Electronics Techniques

Laboratory Exercises

G. O. Sitz

Contents

List of Figures

Chapter 1

Lab 1: Basic Measurements and Oscilloscope Use

Introduction In this lab you will learn how to take basic AC and DC measurements using an oscilloscope and a multimeter, as well as how to use a function generator and a DC power supply. You will also build a basic voltage divider, observe the rise time of a square wave and measure the RMS voltage of various waveforms.

Helpful Info The write ups to be handed in for this lab are meant to be written like homework assignments. Number each problem and include whatever is necessary to answer it. This can include collected data, plots, written sentences, etc. Information to include in your answer to each problem is listed as FYR (for your report), which is what you will be graded against. Report due dates are given on the syllabus.

1. Using a few wires and the multimeter, determine the overall layout of the connections within the breadboard. You can plug the wires into the holes and use the multimeter on the buzzer setting. It looks like this: ●)). Whenever there is a connection between the leads of the multimeter on this setting, it buzzes. FYR: make a simple diagram of the connections inside the breadboard.

2. Output Impedance

   (a) On your breadboard, construct the open circuit as shown in Figs. 1 & 2. Fig. 2 is drawn for pedagogical purposes: it is merely a different (and probably more familiar) way to draw the circuit shown in Fig. 1. Both Fig. 1 and Fig. 2 represent the same physical circuit in reality. The "gap" drawn in both diagrams represents the absence of a wire or any other circuit element. The circuit is literally "open", in the sense that it is not completed and no current flows through it. Set V_{in} to be +5 V using the red (+) terminal of the DC power supply (or the voltage available on the breadboard). The black (-) terminal will serve the purpose of GND. Measure the open circuit output voltage by measuring the voltage difference between V_{out} and GND. Make sure your multimeter is not on AC/Cap mode. Keep in mind that a volt meter is a very large impedance device (and therefore will let very little current flow through it), so touching the leads to V_{out} and GND will not affect the "openness" of the circuit.

   (b) Construct the closed circuit as shown in Figs. 3 & 4. Measure the short circuit output current by measuring the current flowing through the wire connected between V_{out} and GND. Keep in mind that an ammeter is a tiny impedance device, so it "takes the place" of a wire between V_{out} and GND. It will not affect the "shortness" of the circuit.

(c) The output impedance of a circuit is defined as the ratio of the open circuit output voltage to the short circuit output current. Calculate this. FYR: record both measurements and show your calculation of the output impedance. Does this value make sense when you look back at your circuit?

3. Using the setup from Prob. 2, construct the voltage divider as shown in Fig. 5. Measure V_{out}. Keep in mind that the volt meter is held 'across' the 10k resistor. Calculate what V_{out} should be using the voltage divider formula. FYR: record both the measured and calculated values for V_{out}. Are they similar?

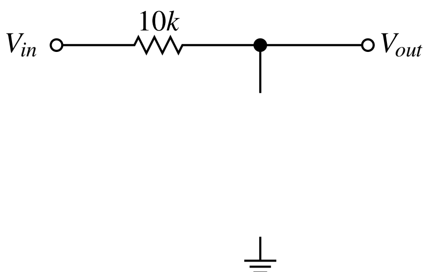

Figure 1.1: Open Circuit.

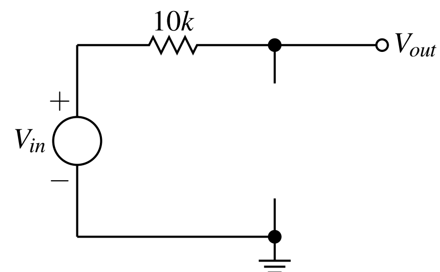

Figure 1.2: Open Circuit. This diagram uses a different convention to represent the same physical circuit drawn in Fig. 1.

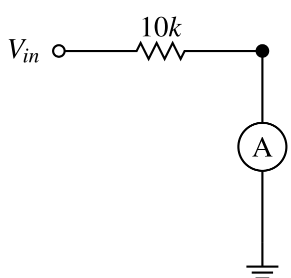

Figure 1.3: Closed Circuit.

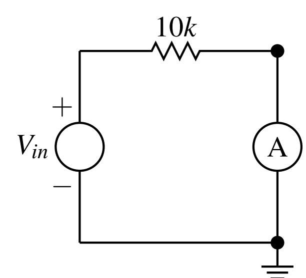

Figure 1.4: Closed Circuit. This diagram is equivalent to that in Fig. 3.

4. FYR: Derive the voltage divider formula for the circuit in Fig. 6. Show your derivation and final answer. Consider two limiting cases: $R_2 \gg R_1$ and $R_2 \ll R_1$. In each case, what does V_{out} become?

5. Connect the output BNC terminal of the function generator to Channel 1 of the oscilloscope, and then press the Ch 1 button to make sure it is on. The signal is sent through the inner wire of the BNC cable, and the outer cylinder is GND. Set the output waveform from the function generator to

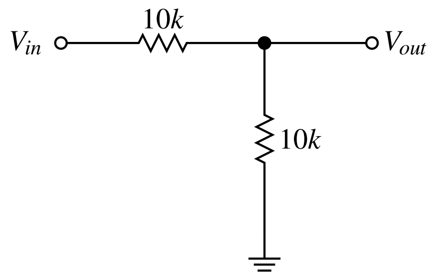

Figure 1.5: Voltage Divider (both resistors equal).

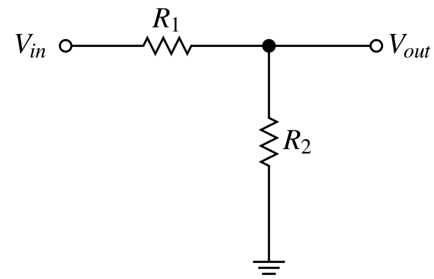

Figure 1.6: Voltage Divider (general).

be a 1kHz sine wave, about 2 Volts peak-to-peak (pp means the difference between the highest and lowest voltages in a signal, which in this case is twice the amplitude). You may have to adjust the horizontal and vertical scales of the scope to see it properly. You can measure volts on the screen by hitting the Cursor button near the top, then choosing ΔV. You can make the horizontal bars move up and down by using the cursor knob and the toggle button. The displayed voltage is the voltage difference between the two horizontal bars. Next, use a T and a 50Ω terminator from your wire box to connect to the scope. The terminator is a resistor that connects the inner wire to the outer. FYR: Sketch and compare your two observed waveforms with and without the terminator. Make a circuit diagram for the function generator, scope and terminator. Explain the difference between your two observed waveforms. Should you think of the scope as a volt meter or an ammeter?

6. Use the same setup as in the above problem but without the terminator. On the scope, go to Vertical Menu, then Coupling on the screen and choose DC. Twist the trigger level knob on the scope and observe the display of the waveform. Take note of what happens when you move the trigger level outside the range of the displayed signal. Also, pull out the DC offset knob on the function generator, and twist it to move the signal completely above and completely below the trigger level. Take note of what happens in this case too. You should consider the possibility that the scope will automatically readjust the trigger level under some circumstances. FYR: Sketch a few waveforms to show how the trigger level affects the display of the waveform. Explain what happens when the trigger level is outside the range of the displayed signal. Next, use another BNC to connect the TTL output of the function generator (which is just to the left of the main output) to Channel 2 of the scope. Hit Trigger Menu, then SRC on the screen and choose CH 2. It should now be triggering on the TTL output. Now adjust the trigger level and take note of the displayed sine wave. Also take note of what happens when the trigger level is above or below the displayed sine wave, either by moving the trigger level or adjusting the DC offset. FYR: sketch what you observe now when you adjust the trigger level up and down. Include what happens when the trigger level is outside the range of the displayed sine wave. Is there an advantage to triggering on the TTL output?

