

# **Electronics Laboratory Workbook**

## **Fall 2018**

**Massachusetts Maritime Academy**

**Dr. Diane DiMassa, editor**

## Acknowledgements

Not all of the information contained in this workbook is my own. Over the course of time, Prof. John Bausch, Prof. Jack Donohoe, Prof. Matt Kane, Prof. Saffir Khan, Prof. Greg Russell, Mr. Art Torino, and Prof. Wei Yu have collaborated to various degrees on the development of the Electronics Lab. Since I did not document our discussions, in many cases it is impossible for me to determine from whom the information originated. Further, some of the information contained herein was garnered from the internet, mostly from [ocw.mit.edu](http://ocw.mit.edu), [www.instructables.com](http://www.instructables.com), [www.electricneutron.com](http://www.electricneutron.com), [www.electronics-tutorials.ws](http://www.electronics-tutorials.ws), and [www.wikipedia.com](http://www.wikipedia.com). Therefore, I list myself as editor rather than author of this workbook.

# Contents

<b>Acknowledgements .....</b>	<b>2</b>
<b>Electronics Laboratory 1 – Prelab .....</b>	<b>5</b>
<b>Electronics Laboratory 1 - Breadboards and Basic Passive Components.....</b>	<b>6</b>
<b>Electronics Laboratory 2 – Pre-Lab .....</b>	<b>10</b>
<b>Electronics Laboratory 2 - Power Supplies, Diodes, Oscilloscopes, and Switches .....</b>	<b>11</b>
<b>Electronics Laboratory 3 – Pre-Lab .....</b>	<b>17</b>
<b>Electronics Laboratory 3 - Voltage Dividers, Wheatstone Bridge .....</b>	<b>118</b>
<b>Electronics Laboratory 4 – Pre-Lab .....</b>	<b>24</b>
<b>Electronics Laboratory 4 – Rectifiers .....</b>	<b>25</b>
<b>Electronics Laboratory 5 – Pre-Lab .....</b>	<b>33</b>
<b>Electronics Laboratory 5 - Transistors.....</b>	<b>34</b>
<b>Electronics Laboratory 6 – Pre-Lab .....</b>	<b>40</b>
<b>Electronics Laboratory 6A – Stroboscope .....</b>	<b>41</b>
<b>Electronics Laboratory 6B – AND Gate .....</b>	<b>45</b>
<b>Electronics Laboratory 7 – Pre-Lab .....</b>	<b>47</b>
<b>Electronics Laboratory 7 – Filters .....</b>	<b>48</b>
<b>Electronics Laboratory 8 – Pre-Lab .....</b>	<b>55</b>
<b>Electronics Laboratory 8 – Operational Amplifiers (Op-Amps) .....</b>	<b>56</b>
<b>Electronics Laboratory 9 – Pre-Lab .....</b>	<b>61</b>
<b>Electronics Laboratory 9 – Inverting Summers .....</b>	<b>62</b>
<b>Electronics Laboratory 10 – Pre-Lab .....</b>	<b>70</b>
<b>Electronics Laboratory 10 – Logic Circuits and the Master Control Relay (CRM) .....</b>	<b>71</b>
<b>Electronics Laboratory 11 – Pre-Lab .....</b>	<b>777</b>
<b>Electronics Laboratory 11 – Programmable Logic Controllers (PLCs) .....</b>	<b>788</b>
<b>Electronics Laboratory 12 – Comparators .....</b>	<b>81</b>
<b>Electronics Laboratory 13 – Comparator and Logic Design Project .....</b>	<b>899</b>
<b>Electronics Laboratory 14 – Third-Order Filters and Op-Amp Isolation .....</b>	<b>94</b>

## APPENDICES

<b>Breadboards .....</b>	<b>102</b>
<b>Capacitors .....</b>	<b>103</b>
<b>Diodes and LEDs.....</b>	<b>104</b>
<b>Figures and Tables.....</b>	<b>106</b>

<b>Laboratory Equipment.....</b>	<b>108</b>
<b>Operational Amplifiers.....</b>	<b>110</b>
<b>Potentiometers.....</b>	<b>114</b>
<b>Pushbutton Switches .....</b>	<b>115</b>
<b>Relays .....</b>	<b>1177</b>
<b>Resistors .....</b>	<b>11919</b>
<b>Soldering .....</b>	<b>1211</b>
<b>Transistors .....</b>	<b>1233</b>

## **Electronics Laboratory 1 – Prelab**

Read and complete the following Appendices:

- Breadboarding
- Capacitors
- Laboratory Equipment
- Potentiometers
- Resistors
- Soldering

# **Electronics Laboratory 1**

## **Breadboards and Basic Passive Components**

### **Breadboards**

Perform continuity tests with your DVM to review the functionality of the breadboard and its busses.

- Set up the DVM to test for continuity. This mode is used to check if two points are electrically connected (to check for shorts and open circuits). For a certain setting on your DVM, if continuity exist, the beeper sounds continuously. Figure out how to set this up. Touch the 2 leads of the DVM together to be sure it is working in this mode.
- Use the continuity test to review the connections on your breadboard. Sticking wires in the holes may help.
- How are the holes in the primary (main) bus connected? How are the holes in the secondary (auxiliary) busses connected?
- How are the holes in the working area connected? Is there continuity across the gap?
- What about the four “pins” at the top?
- Answer the questions below.

1. Are the following holes electrically connected? (Yes or No)

- a. A6 & B6 \_\_\_\_\_
- b. A12 & E12 \_\_\_\_\_
- c. A19 & F19 \_\_\_\_\_
- d. E34 & J34 \_\_\_\_\_
- e. A10 & A11 \_\_\_\_\_
- f. H53 & H61 \_\_\_\_\_

Describe the connectivity of the breadboard. Include comments on the gap, the buses, and the four pins at the top of the board.

## **Resistors**

- Find the  $12\text{ k}\Omega$  resistor and the  $91\text{ k}\Omega$  resistor in your toolbox and verify the nominal values using the Resistor Bands.
- Measure the actual values with your DVM and record them below.
- What is the calculated percent difference between the measured and nominal values?
- Is this within the stated tolerance?

$R1 = 12\text{ k}\Omega$

Measured Resistance \_\_\_\_\_

Band Colors \_\_\_\_\_

% Difference \_\_\_\_\_

Within Tolerance? \_\_\_\_\_

$R2 = 91\text{ k}\Omega$

Measured Resistance \_\_\_\_\_

Band Colors \_\_\_\_\_

% Difference \_\_\_\_\_

Within Tolerance? \_\_\_\_\_

## **Resistors in Series**

- Place your 2 resistors in series in the breadboard.
- Calculate the total resistance using the measured values above.
- Measure the total resistance with your DVM
- How well do your answers agree?

Calculated Series Resistance \_\_\_\_\_

Measured Series Resistance \_\_\_\_\_

## **Resistors in Parallel**

- Place your 2 resistors in parallel in the breadboard.
- Calculate the total resistance using the measured values above.
- Measure the total resistance with your DVM
- How well do your answers agree?

Calculated Parallel Resistance \_\_\_\_\_

Measured Parallel Resistance \_\_\_\_\_

## Isolating components for measurement

1. Using the  $91\text{k}\Omega$  resistor, measure the resistance with the Digital Volt Meter (DVM) when holding the resistor with your hands. This means that you squeeze each the leg of the resistor and the DVM lead tightly together with your thumb and forefinger.

2.

$$R_{91\text{k}\Omega \text{ (hands)}} = \underline{\hspace{2cm}}$$

3. Next, place the resistor in the breadboard isolated from everything else and retake the measurements.

$$R_{91\text{k}\Omega \text{ (breadboard)}} = \underline{\hspace{2cm}}$$

4. How do the 2 values compare?

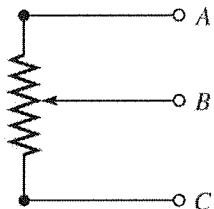
5. Now, measure your own resistance by tightly squeezing only the leads of the DVM.

$$R_{\text{student}} = \underline{\hspace{2cm}}$$

6. Explain why the values above for the  $91\text{k}\Omega$  resistor are different. What is physically happening when the measurements are taken? Explain why resistance measurements should be taken when resistors are isolated. Drawing a picture might help.

## Potentiometers

Place the variable resistor in the breadboard so that each pin is electrically independent, i.e. each pin is in a different row.



Measure the total resistance of the variable resistor, i.e. between leads A and C.

$$R_{AC} = \underline{\hspace{2cm}}$$

Measure the resistance between leads A and B and between leads B and C.

$$R_{AB} = \underline{\hspace{2cm}} \quad R_{BC} = \underline{\hspace{2cm}}$$

Turn the knob on the potentiometer and measure the resistance between leads A and B and between leads B and C again. What happened?

Experiment by turning the knob all the way one way, and then all the way the other way. Explain how a potentiometer works.

## Capacitors

Select two  $0.1\mu\text{F}$  capacitors and, using the DVM, measure and record the capacitance of each. Wait about 20 seconds until you get a stable reading. Calculate the percent error. Capacitors usually have a larger tolerance than resistors, meaning that the actual value can have a wide range around the nominal value. Do these capacitors have values close to the nominal value?

$C_1 =$  \_\_\_\_\_  $\% \text{ error} =$  \_\_\_\_\_

$C_2 =$  \_\_\_\_\_  $\% \text{ error} =$  \_\_\_\_\_

Place the two capacitors in series in the breadboard. Calculate (using measured values above), measure, and compare the total capacitance of these two capacitors in series.

$C_{T\text{-calculated}} =$  \_\_\_\_\_  $C_{T\text{-measured}} =$  \_\_\_\_\_

$\% \text{ error} =$  \_\_\_\_\_

Place the two capacitors in parallel in the breadboard. Calculate (using measured values above), measure, and compare the total capacitance of these two capacitors in parallel.

$C_{T\text{-calculated}} =$  \_\_\_\_\_  $C_{T\text{-measured}} =$  \_\_\_\_\_

$\% \text{ error} =$  \_\_\_\_\_

## **Electronics Laboratory 2 – Pre-Lab**

Read and complete the following Appendices

Diodes and LEDs

Laboratory Equipment

Pushbutton Switches

Relays

## Electronics Laboratory 2

### Power Supplies, Diodes, Oscilloscopes, and Switches

#### DC Power Supply

Choose a pin at the top of the breadboard and using a wire connect it to the negative side of the primary bus. To do this, unscrew the pin and insert the wire through the hole at the bottom. Make sure that you have a good connection when tightening the pin, i.e. be sure that you have NOT tightened down on the insulation. This will define ground. Choose a second pin and connect it to the positive side of the main bus. This will define the input power.

Connect the leads of the DC Power Supply to the appropriate pins on your breadboard. You can vary the output level by turning the adjustment knob. Dial in the *maximum* voltage and record the nominal voltage (shown on the DC Power Supply) and the measured voltage (measured with the DVM.)

Nominal Maximum Voltage \_\_\_\_\_ Measured Maximum Voltage \_\_\_\_\_

Dial in the *minimum* voltage and record the nominal voltage (shown on the DC Power Supply) and the measured voltage (measured with the DVM.)

Nominal Minimum Voltage \_\_\_\_\_ Measured Minimum Voltage \_\_\_\_\_

What is the precision of the DC Power Supply, i.e. the smallest  $\Delta V$  that can be shown on the display?

DC Power Supply Precision \_\_\_\_\_

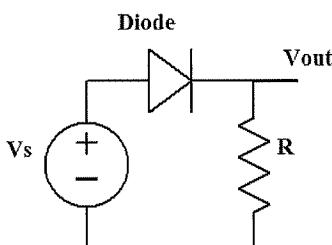
#### LEDs

For this part of the lab, you will setup a simple circuit using the DC power supply, a  $470\ \Omega$  resistor and an LED.

