ENGG1000 Electrical Engineering Stream, Session 1, 2016

Lab Exploration Tutorial 2

Sensors and Filtering Circuits

In this laboratory exploration you'll investigate some electrical circuits used for sensors – converting physical signals and disturbances into electrical signals for processing or some other purpose. You'll also build simple circuits to filter these input signals – essentially to clean them up for more accurate processing.

1. Driving an LED

To begin with, you'll need a circuit to generate the light signals that you'll use to detect with the photodiode. Here you'll construct three different light emitting circuits to assist you in your detection process – one to power the 12V incandescent lamp; one for the mini-red LED; and finally one for the Infra-red LED.

In your component pack you'll find a simple toggle switch. This make turning the light source on and off much easier than connecting and disconnecting the voltage source, as you would have done in Lab 1. Solder some wires to the connectors of the switch to make it easy to attach to your breadboard.

Looking back at your work in Lab 1, design and test three circuits for illuminating the three components of interest:

1. The 12 V halogen lamp, using the DC power supply.
2. The mini-red LED, which expects an operating current of 30 mA and has a forward bias voltage drop of 1.9 V. Power this LED with the 9V battery supplied.
3. The infra-red LED in your component pack. Design a circuit, using the 9V battery as the source, to power this IR LED with 100 mA of current. Its forward bias voltage drop is listed as 1.4 V.

For your final circuit, you won't be able to see the radiation it is emitting so you'll need to think about how you'll demonstrate that your circuit is operating correctly. Using a multimeter, perhaps? Or is the camera on your phone sensitive to Infra-red, perhaps?

For each of your circuits use the toggle switch connected in series to the voltage supply to turn the light source on or off at will.

2. Detecting a Light Signal

You will now use a Phototransistor to detect your light signal. A phototransistor is a device that when light of the appropriate wavelength is shone on it, it will produce a current out of its terminals. We can thus use this Phototransistor to detect when the LED is flashing.

Phototransistors are photodiodes with internal gain (or amplification). Photodiodes are essentially just diodes (devices that allow current to pass in only one direction) that are capable of converting light into a small current. Photodiodes are typically reverse biased when photoconducting, and should be supplied with a small current when attempting to detect light, according to the data sheets, for example using in a circuit like that seen in Figure 1.

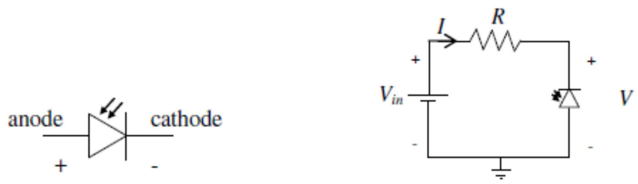

Figure 1 – Photodiodes and photodiode detection circuit

When a light signal (or infrared signal, if it is an infrared photodiode) is detected, the current in the photodiode increases (typically by a small amount), so the voltage *V* will change in response to the increased current.

In your kit you have one clear plastic photodiode, which looks a lot like the incandescent lamp but has legs of different lengths. As usual for diodes, the longer leg is the anode. This photodiode is matched for the wavelengths of light emitted by the incandescent lamp. As stated above, for better sensitivity to incident light, a photodiode is usually reverse-biased, which means that an external voltage is applied to make the cathode at a higher voltage than the anode. The current through a photodiode when reverse-biased will be small until light is incident on the device. Then the current through the device will be proportional to the incident light intensity.

Connect up the photodiode light detection circuit shown in Figure 2. Set V_{CC} = 9V and use R_L = 1 kΩ. Observe the voltage across the load resistor, and hence the current produced in the photodiode, when the lamp is both on and off. Then compare this to the current produced when the LED is shown at the photodiode - please be careful not to overload the diode. As you use it, try to work out which direction it is most sensitive to. (Hint: if you are having trouble detecting current out of the photodiode, use a brighter light source, like a lamp!)

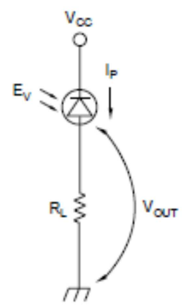

Figure 2 – Photodiode detection circuit

The principle of operation of a Phototransistor is similar, and we'll work through transistors later in this lab exercise. Once again you are given two different types of phototransistors, both sensitive to IR light but each with different characteristics. One of the IR phototransistor resembles a clear diode, while the IR phototransistor has a flat top. You should try both at different times of the experiment. Note that phototransistors are not sensitive to just one single wavelength of light but a range of wavelengths with varying sensitivities. Information on this can be found on the data sheet for the component.