7. Now go back to using only the main output of the function generator, Trigger on it, and switch it to a square wave. Adjust the DC offset knob on the function generator. Switch back and forth between AC and DC mode of the scope (reminder: you can do this by going to Vertical Menu, Coupling). Note where GND is on the scope by adjusting the vertical position. FYR: sketch what you observe when you adjust the DC offset in each mode. What is the difference between the two?

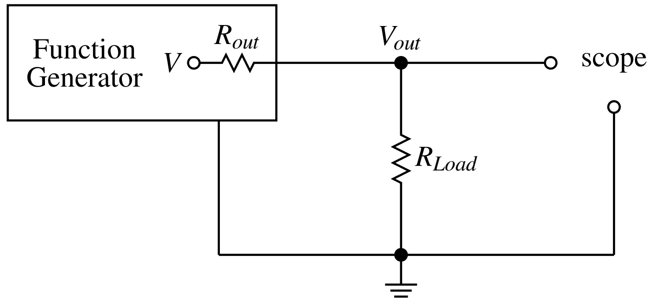

Figure 1.7: Voltage Divider formed using a load resistor and the output impedance of the function generator.

8. The function generator has its own internal output impedance just like that in Problem 2. Using a load resistor across the output of the function generator, make a voltage divider. See Figure 7. Using a given amplitude V, use the scope to measure V_{out} for 10 different values of R_{Load}. It may help to order your resistors in increasing resistance before taking measurements. The voltage divider formula for this circuit is given as:
$$V_{out} = V\frac{R_{Load}}{R_{out} + R_{Load}} \tag{1.1}$$
Do some algebra to show that $1/V_{out}$ is related linearly to $1/R_{Load}$. If $1/V_{out}$ was graphed vs. $1/R_{Load}$, determine what would give the vertical intercept and what would give the slope. Keep in mind that graphing Y (dependent) vs. X (independent) means graph Y on the vertical axis and X on the horizontal. FYR: plot $1/V_{out}$ vs. $1/R_{Load}$ from your collected data. Explicitly show your algebra starting with the voltage divider formula to derive your final equation that you used to make your plot. Give the values for R_{out} and V as determined from your graph.

9. A volt meter is supposed to measure the Root Mean Square (RMS) amplitude of a time varying signal. This RMS amplitude is the DC voltage which would deliver the same average power as the time varying signal. On AC/Cap mode, use the voltmeter to measure the RMS voltage of a 0 DC offset 2 V pp sine wave for the following frequencies: 10 Hz, 1 kHz, and 100 kHz. Then do the same for a square wave and a triangle wave. The theoretical values for RMS amplitudes are as follows: sine wave: $A_{RMS} = A/\sqrt{2}$, triangle wave: $A_{RMS} = A/\sqrt{3}$, square wave: $A_{RMS} = A$. FYR: Report your measured values for the 9 frequencies in a table. For each waveform, give a short comment on the sort of deviation from the theoretical values do you observe at different frequencies.

10. Set the function generator to produce a $\approx$ 1 MHz square wave, and input it to the scope. After triggering on it, zoom in and measure the *rise time* or *fall time* of the square wave. This is defined as the time it takes to rise from 10% to 90% of the max value or vice versa. **Hint:** if you look closely, there are two faint horizontal lines on the scope screen, along with two marks for 90% and 10%. You can use these for a more precise measurement. You can also hit the Cursor button near the top, and choose T (time, instead of frequency) On. You can then use the cursor knob and the toggle button to move the two vertical lines left and right. The displayed time is the time difference between the two vertical lines. FYR: Sketch your observed waveform and report your measured rise (or fall) time.

11. **Bonus** This is somewhat difficult to do on your own. Trigger the scope on the rising edge of a 100kHz square wave. Use the Delay feature of the scope to display the falling edge of the square wave. FYR:

estimate the fall time of the square wave. How does this compare to the rise/fall time of the 1MHz wave? Do you see any *jitter* in the pulse width?

Chapter 2

Lab 2: RC Circuits

Introduction In this lab you will build and characterize the two elementary RC circuits.

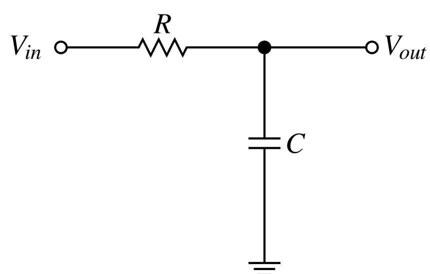

Figure 2.1: Circuit A.

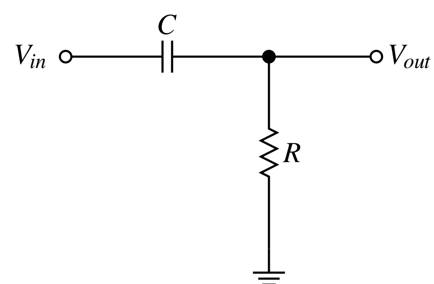

Figure 2.2: Circuit B.

1. Circuit A

   (a) Build circuit A with R = 10 kΩ and C = .01 μF. Drive it with about a square wave with no DC offset. Adjust the driving frequency and horizontal display until you can see the rise and fall times easily on the screen. Use the scope in DC mode to look at the output. Measure the time constant RC from both the rising and falling edges of the waveform. FYR: sketch your observed output waveforms and report your observed time constants. Are the time constants the same? Compare them to what you compute for RC.

   (b) In the regime where $\omega \gg 1/RC$, where $\omega = 2\pi f$, V_{out} is the *integral* of the input voltage:

$$V_{out} = \frac{1}{RC}\int V_{in}(t)dt \tag{2.1}$$

   Hence the term “integrator.” Beginning with a square wave output, increase the frequency in ‘relatively’ small steps. Show how the output gradually changes from a square wave to its integral. You may want to stop and think about what the anti-derivative of a square wave should look like. Adjust the amplitude of the input to verify that the circuit does indeed “integrate.”

FYR: sketch about 4 observed waveforms to show how the output changes from a square wave to its integral. In terms of basic calculus, describe how you confirmed that it does indeed "integrate" when you adjusted the amplitude.

(c) Using a sine wave as input, measure the gain (V_{out}/V_{in}) and phase shift ϕ as a function of input frequency.

Helpful Info: You can use a T to hook up the function generator to the input of your circuit and Ch 1 of the scope simultaneously. You can then hook the output of your circuit up to Ch 2, and then see both the input and output of your circuit simultaneously. You should trigger on the input signal (Trigger Menu, SRC, Ch 1), and move the trigger level to be very close to GND.

For each frequency, you can adjust the horizontal scale to get exactly one period of the input on the screen (Horizontal Menu, VAR, On), which is 10 squares wide. Stretch the screen sideways by twisting the cursor knob. Measure phase difference of the two waves as a fraction of one period (which corresponds to 2π radians). Do not measure absolute times. The displayed time is meaningless when VAR is on. Use the sign convention that a waveform shifted to the right on the screen corresponds to a *positive* ϕ.

