

Chapter 3

Lab 2: Filters and Operational Amplifiers

Objectives:

- Use MATLAB to generate plots.
- Show the I-V characteristics of diodes and incandescent light bulbs.
- Understand low-pass RC filters and design one to remove noise from a signal.
- Understand Operational Amplifiers and use them in the following devices:
 - Construct a voltage follower to solve the input impedance problem in the last lab.
 - Construct a linear amplifier to enhance small signals
 - Construct a differential amplifier to enhance the difference between two signals
 - Construct an integrator. Convert a velocity measurement to displacement.

3.1 Introduction

In Lab 1 we looked at resistors and devices that can be built with them. We showed that the resistance of a resistor is constant and that the current flowing through a resistor can be determined by measuring the voltage across it. In this lab, we will investigate more components: light bulbs, diodes, capacitors, and operational amplifiers. We will use the linear resistor as a tool to show that light bulbs and diodes are not linear, and use the frequency-dependent impedance of a capacitor to remove unwanted frequencies in a signal. We have previously seen that the voltage divider produces a voltage level *less* than that of the input. In this lab, we will introduce linear and differential amplifiers that increase or *amplify* voltages, and the voltage follower which solves the loading problem that we experienced with voltage dividers. Finally, using capacitors and amplifier circuits, we will construct devices that can integrate and differentiate waveforms much more quickly than digital computers.

Operational amplifier circuits tend to be much more complicated than the circuits that were built in lab 1, and can be difficult to debug when not working properly. Therefore, it is imperative that you are very careful when constructing these circuits. Make sure that all connections are made securely, and lay out wires in an orderly fashion. Try to get in the habit of building your circuits so that it is easy to compare them with the schematic. It will make your life much easier.

3.2 Suggested Reading and Reference

Horowitz and Hill

- 1.06 Small Signal Resistance
- 1.07 Sinusoidal Signals
- 1.09 Other Signals
- 1.11 Signal Sources
- 1.12 Capacitors
- 1.13: RC Circuits: V and I versus Time
- 1.14 Differentiators
- 1.15 Integrators
- 1.18 Frequency Analysis of Reactive Circuits
- 1.19 RC filters
- 1.25 Diodes

Hayes and Horowitz

Chapter 1: Overview

3.3 Pre-lab Questions

1. Derivation of the equation for a low-pass RC filter. In Chapter 2, we saw that the equation for a voltage divider is:

$$\frac{V_o}{V_i} = \frac{R_2}{R_1 + R_2} \quad \text{EQ 3.3.1}$$

where R_2 is the resistor in parallel with the load. For a pure sinusoidal input signal, this equation has a more general form that also applies to complex impedances:

$$\frac{V_o}{V_i} = \frac{\mathbf{Z}}{\mathbf{Z} + \mathbf{Z}_2} \quad \text{EQ3.3.2}$$

Here we use boldface text to represent complex numbers. A complex number is a vector in the complex plane. It has a *real* component ‘a’ and an *imaginary* component ‘b’ and is written ‘ $\mathbf{c}=a +jb$ ’ where j is the square root of -1 . Complex impedances result in a time delay between current fluctuations and the voltage fluctuations that cause them. We say that the current and voltage are *out of phase*, and can specify the amount that the current *lags* or *leads* the voltage. The real part of a complex impedance causes power dissipation while the imaginary part only causes a time delay. An ideal resistor has an impedance that is real; hence power is dissipated, and the current flowing through the resistor at any given time is always in exact proportion to the voltage across the resistor at that time. Imaginary impedances do not dissipate energy but withdraw it from the circuit and then release it back to the circuit.

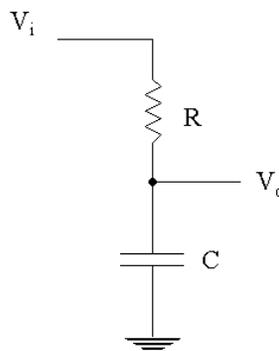


Figure 3.3.1: The Low-pass RC Filter

Figure 3.3.1 contains a schematic diagram of a low-pass RC filter, which is simply a frequency-dependent voltage divider. V_i is the input signal and V_o is the output signal. The component labeled C is called a capacitor. If the capacitor were replaced by a resistor, we would have a voltage divider just like in the first laboratory session.

A capacitor is a device that stores charge. The charge Q is proportional to the voltage difference V across the capacitor: $Q=CV$, where C is the capacitance of the capacitor, expressed in Farads (F). A Farad is a huge quantity and most of the capacitors that we will use will be less than 1 microFarad. Capacitor sizes are usually stated in

units of microFarads (μF) or picoFarads (pF) and there are many different types (Horowitz and Hill present a table in Section 1.12 summarizing the relative properties of each). The labeling convention is even more complicated than that of resistors and is explained in detail in Chapter 1 of Hayes and Horowitz. An ideal capacitor has an impedance of $Z_C = 1/j\omega C$ – its impedance is inversely proportional to the radial frequency (ω – radians /second) of the signal.

Using Ohm's law, derive Equation 3.3.1. Then, with your knowledge of capacitors and resistors, and Equation 3.3.1, show that the transfer function V_o/V_i for a low-pass filter is given by Equation 3.3.3. Note that in this equation, the quantities are not in bold – V_o/V_i is the ratio of signal magnitudes, or amplitudes, not complex (thus time-dependent) quantities.

$$\frac{V_o}{V_i} = \frac{1}{(1 + (\omega RC)^2)^{1/2}} \quad \text{EQ 3.3.3}$$

Hints: $j*j=-1$. Sometimes Z_C can be easier to deal with by multiplying both the top and bottom by j : $Z_C=-j/\omega C$ (now we can see that if $Z_C=a+jb$ then $a=0$ and $b=-1/\omega C$). Plug the impedances into Equation 3.3.3 then make the denominator real by multiplying both numerator and denominator by the complex conjugate of the denominator. The complex conjugate of $a+jb$ is simply $a-jb$. Take the magnitude of the entire function by multiplying the entire expression by its complex conjugate and taking the square root.

Note that as ω goes to 0 the ratio goes to 1, and as ω goes to infinity, the ratio goes to 0. Thus, the low-pass filter *passes* low frequency signals with minimal attenuation, hence the name .

