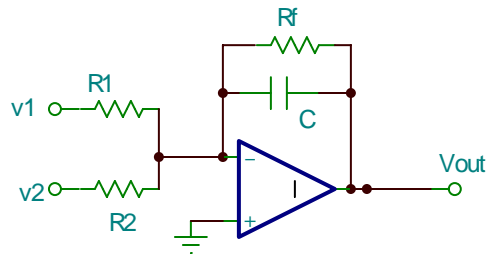
In this case, Kirchoff's current laws say that the total current moving toward the inverting input through the two resistors R1 and R2 will be equal to the current through the parallel combination of RF and C. At low frequencies (where the capacitance has a very large impedance and can therefore be ignored), Kirchoff's Laws give:

$$\frac{V_1}{R1} + \frac{V_2}{R2} = -\frac{V_{out}}{Rf}.$$

Build this circuit using R1 = 1 kΩ, R2 = 1 kΩ, RF = 10 kΩ, and C = 0.001 microfarad. Try using a 100 Hz sine wave with about a 0.1 V amplitude (peak-to-peak) from the oscillator on the breadboard for V1. Use about a 15 mV (peak to peak) sine wave at 1 kHz from the function generator for V2. Look at the composite waveform on the oscilloscope and sketch the waveform in your notebook. Does your result make sense?

Now change the capacitor in parallel with R_F to 0.1 microfarad. Again, look at the signals on the scope and sketch the resulting waveform in your notebook. What difference do you see?

Measure the bandwidth (cut-off frequency) of this circuit with the 0.1 μF capacitor and also with the 0.001 μF capacitor. Use the same input voltage on both inputs when making these measurements.

Be sure to also measure and record the actual values of the resistors and capacitors you use. You will be asked to calculate the expected cut-off frequency for the circuit and compare it to your measured results.

III. Non-inverting Amplifier
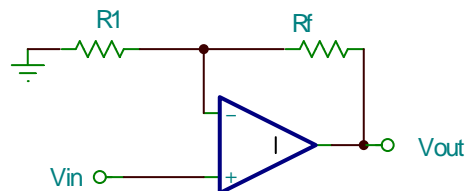
Another commonly used amplifier circuit is the non-inverting amplifier. Note that the signal input is at the noninverting input of the op amp. To derive the gain equation, we

again start with Kirchoff's current law and recall that the op amp allows very little current to flow into the inputs:

$$\frac{v_{out} - v_-}{Rf} = \frac{v_- - 0}{R1} .$$

Substituting in $v_{in} = v_-$ (because the op amp tries to make the voltages at each of its inputs the same) and rearranging the equation yields the gain equation for the noninverting amplifier:

$$v_{out} = v_{in}(1 + \frac{Rf}{R1}) .$$

The non-inverting amplifier's major advantage is that it has the highest input impedance of any op-amp configuration, because the input is applied directly to the non-inverting input.