First determine the true value of the resistor.

Measured Resistance \_\_\_\_\_

Start with the voltage source OFF. Setup the board to minimize the amount of wire handling (and reduce the amount of noise you create). Shorter wires are better. As the circuit below shows, you will connect an LED in series with a resistor. Remember that the orientation of the LED is important.



When you are ready, make sure the voltage is dialed to the lowest possible setting, and turn on the DC Power Supply. SLOWLY increase the voltage until the LED lights up.

What is the minimum voltage needed to turn on the LED?

Minimum Voltage to light LED \_\_\_\_\_

What color is your LED?

LED Color \_\_\_\_\_

Increase the DC voltage to 5V and use the DVM to measure the voltage across the LED

LED Voltage \_\_\_\_\_

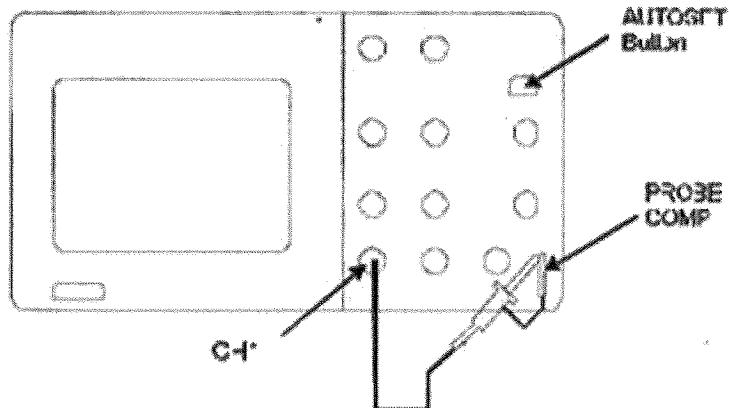
How does this voltage compare to the voltage of the DC Power supply? Why?

Would it matter if the order of the resistor and LED were reversed, i.e. the resistor came after the source and before the LED? Why or Why not?

**Turn off and Disconnect the DC power supply, but leave the rest of the circuit on the breadboard, you will use it again later.**

## **Oscilloscope**

First, check the compensation of your oscilloscope. As shown in the figure below, connect the oscilloscope probe to the Probe Compensation leads on the lower right of the oscilloscope.

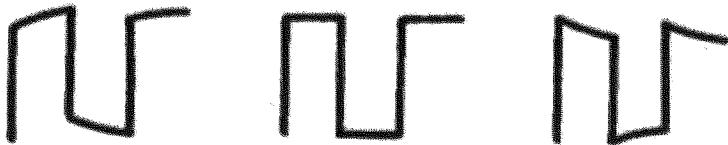


Press the Autoset button and you should see a 5V square wave with a frequency of 1kHz.

Where on the screen does it tell you what each major division is worth?  
Press the Default Setup button. What happens?

Check the shape of the displayed waveform against the figure below.

Over compensated      Correctly Compensated      Under compensated



Use the small screwdriver to change the compensation and see what happens to the square wave. Depending on which probe you have, the screw itself may be on the handle of the probe or on the connection to the oscilloscope. Describe how the square wave changes as you change the compensation of the oscilloscope probe. Explain.

Press the Menu button for channel 1. What happens? Press the Menu button again. What happens?

Press the Measure button. Press the button that lines up with Channel 1 on the screen. Using the Multipurpose button, select Frequency, Peak-to-Peak, and RMS. Note that the displayed amplitude is usually the peak-to-peak value. You will need to dial the multi-purpose button to the measurement type that you want and then press the multipurpose button to select it. Watch for the check marks. When you have the measurements that you want, press the Menu button again.

Adjust the vertical and horizontal scale knobs as needed to answer the questions below.  
What are the smallest/largest voltages possible for a major division?

Smallest V/div \_\_\_\_\_

Largest V/div \_\_\_\_\_

What are the shortest/longest time periods possible for a major division?

Shortest time/div \_\_\_\_\_

Longest time/div \_\_\_\_\_

How many minor divisions are there in a major division?

What is the resolution of the oscilloscope, i.e. minimum voltage that could be measured?

Oscilloscope Resolution \_\_\_\_\_

**Spend some time using the menu, measure, and multipurpose buttons to familiarize yourself with how you can display what you want on the screen. You will need to be good at this as time goes on. Press the cursor button and see if you can figure out what it does and how to use it.**

## **Function Generator**

Use your Function Generator to create a 440 Hz sine wave with a **peak-to-peak** amplitude of 10V. To do this you will need to attach the oscilloscope probes directly to the output cable of the Function Generator, so that you can see the sine wave on the oscilloscope screen and determine if it is the proper frequency and amplitude.

What is the Frequency as shown on the Function Generator?

Function Generator Frequency \_\_\_\_\_

What is the Frequency shown on the oscilloscope?

Oscilloscope Frequency \_\_\_\_\_

Why might these two frequencies be different?

Calculate the signal period based on the frequency on your Function Generator.

Calculated 440 Hz Period \_\_\_\_\_

What is the period shown on the oscilloscope? (use the measure and multipurpose buttons)

Oscilloscope 440 Hz Period \_\_\_\_\_

What is the amplitude shown on your Function Generator.

Function Generator Amplitude \_\_\_\_\_

What is the amplitude shown on the oscilloscope? (use the measure and multipurpose buttons)

Oscilloscope Amplitude \_\_\_\_\_

Why might these two amplitudes be different?

Adjust the Function Generator Amplitude until the Oscilloscope amplitude is 10V peak-to-peak.

Now, measure the same AC signal with the DVM.

DVM Amplitude \_\_\_\_\_

How does the DVM Amplitude compare to the peak-to-peak amplitude shown on the oscilloscope? Why?

Tune a 30Hz sine wave with 6 V peak-to-peak. Remember that you should trust the values on the oscilloscope more than the values on the Function Generator. Once the signal is tuned precisely, connect the output of the Function Generator to the breadboard pins. (**Make sure that the DC power supply has been disconnected.**) Now, you will be sending an AC signal through your LED-resistor circuit. Make sure you connect the ground and the source to the correct pins. What happens to the LED? Why?

What happens if you decrease the FREQUENCY? Why?

Connect the Oscilloscope probes across the Resistor. What does the waveform look like? Why?

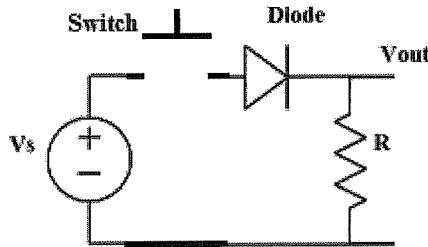
The circuit you have made is called a half-wave rectifier. You will see this again in a later lab. **Disconnect the Function Generator from the circuit and turn it off.**

### **Pushbutton Switch**

Place a **MOMENTARY pushbutton switch** (black and white base) on your breadboard somewhere away from your resistor-LED circuit, leaving the circuit intact. For this course, we will only consider the left side of the switch. Use your DVM to verify continuity between the appropriate pins when the switch is in its normal condition and when the switch is depressed.

You may want to insert wires into the breadboard holes next to the pins to be sure that you have a good connection. Remember that the switches are finicky.

Once you understand how the pushbutton switch works, modify your resistor-LED circuit as shown below. The switch should be wired as Normally Open.



Connect the **DC power supply** as you did before and dial in 5V. When the switch is NOT depressed, there should be no current flowing through the circuit and the LED should be OFF. Gently depress the switch. The LED should turn ON.

Dial the voltage down to the lowest voltage and then turn the DC Power Supply Off.

## Relay

A relay is like the push-button switch, but instead of manually pushing the button, the switching is electrically controlled.

Place your relay on your breadboard somewhere away from your resistor-LED circuit and make sure that the two sides of the relay bridge the breadboard gap. Connect Pin 16 to Ground and Connect Pin 1 to the DC Power. (Remember that the diagram in the Appendix is a BOTTOM VIEW.) With the power OFF, use your DVM to verify continuity between the appropriate pins. Check BOTH sides of the relay. You may want to insert wires into the breadboard holes next to the pins to be sure that you have a good connection. Connect the DVM to check continuity between the Normally Open and Common pins. Turn on the DC Power Supply and SLOWLY increase the power until the DVM detects continuity. This is an indication that the switch is no longer in its Normal condition. Increase the DC Power to 5V, then use your DVM to check continuity between the Normally Closed and Common pins. Check BOTH sides of the relay.

Once you understand how the relay works, turn the DC Power Supply to its minimum voltage and then turn it OFF. Modify your resistor-LED circuit by replacing the pushbutton switch with the Normally Open and Common contacts of one side of the relay. Remember to connect Pin 16 to Ground and Connect Pin 1 to the DC Power.

Turn on the DC Power Supply at its Minimum Voltage. The Relay should remain in its Normal Condition, and the LED should remain OFF. Increase the voltage to 5V. The relay should now have switched and the LED should be ON.

Look carefully at the circuit and trace the current loop that includes the LED. Are pins 1 and 16 in this loop? Make sure you understand this as you will come back to this later.

## Electronics Laboratory 3 – Pre-Lab

Read and complete the following Appendix  
Potentiometers

## Electronics Laboratory 3

### Voltage Dividers, Wheatstone Bridge

**Materials:** Two  $12\text{k}\Omega$ , Two  $91\text{k}\Omega$ ,  $500\Omega$  potentiometer, mystery resistor

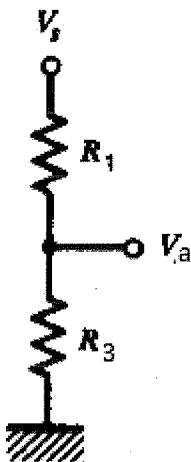
#### Resistance Measurements

Measure and record the values for a  $91\text{k}\Omega$  and a  $12\text{k}\Omega$  resistor. Remember to always measure values with the resistor isolated in the Breadboard. Be sure that the measured values fall within the indicated tolerance.

$R_{91\text{k}\Omega\_1} (R_1) = \underline{\hspace{100pt}}$        $R_{12\text{k}\Omega\_1} (R_3) = \underline{\hspace{100pt}}$

#### Voltage Divider Circuit

Your job here is to set up the Voltage Divider shown below on your Breadboard.  $V_S$  is the source voltage from the DC Power Supply, and  $V_a$  is where you will measure the divided voltage. First, connect the appropriate pins of the breadboard to the primary bus. You will switch back and forth between DC and AC in the lab, so it is easier in the long run to use the power pins.



1. Use  $R_1 = 91\text{k}\Omega$  and  $R_3 = 12\text{k}\Omega$ . (We use  $R_3$  here instead of  $R_2$  so that it is more clear when we do the Wheatstone Bridge later.)
2. What is the Voltage Divider Formula for the circuit as shown?
- 3.

Voltage Divider Formula \_\_\_\_\_

4. What is the Gain of the voltage divider based on the measured resistance values?

Voltage Divider Gain \_\_\_\_\_

Examine the output/input Transfer Function relationship between  $V_a$  and  $V_S$ . What is the shape of this equation?

1. Use your DC voltage supply to input values of  $V_s$  between 2 and 20 volts.
2. Use your DVM to measure both the  $V_s$  input and  $V_a$  divided voltages for each data point.
3. Take at least 10 well-spaced data points across the voltage scale. Put the units in the column heading. (Homework will include both data table and plots in Excel).