Identify the flat topped phototransistor sensitive to visible (red) light and connect the circuit illustrated in Figure 3. Place the phototransistor close to the illuminated red LED, and connect it to a resistor R_L. Initially set V_{CC} = 9V (use the supplied battery) and R_L =270 Ω. Observe the voltage across this resistor using a multimeter. You should see the current in the phototransistor following the signal driving the LED.

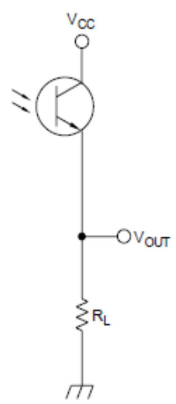

Figure 3 – basic phototransistor circuit

For this phototransistor light detecting circuit, study the following:

1) Compare the input signal to the LED with the output of the phototransistor. Is there any noticeable delay? Obviously light and electricity travel very fast….

2) Measure the current I_C as the LED is moved away from the phototransistor. You should observe that I_C is proportional to the incident light intensity onto the phototransistor. Note the magnitude of these currents – are they sufficient to drive another LED, for instance?

3) Now make R_L = 100 kΩ, and observe the current as the LED is moved away. Note that the current doesn't change in value at all. This is known as 'switching mode', as opposed to 'active mode' above. 'Switching mode means the phototransistor will either be on or off depending on whether there is incident light or not. However, in 'active mode' the phototransistor current is proportional to the incident light intensity. This idea of switching is fundamental to the transistor as a circuit element, as we shall soon see in this lab exercise. Transistors are the foundation of digital logic, and so underpin all . modern computing technology.

4) Finally, drive the LED with a 1kHz signal generated by the Signal Generator. The LED will now flash 1000 times per second, which is far too quick for your eye to be able to resolve. Display the current detected by the phototransistor using the CRO. You will be able to see that detected current oscillates with the same frequency of 1kHz. Verify this experimentally, by calculating the period of the phototransistor current.

Check Point 1 – Show your circuit and your note book, including your calculations, to a demonstrator to be marked off.

3. Microphone

In your component kit you'll find a little microphone. To use it on your breadboard you'll need to solder some wires to the two connectors on the back of the little microphone – they're called 'solder pads' for exactly this reason. The datasheet for this microphone can be found on the subject website. Take note that the terminal on the darker side is the positive (+) connector, and always try to have this to the positive voltage of the power supply.

If the microphone is not connected to anything it is not very useful in detecting sound, and it must be powered to convert sound waves into electrical signals. Its operating principle is very simple: the incident sound wave will modulate the current flowing through the microphone. This means that the electrical current flowing through the microphone will follow the incident sound pressure wave.

On the datasheet you'll see that the microphone expects a supply voltage of 2V for normal operation, and has an impedance of 2.2kΩ. Design a simple circuit, as shown below in Figure 4, to power this microphone from one of your 9V batteries. That is, you'll need to determine a suitable value for the resistor *R*.

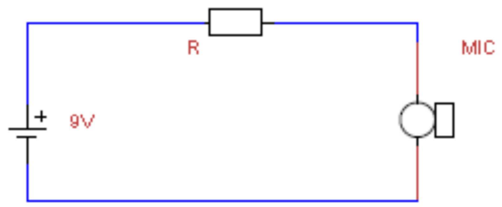

Figure 4 – Microphone circuit

The question is then: how can we tell if our microphone is working correctly? The issue is that the audio signal is not constant in time, and its variation is exactly what we're interested in – this is what reflects the audio signal. Hence, we cannot use a simple multimeter to check that our circuit is operating correctly. Instead, we must use a CRO (Cathode Ray Oscilloscope), a device that allows us to display how signals vary in time.

Connect a channel of the CRO across either the microphone or the resistor, obtaining a representation of the current waveform on the display. Make some sounds near the microphone and verify that the waveform you are observing on the CRO does indeed correspond to the input sound pressure waveform to the microphone. Make sure that the CRO channel you're using here is AC coupled (this is just a little input switch).

To make the operation of the microphone a little more stable than simply detecting random generated sounds, we'll use the function generator to produce a stable sound wave at a fixed frequency. To do this you're going to need the computer speakers. Set the signal generator to produce a 1kHz sine wave, and connect the signal generator outputs to the input of the speakers. Have the volume settings on the speakers very low to begin with, as a one 1kHz sine wave is quite annoying for your colleagues. Depending on the lab you are in, the plugs on the speaker input and signal generator output may not match, but you can easily make this connection using cables with alligator clips – ask a demonstrator if you're unsure how to do this.