A. Noninverting Amplifier with Photoconductive Detector

A common application of the non-inverting amplifier is in the pre-amp circuit for photoconductive detectors. These include the mercury-cadmium-telluride (HgCdTe) and indium antimonide (InSb) detectors often used in FTIR spectrometers, and Cadmium Sulfide (CdS) detectors often used in the visible region of the spectrum. The CdS detectors that you will find in lab have a resistance of about 15 kΩ. In that case, highest sensitivity will be achieved when R1 is also about 15 kΩ. The detector forms a simple voltage divider with the photoresistor. Note that if the photodetector has a resistance of ~15 kΩ (the same as R1), then the voltage across the detector will be about 7.5 V. We don't want to amplify that 7.5 V DC component – we only want to see the *changes* in that value. The easiest way to block the DC component is to use a high-pass filter as shown below. Here, the combination of C1 and R2 form a high-pass filter. The high-pass filter blocks all low-frequency signals from reaching the amplifier. By having the filter connected directly to the non-inverting input of the op amp, there is no "loading" of the filter.
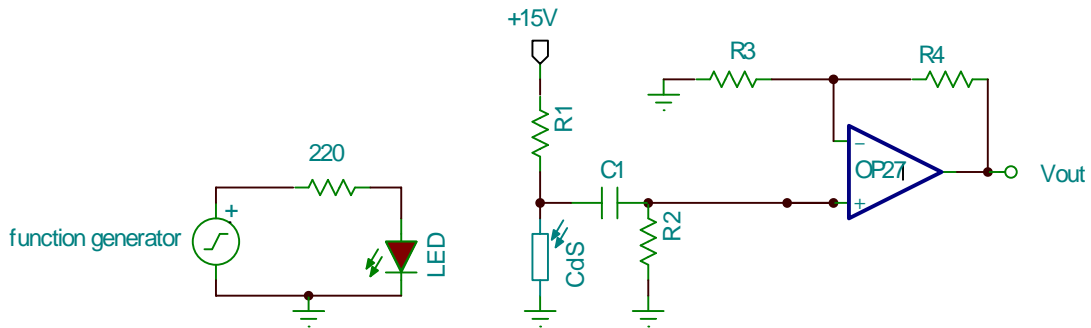
To build the circuit, first set up the function generator so that it applies a 100 Hz square-wave voltage from ground to some positive voltage. About 2 to 4 volts peak (measured from ground) is appropriate. This voltage should be applied through a 220 ohm resistor to the LED. Use a green LED. If it doesn't light up, then the LED is in backwards. Then set up the CdS detector a couple inches away from the LED and use $R1 = 15$ kΩ.

For the rest of the circuit, you need to choose appropriate components. Pick values of R2 and C1 that will give a cutoff of around 1 Hz. The filter will then block the 7.5 V DC but it will pass low frequency signals and low frequency noise (such as the fluctuations in the room lights.). For this kind of application you want to use a ceramic capacitor, which are commonly found only in values up to about 0.1 microfarad. This implies that R2 must be very big. Then try to find values of R3 and R4 that will give you reasonable voltage levels (~1V) at the output. You will need to experiment to find appropriate values, but a gain on the order of 10-100 should be sufficient. You will need something to block the room lights. Be sure to record the values of the components you chose.

Play around looking at the response of the detector and make sure that you are actually seeing the signal from the LED. Sketch the waveforms (noting the time and voltage scales) that you observe with the room light blocked and not blocked. Why does the room light need to be blocked in order to see the signal from the photoconductor?

B. Effect of input bias current

You might expect that when the CdS detector is removed from the noninverting amplifier circuit, that the output of the op amp would be zero. However, you will find even when there is no input signal and you have a blocking capacitor in place (shown below), there is still a small DC offset voltage at the output of the op amp. This comes from the fact that the op-amp actually does draw some current into its inputs, and in the non-inverting configuration this can only come through R2. Since R2 is big, this "input bias current" results in a voltage offset at the non-inverting input that gets amplified just like any other signal.
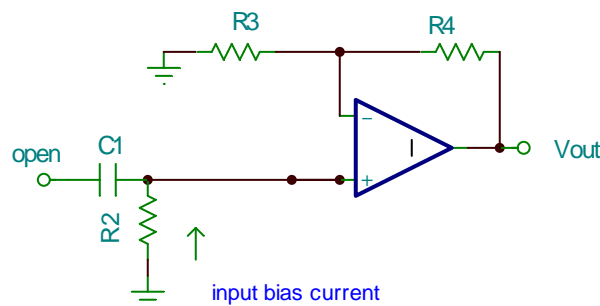
Remove the photoconductor from the circuit, but leave the blocking capacitor and resisters in place. Record the DC offset voltage that you see at the output of the op amp. Also, measure the "fuzz" noise. You might need to switch the scope input to AC coupled in order to filter out the DC offset and focus on the fuzz noise. You will use the measured DC offset voltage to later calculate the input bias current.