$V_s$	$V_a$

**Sketch** a plot of  $V_a$  vs  $V_s$  next to the data table above with  $V_s$  on the x-axis and  $V_a$  on the y-axis.  
 Is this plot linear?  
 Should it be? Why/why not?

What is the slope of the line?

How does the slope compare to the calculated gain?

Now, disconnect the DC power supply from the voltage divider circuit.

Attach the oscilloscope probe directly to the Function Generator output cable and tune a 100 Hz Sine wave with a peak-to-peak amplitude of 10V. Remember to connect the grounds, and remember that you should trust the values on the oscilloscope more than the values on the Function Generator. Adjust the signal so that you get the right values on the oscilloscope.

AC Input Frequency \_\_\_\_\_ Amplitude \_\_\_\_\_

Once the sine wave is established, connect the function generator to the power pins instead of the DC power supply. Connect the oscilloscope to read the divided voltage. Remember to connect the grounds.

Divided Voltage Frequency \_\_\_\_\_ Amplitude \_\_\_\_\_

Compare the input and output frequencies.

Compare the input and output amplitudes.

What can you say about voltage dividers when an AC signal is used as the input? Think about what you learned when you used a DC input signal.

**Disconnect the Function Generator from the breadboard before proceeding.**

### **Wheatstone Bridge Circuit**

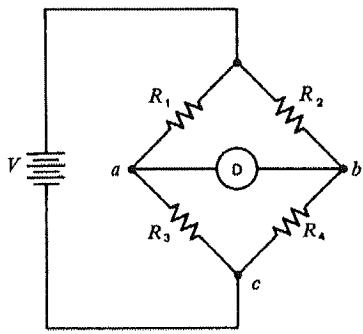
Your job in this section is to build a balanced Wheatstone Bridge. Remember, a Wheatstone Bridge is two voltage dividers connected in parallel to the same source voltage. You have already built the first voltage divider, i.e. the left side of the bridge. Now you will build the right side of the bridge.

Measure and record the values for the second 91k $\Omega$  and a 12k $\Omega$  resistors. Remember to always measure values with the resistor isolated in the Breadboard. Be sure that the measured values fall within the indicated tolerance.

$R_{91k\Omega\_2} (R_2) =$  \_\_\_\_\_  $R_{12k\Omega\_2} (R_4) =$  \_\_\_\_\_

Use these resistors to make the right side of the bridge as shown below. Remember that point a and point b must be in different rows of the breadboard.

The “D” in the circuit below indicates where you would put a Detector or measurement device (for example a DVM) to measure the difference in voltage between points *a* and *b*.



The bridge is considered **balanced** when the voltage is divided the same way on both sides, i.e. the voltage at point "a" is the same as the voltage at point "b."  $V_a = V_b$  or  $\Delta V_{ab} = V_a - V_b = 0$ .

1. **Write the equations for  $V_b$**  as a function of source voltage  $V_s$ , assuming no current flows through the Detector. This is just the voltage divider equation.

$$V_b =$$

2. Write the balance equation **by substituting in the  $V_a$  and  $V_b$  equations** you have determined into the equation below.

$$\Delta V_{ab} = V_a - V_b =$$

3. Use a FIXED 12 VDC supply and verify  $V_s$ , the supply voltage, with your DVM.

$$V_s =$$

4. Calculate the  $\Delta V$  across a and b (**use the formula you wrote above and the measured values of your resistors**)

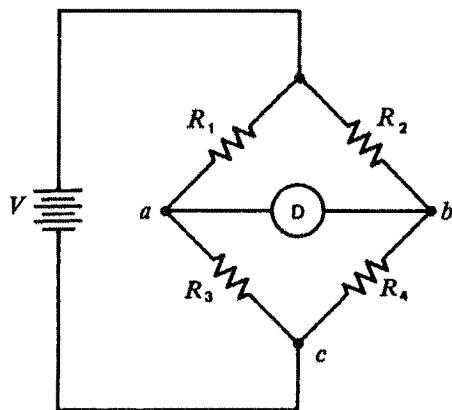
$$\Delta V_{ab\_calculated} =$$

5. Use your Digital Volt Meter (DVM) to **directly measure** the  $\Delta V$  voltage difference between points a and b (the detector, D, in the above figure). How does this relate to the value calculated? Why isn't this value zero?

$$\Delta V_{ab\_measured} =$$

## Finding an unknown Resistance with a Wheatstone Bridge

Your job here is to use a Wheatstone Bridge to determine the resistance of the mystery resistor. Disconnect the LEFT side of the bridge that you have made, but keep the RIGHT side in place. You should have one  $91\text{k}\Omega$  resistor ( $R_2$ ), and one  $12\text{k}\Omega$  resistor ( $R_4$ ). For the LEFT side of the bridge you will use the mystery resistor as  $R_1$  and the  $500\Omega$  potentiometer as  $R_3$ . Be sure to have lead B of the potentiometer as point *a* on the Wheatstone Bridge and lead C of the potentiometer connected to Ground.



Connect the DC Power supply to provide a source voltage of 10V.

Measure the source voltage with your DVM.  $V_s = \underline{\hspace{10cm}}$

Measure the voltage difference across the bridge.  $\Delta V_{ab} = \underline{\hspace{10cm}}$

Now your job is to Balance the Bridge. Carefully adjust the variable resistor until the voltage across the bridge is negligible and the bridge becomes balanced. With a little patience, you should be able to adjust the potentiometer until the  $\Delta V$  is less than 2mV. For the purpose of this lab we will say that any voltage less than 2mV is negligible.

Now, carefully disconnect the circuit from the potentiometer, LEAVING THE POTENTIOMETER IN PLACE AND UNTOUCHED, i.e. disconnect the rest of the circuit from the potentiometer. Now that the potentiometer is isolated, measure the resistance between leads B and C.

$R_{BC} = R_3 = \underline{\hspace{10cm}}$

Use the equation for a balanced Wheatstone Bridge to determine the value of the mystery resistor,  $R_1$ .

$$R_1 R_4 = R_2 R_3$$

$R_{\text{mystery (calculated)}} = \underline{\hspace{10cm}}$

Now, use the DVM to measure the actual value of the mystery resistor and compare your calculated value to this. Remember to isolate the mystery resistor in the breadboard before taking the measurement. How well did you do?

$R_{mystery}$  (measured) = \_\_\_\_\_

## **Electronics Laboratory 4 – Pre-Lab**

Read and complete the following Appendices

Diodes and LEDs

Capacitors

## Electronics Laboratory 4

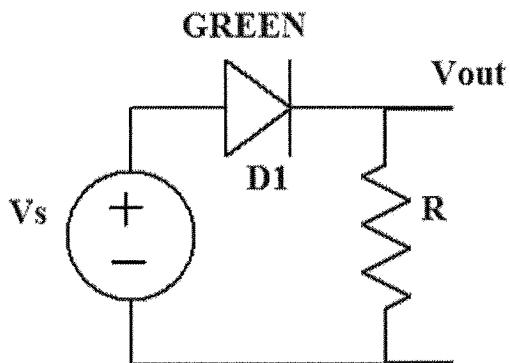
### Rectifiers

**Materials:** Two  $12\text{k}\Omega$  resistors, Four  $10\mu\text{F}$  capacitors, Two Red LEDs, Three Green LEDs, Two Blue LEDs

In this lab, you will use your Function Generator and your oscilloscope to input sine waves of various frequencies into a half-wave rectifier and a full-wave rectifier. A rectifier is a circuit that converts an AC voltage into a DC voltage. Rectifiers are found in all DC power supplies that are driven by an AC voltage source. In Part 1, you will make a half-wave rectifier with only a diode and a resistor. In Part 2 you will add capacitance to the circuit to store energy and release that energy during the time when the diode is reverse-biased. (When the diode blocks the current, we say it is reverse-biased.) You will also investigate the relationship between frequency, time constant, and ripple voltage amplitude. In Part 3, you will make a full-wave rectifier using diodes in a bridge circuit. It is important to use the correct color diodes throughout the experiments so that the full-wave rectifier can be more easily understood.

#### Simple LED-Resistor Half-wave Rectifier

In a previous lab, you built a half-wave rectifier with an LED and one resistor. The diode allows current to flow through in one direction, but blocks the current in the opposite direction. Using a GREEN LED and one  $12\text{k}\Omega$  resistor, re-create the circuit as shown below. Remember that the longer leg of the diode should go towards the higher voltage.



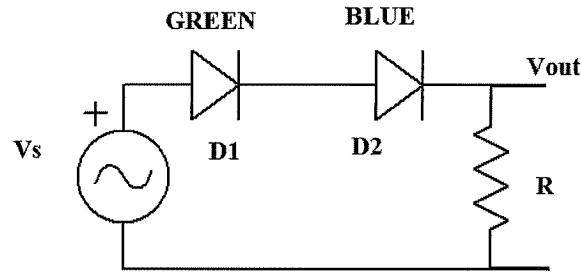
$R_{Measured} = \underline{\hspace{2cm}}$  In Tolerance? Y/N

#### A. $V_s = 10\text{VDC}$

Set up your DC power supply to provide 10V to the circuit and verify you have a working LED. Look carefully as the light may be very faint. Make sure that your LED is a GREEN one.

Test the other diodes at your station and make sure that you have THREE GREEN LEDs, TWO RED LEDs, and TWO BLUE LEDs. Set the diodes aside in a way that you can keep track of which diode is which color.

Modify the circuit so that you have TWO LEDs in series with the resistor as shown. The first diode should be GREEN and the second diode should be BLUE.



Measure the voltage source, the voltage across each LED, and the voltage across the resistor with your DVM.

$V_s = \underline{\hspace{2cm}}$   $V_{LED\_Green} = \underline{\hspace{2cm}}$   $V_{LED\_Blue} = \underline{\hspace{2cm}}$   $V_{R\_DC} = \underline{\hspace{2cm}}$

Measure the total voltage across BOTH LEDs  $V_{LEDs\_DC} = \underline{\hspace{2cm}}$

Disconnect the DC Power supply from the circuit.

**B.  $V_s = 20V$  peak-to-peak at  $f=5Hz$**

Leaving the diode-resistor circuit untouched, set up your function generator to supply a sine wave with frequency 5Hz and Peak-to-Peak Amplitude of 20V. Use your oscilloscope to help tune the signal accurately.

What is the amplitude of this signal?

What should the RMS amplitude be?

Measure the amplitude with the DVM.

What is the period?

Connect the Function Generator to the circuit as the power source. What happens to the LEDs? Use your oscilloscope to measure the voltage across the resistor. This is the output voltage of the Half-Wave Rectifier. Press the Default Setup button on the oscilloscope and then **manually** adjust the voltage scale to 5V per division and the time scale to 50ms per division.

Note that the output signal is never negative. Since the output does not change polarity, it is considered a pulsating DC voltage, not an AC voltage. If necessary, press the Run/Stop button to turn it RED and hold the signal on the screen.

Press the Measure and multipurpose buttons to display the desired voltages on the screen and record the values below.

$V_{min} = \underline{\hspace{2cm}}$   $V_{max} = \underline{\hspace{2cm}}$   $V_{pp} = \underline{\hspace{2cm}}$   $V_{RMS} = \underline{\hspace{2cm}}$

What is the amplitude of the output signal (across the resistor)?

Compare this amplitude to the amplitude of the input signal (source). Explain why they are different.

Compare the output amplitude to the output when you used 10VDC. Explain.

What is the frequency of the rectified signal?