2. Plotting the frequency response of an RC filter. In the MAE 221 laboratory you will use MATLAB frequently for plotting and curve fitting and this exercise will introduce you to MATLAB while demonstrating the behavior of a low-pass RC filter. During this lab session, you will build an RC filter then apply sinusoidal signals of different frequencies to determine its frequency response. For assessment of its performance, the best benchmark is the theoretical transfer function given in Equation 3.3.3. In this pre-lab exercise, you will plot Equation 3.3.3 and save the plot. In the laboratory, you will superimpose your collected data onto this plot. You will need to find a computer running MATLAB 6.1 (any in the lab will do) to conduct the following steps:

- a) In the laboratory, you will construct an RC low-pass filter using $R=10\text{k}$ and $C=0.015\mu\text{F}$ so it is appropriate to use these values now. We want to plot V_o/V_i vs. frequency (Hz, not rad/sec) because we will vary frequency in the lab. Start MATLAB either from the desktop icon or by clicking Start->All Programs->Matlab 6.1->Matlab 6.1. The window is divided into three sections: Launch Pad, Command History, and Command Window. The Command Window is where you can enter commands, and will be used extensively.
- b) **Creating the frequency vector.** To begin, we must generate the data to plot. MATLAB can handle continuous functions such as that given by Equation 3.3.3, but as a first exercise it is easier to use discrete data points stored in arrays or vectors. To create the graph, we must have two vectors. One will contain values for V_o/V_i , and frequency in the other.

We will first create a vector containing frequency values of interest. We want to observe the behavior of this filter for frequencies ranging between 1 Hz and 100kHz so we'll need a vector containing the values: 1, 10, 100, 300, 500, 700, 800, 900, 1000, 1100, 1200, 1500, 2000, 5000, 10000, 20000, 50000, 100000. The selection of these points will become evident later. The simplest way to define a vector is to explicitly declare each element as follows:

```
f=[1 10 100 300 500 700 800 900 1e3 1100 1200 1500 2000 5000 1e4 2e4 5e4 1e5]
```

Note that the vector is enclosed in square brackets and the elements are separated by a space. Exponential notation can be used where it is more convenient. Enter the above line into the command window. If you did not make any errors, MATLAB should have responded by reiterating the *row* vector that you have defined.

When you become comfortable with MATLAB, you may find this verbosity cumbersome and can switch it off by appending a semicolon (;) to each command.

- c) **Calculating the frequency response.** We must now calculate a value of V_o/V_i for every frequency value that we've specified in the vector f . First we must define the scalar variables R and C :

```
R=1e4
C=1.5e-8
```

Note that MATLAB is case sensitive. If you make a mistake entering data, just re-enter it. The new entity will over-write the old (as long as you give it the same name). Use the 'clear' command to delete unwanted variables (ie. `clear R`). Equation 3.3.3 contains the angular frequency ω , but in the lab it will be much easier to measure the frequency f in Hz, so when we define the function we will replace ω by $2*\pi*f$ so that we can use f as the independent variable:

```
V=1./sqrt(1+(R^2*C^2*2^2*pi^2*f.^2))
```

The dots specify an element by element operation. By writing $f.^2$, we square each element in f . The denominator of V is an array, and we invert each element individually by using the `./` notation. Note that this is not necessary with R and C and the constants 2 and π because they each have only one element so the operation is unambiguous. This may seem complicated, but when you are comfortable with the notation, you'll appreciate its efficiency.

Rather than using the '^' notation, we could have wrote ' $f.*f$ '. You may want to explore other ways of multiplying vectors in MATLAB. If two vectors are directly multiplied (ie. $w*w$), MATLAB returns an error; the operation is not defined. But if one vector is transposed (making it a column vector) then the operation is defined:

```
w*w'    -> scalar (this is the scalar product)
w'*w    -> matrix (following the rules of matrix multiplication).
```

- d) **Plotting the frequency response.** We now have the independent and dependent arrays: f and V . Now we can plot the the theoretical response. The syntax is very simple:

```
plot(f,V)
```

There it is! You may have noticed that this doesn't look much like Figure 1.59 in Horowitz and Hill. Our plot seems to have an asymptote on the vertical axis whereas the figure in the text shows the curve approaching the axis at a right angle. Our plot is in fact correct (assuming you entered the equation correctly) but the low frequency part is highly compressed compared to the image in Hill and Horowitz so we can't really see it. To show that our data agrees with Hill and Horowitz, we will plot the same data on a log-log plot, but first, let us finish this one by adding a title, and axis labels:

```
title('Frequency Response of Low-Pass Filter R=10k, C=0.015uF');
xlabel('Frequency (Hz)');
ylabel('Vo/Vi');
```

Now we are ready to generate the log-log plot. If we issue the command now, our existing plot will be overwritten with the new plot so we must issue the `figure` command to generate a new figure window. But first we should formally record the figure number of our first figure for reliable reference later. We do this by using the `gcf` function (get current figure), and assigning the returned value to the variable `LPlin` (the name of this variable is arbitrary, but should be something intuitive):

```
LPlin=gcf;
```

Since we have only one figure, this will return the value '1'. You can confirm this by entering the variable name: RClin.

To create a new window for the log plot, we could just issue the *figure* command by itself, then use the *gcf* command to assign its value to a variable. But we can do both steps in one line:

```
LPlog=figure;
```

A blank figure window should have popped up. This is now the current figure window so we can generate the log-log plot:

```
loglog(f,V);
```

This is called a Bode plot and should have approximately the same shape as Figure 1.60 in Horowitz and Hill. Now apply the same labels as you did for the linear plot. The syntax is exactly the same.

Before analyzing these figures, they should be saved because we'll need them later. From the 'File' menu on each figure, select 'Save As' and name each figure, storing it in your Documents and Settings directory.

The jagged line in the linear plot is simply a result of using discrete points rather than plotting the function directly. The elements of the frequency vector were chosen so as to make the curves as smooth as possible.

Associated with this circuit is the time constant, RC . RC has units of seconds. If we multiply by 2π and invert this value, we get a frequency of 1.06 kHz. By examining the Bode plot we can say roughly that frequencies below this value are left relatively unchanged, and that frequencies above this value are attenuated, with the output going to zero as the frequency goes to infinity. It is a low pass filter.

3. Design of a low-pass RC filter for a specific application. Before getting into the details of designing an appropriate low-pass filter, let's clarify the characteristics we need to achieve. The signal produced by a sensor can be considered the sum of a base signal (the data) with superimposed unwanted *noise* that typically contains higher frequencies than the base signal. We want to attenuate the noise without distorting the base signal. It therefore makes sense to talk about the *signal to noise ratio (SNR)*. This is simply the ratio of the amplitude of the base signal to the amplitude of the noise. Ideally we'd like this to approach infinity. It won't. What we hope to achieve is to increase it enough so that the features of our primary signal can be clearly resolved. To accomplish this, we must attenuate the noise much more than the signal of interest, and in designing the filter, it is essential to develop an understanding of the behavior of the low-pass filter as we have done in questions 1 and 2.