Place the speaker near the microphone and increase the volume. You should observe a stable sine wave on the CRO display. Alter the frequency produced by the signal generator and record the changes that you observe on the CRO. For a frequency of 2kHz measure the period of the current waveform you're observing on the CRO, and hence verify it is indeed the value you expect.

4. Filters

The problem with many sensors is that they will pick up more than just the signal you are interested in. For example, a microphone will pick up all sound waves incident on it and convert them into electrical current, not just the sound signal you are interested in. In many applications we are only interested in a part of the signal picked up by a sensor – say a particular frequency or frequency region. A filter is an electronic device that allows us to remove the unwanted components in a signal, leaving only the parts we're interested in.

There are many different types of filters design for numerous different applications, but we'll only introduce you to some of the simplest types here. The key element in a filter is the capacitor, which you met in Lab 1, since its impedance varies with frequency. With this simple building block we can design electronic circuits that treat the various frequency components in a signal differently, ultimately allowing us to reject some frequencies (those we're not interested in), and enhance or amplify others (those that we are interested in).

Construct the very simple circuit shown below in Figure 5. Use a 10kΩ resistor and a 10nF capacitor – you can find these on the shelves in the lab. Use the signal generator as the input and observe the voltage waveform across the capacitor (this represents the output).

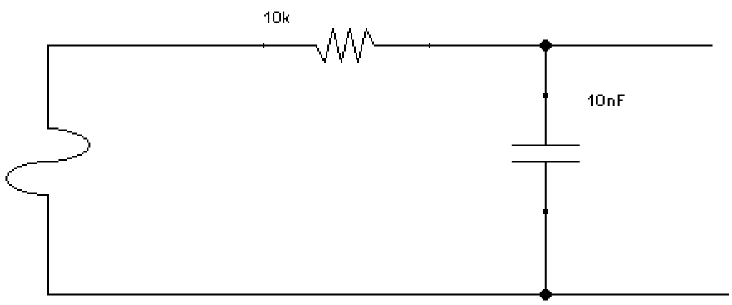

Figure 5 – First-order Low-pass Filter

Begin with a frequency of only 10Hz or so, and observe the amplitude of the output. Slowly increase the frequency generated by the signal generator, observing the effect of the amplitude of the output signal that you are displaying on the CRO. Keep on increasing the frequency until you reach say 3kHz. What you would have observed, if you had connected the circuit correctly, was that the output amplitude would have remained roughly constant at low frequencies, but when the frequency neared 1.6kHz the output amplitude would have begun to decrease, becoming nearly undetectable at high frequencies.

This simple circuit is an example of a low-pass filter: it will pass low frequencies with very little attenuation, but remove high frequency components. The important parameter is the frequency at which it transitions from passing to attenuating, which is called the 'cut-off' frequency. There are actually several ways this frequency is defined, depending on the particular application (communications, audio signal processing, etc.). In electronics one tend to talk about the 3dB frequency, which is the frequency at which the output power at that frequency is reduced to one-half of the input (and a factor of ½ is equivalent to 3 decibels, hence the name).

For this simple circuit, the 3dB frequency is determined by the capacitance and the resistance used by the simple equation:

$$f_{3dB} = \frac{1}{2\pi RC}$$

Calculate, using this equation, the expected cut-off frequency for the above circuit. Now, experimentally try to measure the cut-off frequency. On the CRO you can observe the output signal amplitude, not the output power, so the cut-off frequency is the frequency at which the amplitude is reduced by a factor of $1/\sqrt{2}$, about 0.707 (this is because the signal power is proportional to the square of the signal amplitude).

Now, design a simple low-pass filter to have a cut-off frequency of 1kHz. That is, select values for the resistance and capacitance to produce a circuit with a cut-off frequency of about 1kHz. Construct and verify your circuit design behaves as expected.

The one thing you would have noticed in your filters is that the transitions between the pass-band (or frequency range over which it allows signal components through) and the stop-band (or frequency range over which it removes signals) is very slow. This is a characteristic of a simple circuit that we constructed – something called a first order filter. It is possible to build more complex, higher order filters that have much sharper transitions, but we'll leave these to your later studies or personal research.

The natural analogy to a low-pass filter is of course a high pass filter: a circuit that passes high frequencies but removes low frequencies. This is very easy to construct by simply swapping the position of the resistor and the capacitor, to create the circuit as shown in Figure 6. The input comes from the signal generator and the output is across the resistor – the waveform displayed on the CRO.