Remember to measure phase difference and amplitude for each data point (about 12 total is fine). Don't forget to measure the input amplitude (which should stay constant the whole time) before or after you take data. FYR: plot the gain vs. f and the phase shift ϕ vs. f. Determine the 3 dB point from your gain plot ($\omega_{3dB} = 1/RC$). Remember: $\omega = 2\pi f$.

2. Circuit B

(a) Build circuit B with the same R and C as before. Drive it with a square wave with no DC offset. Adjust the driving frequency and horizontal display until you can see the rise and fall times easily on the screen. **Note:** the output in this case will not look like the output from Prob. 1(a) above. In fact, you may decide that your output has no rise times at all, only fall times. This is really a matter of nomenclature. Measure the time constants for both decays (make sure VAR is off). FYR: sketch your observed output waveforms and report your observed time constants. Are they the same? Compare them to what you compute for RC.

(b) In the regime where $\omega \ll 1/RC$, V_{out} is the *derivative* of the input voltage:

$$V_{out} = RC\frac{dV_{in}(t)}{dt} \tag{2.2}$$

Hence the term "differentiator." Beginning with a square wave output, decrease the frequency in 'relatively' small steps. Show how the output gradually changes from a square wave to its derivative. You may want to stop and think about what the derivative of a square wave should look like. Adjust the DC offset of the input to verify that the circuit does indeed "differentiate." FYR: sketch about 4 observed waveforms to show how the output changes from a square wave to its derivative. In terms of basic calculus, describe how you confirmed that it does indeed "differentiate" when you adjusted the DC offset.

(c) Using a sine wave as input, measure the gain (V_{out}/V_{in}) and phase shift ϕ as a function of input frequency. The Helpful Info from above applies here too. FYR: plot the gain vs. f and the phase shift ϕ vs. f. Determine the 3 dB point from your gain plot ($\omega_{3dB} = 1/RC$). Remember: $\omega = 2\pi f$.

3. For circuit A, derive the ratio V_{out}/V_{in} (gain) as a function of frequency. Do the same for the phase difference ϕ. You can use complex numbers, like that done in H&H on pp. 35-38, or you can use another method if you like. Take note that the graph displayed in H&H uses the sign convention that a shift to the right corresponds to a *positive* phase shift. FYR: show your derivation and final answer for both the gain and phase difference as a function of frequency.

4. Do the same for circuit B. FYR: show your derivation and final answer for both the gain and phase difference as a function of frequency.

Chapter 3

Lab 3: Passive Filters

Introduction In this lab you will build and characterize a two stage *RC* bandpass filter, and two *LRC* bandpass filters.

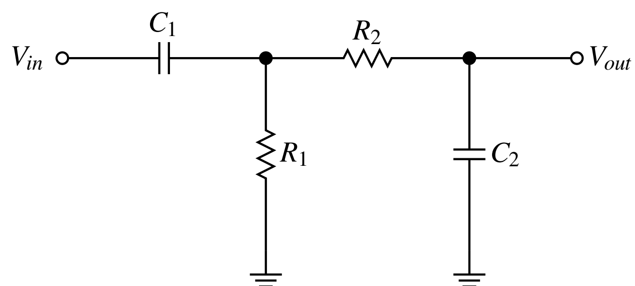

Figure 3.1: RC filter.

1. Set up the two stage *RC* bandpass filter shown in Fig. 1. Choose $1\text{M}\Omega \gg R_2 \gg R_1 \gg 50\Omega$, and $1/R_1C_1 \approx 1/R_2C_2 \approx 2\pi \times 10\text{kHz}$. Measure the gain ($V_{out}/V_{in}$) as a function of frequency. You want to show both the rise and fall of the 'resonance.' Take at least 12 data points total. FYR: plot the gain vs. f. Report the 4 component values you chose to use.

2. By using the above restrictions on R_1 and R_2, we can make the following approximation: the first and second stages are independent of one another, and can be treated separately. Derive the theoretical gain as a function of frequency. It may help to define an intermediate voltage to serve as the output from the first stage and input into the second stage. FYR: Show your derivation and final answer. Explain why this choice of R_1 and R_2 enables you to use this approximation.

3. Build the bandpass *LRC* circuit shown in Fig. 2. Use $R \approx 220\Omega$ and choose C such that $2\pi \times 150kHz \approx \omega_0 = 1/\sqrt{LC}$. Your inductor should be close to .1mH. Don't forget that $\omega = 2\pi f$. Attempt to 'see' the resonance peak for this circuit by continuously varying the frequency near ω_0. **Warning:** This circuit will not work. FYR: explain why this circuit doesn't work. Come up with and outline a scheme that would enable you to determine $V_{out} = V_R$ for this circuit. **Hint:** $V_{in} = V_R + V_{LC}$.

4. Build the bandpass *LRC* circuit shown in Fig. 3. Use the same R, L and C as above: $R \approx 220\Omega$, and C such that $2\pi \times 150kHz \approx \omega_0 = 1/\sqrt{LC}$. Your inductor should be close to .1 mH. 'See' the resonance

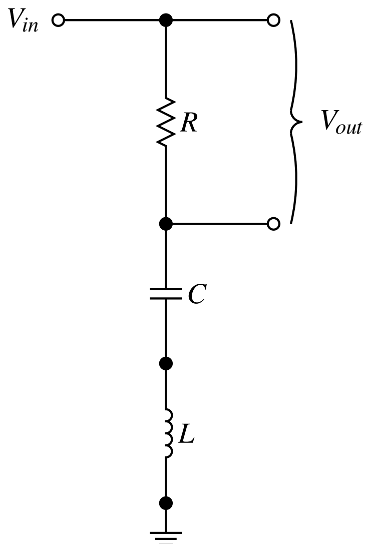

Figure 3.2: LRC Bandpass filter (Prob. 3).

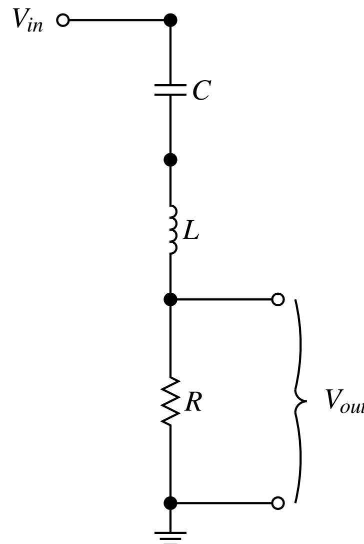

Figure 3.3: LRC Bandpass filter (Prob. 4).

peak for this circuit by continuously varying the frequency near ω_0. Determine a measured resonant frequency, and use this to determine the inductance. FYR: sketch a few wave forms to show how you could see the resonance. Show your calculation of L using your measured resonant frequency.

5. For the same circuit in the previous problem, observe the resonance again using a 500Ω resistor and then a 100Ω resistor. Compare this to what you observed using the 220Ω resistor. FRY: sketch waveforms to show how you could see the resonance using the 100Ω and 500Ω resistors. Using these sketches and those from the previous problem, compare the three cases. If you want a sharper resonance, should you use a smaller or larger resistance?