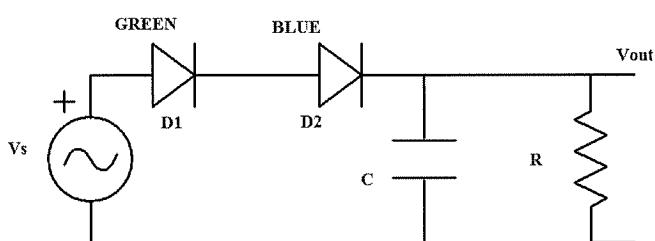
Compare the frequency of the source voltage to the frequency of the rectified voltage.

### **Half-wave Rectifier with Capacitor Filter**

In this circuit, you will add capacitance to the rectifier. The capacitor will act like a filter to reduce the fluctuations in the rectified voltage. Measure the capacitance of a  $10\mu\text{F}$  capacitor.

$C_{\text{Measured}} = \underline{\hspace{2cm}}$

Add the capacitor in parallel with the resistor as shown.



Reset the function Generator to provide a source voltage of 20V peak-to-peak with a frequency of 5Hz.

How is the shape of the output signal different with the capacitor in the circuit?

During the first quarter-period of the input, the diode is forward-biased, i.e. it lets current through from the voltage source to the resistor and capacitor. The voltage across the resistor (same as the voltage across the capacitor) tracks the voltage from the input source. Why does it have a value smaller than the input voltage?

During the second quarter-period, the voltage from the voltage source begins to decrease; however, the capacitor holds its charge. When the capacitor has a higher voltage than the input source, it tries to “push current” back towards the source. The diode becomes reverse-biased and blocks this current from flowing. The current does however flow out of the capacitor and through the resistor. The capacitor begins to discharge through the resistor. The discharging will continue until the voltage across the source is greater than the voltage across the capacitor and the diode becomes forward biased again and lets current through. The voltage across the capacitor once again tracks the voltage from the input source. When does this happen?

The variation in the output voltage due to the charging and discharging is called the **Ripple Voltage**. The smaller the ripple, the better the filtering action. The ripple size depends on how quickly the capacitor discharges and the frequency of the source voltage. This will be easier to understand as you see the different signals on the oscilloscope screen.

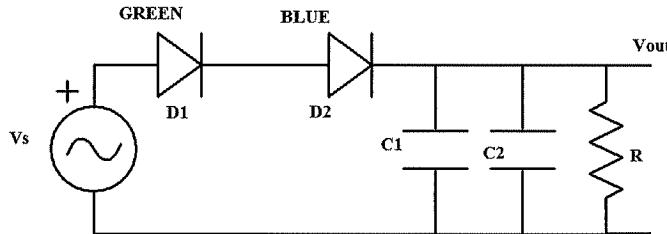
Calculate the period of the input signal and record it in Table 1.

Next, calculate the discharge time constant,  $\tau=RC$ , and record it in the table.

Measure the output signal with the oscilloscope as you did before and record the voltage values in the table

Measure the capacitance of a second  $10\mu\text{F}$  capacitor.  $C_{2\text{Measured}} = \underline{\hspace{10mm}}$

Disconnect the Function Generator from the circuit and add this capacitor in parallel with the first as shown.



Calculate the total capacitance and discharge time constant,  $\tau=RC_{\text{Total}}$ , and record the time constant in the table.

$C_{\text{Total}} = \underline{\hspace{10mm}}$        $\tau=R C_{\text{Total}} = \underline{\hspace{10mm}}$

Table 1. Half-Wave Rectifier with Capacitor Filter (time constant)

	Input Signal Period	Time Constant	$V_{\min}$	$V_{\max}$	$V_{\text{pp}}$	$V_{\text{RMS}}$
<b>1 Resistor 1 Capacitor</b>						
<b>1 Resistor 2 Capacitors</b>						
<b>1 Resistor 3 Capacitors</b>						
<b>1 Resistor 4 Capacitors</b>						

Reconnect the Function Generator, keeping the source voltage at 20V peak-to-peak with a frequency of 5Hz. Record the period in the table.

Measure the output signal with the oscilloscope as you did before and record the voltage values in the table. Be sure you are measuring the output across the resistor.

How is the shape of the output signal different with two capacitors in the circuit?

Follow the same procedure to complete the table with 3 capacitors in parallel and then 4 capacitors. How does the shape of the output signal differ?

Think about the trends in the voltage values.

What happened to the ripple voltage as you increased the time constant? Why?

What happened to  $V_{RMS}$  as you increased the time constant? Why?

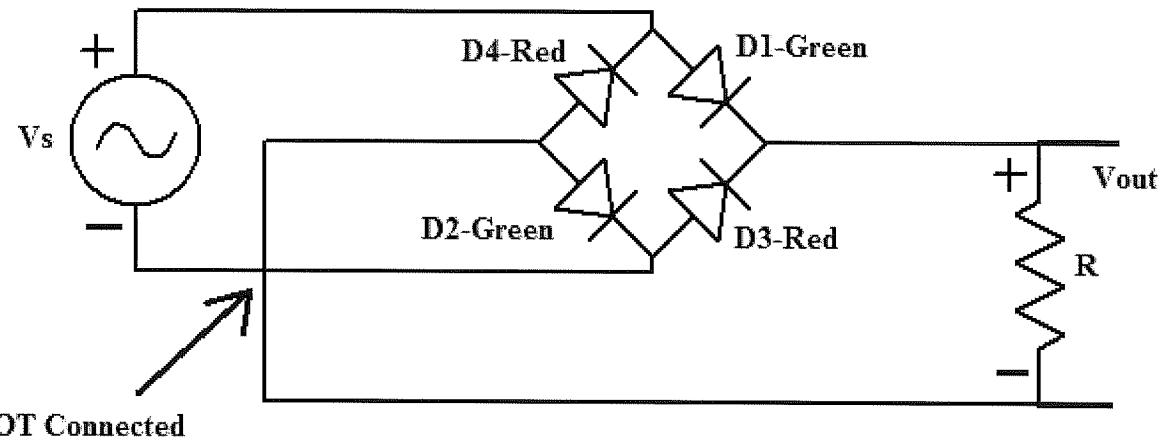
How would the signal change if you increased the frequency?

Make a sketch the output of the half-wave rectifier with and without capacitor filtering that shows the filtering effect.

### **Full-Wave Rectifier.**

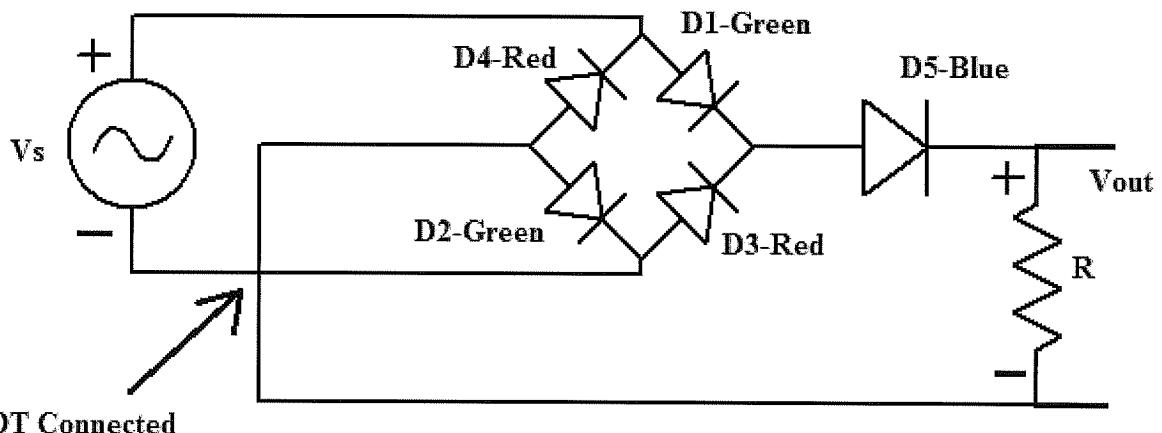
*Before proceeding with the Full-Wave Rectifier, remove ALL of the capacitors from the previous circuit going back to the case when there was a GREEN diode, a BLUE diode, and a resistor. Leave the circuit intact and use a different part of the breadboard for the Full-Wave Rectifier. You will compare the two circuits at the end.*

In a half-wave rectifier, only the positive half of the input sine wave is passed and the negative half is blocked. In the full-wave rectifier, the whole signal is converted to a positive signal and thus passes through the circuit. Using TWO GREEN and TWO RED LEDs, create the full-wave rectifier circuit below on a different part of your breadboard. Diodes 1 and 2 should be GREEN and Diodes 3 and 4 should be RED. Be very careful to properly orient (bias) your diodes. The four diodes in the diamond shape are called a diode bridge and, configured in this way, they convert the AC signal to a DC signal. *Note that the “ground” of the source is NOT connected to the 0V reference side of the resistor.*



Set up the function generator to provide a 1 Hz sine wave with 20V peak-to-peak. Connect the source as shown. You should see the resistors flash in pairs, i.e. the Greens should flash together and the Reds should flash together. When the source is outputting the positive part of the sine wave, the current path is through diode D1, then resistor R, and then diode D2 (two GREEN diodes and a resistor). When the source is outputting the negative part of the sine wave, the current path is through diode D3, then resistor R, and then diode D4 (two RED diodes and a resistor). Note that no matter the sign of the input signal, the current is always passing through the resistor in the same direction, i.e. from +Vout to 0V. Thus, the signal is fully rectified and not blocked at any time.

Now, insert the BLUE diode (Diode 5) between the bridge and the resistor as shown.



Compare the flash rate of this BLUE diode to the flash rates of the RED and GREEN diodes respectively. The BLUE LED represents the output of the circuit. Explain.

Compare the flash rate of the two BLUE diodes – one from the FULL-WAVE rectifier you just built and one from the HALF-WAVE rectifier you built earlier. Explain.

Estimate the frequency of the signal going through the resistor of the FULL-WAVE rectifier, i.e. the rectifier output.

Sketch both the input signal and rectified signal on the same plot. Remember to carefully label the axes, the amplitudes and the periods.

What would happen to the rectified signal if you added a capacitor in parallel with the resistor? Sketch the rectified signal with and without the capacitor.

## **Electronics Laboratory 5 – Pre-Lab**

Read and complete the following Appendices

- Potentiometers
- Pushbutton Switches
- Transistors

## Electronics Laboratory 5

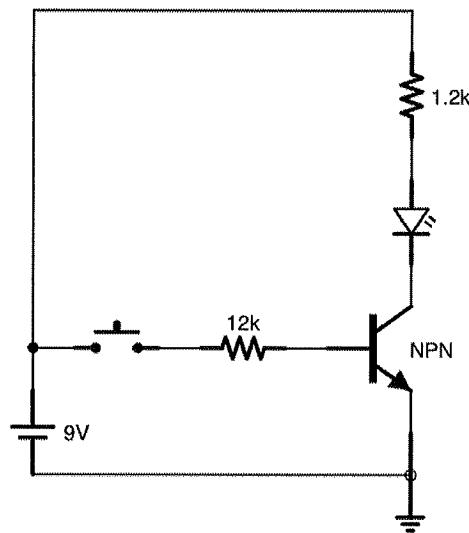
### Transistors

**Materials:** 2N2222 Transistor, LED,  $200\Omega$  resistor,  $750\Omega$  resistor, Two  $1.2k\Omega$  resistors,  $10k\Omega$  resistor,  $12k\Omega$  resistor,  $10\mu F$  capacitor,  $500\Omega$  potentiometer

#### The Electronic Switch

In a previous experiment, you learned the function of a push button switch and that switches can be wired as normally open or normally closed. The pushbutton switch that you used was a mechanical switch controlled by the action of your finger on the button. Transistors are often used as *electronic* switches, but in a different way than relays.