Build this simple high-pass filter on your breadboard and verify that it removes low frequency inputs but passes high frequency inputs. Measure the 3dB cut-off frequency, and show that the cut-off frequency equation above also holds for this high-pass filter too.

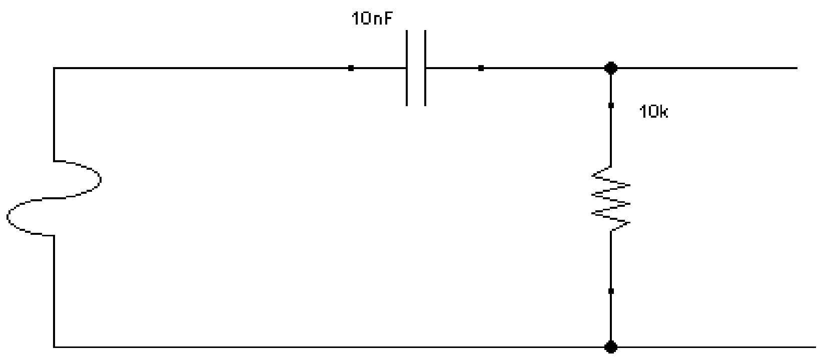

Figure 6 – First-order High-pass filter

The next type of filter that you'll find very useful is a band-pass filter: one that allows frequencies through within some finite region, $f_1 < f < f_2$., and removes components outside of this range. The simplest way to construct a band-pass filter is to cascade a low-pass filter with a high-pass filter. For our simple first order filters, this results in a circuit as shown in Figure 6.

Design a simple band-pass filter to pass frequencies in the range of 700Hz up to 1500Hz. All you need to do is to select suitable values for R1, R2, C1, and C2 in the circuit shown in Figure 7. The easiest way to do this is to think of this circuit as a low-pass filter, like that in Figure 5, followed by a high-pass filter, as in Figure 6. Design R1 and C1 as if you were designing a low-pass filter with a cut-off frequency of 1500Hz, and then choose R2 and C2 to make a high-pass filter with a cut-off frequency of 700Hz.

Build and then test that your circuit behaves as expected.

These filters are very, very simple, and consequently do not perform particularly well. We'll now introduce the operational amplifier, or 'op-amp' for short, which is the building block of more sophisticated and higher-order filters.

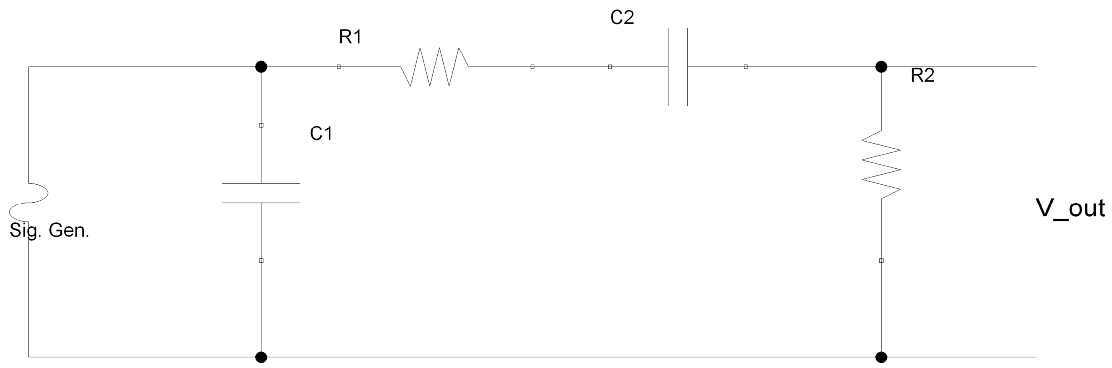

Figure 7 – Band-pass filter

Check Point 2 – Show your circuit and your note book, including your calculations, to a demonstrator to be marked off.

5. Op-amp Filters

In lectures transistors were discussed, and then we saw how they could be used to provide current or voltage amplification. Amplifiers have turned out to be very important in analog electronics, and for this reason the very simple transistor amplifier circuit we looked at has been superseded in most practical cases by hundreds of different amplifier designs, often including many transistors rather than just one. Over the years, one of these designs, known as the operational amplifier (op-amp) has become a standard component of a wide range of circuits. This very useful device, only available as an integrated circuit, is commonly seen in circuit diagrams as the symbol shown in Figure 8. You were in fact given one of these in your component pack – the IC chip labeled LM741. Note that two important connections are not shown: the positive and negative supply voltages to the chip, which are either positive (*VCC*) and ground or positive (*+VCC*) and negative (*-VCC*). Conventionally, these connections are not shown, however they are always implied! (It won't do anything unless the supply voltage is independently supplied)