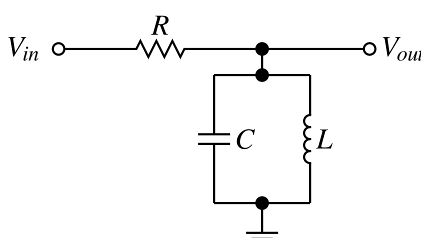

Figure 3.4: LRC Bandpass filter.

6. (a) Build the bandpass *LRC* circuit shown in Fig. 4. Use the same L and C as you did in Prob. 5, and use the value of R that gave you the sharpest resonance. Measure the gain as a function of frequency near ω_0, and make sure you get both the rise and fall of the resonance. Take at least 12 data points total. From your data, determine $Q = \omega_0/\Delta\omega$. $\Delta\omega$ is the Full Width at Half Max (FWHM) of the resonance. FYR: plot the gain vs. f. Give your measured value for Q.

(b) Using the data from part (a), obtain a plot of the current I through R vs. f. FYR: plot I vs. f. Give the formula you used to calculate I from V_{out}, V_{in}, and R.

(c) Derive the theoretical gain as a function of frequency for this circuit. Using your component values, compute the theoretical value for $Q = \omega_0 L/R$. FYR: show your derivation and final answer for the gain. Compare your theoretical Q to your measured Q. Don't worry if your measured Q is much smaller than theory predicts.

Chapter 4

Lab 4: Diodes

Introduction In this lab you will build and characterize several diode circuits.

Helpful Info: You will build many circuits with the same structure: an input, an output, and two components in a 'voltage divider' configuration. It will save time to build a 'station,' keeping the input, output and connecting wires set in place. You can then take the components in and out with little trouble.

1. Using the diode setting on your multimeter, determine the anode and cathode of a Silicon diode from your wire box (see labels in Fig. 1). When the red lead of the multimeter is connected to the anode and the black to the cathode, current will flow through the diode. The multimeter will then display the measured voltage drop in mV across the diode. FYR: make a drawing of an actual diode and its schematic. Label the anode and cathode on each to show how the markings on an actual diode correspond to the schematic. Report your measured voltage drop across the diode you have used.

2. Using your diode from Problem 1, construct each circuit shown in Figs. 1-4. Keep V_{in} the same for all: an 8 V pp 1 kHz sine wave with no DC offset. Observe V_{out} using the scope with both channels in DC mode, and make sure that you have GND for both channels at the same level. FYR: For each circuit, sketch your observed V_{out} as well as V_{in} together in the same graph. Based on your observations for all four circuits, give a rule for how the diode behaves depending on the voltage conditions applied to it. It should be similar to an if/else statement: if this happens, then the diode does that, otherwise something else happens... etc. Use this rule to label the regions in each graph where the diode is "on" (conducting) and "off" (not conducting). It may help to think back to how a voltage divider works.

3. Still using the same V_{in}, build the circuit shown in Fig. 5. Observe V_{out} using the scope. Remember that V_{out} and +1 V are in relation to GND. You may have to stop and think about how to connect the GND's for the function generator, scope, and power supply (you always want all of the GND's connected together). To help understand what you observe, vary the +1 V up and down a little. FYR: sketch your observed V_{out} as well as V_{in} together in the same graph. Indicate what happens when you vary the +1 V. Explain what is happening in terms of the diode rule. You may have to reevaluate the rule based on what you observe here. Indicate on the graph where the diode is "on" and "off."

4. Using the same V_{in} from above, construct the circuit shown in Fig. 6. Observe V_{out} using the scope. FYR: sketch your observed V_{out} as well as V_{in} together in the same graph. Using the rule for a diode,

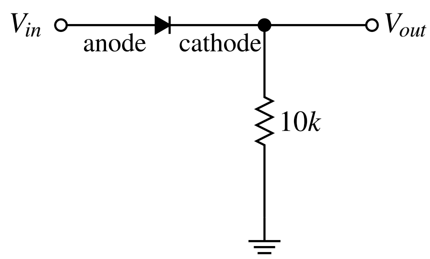

Figure 4.1: Diode Circuit 1.

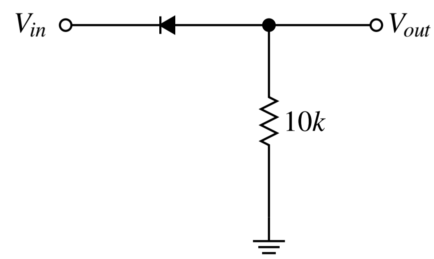

Figure 4.2: Diode Circuit 2.

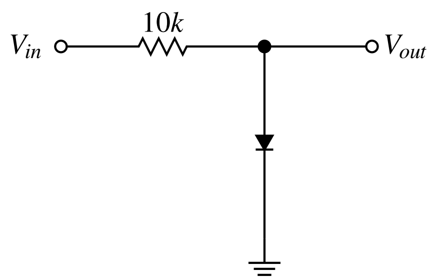

Figure 4.3: Diode Circuit 3.

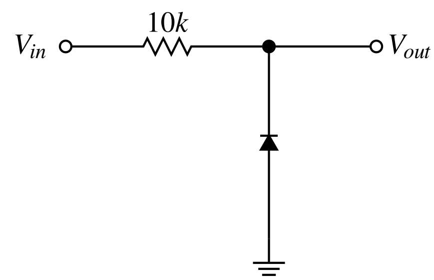

Figure 4.4: Diode Circuit 4.

give a brief explanation of the output. Label the regions where D_1 and D_2 are "on" and "off."

Bonus: If you want, use an input frequency of about 10 Hz and replace both of the diodes with LED's (preserving the polarity). An LED is a diode that lights up when it is on. You might like to flip one of them and observe what happens. How does the intensity compare in both cases? FYR: report what you observe and explain this in terms of the diodes being on and off. Also compare what happens when both of the diodes have the same polarity to when they are reversed, in terms of when they are on and their intensity.

5. Still using the same V_{in}, build the circuit shown in Fig. 7 using an electrolytic capacitor. The capacitor in this context is called a low pass *filter capacitor*. Keep in mind that capacitors this large have a *polarity*. If you hook up a capacitor with polarity the wrong way, it may explode. Literally. Look at the markings on the capacitor itself to determine which end is which.
Observe V_{out} using your scope in DC mode. V_{out} will have a max and a min: the difference between these is called V_{ripple}, and the average of these is called V_{DC}. Think about the analogy of a sine wave with a DC offset. For 6 different values of R, measure V_{ripple} and V_{DC}. FYR: plot the ratio V_{DC}/V_{ripple} vs. R. What could this circuit be used for? In order to accomplish this, should you use a very large or a very small R?

6. Use a pair of multimeters to measure the I-V curve for a diode using the circuit shown in Fig. 8. Use a Zener diode from your wire box that is labeled to have a breakdown voltage of about 5 V (you have other Zener diodes in your wire box, but their breakdown voltages are probably too large to see with

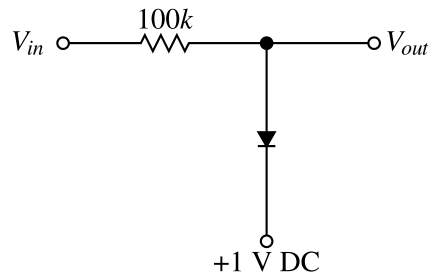

Figure 4.5: Diode Voltage Clamp Circuit.