To demonstrate the transistor as an electronic switch, connect the circuit according to the schematic below. Use the DC power supply to provide the 9V. Be careful to make sure that you use the proper pins of the transistor.



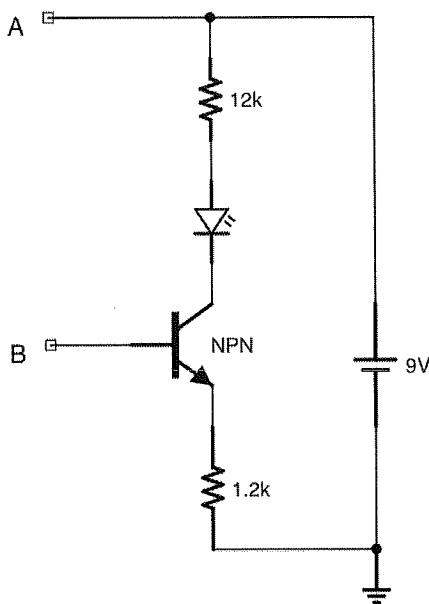
Note that although there appears to be a closed circuit with the power supply,  $1.2k\Omega$ , LED, and transistor, no current will flow since the transistor is acting like an open circuit (with no base current the gate remains shut). When you press the switch, a base current flows and opens the gate, resulting in a large collector current which lights the LED. The transistor is being used as an electronic switch in the outer loop of the diagram above. The pushbutton switch is used in the inner loop to activate the electrical switch.

Demonstrate this working circuit to your instructor. \_\_\_\_\_

What color is your LED? \_\_\_\_\_

## **The Two Hand Touch Lamp**

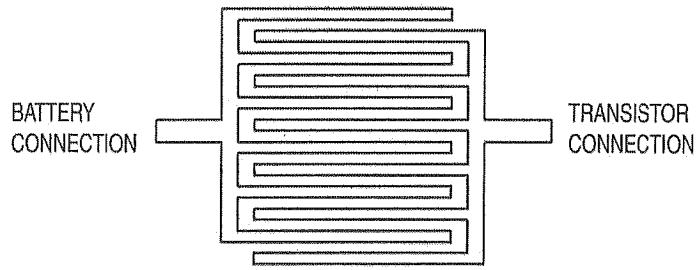
The next circuit you will make is shown in the schematic below. Now that you know how transistors work, you may be wondering how this circuit can work, since nothing is connected to the transistor base. It can't, but there is another component that isn't shown in the schematic. That component will be you!



Connect the circuit according to the schematic above, but include two loose wires, one connected at point A and the other at the Base of the transistor – point B. To do the experiment, pinch the loose battery wire (the one connected to point A) with one hand and the loose transistor wire (the one connected to point B) with the other. The LED may be dimly lit. If this is the case, your fingers aren't making good enough electrical contact. Wet your fingers with water or saliva and pinch the wires again. Your body's resistance varies a lot, but is typically a few hundred kilohms as you saw in a previous experiment. Think of this circuit as a touch lamp since when you touch it the LED lights.

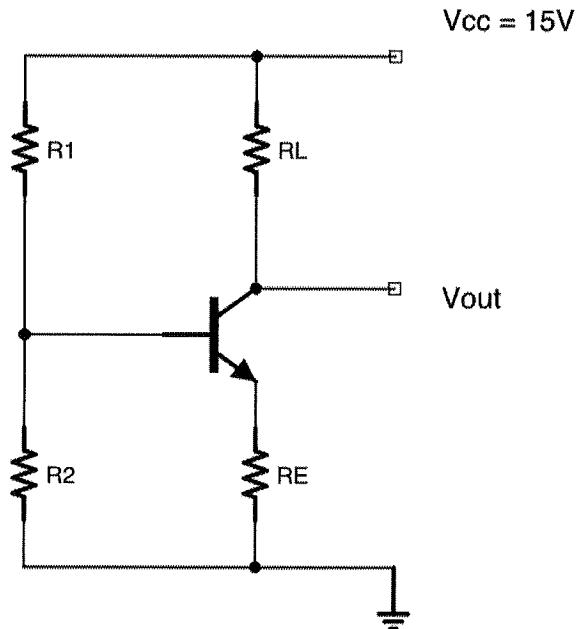
Demonstrate this working circuit to your instructor. \_\_\_\_\_

Actually, the touch lamps you see in the stores only need to be touched by a single finger to light. To make it possible for one finger to touch the two contacts, touch lamps or other touch devices will have the metal contacts interwoven as shown below and will also be more sensitive so that you don't have to wet your finger to make good contact. (This circuit is still different from the touch lamps sold in stores because in our circuit the LED goes dark when you let go.)



## The NPN Common Emitter Amplifier

The aim of any small signal amplifier is to amplify all of the input signal with the minimum amount of distortion possible to the output signal, in other words, the output signal must be an exact reproduction of the input signal but only bigger (amplified). This means that the Gain should be greater than 1. An example of a transistor amplifier is an NPN Common Emitter Amplifier. Shown below is a portion of the circuit that you will use first to create the bias necessary for the amplifier to work. Note that the small square boxes in the diagram show measurement points. These voltages should NOT be connected to ground.



## Biassing

All types of transistor amplifiers operate using AC signal inputs which alternate between a positive value and a negative value, so some way of “presetting” the amplifier circuit to operate between these two values is required. This is achieved using a process known as Biassing. Biassing is very important in amplifier design as it establishes the correct DC operating point of the transistor amplifier ready to receive signals, thereby reducing any distortion to the output signal.

Resistors  $R_1$  and  $R_2$ , arranged as a voltage divider at the Base of the transistor create the biasing necessary to operate the transistor in a linear range. This is commonly called “Voltage Divider Biasing”.

Then the potential divider network used in the common emitter amplifier circuit divides the input signal in proportion to the resistance. This reference bias voltage can be easily calculated using the simple voltage divider formula. The reference bias is the bias BEFORE amplification.

$$V_B = \frac{V_{CC}R_2}{R_1 + R_2}$$

The resistors you will use are

$$R_1=10\text{k}\Omega \quad R_2=1.2\text{k}\Omega \quad R_L=1.2\text{k}\Omega \quad R_E=200\text{\Omega}$$

Measure your resistor values and then build the circuit above.

R1=\_\_\_\_\_ RL=\_\_\_\_\_

R2=\_\_\_\_\_ RE=\_\_\_\_\_

Use the DC power supply to provide  $V_{CC}=15V$ .

Use the equation above to determine the expected value of the Base voltage and then measure the actual value

$V_{B\_calculated}$  \_\_\_\_\_  $V_{B\_measured}$  \_\_\_\_\_

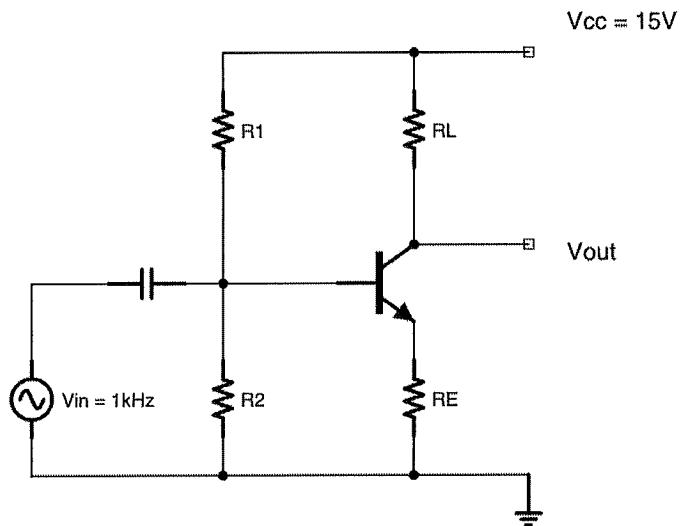
What is the measured value of the Collector ( $V_{out}$ )? \_\_\_\_\_

This is your ACTUAL Bias Voltage, the bias AFTER amplification, i.e.  $V_{out} = GV_B$ .

What value do you expect for the Gain? G

### *Coupling Capacitor*

Coupling Capacitor  
In Common Emitter Amplifier circuits, (complete circuit diagram shown below) capacitor C1 is used as a Coupling Capacitor to separate the AC signals from the DC biasing voltage. This ensures that the bias condition set up for the circuit to operate correctly is not affected by any DC offset from the AC signal source, as the capacitors will only pass AC signals and block any DC component. The output AC signal is then added to (superimposed upon) the biasing of the following stages. Generally, the value of the coupling capacitor, is chosen to provide a reactance (impedance) of at most, 1/10th the value of  $R_1$  in parallel to  $R_2$  at the lowest operating signal frequency. When connecting the circuit as shown, be sure that all of the grounds are connected making a single common ground for the entire circuit.



### Common Emitter Voltage Gain

The Voltage Gain of the common emitter amplifier is equal to the ratio of the change in the input voltage to the change in the amplifiers output voltage. In the common emitter configuration,  $V_{in}$  is the voltage at the base of the transistor and  $V_{out}$  is at the collector of the transistor. Then  $\Delta V_B$  is  $V_{in}$  and  $\Delta V_L$  is  $V_{out}$ . The voltage gain is also equal to the ratio of the signal resistance in the Collector to the signal resistance in the Emitter and is given as:

$$Gain = \left| \frac{V_{out}}{V_{in}} \right| = \left| \frac{\Delta V_L}{\Delta V_B} \right| = \left| \frac{R_L}{R_E} \right|$$

What is the expected (calculated) value of the voltage gain? \_\_\_\_\_

Install capacitor  $C_1$  as shown in the circuit diagram. Use  $C_1=10\mu F$ .

Set up the function generator to create the input sine wave signal  $V_{in}$ :  $1V_{pp}$  @ 1KHz

**Note that this is  $V_{in}$  in the circuit diagram and NOT  $V_{cc}$ .**

Before connecting the input directly, Measure the frequency and peak-to-peak amplitude of  $V_{in}$  using the oscilloscope with the coupling setting on AC. You will need to use the button near the screen that corresponds to coupling as well as the multipurpose button.

$V_{in}$  peak-to-peak amplitude \_\_\_\_\_  $V_{in}$  frequency \_\_\_\_\_

Connect the function generator to the circuit.

Measure the frequency and peak-to-peak amplitude of  $V_{out}$

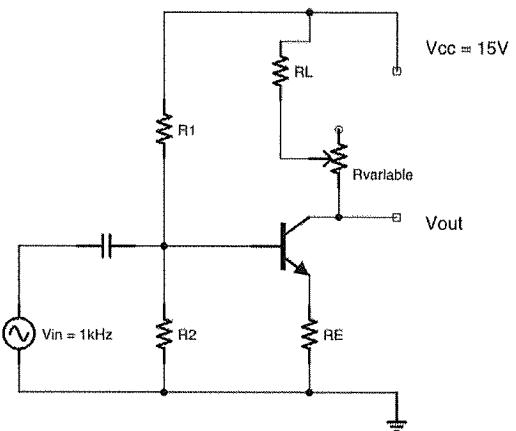
$V_{out}$  peak-to-peak amplitude \_\_\_\_\_  $V_{out}$  frequency \_\_\_\_\_

What is the value of the voltage gain based on the measurements above? \_\_\_\_\_

How do the measured and expected gain values compare?

### Tuning the Amplifier

In this section you will replace the load resistor with a series combination of a resistor and a potentiometer. By changing the value of the variable resistor, you will be able to tune in the desired gain. Change  $R_L$  to  $750\ \Omega$  and add a  $500\ \Omega$  or  $1k\Omega$  potentiometer to the collector circuit as shown:



What are the *expected* (calculated) gain and peak-to-peak values of  $V_{out}$  when the potentiometer is as its minimum setting?