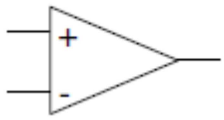

Figure 8 – Circuit symbol for an Op-amp (such as the LM741)

The op-amp is a two-input amplifier, whose output voltage *VO* is an amplified version of the *difference* of the two inputs *V+* and *V-* , i.e. $V_o = A(V_+ - V_-)$. (Note that all voltages discussed here are relative to the ground point in the circuit) The positive terminal is known as the non-inverting terminal, and the negative terminal is known as the inverting terminal. Of course one input of the op-amp could be connected to ground to obtain a single-input amplifier; the reason this is not common will be discussed shortly.

The op-amp is also approximately an ideal amplifier, in that its gain *A* is extremely large (typically ≈ 10^5), its input resistance (the resistance between the two input terminals) is also very large (typically 10MΩ or so), and its output resistance is small (typically <500Ω). These features are extremely attractive – we saw earlier that the impedance of a device will alter the circuit when you connect it to it, when you attempted to drive the LED. Very high input impedance means that it can be thought of like an ideal voltmeter, and be connected to a circuit across a device without drawing significant current (and hence changing the circuit). Low output impedance means the output voltage can look a bit like an ideal voltage supply, producing the same voltage regardless of the circuit that is connected to it.

Although an op-amp is fundamentally an amplifier, it turns out to be useful in a vast number of other types of circuits, some of which are introduced in this lab program.

When analyzing op-amps, we assume that the op-amp actually is ideal, i.e. it has infinite gain *A* = ∞, and the input resistance is also infinite. These assumptions lead to two rules of analysis:

- Because the input resistance is infinite, *no current enters either input terminal*
- Because the gain *A* is infinite, if the output voltage is **finite**, then *both input terminals have identical voltage*

These might seem like idealistic assumptions, however in practise they are very accurate for many applications, including the circuits that you will meet in this lab program and throughout your career. The effects of deviation from this ideal behavior on a circuit can be modeled separately in terms of non-ideal effects.

The application of this extremely useful and versatile integrated circuit (IC) will be to construct better filters then we did in section 4 above. A simple band-pass filter using an op-amp is shown in Figure 9. The values of the resistors and the capacitors here have been chosen to produce a band-pass filter with the centre of its pass-band at 1700Hz.

Construct this simple band-pass filter on your breadboard using the LM741 op-amp. Remember that you'll need to power the op-amp for the circuit to work (and these are not shown, but are always implied). The input voltage comes from the signal generator and the output signal should be viewed on the CRO. Experimentally determine the cut-off frequencies for this filter.

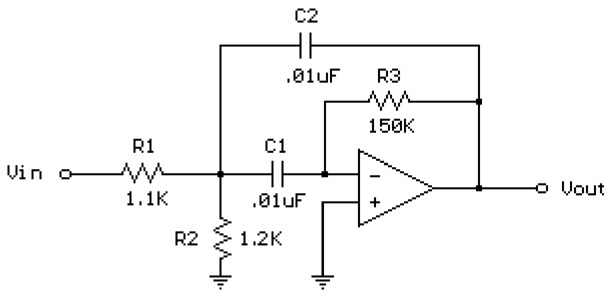

Figure 9 – Band-pass filter using an op-amp

As discussed before, a band-pass filter passes a range of frequencies while rejecting frequencies outside the upper and lower limits of the pass-band. The range of frequencies to be passed is called the pass-band and extends from a point below the centre frequency to a point above the centre frequency where the output voltage falls about 70% of the output voltage at the centre frequency – the 3dB points. These two points are not equally spaced above and below the centre frequency but will look equally spaced if plotted on a log graph. The percentage change from the lower point to the centre will be the same as from the centre to the upper, but not the absolute amount. This is similar to a musical keyboard where each key is separated from the next by the same percentage change in frequency, but not the absolute amount.

The filter bandwidth (BW) is the difference between the upper and lower pass-band frequencies. A formula relating the upper, lower, and centre frequencies of the pass-band is:

Centre Frequency = Square Root of (Lower Frequency * Upper Frequency)

The quality factor, or Q, of the filter is a measure of the distance between the upper and lower frequency points and is defined as (Centre Frequency / BW) so that as the pass-band gets narrower around the same centre frequency, the Q factor becomes higher. The quality factor represents the sharpness of the filter, or rate that the amplitude falls as the input frequency moves away from the centre frequency during the first octave. As the frequency gets more than one octave away from centre frequency the roll-off approaches 6 dB per octave regardless of Q value.