Now you can try replacing the OP27 with an OPA627. They have identical pin configurations, so you can just pop out the OP27 and pop in the OPA627. Again record the DC offset and "fuzz" noise. Does the result change?

The OP27 is a very low-noise op amp, but it draws some current into the inputs. This "input bias current" produces a voltage at the inputs. The OPA627 uses field-effect transistors at its inputs. These draw less current, so that your input (and output) voltage should be closer to zero. However, the "fuzz" noise is not likely to improve with the OPA627.

IV. Current-to-Voltage (I-V) Converters: Stability, ringing, and noise

Many different analytical measurements use devices that inherently produce an output current. Examples include photomultiplier tubes, channeltrons and channel plates (for detecting electrons, anions, and cations in the gas phase), and photodiodes. For measurement purposes, these currents are often converted to voltages.

The basic circuit for a current-to-voltage converter (more correctly called a "transimpedance amplifier") is shown below.
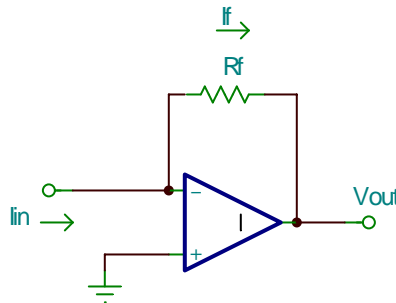
According to Kirchoff's laws, $I_{in} = I_F$ since the inputs to the "ideal" op amp do not draw any current. This yields the equation:

$$I_F = \frac{v_- - v_{out}}{R_F} = I_{in} \;.$$

Also, since the op amp keeps the voltage at the non-inverting and inverting inputs the same, we have $v_- = v_+ = 0$. Substituting in this result and solving for v_{out} gives the equation for a current to voltage converter:

$$v_{out} = -I_{in} R_F \;.$$

So, the amplifier takes a current as an input, and produces an output voltage proportional to that current (but inverted in sign). This is one of the most important op amp circuits around, and also one of the most problematic in practice. One of the biggest problems in practice in using an op amp in this configuration is that they tend to "ring" and/or even oscillate under some conditions. Here, you will be looking at the current-to-voltage converter and the factors that control its stability. In a later part of this lab you will be hooking the I-V converter to a photodiode.

A. Looking at Op Amp Instability

Op amps are least stable when used at the *lowest* gain. In an earlier part of the Op Amp Lab Unit you saw that the effective bandwidth of the op amp circuit was higher when used with lower gains. The current-to-voltage converter is essentially a "unity-gain" configuration because all of the output voltage is fed back to the input. The major cause of instability and oscillation in op amps arises from uncontrolled phase shifts; these phase shifts usually come from capacitance on the inputs that alters the phase of the feedback signal coming from the op amp output back to the inverting input. In the case of the current-to-voltage converter, the inverting input is where the signal is applied (*i.e.*, where you hook up to your photomultiplier tube or photodiode) and also where the feedback signal comes back to. Consequently, capacitance on the input cables causes major problems. One of the most common situations encountered is that you build a circuit and it looks fine. However, when you hook it up to the transducer of interest (photomultiplier tube, photodiode, etc.) it starts to oscillate or otherwise behave strangely – this happens because the extra capacitance on the input shifts the phase of the feedback signal. Below, you will look at this phenomenon, how it affects the noise on your signal even if it isn't oscillating, and how you can reduce this undesirable affect.