Observing the Bode plot that you've generated, note that the output amplitude begins to rapidly drop near an angular frequency (ω) equal to about $1/RC$ radians/second. It would therefore seem that to obtain the best SNR, we should design a filter such that our signal frequency is adjacent to that point. This would be a good choice if the signal was a simple sinusoid containing only one frequency; however, most signals contain more than one frequency, and if $1/RC$ is too low, we may attenuate some of those frequencies more than others, resulting in a distorted signal. **A good rule of thumb therefore is to build a filter such that the primary angular frequency (ω) in your signal is about $1/2RC$ (or $f=1/4\pi RC$).**

Now that the goal is defined, we can discuss how to achieve it. First, choose R . Recall that an RC filter is just a frequency-dependent voltage divider, and that in our derivation of the behavior of a voltage divider, we neglected any load applied to the output. In other words, the input impedance of the load (or data acquisition instrument) must be much larger (by at least a factor of 10) than the output impedance of the filter. For the filter, at very low frequencies, Z_C goes to infinity and the output impedance is R . At higher frequencies, the output impedance is

less since Z_C decreases. Therefore, **R must be much less than the input impedance of the measuring instrument.**

There is a second requirement on R. To show what it is, consider the input impedance of the filter. Assume that the data acquisition instrument has an infinite impedance so that for the filter the input impedance is $Z_{in} = R + Z_C$ (R and C are in series so we add the impedances directly). At low frequencies, Z_C is large, so the input impedance is least at high frequencies, but is never less than R. If the filter's input impedance is too low, the signal transducer will generate large currents which may damage the transducer (the device generating the signal) or distort the signal. To avoid such problems, **make sure that $I_{max} = V_{transducer, max}/R$ is greater than the maximum current capability of the signal transducer** (this information can usually be found in the transducer's documentation). Our signal will be generated by the LabVIEW DAQ card which has a maximum current of 5mA and maximum output voltage of 10V.

Having determined RC and R, C is therefore defined. However, this is not as trivial as it may sound because there is generally a range of possible values for R, which leaves some room for optimization. Also, at lower frequencies, (say <100 Hz), the product RC may be large, and you will find that large capacitors (ie. 100's of μF) can be difficult to find.

Using the above information, choose a suitable resistance and capacitance for a low-pass filter that will be used to attenuate noise at a frequency of 1kHz without distorting the base signal at 100Hz. Note that resistors and capacitors come in discrete values so you should stop by the lab and look at the selection to see what choices you have. In the laboratory session, you will construct the circuit that you design, and test its performance on the specified signal. You can also exchange components and observe the effects.

Although you will be working with a partner in the lab, you must design your own filter and test it in the lab.

4. Integration of a square wave. During the laboratory, you will construct a circuit that integrates a waveform – the output signal will be the integral of the input signal with respect to time. Using a square wave input such as that shown in Figure 3.3.2, you will be able to test the integrator, and learn its limitations. The performance of an integrator is assessed by comparing the generated output to the theoretical output. Copy the square wave signal from Figure 3.3.2, then calculate and draw the theoretical integrated signal on a set of axes beneath the square wave.

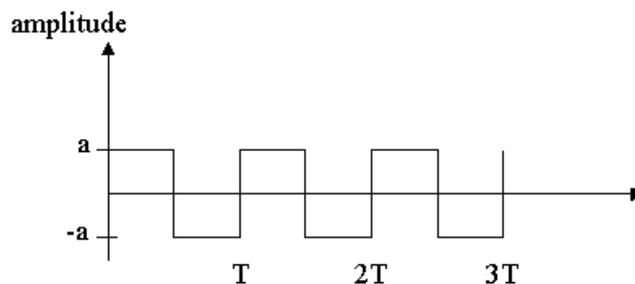


Figure 3.3.2: A Square Wave of Amplitude a and Period T

5. Calculation of the transfer function of a linear amplifier. The linear amplifier is a very fundamental and important op-amp device. You will encounter it frequently in this course so it is worthwhile spending some time understanding how it works. It is also a good place to begin studying operational amplifier circuits.

Important notes about op-amp behavior:

The schematic symbol for an operational amplifier is shown in Figure 3.3.3. It consists of a non-inverting and an inverting input (+ and -), an output (V_o), and two power supply inputs (V_{CC} and V_{EE}). The op-amp is a differential amplifier with very high gain. More specifically,

$$V_o = G*(V_+ - V_-)$$

EQ 3.3.4

where $G > 10^5$ is the gain, and V_+ and V_- are external voltages applied to the inputs. Therefore, when V_+ exceeds V_- , the output voltage is positive, and when V_- exceeds V_+ , the output is negative. However, the output voltage cannot be greater than V_{CC} or less than V_{EE} , and because of the high gain, small voltage differences between the two inputs quickly *saturate* the output, meaning that the output goes to V_{CC} or V_{EE} . As the difference $(V_+ - V_-)$ increases, the gain is reduced (G is not constant), making the op-amp a nonlinear device. These properties of nonlinearity and rapid saturation make the op-amp suitable for only a few applications on its own, but the real strength of op-amps is achieved when they are used in feedback loops. When feedback is used, the output is in some way connected to the inverting input (V_-) creating a *negative* feedback loop. If the output were connected to the non-inverting input (V_+), the circuit is usually unstable (positive feedback), and is not as common.

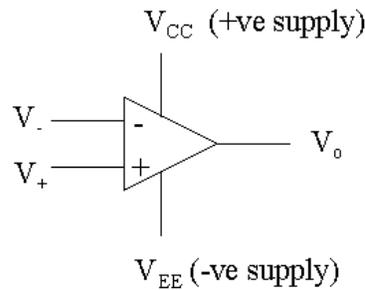


Figure 3.3.3: An Operational Amplifier

Non-inverting and inverting linear amplifiers are shown in Figures 3.3.4 and 3.3.5. A linear amplifier may be characterized by its gain, g , ie. $V_o = gV_i$. As we will see, because the op-amp is used in a negative feedback circuit, the gain is constant and therefore the device is linear. In this case we will only consider the inverting amplifier due to its frequent use in electronic circuits. We want to calculate the gain (or V_o/V_i) based on the values of the resistances R_i (the input resistance) and R_f (the feedback resistance).

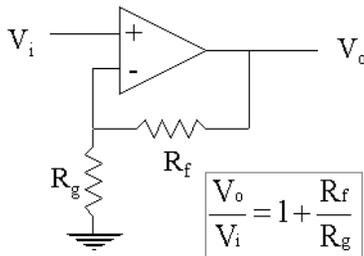


Figure 3.3.4: A Non-inverting Linear Amplifier

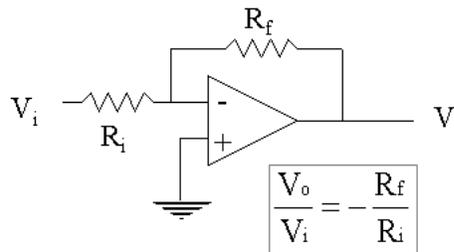


Figure 3.3.5: An Inverting Linear Amplifier

The impedance of the op-amp inputs is very high ($> 10^6 \Omega$) so in most cases we can make the accurate assumption that **the inputs draw no current**. This assumption will be valid as long as the impedance of the feedback loop is less than $\sim 10\%$ of the input impedance. The output impedance of an op-amp is orders of magnitude lower – usually 100Ω or less.

To understand the effects of feedback, first consider the inverting amplifier with the feedback loop broken, as shown in Figure 3.3.6.