For a single op-amp band-pass filter with both capacitors the same value, the Q factor must be greater than the square root of half the gain, so that a gain of 98 would require a Q factor of 7 or more. These are some of the more detailed design issues associated with op-amp based filters which we do not have time to explain in detail in this course. There are many filter design out there in the literature, and it should be very easy to find an existing pre-designed circuit to achieve your design goals.

The circuit above is a 1700 Hz band-pass filter with a Q of 8 and a gain of 65 at centre frequency (1700 Hz). Resistor values for the filter can be worked out using the three formulas below. Both capacitor values need to be the same for the formulas to work and are chosen to be 0.01µF which is a common value usable at audio frequencies.

$$R_1 = \frac{Q}{2\pi GCf} = 8/(65 * .00000001 * 6.28 * 1700) = 1152 \text{ or } 1.1K$$

$$R_2 = \frac{Q}{2\pi(2Q^2 - G)Cf} = 8/((128-65) * .00000001 * 6.28 * 1700) = 1189 \text{ or } 1.2K$$

$$R_3 = \frac{2Q}{2\pi fC} = 16 / (.00000001 * 6.28 * 1700) = 150K$$

Using the above equations, design a band-pass filter with a centre frequency of 1kHz, a Q-factor of 20 and a Gain of 20. Build and verify that your circuit works as expected.

Check Point 3 – Show your circuit and your note book, including your calculations, to a demonstrator to be marked off.

5. Voltage Comparator

A very simple application for an op-amp is a voltage comparator. The idea of a voltage comparator leads very nicely into an introduction to digital logic circuits. A voltage comparator compares two input voltage signals and produces one of two outputs: the high output is produced if input A is greater than input B; or the low output if input A is less than input B. This means that the output is binary, and one can associate these two voltage levels as either ‘1’ or ‘0’ – the two digital logic states.

An op-amp without feedback is the simplest form of comparator one can conceive. Recall the ideal op-amp depicted in Figure 10. For a practical op-amp the gain *A* is very large, at least much larger than 1000 at least. If feedback is not employed, consider what happens if the V+ input is greater than V-. The output voltage rises, and keeps on rising, but practically will saturate at slightly less than Vcc – the maximum supply voltage, as the op-amp cannot produce any greater output than this. On the other hand, if the V+ input is held at less than V- the output will saturate at –Vcc, the minimum output voltage that this op-amp can produce.

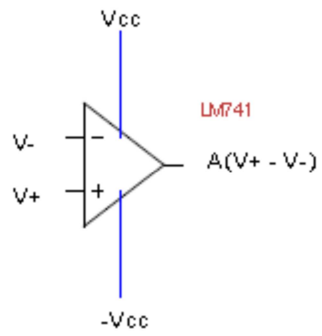

Figure 10 – Ideal Op-amp

This circuit thus produces one of two outputs, which changes depending on which of the input signals is larger – our first foray into the world of digital logic.

Now not every op-amp is well-suited to be used as a comparator, for reasons that are beyond the realm of our interest here. Particularly for us, the LM741 is one such op-amp. This is why you've been supplied with the CA3260 op-amp – an op-amp that is well-suited for this application. In fact, dedicated op-amp based circuits are readily available and packaged as 'voltage comparators' themselves.

Connect up the basic op-amp comparator, using the CA3260, as shown in Figure 11. Connect the output to the digital multimeter and the inputs to the two outputs of the DC power supply. See that the output switches between two values, depending on which of the inputs is larger. You may use one of the 9V batteries to power this chip.

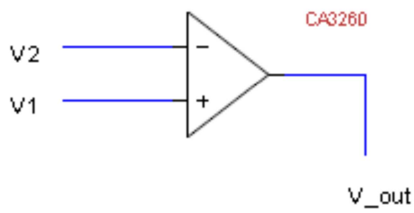

Figure 11 – CA3260 voltage comparator

Next connect the output to an LED, so that the LED will light-up if V+ is larger than V-, and be off otherwise. Then try to design a circuit with two LEDs at the output, one of which will light up corresponding to which input voltage is higher (hint: you might find a transistor helpful here, or perhaps not – can the comparator generate enough current to power the LED?).

The comparator circuit is often used with a single voltage input and a fixed voltage reference. Here, the V- input is set to V_ref, some fixed voltage reference level, and the output will switch whenever V+ is greater than the fixed voltage reference.