Start by making the op amp circuit below using an OP37 (or OPA37) op amp and RF = 100 kΩ. The OP37 is an unusually unstable amplifier, so it works well for demonstration purposes. However, the same thing will happen with any op amp under the right (or wrong) circumstances. With just a resistor across the op amp, this circuit forms a simple current-to-voltage converter. At this point you might see the circuit as being stable (producing an output voltage near zero volts) or it might be unstable, simply oscillating between ~-15 V and +15 V.
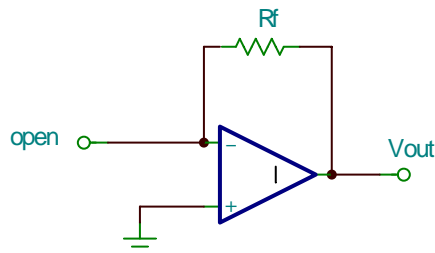
As we will see later, op amps can be used to make oscillators. But, in applications such as photodiode amplifiers and photomultiplier tube amplifiers, you do not want the circuit to oscillate. This undesirable tendency to oscillate is made even worse by capacitance on the inputs. Connect a 100 picofarad capacitor as shown below. Since coaxial cable has ~ 30 pf/ft of capacitance, this represents the capacitance associated with about 3 feet of coax cable – the kind of cable lengths you might be tempted to use in your research.
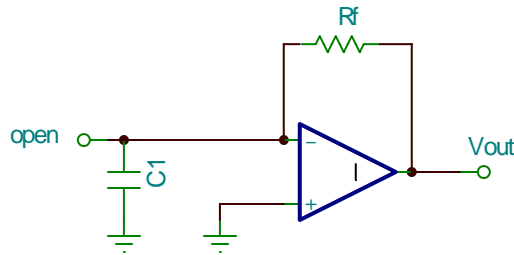
After connecting C1, you should definitely be seeing the op amp circuit oscillating. This (undesirable!) oscillation happens because the combination of capacitance (C1) and feedback resistor RF shifts the phase of the feedback system. You can think of this as a low-pass filter, in which the output of the op amp gets coupled back to the input through RF and then C1. When C1 is large, this introduces a phase shift into the circuit that turns the "negative feedback" (which stabilizes the op amp) into "positive feedback" (which turns it into an oscillator).

B. Illustration of Phase Shifts from Input Capacitance

To illustrate the phase shifts due to the input capacitance, set up the simple RC filter shown below on the left. Leave your op amp circuit intact and simply use another region on your prototyping board.
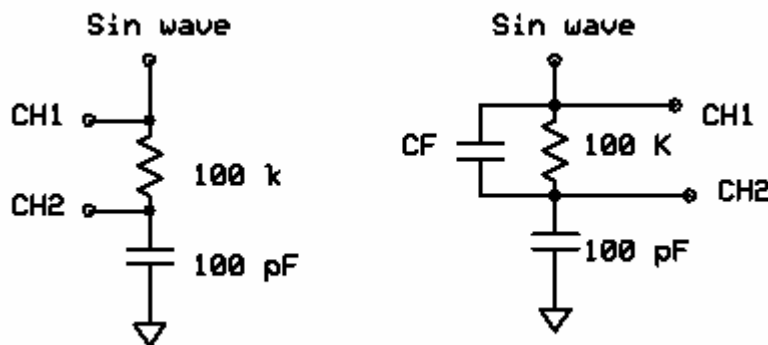
At high frequencies, you will see a phase shift comparing channel 1 and channel 2 on your scope. (Channel 1 represents the op amp output, and channel 2 represents how changes in V_{out} are sensed at the inverting input in your op amp circuit). Now connect

a small (on the order of 10-50 pF) capacitor C_F in parallel with the 100 kΩ resistor, as shown at above right. Now you will see that the phase shift between channels 1 and 2 is smaller than it was without the small capacitor. So, while the 100 pF capacitor is trying to shift the phase one way, adding a second, small capacitor in parallel with the resistor shifts the phase partially back.

The really important thing here is that in your op amp circuit, it is the phase shift associated with the input capacitance (often uncontrolled, or associated with cables, etc.) that tends to make the circuit unstable; because of this capacitance, the inverting input is not sensing the actual output voltage, but a phase-shifted version of it. If the op amp circuit is oscillating or "ringing", you can restore stability to the op amp circuit by shifting the phase back. A capacitor in parallel to the feedback resistor effectively does the job.