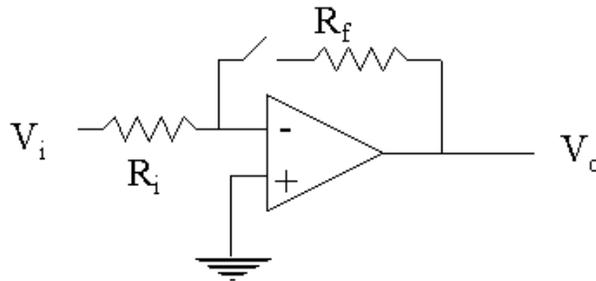


Figure 3.3.6: The Inverting Amplifier with Feedback Removed

It is then clear by Equation 3.3.4 that a positive V_i will cause V_o to achieve its negative saturation voltage because of the large (often idealized as infinite) gain. We assume that no current flows through R_i . Now if we re-connect R_f , a current is generated through R_i and R_f (which are in series):

$$I = \frac{V_i - V_o}{R_i + R_f} \quad \text{EQ 3.3.5}$$

There is now a voltage drop through R_i which reduces V_- . By this mechanism, V_- is brought to the same level as V_+ . In other words, **the output attempts to do whatever is necessary to make the voltage difference between the inputs zero.**

Equation 3.3.4 above, and the two ‘golden rules’ stated above in bold are everything you will need to know about op-amps to analyze op-amp circuits in this laboratory. Using this information and your knowledge of circuit analysis, show that the gain for the inverting linear amplifier shown in Figure 3.3.5 is:

$$g = \frac{V_o}{V_i} = -\frac{R_f}{R_i} \quad \text{EQ 3.3.6}$$

3.4 Laboratory Activities

3.4.1 A look at nonlinear components: I-V Characteristics.

What is the voltage drop across R_1 and R_2 of Figure 3.4.1(a)? You have several ways to determine this: One is by Ohm's law - divide the applied voltage by the sum of the resistances to calculate the current, and then multiply the current by each resistance. Another method is by observation: the power supply is 6 volts and the resistors sum to 600Ω , so R_1 must be 4 V, and R_2 must be 2 V.

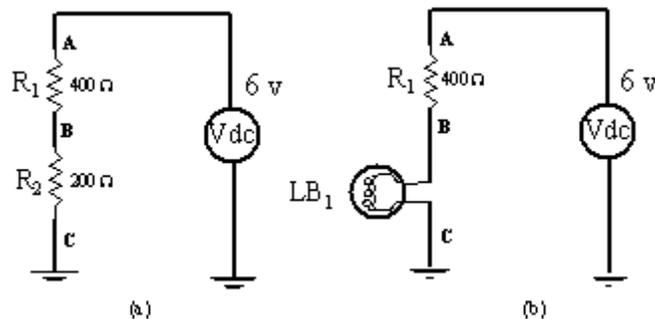


Figure 3.4.1: Resistor and light bulb series circuits.

Now what is the voltage drop across R_1 and the light bulb of Figure 3.4.1(b)? The first question you might ask is “what's the resistance of the light bulb?” The problem is, unlike a resistor, it is not constant.

mostly with resistors, which behave linearly under all conditions. If you double the voltage through a given resistor, the current through that resistor also doubles. But many electronic components do not behave linearly. If you double the voltage through a light bulb, the filament burns hotter which increases the resistance of the circuit, so the current may actually increase only a small percent. This makes predicting circuit parameters difficult unless we know the relationship between the voltage applied across a circuit element and the current that flows through it. In this exercise, we will determine the relationship between current and voltage for a light bulb and a diode. These are called I-V characteristics.

We begin by putting the component of interest into a voltage divider, as Figure 3.4.2 shows. A voltage is applied across the resistor and component, generating a current. If we know V_a and V_b , then the current through the resistor and the component is simply $(V_a - V_b)/R$. The voltage across the component is simply V_b . Therefore, by generating various levels of V_{applied} and recording V_a and V_b for each level, we can trace out the I-V characteristic of the component of interest.

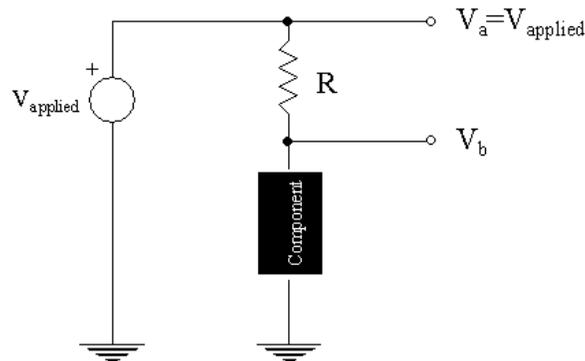


Figure 3.4.2: Circuit for Generating I-V Characteristics

The applied voltage will be generated by the program **ivtracer.vi**. It will also read V_b , calculate the current, and plot the I-V characteristic on the screen. You can find it in the MAE 221 Software folder on the desktop. Although you are not expected to write LabVIEW code of this level, it is a good example of what LabVIEW can do. Open the program. The documentation in the yellow boxes will lead you through the items on the panel and the diagram, including some short exercises. Go through this before continuing. In your notebook you can record the names of sub-vi's used, or any other information that you think may help you with LabVIEW programming.

Let's start with the light bulb:

1. The supply voltage will come from the Analog Output of the DAQ board; however, the light bulb draws more current than the DAQ board can safely provide so the current must be amplified. In the next lab we will build our own current amplifiers, but for now, we will use the one provided in the break out box. The circuit that you must construct is shown in Figure 3.4.3. The positive (red) and ground (black) terminals of the Analog Output Channel 0 must be connected to the Power Amplifier Input as the figure shows. The amplifier output, labeled '5 Amps Max', will then deliver the same signal, but is capable of generating 1000 times the current. We don't need nearly that much, but the 5 mA of the DAQ board is not sufficient. The power supply will power the current amplifier and can be connected via the 'Power Supply' terminals directly above the DAQ data cable (this is not shown in Figure 3.4.3). Ensure that the power supply shunts are correctly positioned so that the power supply can provide positive and negative voltages from the two supplies. Then set the supplies to produce +15V and -15V. Turn off the power supply and connect it to the breakout box. Make sure the ground terminal is also connected to a ground terminal on the power supply.

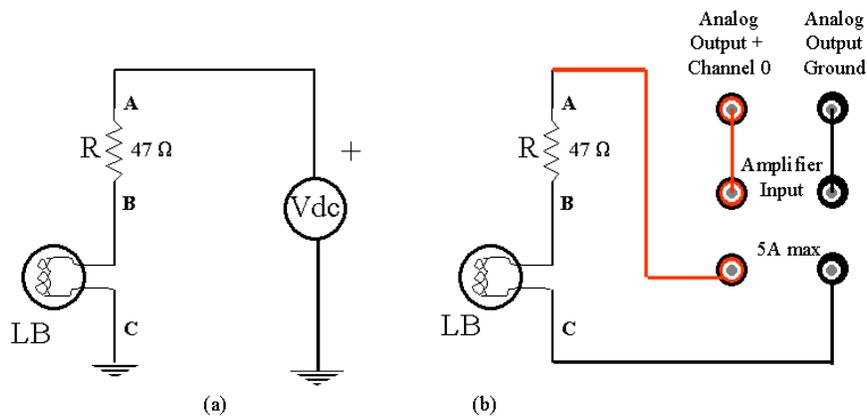


Figure 3.4.3. Light bulb schematic, and the actual connections.