Gain<sub>min</sub> \_\_\_\_\_  $V_{out\_pp\_min}$  \_\_\_\_\_

What are the *expected* (calculated) gain and peak-to-peak values of  $V_{out}$  when the potentiometer is as its maximum setting?

Gain<sub>max</sub> \_\_\_\_\_  $V_{out\_pp\_max}$  \_\_\_\_\_

Vary the potentiometer to its minimum and maximum values and *measure* the peak-to-peak output voltage at each of these end points to make sure that your calculations above are reasonable.

$V_{out\_pp\_min\_measured}$  \_\_\_\_\_  $V_{out\_pp\_max\_measured}$  \_\_\_\_\_

What can you say about the range of gains?

Tune the circuit to provide a gain of 4.

What should the value of the variable resistance be?

Disconnect the circuit from the potentiometer and measure the resistance  $R_{variable}$  \_\_\_\_\_

Check out this website for more information:

[http://www.electronics-tutorials.ws/amplifier/amp\\_2.html](http://www.electronics-tutorials.ws/amplifier/amp_2.html)

## **Electronics Laboratory 6 – Pre-Lab**

Read and complete the following Appendices

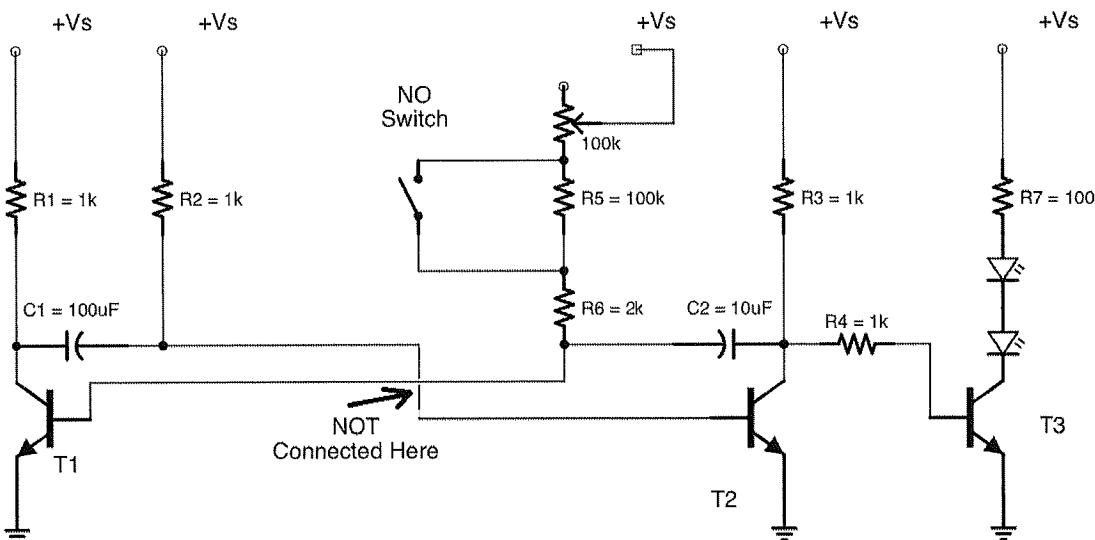
- Diodes and LEDs
- Capacitors
- Potentiometers
- Transistors
- Pushbutton Switches

# Electronics Laboratory 6A

## Stroboscope

**Materials:** three 2N2222 Transistors, *two* LEDs,  $2\text{k}\Omega$  resistor,  $100\Omega$  resistor,  $100\text{k}\Omega$  resistor, *four*  $1\text{k}\Omega$  resistors,  $10\mu\text{F}$  capacitor,  $100\mu\text{F}$  capacitor,  $100\text{k}\Omega$  potentiometer, pushbutton switch

In this lab you will build a stroboscope, sometimes just called a strobe, using the components that you have used in the previous labs. The complete circuit is shown below but you will build it systematically. Looking at the circuit, you can roughly visualize 6 columns of components. To keep the circuit organized, each of these “columns” will be set up using one of the 6 columnar groupings on the breadboard. For ease of explanation, we will number each of these areas 1 through 6, where area 1 is on the left and area 6 is on the right. We will also number the four auxiliary busses the same way with bus #1 on the left and bus #4 on the right.

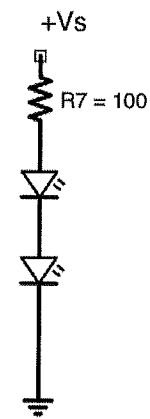


### Part 1. Breadboard setup

First, setup your breadboard with source voltage and ground on the pins of the breadboard by connecting the pins to the primary bus putting source voltage on the + side of the bus and ground on the - side. Next, connect auxiliary bus #4 (the one on the far right) to the primary bus so that source voltage is on the + side and ground is on the - side. Do the same for auxiliary busses 2 and 3.

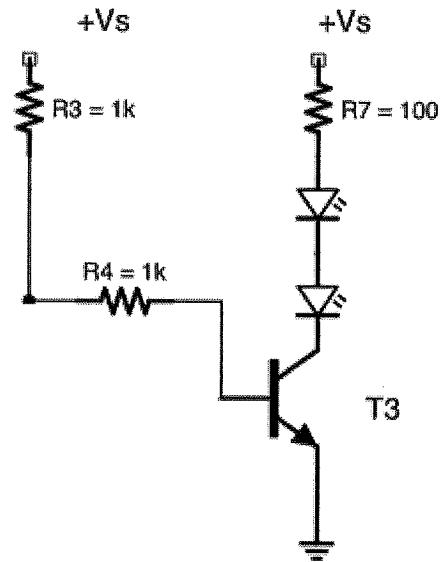
### Part 2. Systematic Circuit Building

Using the RIGHT-MOST columnar area of the breadboard (area #6), set up the circuit below.



Be careful to properly orient the diodes. Connect the DC power supply to the appropriate pins and dial in 9 V. At this point the two LEDs should turn on. When you are sure that you have 2 working LEDs, turn off the DC power supply.

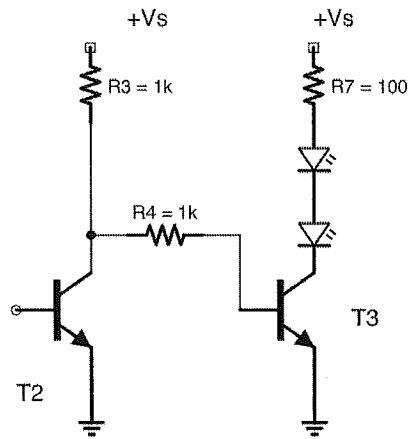
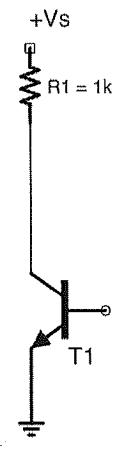
Next, you will expand the circuit as shown.



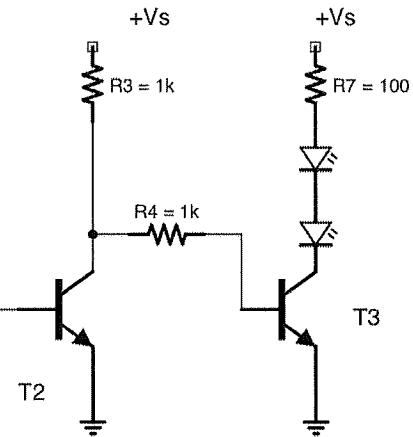
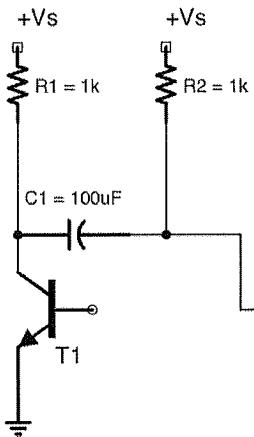
Be careful to use the correct pins of the transistor. Resistor R3 should be placed in area 5 and resistor R4 should go over the gap between areas 5 and 6. Turn on the DC power supply. The current should travel from the source through resistors R3 and R4 and into the base of the transistor. This turns the transistor on and allows the current to flow from the source through R7, both LEDs, and the transistor. Thus, the two LEDs should be on. When you are sure this part of the circuit is working, turn the DC power off.

The next step is to complete the circuit in area 5 by adding in transistor T2 as shown below and to create the circuit in area 1. Again, be careful to use the correct pins on the transistors. Note

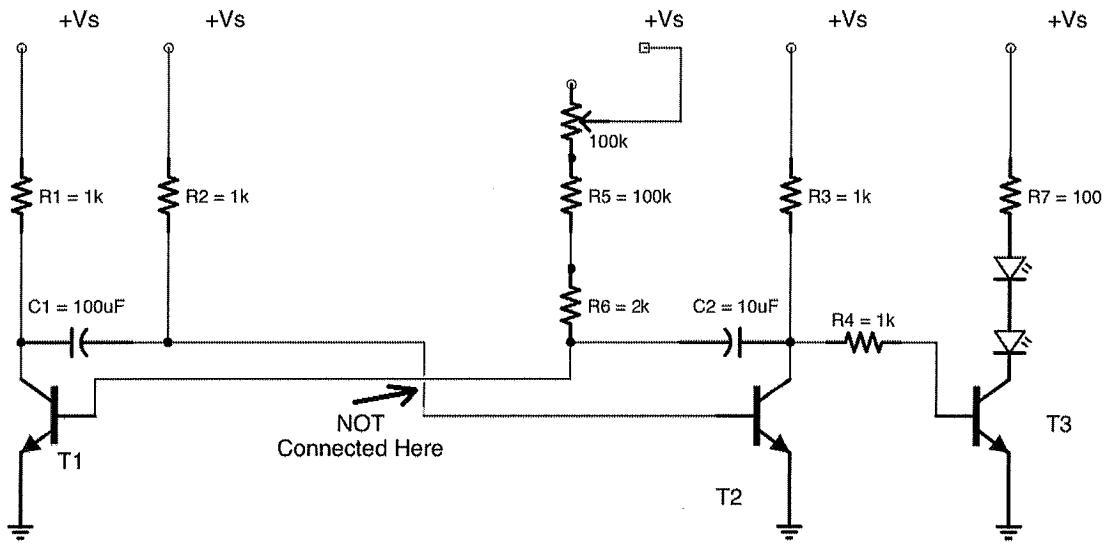
that transistor T1 is drawn differently than transistors T2 and T3.



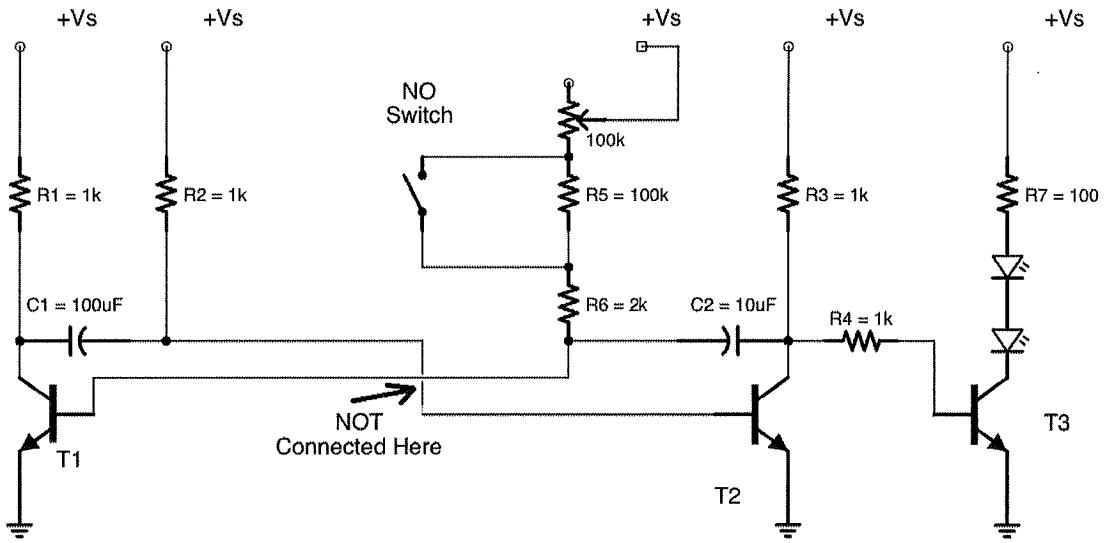
Now, complete the left part of the circuit. Resistor R2 should be in area 2 and capacitor C1 should go over the gap between areas 1 and 2. Note that the straight line of the capacitor symbol represents the positive side of the capacitor. Make sure that the capacitor is oriented properly. Then, make the appropriate connection with transistor T2.