C. Reducing Op Amp Instability by Reducing the Phase Shift in the Feedback

Return to your op amp circuit and try to restore stability by connecting a capacitor in parallel with R_F. By doing so, the voltage sensed by the inverting input is closer in phase to that produced by V_{out}. Keeping C1 = 100 pF, try connecting various capacitors in parallel with R_F. Try to find the minimum capacitor value that keeps the circuit stable.

Now, look at the noise level. You will probably see some small oscillations, perhaps on the order of millivolts or tens of millivolts in amplitude. Record the amplitude and frequency of these oscillations, along with the value of the capacitor you need to keep the circuit stable. Once you have done this, look at the frequency response by using your signal generator in series with a 100 kilohm resistor as a current source (shown below). What is the bandwidth of the circuit?
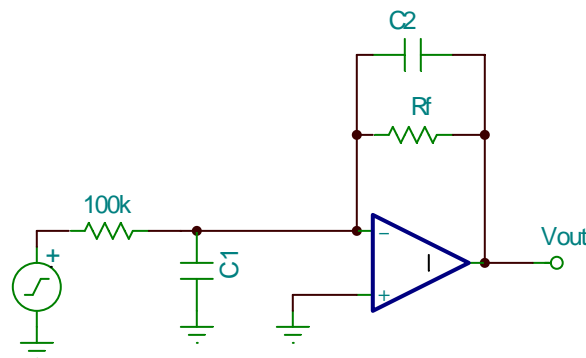
Now, try to increase the value of C2. Again, look at the noise level (amplitude) and the frequency of the remaining "ringing" or small oscillations, and measure the frequency response. What is the bandwidth of the circuit now?

In a circuit such as this, the oscillations are "noise" that interferes with your ability to measure small signals. Without doing anything fancy (such as lock-in amplification), the best signal-to-noise ratio that you could obtain is the maximum output voltage of the op amp (typically about 15 V) divided by the magnitude of these oscillations. In your report, you should be prepared to give your estimate of the maximum signal-to-noise that you obtained, along with the "bandwidth", and to describe the conditions under which stable vs. unstable operating is achieved.

At the end of this experiment you should have at least a qualitative understanding of how to "compensate" an op amp circuit (by putting capacitance in parallel with RF). You should also understand that the compensation to achieve stability also decreases the bandwidth, making the circuit slower to response. You should also understand that because the origin of the instability is the stray capacitance at the inputs, the signal-to-noise ratio that you get in a "real" measurement is high only when the input capacitance is low. So, this means moving the amplifier very close to the signal source, using short cables, and avoiding capacitance on input signals wherever possible.

V. Photodiode Circuits

One of the most common applications of op amps is as current-to-voltage converters for photodiodes, "channeltron" electron multipliers (in mass specs, etc.) photomultipliers, and other similar devices. Each of these devices tends to act as a current source. In the particular case of the photodiode, it can act as either a current source or a voltage source. In the case of photodiode there are a number of ways the signal can be read out. Here you will look at a few of these.