2) Ivtracer.vi will send incremental voltages to the circuit starting at 0 volts, and ramping up to 10 volts. When it reaches 10 volts, the program will send out a zero volt signal (to depower the circuit) and stop the program.

To measure the current flowing through the bulb, the computer will subtract the voltage at B from the output voltage and divide by the value of R . The voltage across the light bulb will simply be the voltage at B. Therefore we only need to measure the voltage at B. We will do this using Analog Input Channel 0. Figure 3.4.4 shows how this is connected to the light bulb circuit. Complete the wiring and you will be ready to generate the I-V characteristic using LabVIEW.

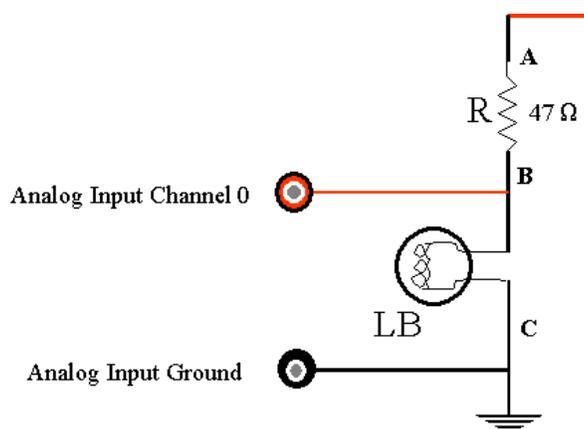


Figure 3.4.4: The analog input connection

3) Turn the power supply on (it should be connected to the current amplifier in the breakout box). Make sure that the **resistor value** on the panel of ivtracer is set to 47 ohms. Press the run button to start the program. As the voltage level increases, the light bulb should begin to glow. The program will generate a plot of current vs. voltage in real time. Notice that the curve becomes less steep with increasing voltage – the resistance is increasing as the filament gets hotter. This is, in general, true for metals. We increase the voltage slowly so that the filament is in a state of quasi-equilibrium. If the voltage were increased quickly, the shape of the curve would be different because the temperature rise would lag the current. Copy and paste the output graph into another document and print it. You can paste the printout in your notebook.

4) Repeat the experiment with a large LED. To limit the current, you should increase the series resistance to 220Ω , also changing the **resistor value** control to match. Note that at low voltage the resistance is essentially infinite, but abruptly changes and approaches 0 as the voltage rises. This is why it is critical that LED's and other diodes have a resistor in series. Since the impedance of the LED is not dependent on temperature, it is not necessary to increment the voltage so slowly, but for convenience, we will anyway.

5) Replace the LED with a 100Ω resistor, and repeat the experiment. We have assumed resistors to be ideal, ie. constant R. What does the output tell us?

3.5.2 Removing noise from a signal – the RC low-pass filter.

The ultimate goal of this activity is to design and build a low-pass filter to attenuate random fluctuations on a signal created by a tachometer. These fluctuations, or *noise*, are a result of the imperfect nature of the tachometer's construction and can obscure the signal. Therefore they must be removed. If the frequency of the noise is significantly higher than the frequency of the signal you want to measure, the low-pass filter will do a good job of attenuating the noise without distorting the signal. We will begin with some fundamental exercises.

a. Experimentally testing the response of a low-pass filter. In the second pre-lab question, you plotted the theoretical frequency response of a low-pass filter with $R=10k$ and $C=0.015\mu F$. Construct this filter on your trainer board and provide the input signal with the function generator on the trainer. Set the amplitude at about 3V. Test at least 5 or 6 frequencies between 10 Hz and 100kHz, with at least 3 near 1kHz. Use the oscilloscope to record both input and output signals and record these with their respective frequencies.

You can measure voltage amplitudes by counting squares on the oscilloscope display, by using cursors, or by simply configuring the oscilloscope to calculate and display the amplitude. These techniques are all described in the document, 'Using the Tektronix TDS 210'. You can find it on the course website and in the 'MAE 221 Lab Reference' binder on your bench.

When you have collected your data, you must superimpose it on the theoretical Bode plot that you constructed in the second pre-lab question. To do this, execute the following steps:

- i. Start MATLAB.
- ii. Create two new arrays containing your experimental frequencies (f_{exp}) and output amplitudes (V_o). Note that each element in f_{exp} should correspond to the same element in V_o . Define a scalar V_i with the value of the input amplitude.
- iii. The required data is the transfer function of the filter, so divide each element of V_o by V_i and put this into a new array, V_{exp} .
- iv. Open the Bode plot using 'File -> Open'. Find and select your Bode plot (the filename extension will be .fig).
- v. This will be the current figure window, but anything plotted to it will overwrite the current data, so the `hold` command must be issued to prevent modification of the existing data:

```
hold on
```

- vi. Now you can superimpose the experimental data. You can this by issuing the `loglog` command with f_{exp} and V_{exp} as arguments. However, if you use the same syntax as before, you will get a continuous line. Since these are discrete data points, we want to plot symbols that are not connected. To do this, we must include the 'linetype' parameter. Our choices of symbols are: point (.), circle (o), x-mark (x), plus (+), and star (*). We can also use a different color by preceding the linetype character with a letter representing the color. Choices include y,m,c,r,g,b,w, and b. They are the first letter in the spelling of the color s they represent. You can try them to find out what they are. For example, to plot the points as red pluses, you would enter:

```
loglog(fexp,Vexp,'r+')
```

If you make a mistake in your plotting and want to redo it, you can delete a line or set of points by selecting the 'edit plot' (arrow) tool in the plot window then clicking on the data to highlight it. Pressing the 'delete' key will remove it.

- vii. You should save and print a copy of this plot. Paste this plot and your theoretical linear plot in your notebook. After completion of this lab, you will be required to generate a reference sheet on RC filters. You will need to include these plots in that document so you may want to **export** them as bitmap (.bmp) files (some other formats do not work well).

b. Attenuating a high-frequency signal superimposed on a low frequency signal. In the pre-lab exercises, you designed an RC filter to filter noise consisting of a 1 kHz sinusoid from a 100 Hz sinusoidal base signal. Now you will have an opportunity to test it. Begin by opening the LabVIEW program **Sine Generator.vi**. This program generates the noisy signal. The front panel contains three charts, displaying the base signal, the noise, and the base+noise. When you run the program, the combined signal will appear on analog output channel 0. Connect channel 1 of your oscilloscope to the analog output and run the program. Adjust your settings so that you can see 2 or 3 wavelengths of the base signal on the display. The trace should look very thick. Now increase the scan rate (SEC/DIV) until you can resolve a few wavelengths of the noise on the display.