Next, complete the circuit that is in column 4 using area 4. Resistors R5 and R6, and the potentiometer should be in area 4 and capacitor C2 should go over the gap between areas 4 and 5. Make sure that the capacitor is oriented properly. Then, make the appropriate connection with transistor T1.



The final piece is to connect the pushbutton switch. Place the switch in area 3 and make sure it is wired Normally Open.



Turn on the DC power supply. You should see the LEDs flash. If they do not, press the pushbutton switch. If they still do not flash, something is not connected correctly.

Turn the knob on the potentiometer and you should see that the flash rate changes. Press the pushbutton switch and you should also see that the flash rate increases.

Congratulations! You have built a strobe. Now go have a dance party.

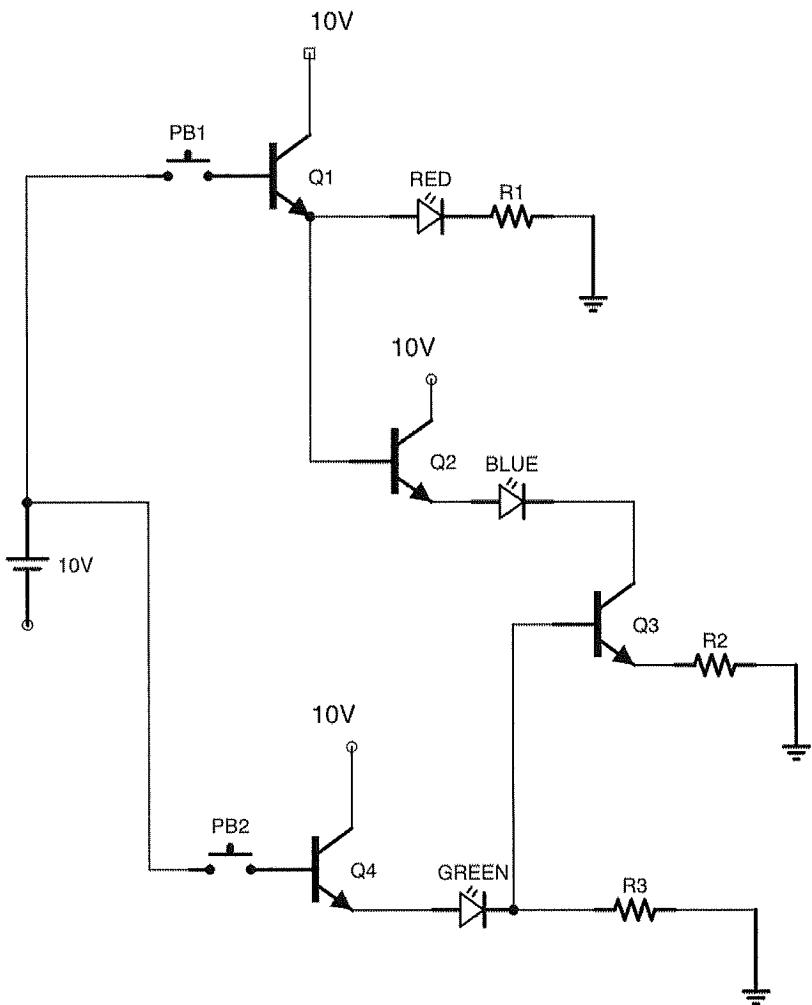
## Electronics Laboratory 6B

### AND Gate

**Materials:** four 2N2222 Transistors, 1 RED LED, 1 Green LED, 1 Blue LED, three  $1\text{k}\Omega$  resistors, 2 pushbutton switches

In the Electronics lecture course you will learn about digital electronics and logic gates. Your job in this experiment is to make an AND gate using transistors as switches. As you did in the transistor lab, you will use pushbutton switches to control transistors and transistors to control LEDs. However in this circuit you will also use transistors to control other transistors to make an AND gate. In digital electronics, the 3 basic building blocks are and AND gate, an OR gate and a NOT gate.

Create the circuit shown in the diagram below.



In this circuit Pushbutton switch 1 (PB1) controls transistor Q1, and transistor Q1 controls both the Red LED and transistor Q2. Pushbutton switch 2 (PB2) controls transistor Q4, and transistor Q4 controls both the Green LED and transistor Q3. The only way that the Blue LED will light is if BOTH Q2 and Q3 are activated and this will only happen if transistors Q1 and Q4 are activated. Thus, this circuit is an AND gate because you need both Q1 AND Q4 to be activated to turn on the Blue diode.

## Electronics Laboratory 7 – Pre-Lab

Read and complete the following Appendices

Capacitors

Resistors

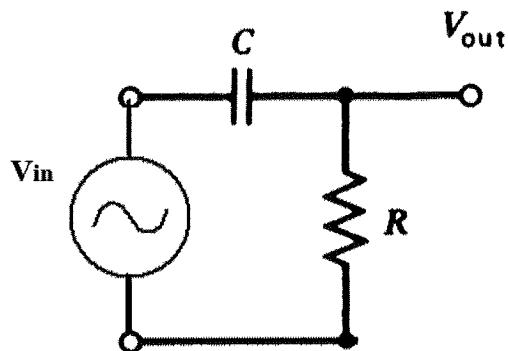
## Electronics Laboratory 7

### Filters

**Materials:** 12k $\Omega$  resistor, 1k $\Omega$  resistor, Two 0.1 $\mu$ F capacitors

In these experiments, you will use your function generator and your oscilloscope to input sine waves of various frequencies and to measure the effects of high-pass, low-pass, and band-pass filters. Filters are designed to pass and to attenuate signals of different frequency. Depending on the type of filter and the frequency of the input signal, the magnitude of the output signal may remain relatively unchanged or may be attenuated significantly.

#### 1st Order High-Pass RC Filter



The first filter you will make is a High Pass Filter using the 12 k $\Omega$  resistor and the 0.1  $\mu$ F capacitor. What is a High Pass Filter and what does it do?

The circuit above shows a high-pass filter. The  $V_{in}$  is the source signal created by the function generator, and the  $V_{out}$  is the filtered signal measured by the oscilloscope, with respect to ground but NOT connected to ground. Before building the circuit, measure the actual resistance and the actual capacitance, and calculate the corner frequency,  $f_c$ , for this filter.  $f_c = \frac{1}{2\pi RC}$

What is a corner (critical) frequency?

$R_{measured}$  \_\_\_\_\_  $C_{measured}$  \_\_\_\_\_

Calculated corner frequency  $f_c =$  \_\_\_\_\_

Build the circuit above on your breadboard, remember to bring the input signal into the pins first, and then onto the breadboard itself.

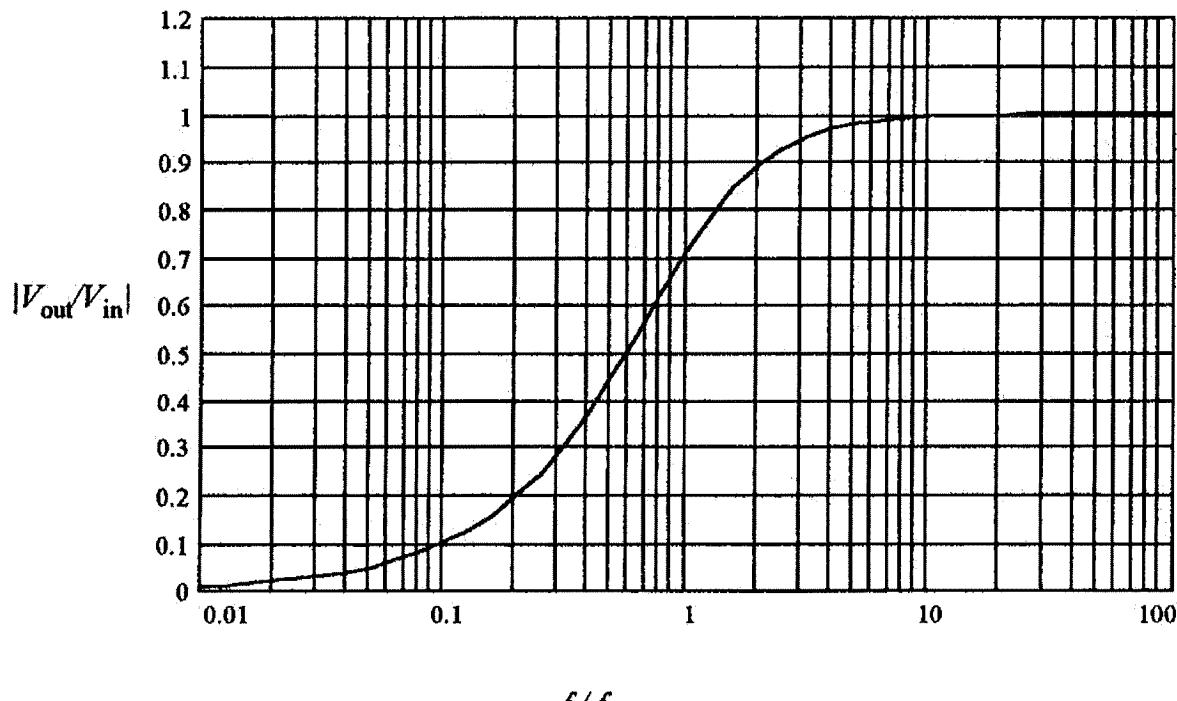
Setup your function generator to output a fixed sine wave with a frequency equal to the calculated corner frequency and with Peak-to-Peak voltage of 16 volts. Measure the output peak-to-peak value on your oscilloscope at various frequencies (it should remain relatively constant). Once you have set up the input sine wave, **LEAVE the AMPLITUDE of the input wave FIXED at 16 Volts Peak-to-Peak.**

Now, move the oscilloscope probe to measure the output signal of the High Pass Filter. Vary the input frequency around the corner frequency and measure the output signal. Take 11 well-spaced data points across the FREQUENCY scale and record your data below. Use the frequency ratios given in the data table to determine the input frequencies to use. As you fill in the data table, you should occasionally check to make sure your input is still at 16V peak-to-peak. **I strongly recommend doing the data points in the top two rows LAST.**

**High Pass Filter Data Table**

$V_{in}=V_s$ (volts, P-to-P)	Freq Ratio $F_R$ = $f/f_c$	Target Freq $f$ (Hz)	Actual Freq $f$ (Hz)	$V_{out}$ (volts) Peak-to-Peak	Mag Ratio = gain $= \frac{ V_{out} }{ V_{in} }$
16	0.1				
16	0.2				
16	0.3				
16	0.5				
16	0.7				
16	1.0				
16	2				
16	3				
16	5				
16	7				
16	10				

The figure below shows the ideal curve of a first-order high pass filter. Make a **rough sketch** of your high pass filter data on the plot below. Note that you are plotting frequency ratio on the x-axis with a LOG scale and GAIN on the y-axis. How well did your filter match the expected curve for a first order High Pass Filter?