A. Photodiode in Photovoltaic Mode with Noninverting Amplifier

The electron-hole pairs generated by the photodiode are separated by the high electric field that exists in the space-charge region at the p-n junction of the diode. This separation of charge produces a voltage known as a photovoltage. One can measure the photovoltaic response directly using the circuit below.
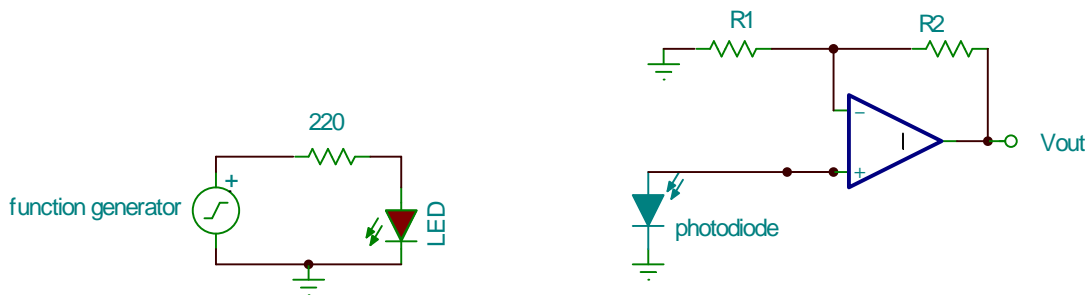
Build the circuit above using an OP627 op amp. Use a red LED and a 0 to 5 V square wave from the function generator to power the LED. Arrange the LED and photodiode so that they can maintain a fixed position throughout these activities.

Note that the op amp circuit is in the noninverting configuration and to have any amplification greater than 2, you need R2 > R1. A ratio of about 10:1 is a probably good. Set up a good optical alignment and try to get V_{out} at least 100 mV due to the LED light.

Determine the time response of your circuit by measuring the rise time of V_{out} at a suitable frequency. The bandwidth of the circuit is then calculated from:

$$\tau_r = \frac{1}{3BW}$$

To determine the signal-to-noise ratio, set the function generator to some reasonably low frequency (may be a few hundred Hz) and measure the amplitude of the LED signal you see on the oscilloscope. Then shut off the function generator, cover up the photodiode so that it is in the dark, and measure the noise. Dividing these two will give you a measure of the signal-to-noise ratio for comparison with the circuits below. (As long as your alignment is maintained)

This photovoltaic response of a photodiode is often used because voltage can be easily measured with a simple circuit. However, in this circuit the time response for silicon diodes is poor because the electron-hole pairs created in the silicon diode have a long lifetime and the electric field at the p-n junction is not big enough to sweep the electrons out of the depletion region quickly. With some other types of semiconductors (GaAs, for example) where the electrons have very short lifetimes, this circuit can give better time response. An even faster time response can be obtained using a reversed biased photodiode.

B. Reverse-biased Photodiode Coupled to the I-V Converter

A reversed-biased photodiode circuit is shown below. In this circuit you must be sure that the photodiode is reverse-biased – otherwise it will pass large amounts of current, heat up, and be destroyed very quickly (within a matter of seconds).
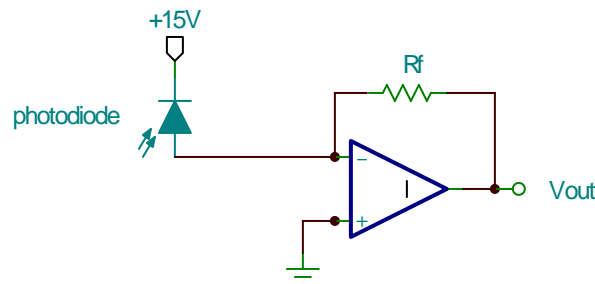

Hook up the circuit as above with an OP627 op amp. A value of 100 kilohms is probably good for Rf, but the best value will depend on the LED and photodiode you are using, and you should change it as appropriate. Try to choose Rf so that the output is again at least 100 mV.

Again, measure the rise time and see how it compares with what you saw in Part A. This circuit is subject to "ringing" and you may need to add a small 10 pF capacitor in parallel with Rf. This does slow the detector response but it may help you get a better rise time measurement. Measure the signal-to-noise ratio in the same way that you did before. How does the signal-to-noise ratio of this setup compare with that in Part A?