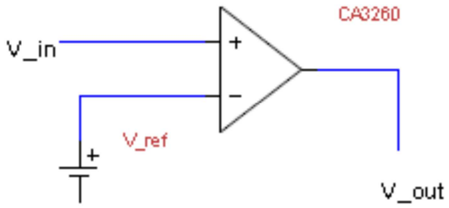

Figure 12 – Voltage comparator with reference input

Wire up the circuit in Figure 12, using the DC power supply as the reference and the signal generator at a low, visible frequency as the input voltage. Vary the reference voltage produced by the DC power supply (and possibly the amplitude of the signal generator) to create different LED patterns at the output – the LEDs will alternately flash on and off at different rates with different duty cycles (staying 'on' for different proportions of the cycle period). Determine your personal favourite pattern.

Finally, in our exploration of the wonders of the voltage comparator, think back to the first experiment and use the photodiode received signal to switch the comparator output on and off, depending on whether the photodiode is detecting the light source or not. You can choose any pair of LEDs and photodiodes for this circuit that you like. Your design goal here is to select the reference voltage level at an appropriate point to be able to distinguish between the 'detecting light' states and the 'not detecting light' states. The output LEDs should provide a nice visual confirmation that your circuit is indeed working as you intend it.

6. Light-activated Switch

The next natural stop on our journey is to ask whether we could use the presence of a signal to switch the output of a voltage comparator. At this point it is worth pointing out some issues with various sensor signals. The difference between the input signal from, say, a photodiode, and a microphone is that the microphone signal is a waveform that oscillates around a constant value, whereas the photocurrent level rises in response to the detected light signal. In essence, the light signal is detected as a consistent disturbance while the audio signal is by its very nature a fluctuating signal. An audio signal thus requires slightly different treatment than a basic, naive voltage comparator.

One of the big problems with the simple voltage comparator shown above is its resistance to noise. Imagine a noisy input signal, that transitions between the above the reference voltage level to above the reference voltage level. If there's noise present, as the signal transitions from to the high the output level will rapidly switch states as it does so. This is depicted in Figure 13.

A simple circuit design to deal with exactly this problem was the Schmitt trigger. The basic Schmitt trigger circuit is shown below in Figure 14. The idea was to use positive feedback on the op-amp to stabilise the output waveform. After the input voltage has switched above 0 (as the reference here), it will not switch back until it returns below

$$-\frac{R_2}{R_1 + R_2} V_{sat}$$

This reduces the noise on transition, and the Schmitt trigger is an important little circuit with many applications, like de-bouncing noisy input signals.

Wire up a simple Schmitt trigger on your breadboard and examine its behaviour. Design the circuit so that it can switch on the signal from your I/R diode at a distance of about 1m, and then turn off when the I/R signal moves beyond about 2m. Could you think of any other uses for this circuit?