Stop the LabVIEW program and construct the filter that you designed. Connect it to the breakout box and run the VI again, observing the filtered and unfiltered signals on the two channels of your oscilloscope. By adjusting the scan rate, you can focus either on the base signal or the noise. Measure the amplitude of each, recording your observations in your notebook. See if you can improve the performance by changing components.

Demonstrate your working filter to an instructor.

c. The tachometer: removing noise from the signal. In this exercise we will take a brief look at a DC motor and a tachometer for the purpose of generating a noisy signal. In the next lab, we will investigate them more formally. As we will see, motors and generators are fundamentally the same, and a tachometer is simply a generator. A DC motor converts a DC current into mechanical motion; whereas a generator converts the mechanical motion into a current. In the case of a tachometer, the output voltage is related to its speed of rotation, and therefore by measuring the output voltage of the tachometer, we can determine the rotational speed of the device to which it is connected. The problem is that our tachometer generates a noisy signal that needs to be filtered before sending the output to another device. In this exercise, we'll use a motor/tach – a motor coupled directly to a tachometer – to generate rotational motion and measure the rate of rotation.

1. You should have found a motor/tach at your workstation when you arrived. It has two sets of leads: the ones emerging from the side supply power to the motor, and the two at the end carry the output signal of the tachometer. With the power supply turned off, connect the motor leads to the 5V fixed supply terminals. Note the polarity of the motor leads.
2. Connect one of the oscilloscope probes across the tachometer output leads and set up the oscilloscope to read 0 to about 10 volts.
3. Turn on the power supply. The motor will spin and the oscilloscope will display a noisy signal with a nominal level of about 7 volts.
4. Your task is to design and build a low-pass filter that will remove as much of the noise as possible. Only the DC signal is desired. Use the SEC/DIV knob to adjust the scan rate so that you can estimate the noise frequency. There may be multiple frequencies. Record the results in your lab notebook.

d. The low-pass filter as an integrator. In the first pre-lab question, you determined the response of the low-pass filter as a function of frequency. If we recognize that the current through the resistor and capacitor at any given time is the same, we get the following time-dependent equation:

$$C \frac{dV_o}{dt} = \frac{V_i - V_o}{R}. \quad \text{EQ 3.4.1}$$

Integrating both sides and rearranging, we get:

$$V_o(t) = \frac{1}{RC} \int (V_i - V_o) dt + \text{constant.} \quad \text{EQ 3.4.2}$$

So when V_o is small compared to V_i , the output is the integral of the input signal with respect to time. This is typically true for periodic input signals when the frequency is greater than $1/2\pi RC$. Using any of your filters, confirm that the circuit acts as an integrator by putting a square wave into the input, and observing the output signal. Use three different frequencies: one approximately equal to $1/2\pi RC$, and frequencies approximately half and double that frequency. Adjust the signal amplitude. Does this change anything? Record all of your observations in your notebook.

e. Introduction to the high-pass filter. If we take the low pass filter and interchange the resistor and capacitor, then low frequency signals are greatly attenuated and higher frequency signals are less so. We call this the *high pass filter*. In the same way as you have done, it is easily shown that the transfer function for the high pass filter is:

$$\frac{V_o}{V_i} = \frac{1}{[1 + (1/\omega^2 R^2 C^2)]^{1/2}}. \quad \text{EQ 3.4.3}$$

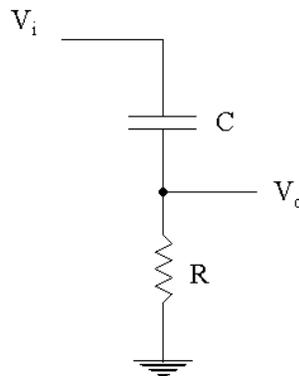


Figure 3.4.5: A high-pass filter

Build a high pass filter with $R=10k$, $C=0.0022\mu F$. Obtain a few experimental data points for this filter and confirm that they satisfy Equation 3.4.3.

The high-pass filter will be revisited in the following lab where it will be used to view *ripple* (noise) on a DC power supply. The filter will remove the DC component, allowing us to greatly amplify the remainder of the signal to better estimate the magnitude of the noise.

3.5.3 The voltage follower – a better solution to the input impedance problem.

For the remainder of the lab, we will focus on operational amplifiers. In Chapter 2, we found that if two identical voltage dividers are connected in series the voltage ratio of the combined circuit is not the product of the original two ratios. We reduced the effect of loading by reducing the output impedance of the first voltage divider and increasing the input impedance of the second one (the load). However, we used small resistances in the first voltage divider resulting in increased current, and hence power dissipation. In this exercise, we will re-address this problem, and solve it with the use of a voltage follower.

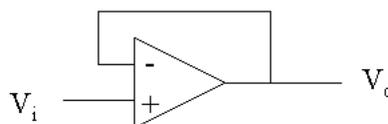


Figure 3.4.6: The voltage follower

A schematic of the voltage follower is given in Figure 3.4.6. The input voltage is applied to the non-inverting input and the output is connected to the inverting input.

Recall that in Chapter 2 we required that for a voltage divider to obey the voltage divider equation $V_o/V_i = R_2/(R_1 + R_2)$, the load must have a high input impedance compared with the output impedance of the voltage divider. If it does not, we can separate the voltage divider and its load by inserting a voltage follower as shown in Figure 3.4.7. The voltage divider will then see the input impedance of the op-amp, which is very large. Because of the feedback loop, the voltage follower has essentially no output impedance as long as the output does not saturate, and can drive much more current than the voltage divider.

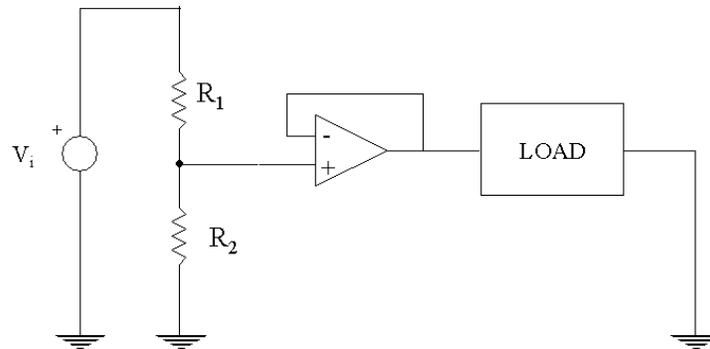


Figure 3.4.7: An application of the voltage follower

In this lab we will use the UA747CN dual operational amplifier. On the course web site and in the Lab Reference binder, you will find a data sheet for the 747 op-amp. Data sheets contain much useful information. The first page will generally contain a diagram showing the pin connections. Most important are the inputs (pins 1 and 2, 6 and 7), the outputs (pins 12 and 10), and the positive and negative power supply pins (pins 9 or 13 and 4). There are four inputs and two outputs because this chip actually has two op-amps in it. The inputs and outputs are independent, but both use the same power supplies. Further into the document, you should find the ‘absolute maximum ratings’. If you exceed these, you will damage the op-amp. Electrical characteristics are listed, such as the input and output impedances and maximum output current. There are also graphs and tables showing output characteristics. One of the most important is the *slew rate*, which is the rate at which the op-amp can change its output voltage. Its units are $V/\mu s$. There are usually also some sample circuits which demonstrate how to connect the device to perform common tasks.