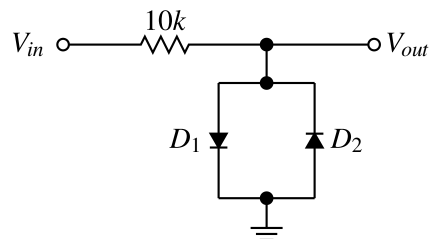

Figure 4.6: Two Diode Circuit.

the equipment that you have). The Zener diodes are very small and are orange and black. We are using a Zener here because they are designed to have a low reverse breakdown voltage and to operate in the reverse bias configuration without burning out. Remember that a Zener diode has polarity, just like a regular diode. You can figure out how it is oriented using the diode function of your multimeter. You will need to use one of the variable DC power supplies, the ones on the breadboards don't go to low enough voltages. Choose a resistor to limit the current through the diode to less than 50 mA.

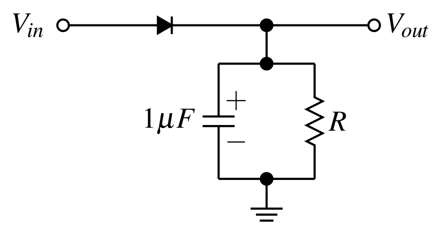

Figure 4.7: Capacitor Diode Circuit.

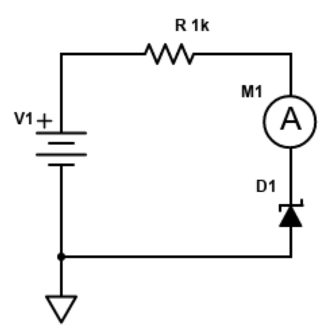

Figure 4.8: Diode Circuit to measure the current-voltage characteristics.

Chapter 5

Lab 5: BPJ Transistors

Introduction In this lab you will build the simplest bipolar transistor circuit and characterize it.

1. You should have 4 transistors in your wire box: 2N2222, 3904, 3906, and 2907. Use your multimeter on the diode setting to measure the two diode drops for one pnp and one npn transistor. See Figs. 1 and 2. Remember from last week: When the red lead of the multimeter is connected to the anode (p) and the black to the cathode (n), current will flow through the diode. The multimeter will then display the measured voltage drop in mV across the diode. The notation is V_{ab} where the red lead is connected to lead a of the transistor, and the black to b. FYR: report your two measured diode drops for 1 npn and 1 pnp transistor. Make sure you label your diode drops properly and which transistors they correspond to.

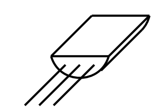

Figure 5.1: a transistor (both npn and pnp).

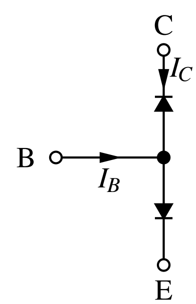

Figure 5.2: a multimeter's view of an npn transistor.

2. Using your 2N2222 transistor, set up the circuit shown in Fig. 3. It may help to review how a transistor works (also see p. 63 of H&H):

   The junction between B (base) and E (emitter) is a diode. This diode will be on (conducting) when V_B is bigger than $V_E + .7V$. When it is on, the voltage drop V_{BE} will be the usual .7V ($V_{BE} = V_B - V_E$). The current going into this conducting diode (better yet, the base-emitter voltage) will control a much larger current going into C (collector). These two currents are related by $I_C = \beta I_B$.

   β for a 2N2222 is on the order of 100 or so. Since we don't want I_C to be too large, choose I_B to be 20 μA. You can do this by choosing an appropriate combination for V_{in} (a DC voltage) and R_B. When

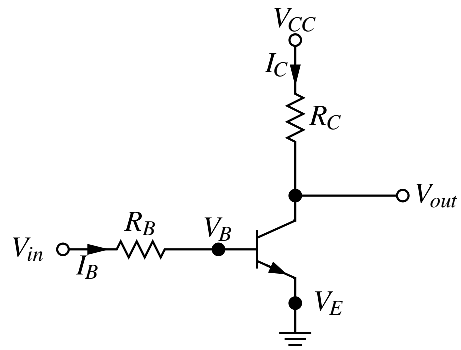

Figure 5.3: basic transistor circuit.

you do your calculation, don't worry about what is happening at the collector, just consider the circuit from $V_{in} \rightarrow R_B \rightarrow base \rightarrow emitter/GND$. Remember that the drop V_{BE} is about .7 V.

Set R_C to be $1k\Omega$ and V_{CC} to be +10 V DC. Keeping V_{in} set, vary V_{CC} from +10 V down to zero. For each value of V_{CC}, measure V_{CE} (in general, $V_{CE} = V_C - V_E$, but for this circuit $V_{CE} = V_C = V_{out}$ since $V_E = 0$). Since you are measuring a DC voltage, and you would like a decent amount of accuracy, decide whether you should use the scope or the voltmeter to measure V_{out}.

Use your collected data to calculate I_C. You want to make a graph of I_C vs. V_{CE}. You want more data points where I_C changes rapidly, and fewer where it doesn't change much.

Repeat this procedure for $I_B = 40\mu A$. FYR: plot I_C vs. V_{CE} for both values of I_B on the same graph. Remember to label which is which.

3. FYR: from each plot in the previous problem, estimate V_{CE-SAT} and β.

4. Last week the measured the current-voltage relationship for a diode. This week measure the collector current as a function of the base-emitter voltage. These are related by the Ebers-Moll equation:

$$I_C = I_S \left(\exp^{V_{BE}/kT} - 1 \right)$$

where the exponential term is usually much larger than 1, I_S is the saturation current (often nA or less), and kT (as we all remember) is about 25 meV at room temperature. FYR: Make a plot of I_C vs V_{BE} where the current is plotted on a $\ln$-scale. What is the slope of this line (i.e., what is the value of kT)?

5. Set $R_B = 10k\Omega$ and set V_{in} to be a .5 V pp 1 kHz sine wave with no DC offset. You should use a V_{CC} somewhere between +10 V and +15 V DC. Hook up V_{out} and V_B to the scope and look at them simultaneously, with both channels in DC mode. Then vary the amplitude up to be larger than .7 V. FYR: for the initial input amplitude, sketch V_{out} and V_B together on the same graph. Then make another sketch to show what happens when the input amplitude gets larger than .7 V. Explain what is happening in terms of the diode between B and E, the voltage drop across R_C, the constant voltage V_{CC}, and the relation between I_C and I_B.

6. Try to produce an undistorted sine wave at the output by varying the DC offset and amplitude of the input sine wave. You can also vary V_{CC} if you need to. FYR: report the DC offset and amplitude you used for the input that makes the output undistorted (they are not necessarily unique). Report the amplitude of the output sine wave. Compute the gain for this circuit.

7. Without going through all of the gory details, think of a transistor circuit that would automatically produce a nice magnified sine wave like that in the above problem. Assume the input has no DC offset. FYR: draw a diagram of this circuit.

Chapter 6

Lab 6: Biased Transistors

Introduction In this lab you will build and characterize the first basic biased transistor circuit: the AC coupled common emitter amplifier.

1. Build the circuit shown in Fig. 1 using a 3904 npn transistor. It will serve as the beginning of your first common emitter amplifier.

   Helpful Info: The circuits we will make from now on get rather complicated. You definitely want to avoid the 'rat's nest' type of circuit. It will help you a great deal to make an organized circuit layout that "looks like" the diagram on this handout. It will then be much easier to find errors and take measurements.