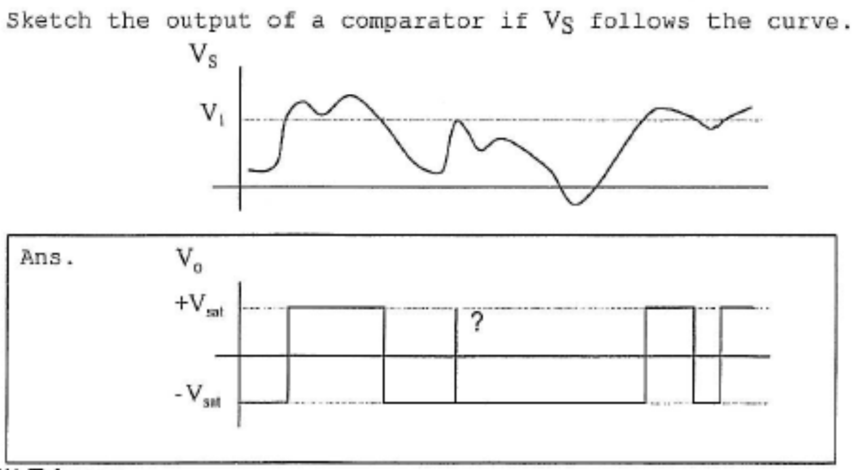

Figure 13 – Noisy comparator transition

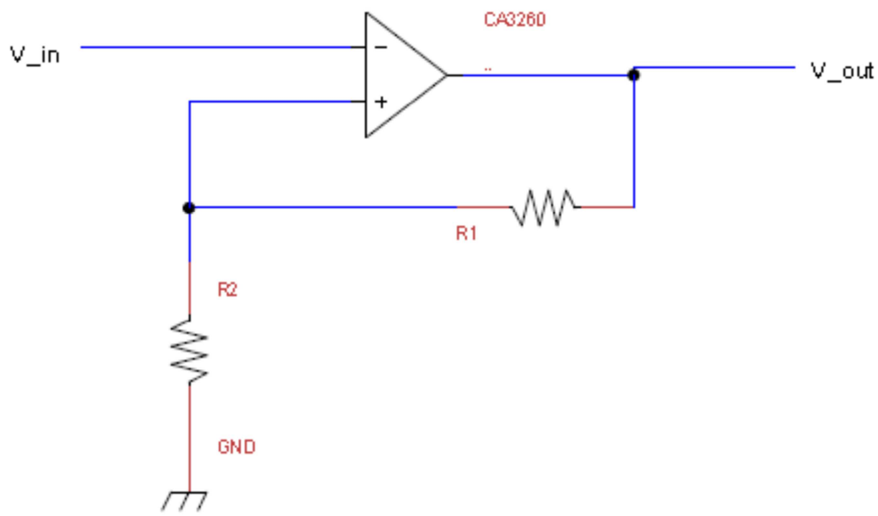

Figure 14 – the Schmitt trigger

Check Point 4 – Show your circuit and your note book, including your calculations, to a demonstrator to be marked off.

7. Temperature Sensing

The basic voltage divider circuit you looked at in Lab 1 and the principles underlying it have many practical applications in electronic circuits. A very simple application of a voltage divider circuit is to power a thermistor. A thermistor is a very simple sensor used to measure temperature – it is nothing other than a temperature dependent resistor. Most thermistors have Negative Temperature Coefficients (NTC), which means that its resistance drops as the temperature increases. Positive Temperature Coefficient (PTC) thermistors are possible, however they are much rarer.

The thermistor you are provided with has a nominal resistance of 22kΩ at 25°C. The resistance, *R*, of a thermistor as a function of temperature, *T*, is typically give by

$$R = Ae^{B/T}$$

The value of the parameter *B* can be found on the datasheet for the thermistor. Once this is known the parameter *A* can be determined from the nominal resistance at 25°C. Consult its datasheet, which you'll find on the Course Website. Examine the data sheet and determine the approximate expected resistance of the this thermistor at 0°C, and also 50°C.

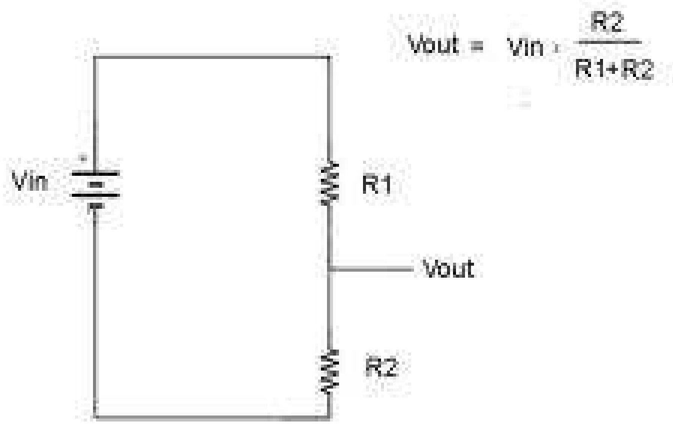

Figure 15 – Voltage Divider Circuit

Replace the resistor R2 in the circuit in Figure 15 with the thermistor. Warm the thermistor in your hands and observed how the voltage measured at V_out changes.

Now, for a little bit of an exercise in circuit design. Building on the basic circuit shown in Figure 15, add the thermistor in parallel to R2, and V_out is now across this parallel combination of R2 and the thermistor. Your aim is, with a 9V battery power source, to determine the values of R1 and R2 such that the output voltage V_out is 5V when the temperature is 50°C and 1V when the temperature is 0°C.

Select the closest values of resistances for R1 and R2 that you can find, wire up the circuit, and measure the output voltage. Use this voltage, along with your equation from before, to determine the approximate temperature of the thermistor. Does this value seem reasonable?

Check Point 5 – Show your circuit and your note book, including your calculations, to a demonstrator to be marked off.

Acknowledgements

We hope that the above laboratory exercise have been both fun and useful for you, and give you some ideas for how to design electronic circuits for your project.

The following references have been used to construct these exercises:

'All about Electronics', http://www.allaboutcircuits.com/vol_6/index.html

J. Epps, and D. Taubman, "ENGG1000 Lab Exercises: 2007-2009".

P. Horowitz and W. Hill, "The Art of Electronics", Cambridge University Press, 1989.

'Basic electronics tutorials", http://www.electronics-tutorials.ws/