To begin to understand operational amplifiers, we will first build a voltage follower. Figure 3.4.8 shows the 747 connections. The +15 and -15V supplies come from the power supply and are referenced to the same ground as the input and output signals shown on the figure. Create an input signal with the function generator on your trainer. View both the input and output signals with the oscilloscope. Change the frequency and amplitude and ensure that the output signal is the same as the input. Now obtain four 1k resistors and construct two identical voltage dividers as in Figure 3.4.7 (the load is the second voltage divider). Ensure that the circuit now divides by four.

Demonstrate your /4 voltage divider to an instructor.

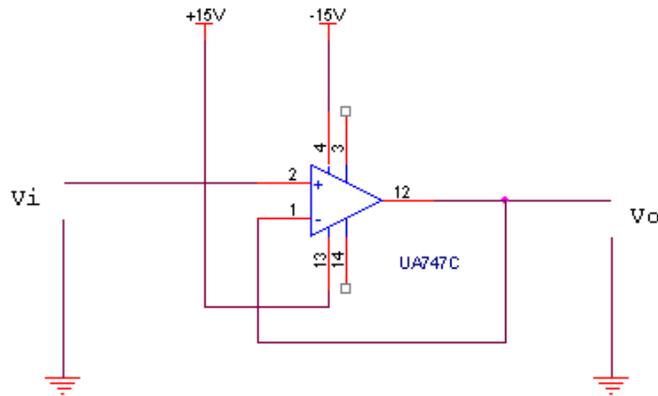


Figure 3.4.8: Construction of the voltage follower

3.5.3 The linear amplifier – Amplifying small signals.

Many of the sensors that you will encounter in the MAE 221 lab produce tiny data signals (voltages). Somehow these tiny signals must be stored and processed in order to extract meaningful data. Digital computers are often used for this purpose, but in order to do so, the tiny analog signal must be converted to a digital representation using an analog to digital converter (ADC). This is precisely what is done when we acquire data using LabVIEW. A typical ADC has a specified input range, say 0 to 10V. It will read an analog voltage value and convert it into a binary number of a given number of digits, or *bits*. For example, if we have a 12 bit ADC with an input range of 0 to 10 volts, it is capable of distinguishing $2^{12}=4096$ discrete voltage levels between 0 and 10 volts. This means it has a resolution of $10V/4096=2.4$ mV. However, the analog signal that we wish to convert may only vary between 0 and 0.1 V. We are only using one hundredth the input range of the ADC, and as a result, our digital signal will only contain about 40 distinct voltage levels. We have paid a lot of money for our 12-bit ADC but we are only getting about 6 bits worth of data! To improve the resolution of our digital signal, we must amplify the analog input signal by a factor of 10 so that it spans the entire input range of the ADC.

Begin by constructing an inverting linear amplifier with a gain of 10. Make sure that the input impedance is at least 1k, and that the resistance of the feedback loop is much less than the input impedance of the op-amp. Create a small amplitude sinusoidal signal with the function generator on your trainer. Amplify the signal with your amplifier and observe the initial and amplified signals on the oscilloscope. Ensure that the gain of your amplifier is correct, and that the amplified signal is not distorted.

A cadmium sulfide photo-resistive cell is a two terminal device with resistance dependent on the intensity of light incident on its surface. We will look at these devices in more detail in Lab 4, but for now, we will use one to generate a small signal that we can amplify. This type of cell is only a resistor so we will use the power supply to provide power for the signal, then put the cell in a voltage divider so that the output is inversely related to the light intensity. To prevent loading, our signal source will be isolated using a voltage follower. An inverting linear amplifier will then be used to amplify the signal by a factor of 10. The circuit is shown in Figure 3.4.9.

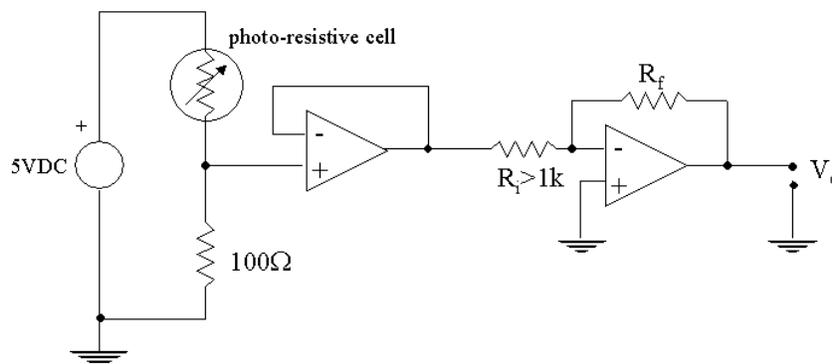


Figure 3.4.9: An amplifier application

First build only the voltage divider and look at its output on the oscilloscope. Then build the circuit in Figure 3.4.9 and observe the output on the oscilloscope while you wave your hand over the detector. Note that the 747 is a dual op-amp so you can build the entire circuit with only one chip.

Demonstrate your linear amplifier circuit to a lab instructor.

3.5.4 The differential amplifier: amplifying the difference between two signals.

The linear amplifier amplifies the level of its input signal by a factor determined by the values of the two resistors (see Figures 3.3.4 and 3.3.5). Stated differently, the linear amplifier amplifies the difference between V_i and ground. However, in many applications we need to amplify the difference between two arbitrary signals. If, rather than connecting the non-inverting input of the inverting amplifier (Figure 3.3.5) to ground, we connect it to the output of a voltage divider as in Figure 3.4.10, we have a differential amplifier.