   Use +20 V DC for V_{CC}. You want the quiescent current, meaning the current into the collector with no input signal into the base, to be about 1 mA. Keep in mind that since $I_B \ll I_C$, and $I_B + I_C = I_E$, you can make the approximation $I_C \approx I_E$. You also want the DC component of V_C corresponding to the quiescent current to be half way in between V_{CC} and GND. This will give the largest possible symmetric swing of the output at V_C for a signal input into V_B. Use Ohm's Law, the desired voltage drop across R_C, and the desired quiescent current to determine the value of R_C.

   R_E serves as a way to stabilize any temperature drift in the diode drop $B \rightarrow E$. It is necessary to choose $R_E \ll R_C$, so it will therefore not significantly affect the total series resistance $R_C + R_E$ (and therefore the quiescent current of 1 mA). About a factor of 10 is good enough for our purposes, meaning you should choose $R_E = R_C/10$. This will make the voltage drop across R_E to be about 1 V. Since $V_E = 1V$, and $V_C = 10V$, the total quiescent drop across the transistor will be $V_{CE} \equiv V_C - V_E = 9$ V.

   R_1 and R_2 serve as a voltage divider to "bias" the base to a certain constant DC voltage. You want very little current going into the voltage divider from V_{CC}. About a factor of 10 should be good enough, meaning $I_1 = I_C/10$. You can use the corresponding ratio for the resistors R_1 and R_C to determine the necessary value of R_1.

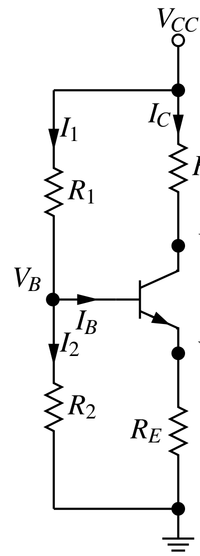

Figure 6.1: Biased Transistor circuit with no input or output.

You want to have the base voltage V_B to be 1 diode drop (for ease of calculation, just say 1 V) higher than V_E at all times. This will ensure that the diode drop $B \to E$ (and therefore the transistor) is always on. You can use this desired voltage for V_B, your calculated value of R_1, V_{CC}, and the voltage divider formula to determine R_2.

A few things to notice: 1) Your calculated value for R_2 is much less than R_1, so it doesn't significantly affect the total series resistance ($R_1 + R_2$) of the voltage divider. 2) The input impedance looking into the base of the transistor is $\approx \beta R_E$. The correct value for R_2 will make $R_2 \approx \beta R_E/10$, and therefore $I_B = I_1/10 = I_C/100$. The transistor condition is satisfied: $I_C = \beta I_B \approx 100 I_B$. The voltage divider feeds only 1/10 of its current into the base, so it is referred to as relatively "stiff."

Once you have computed your values for R_C, R_E, R_1, and R_2, set everything up, and turn on V_{CC}. Measure V_C, V_B, and V_E. FYR: record your choices for R_C, R_E, R_1, and R_2. Report your voltage measurements for V_C, V_B, and V_E.

2. With the same setup from the previous problem, use your multimeter as an ammeter to directly measure I_B. You should stop and think about how to do this in terms of the connections on your breadboard. Remember that you want your ammeter to be in series between the transistor base and the node between R_1 and R_2. In other words, make sure you don't measure I_1 or I_2 instead of I_B.

   Calculate I_C using your measured values for V_{CC}, V_C, and R_C. Use this and your measured value for I_B to calculate β. FYR: show your calculation for I_C, report your measured value for I_B, and show your calculation for β. Does your value for β fall within the reasonable range?

3. Now build the circuit shown in Fig. 2 by adding on to your circuit from the previous problems.

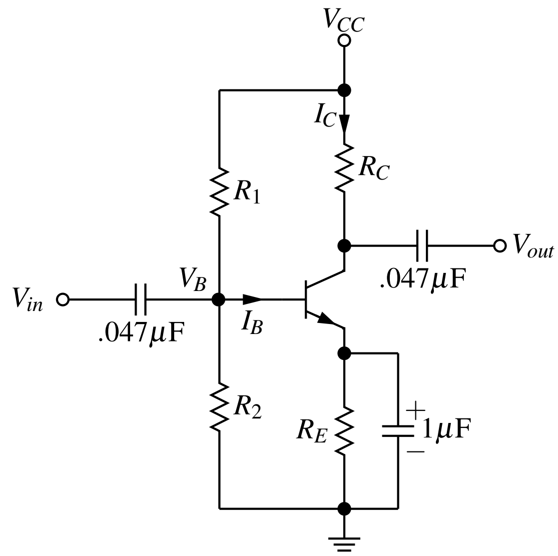

Figure 6.2: AC Coupled Common Emitter Amplifier.

You can make a $.5\mu F$ capacitor from 2 $1\mu F$ capacitors. This is your AC coupled Common Emitter Amplifier with a biased input. Set V_{in} to be a sine wave. Remembering β that you calculated from the previous problem, think about the voltage gain you expect to obtain from this amplifier. You may want to consider making your input amplitude very small to insure no 'clipping' of the output sine wave. You can do this by pulling out the amplitude knob of your function generator. Measure the gain for this circuit as a function of frequency from about 100 Hz to about 500 kHz. FYR: plot the gain vs. frequency.

4. Thinking back to how capacitors act like short circuits for AC voltages and open circuits for DC voltages, determine the function of the $.047\mu F$ "coupling capacitors." Do the same for the $.5\mu F$ "bypass capacitor." You should think about how each capacitor will work for just an AC or just a DC signal, and then apply the principle of superposition. FYR: give an explanation for the function (meaning their purpose, and how they achieve this purpose) of both the coupling and bypass capacitors.

5. Now measure the output impedance of this circuit at 1.5 kHz.

Helpful Info: Get a potentiometer from the envelope in your drawer. A potentiometer is a three terminal device that functions as a variable resistor. The resistance between the outer two leads is constant, while that between the middle lead and an outer lead will change when the screw is turned. Turn the screw with a screw driver while simultaneously measuring the resistance between any two of its leads to get a better understanding of this.

Think about the function generator output impedance measurement from Lab 1. You hooked up a resistor across the output terminals which then changed the output amplitude depending on the size of the resistor. Do something similar for this case: connect a potentiometer from V_{out} to GND. Measure the output amplitude across this potentiometer. When the resistance of the potentiometer equals the output resistance of the circuit, the output amplitude will be half of what it should be (think about a voltage divider with both of the resistors equal). If you have no idea what size of potentiometer is appropriate, first use a few resistors to see if you can get the output to change by about a factor of 2. Then switch to the potentiometer that matches this size resistor to get a more precise measurement. FYR: record your measured output impedance.

6. Now do the same for the input impedance of this circuit. Instead of putting the potentiometer between the output and GND, put it between the input coupling capacitor and V_B. When its resistance is equal to the input impedance of the circuit, the output will be half of what it should be (again, analogy of a voltage divider). You might also want to use a few resistors first to get a feel for how big of a potentiometer you should use. FYR: record your measured input impedance.

Chapter 7

Operational Amplifiers I

Introduction In this lab you will build and characterize basic operational amplifier circuits: 1) a basic comparator, 2) an inverting amplifier, 3) and a summing amplifier.