Some advantages of this configuration include the highest signal-to-noise ratio of all the configurations, along with good time response up to ~ MHz. However, the input impedance of the current-to-voltage converter is nominally zero (the inverting input is held at ground potential by the op-amp), so that for fast signals a reflection may be generated when the pulse reaches the amplifier. As you saw earlier, capacitance at the input of the I-V converter can also make the circuit unstable. So, usually it is very important to put the amplifier very close to the photodiode, and to use low-capacitance photodetectors.

C. Reverse-Biased Diode Coupled to a Noninverting Amplifier

In this next circuit (shown below) the diode is again reverse-biased, and the current through the diode is dropped across a 50-ohm resistor. The voltage across this resistor is then measured with a non-inverting amplifier. The gain of the amplifier will probably need to be increased compared to part A since the induced photocurrent is quite small. This circuit has the advantage that it provides 50 ohms of input impedance. This configuration is usually used when the absolute fastest time response is needed. Commercial voltage amplifiers for photodiodes of this type are available from companies such as Newport/Oriel, with bandwidths on the order of 350 MHz and voltage gains of ~30. This fast time response is accompanied by a loss in signal-to-noise ratio.
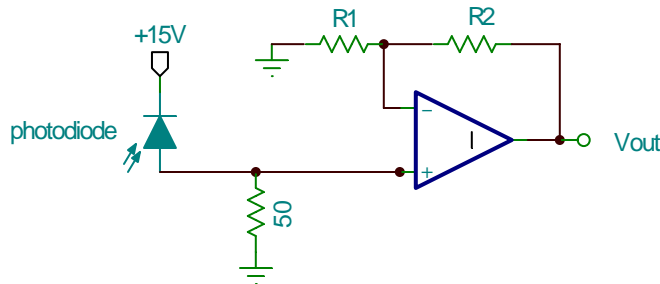

Build this amplifier and determine its rise time and bandwidth. Also try to measure theS/N ratio in the same manner that you used for the other two photodiode circuits.

VI. Active Filters (Do this only if you have time)

Another application of op amps is to create various types of "active filters". You can make active filters simply by sandwiching simple RC filters between two op amps. However, you can also do much more interesting things. The example below uses two op amps. The first one is a simple non-inverting amplifier that provides very high input

impedance and very low output impedance. The second stage, consisting of the two resistors R3, C1, and C2, is one example of an active filter, called a "Sallen-Key" filter. One property of the Sallen-Key filter is that it is a second-order filter, meaning that above the cutoff frequency, the response decreases like $1/f^2$ rather than like $1/f$. In order for the Sallen Key filter to work correctly, it is very important for the two resistors R3 to be as identical as possible, and for the resistor C2 to be exactly twice the value of C1. The easiest way to do this is to use three capacitors of a single value; use one for C1, and then put two capacitors in parallel to form C2.
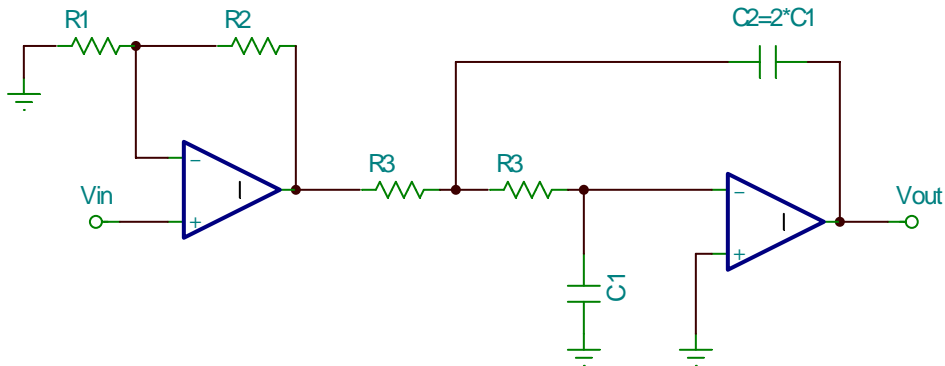
Measure the frequency response (transfer function, phase shift) of this circuit.