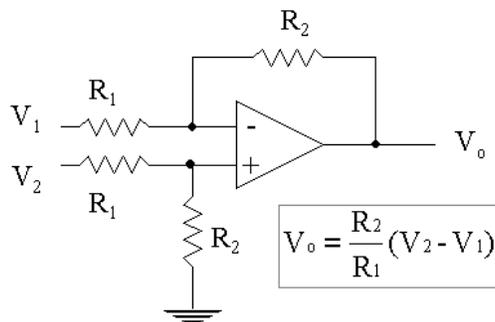


Figure 3.4.10: Differential Amplifier

To analyze the differential amplifier, first consider the voltage divider on the non-inverting (+) input. Since the op-amp controls its output such that $V_- = V_+$, we have:

$$V_- = V_+ = \frac{R_2}{R_1 + R_2} V_2.$$

And because the inputs draw no current, the current is the same in the upper R_1 and R_2 :

$$I = \frac{V_1 - V_-}{R_1} = \frac{V_- - V_o}{R_2}$$

Combining these two equations and manipulating, the transfer function for a differential amplifier emerges:

$$V_o = \frac{R_2}{R_1} (V_2 - V_1) \quad \text{EQ 3.4.4}$$

Figure 3.4.11 shows an application of the differential amplifier: a DC motor speed controller. You will build this circuit to demonstrate the operation of a differential amplifier. The input signal will be generated by the 5V fixed power supply. One input of the differential amplifier will take 2.5V from the discrete voltage divider (R_3) and the other input will be variable, controlled by a pot. Both signals will pass through voltage followers to prevent loading on the voltage dividers. Choose R_1 and R_2 to produce a gain of about 5. Make sure the values that you choose are not too low so that current is excessive (keep the op-amp output current below about 5 – 10 mA), and also make sure that the resistances are not so large that they are on the same order of magnitude as the op-amp input impedance. Use +15V and –15V supplies on the op-amp. Before connecting the motor, look at the output

voltage with your oscilloscope. Then connect a low-current motor supplied by a lab instructor. Demonstrate control of motor speed in both the forward and reverse directions.

Demonstrate your working motor speed controller to an instructor.

Before continuing, take a different motor – one that requires higher current - also supplied by the lab instructor, and replace the present motor with it. Try to run it. It does not turn because the op-amp is not capable of generating enough current. In the next lab, we will build current amplifiers that will solve this problem.

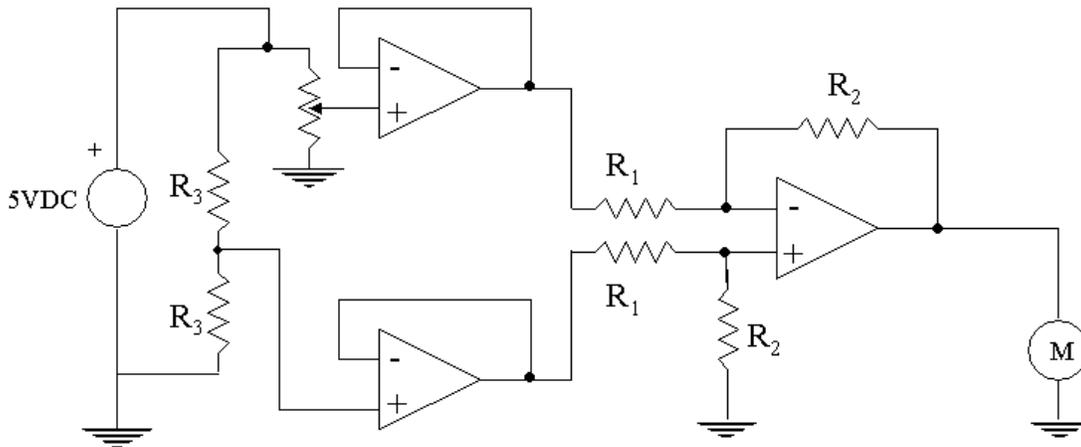


Figure 3.4.11: Motor speed control using a differential amplifier

3.5.5 Integrators and differentiators: Velocity to displacement and displacement to velocity

If either the input or feedback resistor in an inverting amplifier is replaced with a capacitor, the result is an integrator or differentiator. Figures 3.4.12 and 3.4.13 show the circuits and the equations that describe the output signal as a function of the input.

$$V_o = -\frac{1}{RC} \int V_i(t) dt + \text{constant}$$

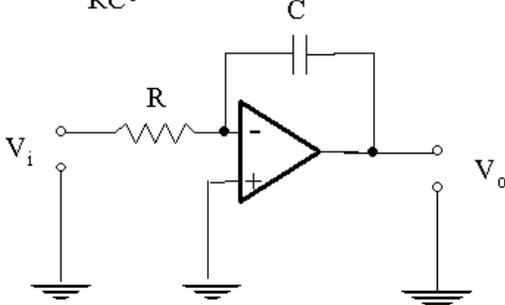


Figure 3.4.12: An integrator

$$V_o = -RC \frac{dV_i}{dt}$$

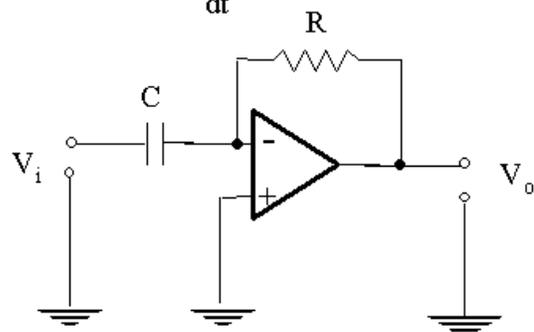


Figure 3.4.13: A differentiator

Analysis of these circuits is straightforward if we recall that a capacitor stores a charge given by $Q=CV$, where V is the voltage across the capacitor. Differentiating the equation with respect to time, we obtain an equation for the current through a capacitor:

$$I = \frac{dQ}{dt} = C \frac{dV}{dt} \tag{EQ 3.4.5}$$

The current through the resistor and capacitor must be the same if the input draws no current so:

$$\frac{V_i}{R} = -C \frac{dV_o}{dt}.$$

Integrating, we obtain:

$$V_o = -\frac{1}{RC} \int V_i(t) dt + \text{constant} \quad (\text{integrator}) \quad \text{EQ 3.4.6}$$

For the differentiator, the analysis is similar. Equating the current through R and C and rearranging the variables gives:

$$V_o = -RC \frac{dV_i}{dt} \quad (\text{differentiator}) \quad \text{EQ 3.4.7}$$

In this final exercise, build an integrator and convert a square wave into a triangle wave as you did with the RC circuit. An integrator can be used to convert a velocity signal into a displacement signal. For example, you could integrate the output of a tachometer to obtain the angular displacement. However, for a dc signal like that the op-amp would soon saturate so it could not be used indefinitely. Likewise, a differentiator will convert a triangle wave into a square wave or could be used to convert a displacement signal into a velocity signal. If a differentiator and an integrator were placed in series, the output would be the same as the input – perhaps with the exception of a factor and a constant offset.

Demonstrate your integrated square wave to an instructor.

3.5.6 Clean Up

Again, before leaving, make sure that all tools and components are put in their proper places and lab instruments are turned off. Do not put broken components back in their containers – throw them out. You may desire to keep voltage followers or linear amplifiers assembled. Remove your pinboard from the trainer, label it with your name(s) and store it in the cabinet at the side of the lab.

3.5.7 Generation of the technical reference sheets

This week you will be required to produce two reference sheets; one for RC filters and one for op-amp circuits.

For filters, include all relevant plots, equations, and diagrams.

For op-amp circuits, discuss some important parameters of op-amps, citing the 747 in particular. Be sure to discuss voltage followers, linear amplifiers, differential amplifiers, integrators, and differentiators.