Helpful Info: You can analyze an Op Amp circuit using the following two rules:

1. The output attempts to do whatever is necessary to make the voltage difference between the two inputs zero.

2. The inputs draw no current.

An Op Amp does not actually change the voltage at its inputs. It "looks" at its input terminals and swings its output terminal around so that the external feedback network brings the input differential to zero. It should be noted that this only happens when there is feedback (connecting the output to one of the inputs). In the case of a comparator (Prob. 1), where there is no feedback, Rule 1 doesn't apply.

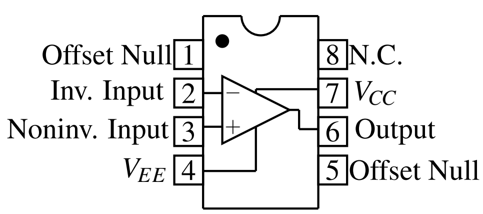

Figure 7.1: LM741CN pin connections.

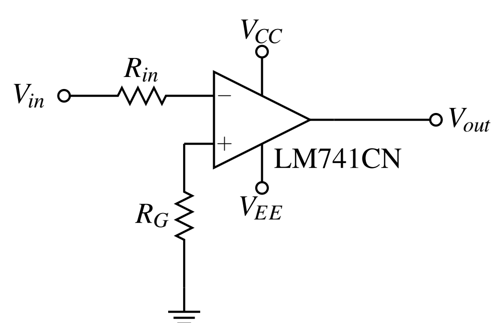

Figure 7.2: Basic Op Amp circuit (inverting comparator).

1. Build the circuit shown in Fig. 2. Think about how to place the Op Amp on the breadboard without shorting any of the pins together. The Op Amp needs at least $V_{CC} = +15V$ DC and $V_{EE} = -15V$ DC

to be powered properly.

You can get $+15V$ in the usual way by connecting the red lead from the power supply to V_{CC} and the black to GND. You can get $-15V$ by reversing the polarity from the second output of your power supply: hook the red up to GND and the black up to V_{EE}.

The values of R_G and R_{in} aren't so critical for this circuit, so something like $10K$ would be fine. They are merely meant to dissipate a very small amount of power from very small currents going in and out of the input (which doesn't happen ideally). Don't worry about the Offset Null or the N. C. (not connected) pins. For V_{in}, use a tiny sine wave at a relatively low frequency with no DC offset (peak to peak voltage about 50 to 300 mV, depending on the particular op amp you have, see below). Look at V_{out} with the scope to verify that this circuit 'comparates.'

Comparating is defined as the following: the output will be V_{CC} (also called the positive rail) when the noninverting input (+) is more positive than the inverting input (-), and vice versa. For the circuit shown in Fig. 2, the noninverting input (+) can be considered to be GND. Therefore, if the input voltage is negative, the noninverting input (+) will be more positive than the inverting input, so the output will be V_{CC}. When the input is positive, the opposite happens. If the open loop gain of your op amp isn't so large and you don't see comparating, increase the amplitude of the input.

Observe the output for a wide range of frequencies. FYR: make a few sketches of the input sine wave and output together for the different frequencies you observed. As you increase the frequency while keeping the amplitude fixed, does the op amp ever stop being able to comparate? At about what frequency does this happen? Why does this happen? Give an explanation of comparating in terms of the rails (V_{EE} and V_{CC}) and the huge gain of the op amp.

2. Build the inverting amplifier shown in Fig. 3. The gain of this circuit is given by $-R_f/R_{in}$. First use a ratio of 10, and measure the gain as a function of frequency for an input sine wave with no DC offset.

   Note: In contrast to the comparator in the previous problem, the particular value of R_{in} matters because it in part determines the gain. However, R_G is again merely meant to dissipate a small amount of power, so its value doesn't matter very much.

   Repeat with a ratio of 100. FYR: plot the gain vs. f for both ratios together on the same log-log plot. After you have made this plot, extrapolate from these curves (by hand using a ruler, if you like) to get a rough estimate of the open loop gain as a function of frequency.

3. FYR: Use the two rules for an Op Amp to explain/derive how the gain is produced for the inverting amplifier in the above problem.

4. Build the circuit shown in Fig. 4. To get V_{CC}, V_{EE}, plus two DC inputs, it may be more convenient for you to use a second DC power supply. Or, you can think about using the single 5 V output on your power supply and some resistors to make some voltage dividers. Some creativity would be helpful here. First use $V_1 = V_2 = +5$V DC and $R_1 = R_2 = R_f = 10k\Omega$ and observe V_{out} using your multimeter. To help see what this circuit does, change 1 or 2 of the input voltages and observe the output. Then,

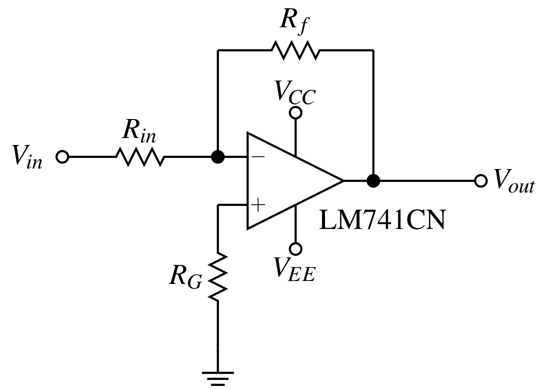

Figure 7.3: Op Amp inverting amplifier.

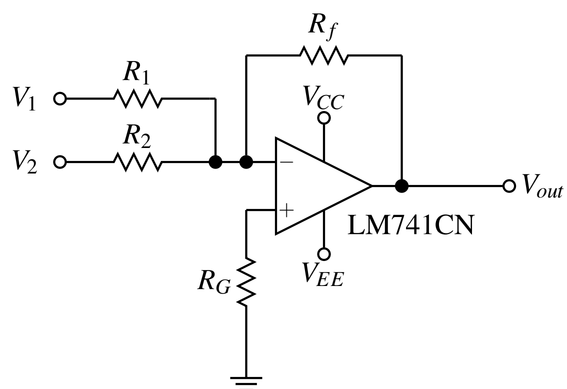

Figure 7.4: Summing (also inverting) amplifier.

using all of the input voltages equal, use different values for some of the resistances (including R_f) and look at the output. Use as many different combinations of resistors and voltages to be able to answer the next question. FYR: report the voltage and resistance combinations you used along with the corresponding output for each.

5. Use what you observed in the previous problem to write down an equation relating the output voltage, input voltages and the 3 resistor values. FYR: Give the equation you came up with. Generalize this equation for N inputs. What happens if your resistor and input voltage combinations give a V_{out} such that $|V_{out}| > |V_{EE}|$?

You can do 1 of the following 2 bonus problems, but not both:

1. **Bonus:** Use the summing amplifier from the above problems with $R_1 = R_2 = R_f$ for simplicity. Instead of using a DC voltage for each input, use a sine wave for each (comparable amplitudes). At first make both inputs very close to the same frequency, then vary one. Observe the output using the scope. FYR: sketch what you observe for various relations between the two input frequencies and explain how each is produced.

2. **Bonus:** Previously, you used one Op Amp that added (and also inverted) two voltages. Using two Op Amps, make a circuit that takes as input two voltages, negates one of them, and then adds them together. For simplicity, use a gain of 1. FYR: Draw a diagram of your circuit. Record your two inputs and the output to show that the circuit "subtracts."

Chapter 8

Operational Amplifiers II

Introduction In this lab you will build three Op Amp circuits: 1) an integrator, 2) a comparator w/o feedback, and 3) a comparator w/ feed back (Schmitt trigger).

1. Build the circuit shown in Fig. 2. Use a value of somewhere between 1 to 50 nF for C_f. R_G isn't so critical, and you can choose your own value for R_{in}. Use a square wave input and observe the output using the scope. Notice that this circuit is similar to the inverting amplifier, except that the feedback resistor is replaced with a capacitor. Also notice that V_{CC} and V_{EE} are not drawn explicitly on the op amp. This is the usual way of drawing one: it is understood that the Op Amp is powered by V_{CC} and V_{EE}. Determine the frequency range over which the circuit does in fact produce the integral of V_{in}. FYR: sketch a few output waveforms at various frequencies (including low and high) to show over what range the Integrator works.

2. FYR: using what you know about RC circuits and the two rules for an Op Amp, derive the relationship between the input and output for the integrator in the previous problem. Compare the integrator made with this Op Amp to the simple RC circuit integrator from Lab 2 in terms of the conditions necessary for it to work properly and the magnitude of the output.

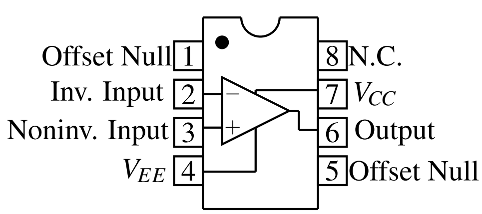

Figure 8.1: LM741CN pin connections.

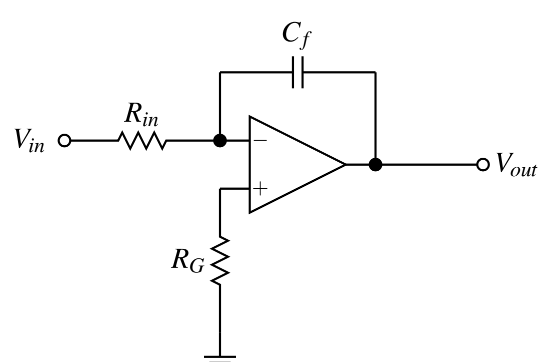

Figure 8.2: Op Amp Integrator.

3. Build the comparator shown in Fig. 3. Notice that this circuit is very similar to the comparator built in last week's lab, except its threshold voltage is something other than GND: R_1 and R_2 merely serve as a voltage divider for V_{DC}. As given in last week's lab, the rule for a comparator is as follows: the output goes to the positive rail (V_{CC}) when the noninverting input (+) goes more positive than the inverting input (-), and vice versa. In this case, the bottom rail is GND, as indicated in the drawing. V_{CC} is still connected to + 15 V.

   It would be best to connect the TTL output of the function generator to the Ext. input of the scope and tell it to trigger on this. That way you won't run into triggering problems.

   Use $R_1 = R_2 = 10k$, and set V_{in} to be a sine wave with a pp amplitude of about 4 V and with no DC offset. Use your DC power supply to apply a positive DC voltage to V_{DC} (and also therefore to V_{th}). Observe V_{out} for a few different values of V_{DC}, starting from 0 V and ending with a voltage such that V_{th} is larger than the input amplitude. FYR: sketch V_{in}, V_{th}, and V_{out} for each of the values of V_{DC} that you chose.

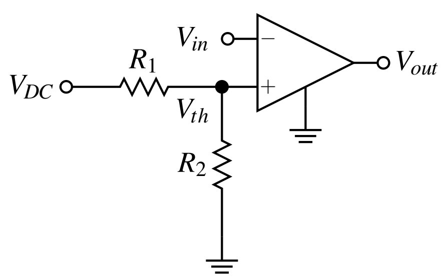

Figure 8.3: Comparator with reference voltage.

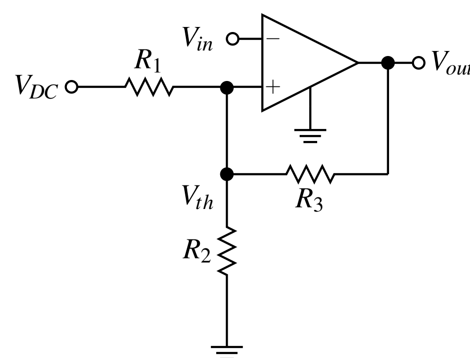

Figure 8.4: Schmitt Trigger.

4. Add a 100k feedback resistor (R_3) to your comparator from the previous problem to make the Schmitt trigger shown in Fig. 4. This circuit has a similar function as above, except it exhibits hysteresis: the voltage input at which the output switches from high to low is different from that at which it switches from low to high. **Note:** If you happen to have an Op Amp that exhibits hysteresis on its own without any feedback, it would be best to change it out for another one that does not.

   Input a triangle wave with a pp amplitude of about 4 volts and no DC offset. If you use a frequency that is too high, you might see some distortion, so use something like 500 Hz. You can choose what value for V_{DC} to use, as long as it is not zero and V_{th} is not above the input. As in the previous problem, it would be best to trigger on the TTL output. Observe V_{out}, and record the voltage levels of the input when the output transitions up and down. Then do the same after disconnecting the feedback resistor R_3.

FYR: make sketches of both the input and output together on the same graph for both with and without the feedback resistor R_3. Also label both threshold voltages in both cases. Point out the hysteresis that you observe when there is feedback.

5. Using the same circuit as above with R_3 connected, observe the hysteresis curve of the circuit directly by connecting both the input and output of the circuit to the scope in XY mode. Remember that XY mode graphs Ch 1 on the horizontal axis, and Ch 2 on the vertical. Start with the same triangle wave input that you used in the previous problem and observe the hysteresis curve. Then observe what happens when you vary the DC offset of the input triangle wave such that the input gets completely above and below the thresholds. FYR: make a sketch of the graph you observe with the initial no DC offset input, making sure you label which axis is input and which is output. Explain how the output is traced out as the input is traced out including the two thresholds. Resketch the output you observed in the previous problem using the regular scope mode and match the transitions you observed there to the transistions you observe here using XY mode. Make a few sketches of the hysteresis curve to show how it changes as you vary the DC offset.

6. FYR: use the Op Amp rules to write down an equation relating V_{th}, V_{out}, V_{DC}, R_1, R_2, and R_3. Use this and your resistor and voltage values above to determine the two theoretical threshold voltages for the circuit built in the previous problem. Do the two calculated values correspond to what you observed in the previous problem? Explain conceptually the operation of a Schmitt trigger and why it exhibits 'hysteresis' in terms of its feedback configuration and the rules of an Op Amp.

7. Build the oscillator shown in Fig. 8.5. How does the frequency relate to the component values you used?

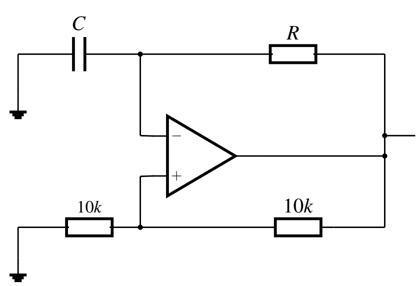

Figure 8.5: Relaxation Oscillator