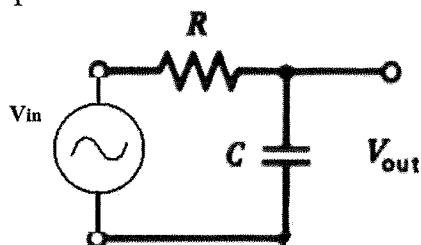


$$f/f_c$$

LEAVE the High Pass Filter circuit on the Breadboard, you will come back to it later.

### 1st Order Low-Pass RC Filter

The next filter you will make is a Low Pass Filter using the 1 k $\Omega$  resistor and the 0.1  $\mu\text{F}$  capacitor.



What is a Low Pass Filter and what does it do?

The circuit above shows a low-pass filter. Notice that it looks a lot like a high pass filter, but instead you are measuring across the capacitor. As before, the  $V_{\text{in}}$  is the source signal created by the function generator, and the  $V_{\text{out}}$  is the filtered signal measured by the oscilloscope, with respect to ground but NOT connected to ground.. Before building the circuit, measure the actual resistance and the actual capacitance, and calculate the corner frequency,  $f_c$ , for this filter.

$$f_c = \frac{1}{2\pi RC}$$

What is a corner (critical) frequency?

$R_{\text{measured}} =$  \_\_\_\_\_

$C_{\text{measured}} =$  \_\_\_\_\_

Calculated corner frequency  $f_c =$  \_\_\_\_\_

How does this value of corner frequency compare to that of the high pass filter? Why?

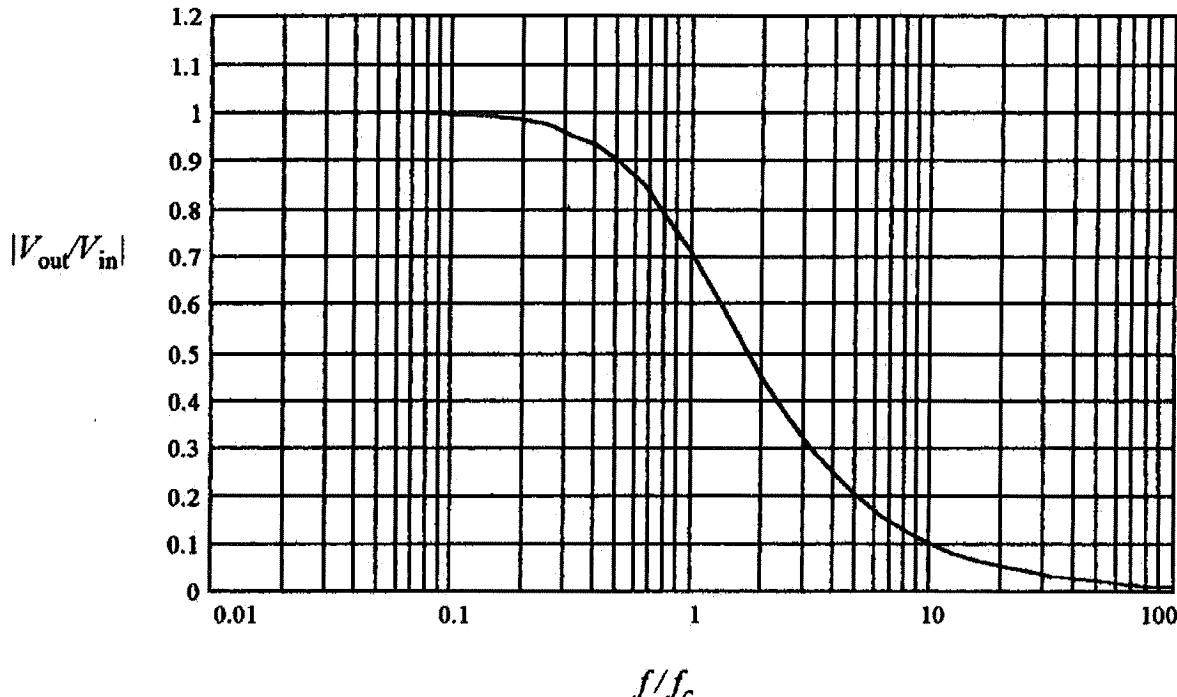
Build the circuit above on your breadboard somewhere not too far away from your high pass filter. Setup your function generator to output a fixed sine sine wave with a frequency equal to the calculated corner frequency and with Peak-to-Peak voltage of 16 volts as before. Measure the output peak-to-peak value on your oscilloscope at various frequencies (it should remain relatively constant). Once you have set up the input sine wave, **LEAVE the AMPLITUDE of the input wave FIXED at 16 Volts Peak-to-Peak.**

Now, move the oscilloscope probe to measure the output signal of the Low Pass Filter. Vary the input frequency around the corner frequency and measure the output signal. Take 11 well-spaced data points across the FREQUENCY scale. Use the frequency ratio given in the data table to determine the input frequencies to use. This is the same procedure you used for the High Pass Filter. As you fill in the data table, you should occasionally check to make sure your input is still at 16V peak-to-peak and record your data below.

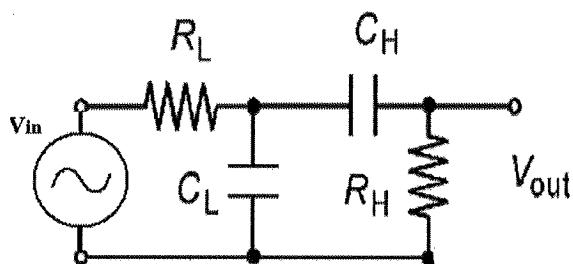
**Low Pass Filter Data Table**

$V_{in}=V_s$ (volts, P-to-P)	Freq Ratio $F_R$ $= f/f_c$	Target Freq $f$ (Hz)	Actual Freq $f$ (Hz)	$V_{out}$ (volts) Peak-to-Peak	Mag Ratio = gain $= \left  \frac{V_{out}}{V_{in}} \right $
16	0.1				
16	0.2				
16	0.3				
16	0.5				
16	0.7				
16	1.0				
16	2				
16	3				
16	5				
16	7				
16	10				

The figure below shows the ideal curve of a first-order low pass filter. Make a rough sketch of your data on the plot below. Note that you are plotting frequency ratio on the x-axis with a LOG scale and GAIN on the y-axis. How well did your filter match the expected curve for a first order Low Pass Filter?



### Band Pass Filter



The next filter you will make is a Band Pass Filter, as shown above. A Band Pass Filter is a simple combination of a low pass filter and a high pass filter.

What do you expect a band pass filter to do?

As before, the V<sub>in</sub> is the source signal created by the function generator, and the V<sub>out</sub> is the filtered signal measured by the oscilloscope. Modify the circuit on the breadboard so that you make a band pass filter, i.e. connect the capacitor of the High Pass filter so that it takes the output of the Low Pass filter as its input instead of the signal from the function generator. Be very careful to not mix up your resistors and capacitors when doing this.

Once again, setup your function generator to output a fixed sine wave with Peak-to-Peak voltage of 16 volts as before. Measure the output peak-to-peak value on your oscilloscope at various

freqencies (it should remain relatively constant). Once you have set up the input sine wave, **LEAVE the AMPLITUDE of the input wave FIXED at 16 Volts Peak-to-Peak.**

Now, move the oscilloscope probe to measure the output signal of the Band Pass Filter. Vary the input frequency and measure the output signal. However, this time, you will decide which frequencies to use. Use the calculated corner frequencies for the low pass and high pass filters to help make your choices. The first frequency should be about 1/10 of the high pass filter corner frequency and the last frequency should be about 10 times the low pass filter corner frequency. Take 16 well-spaced data points, and occasionally check to make sure your input is still 16V peak-to-peak. Record your data below.

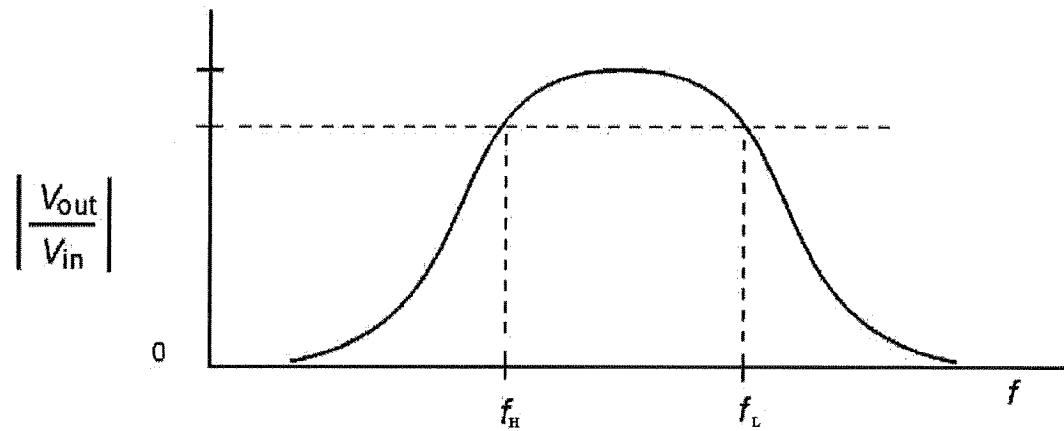
High Pass Filter  $f_{c-HPF}$  \_\_\_\_\_ Low Pass Filter  $f_{c-LPF}$  \_\_\_\_\_

**Band Pass Filter Data Table**

$V_{in}=V_s$ (volts, P-to-P)	Actual Freq $f$ (Hz)	$V_{out}$ (volts) Peak-to- Peak	Mag Ratio = gain = $\frac{ V_{out} }{ V_{in} }$
16	$\sim 1f_{c-HPF}$		
16			
16			
16			
16	$\sim f_{c-HPF}$		
16			
16			
16			
16			
16			
16			
16	$\sim f_{c-LPF}$		
16			
16			
16			
16			
16	$\sim 10f_{c-LPF}$		

By looking at this data can you predict if it will follow the expected curve?

The figure below shows an ideal curve of a band pass filter. Note that this figure plots Gain vs. frequency instead of vs. frequency ratio.



## **Electronics Laboratory 8 – Pre-Lab**

Read and complete the following Appendix  
Operational Amplifiers

## Electronics Laboratory 8

### Operational Amplifiers (Op-Amps)

**Materials:** 741 Op-Amp, two 9V batteries, two  $12\text{k}\Omega$  resistors, one  $1\text{k}\Omega$  resistor

**FIRST, review the Hitachi Spec Sheet Data in the Appendix.** All the wiring diagram information you will need is spaced throughout the packet, but **IT IS IMPORTANT TO KNOW WHICH PIN OF THE OP-AMP IS WHICH.**

#### Part 1. Board Setup and Grounding

#### Part 2. Non-Inverting Amplifier

#### Part 3. Inverting Amplifier

**Caution:** In this lab we are increasing the complexity of the setups. Your breadboard will use two different voltage systems: one to power the Op-Amp and one to generate the input signals.

### **Board Setup and Grounding**

In this lab, you will be using multiple supplies: two 9-Volt batteries and the DC Power Supply. All Power Sources will need to be connected to a Common Ground, so they share the same 0 Volt reference and measure from the same starting point.

The TWO 9-Volt batteries will be the power supply to drive your Op-Amp (Operational Amplifier). As an active circuit, these amplifiers need a separate power source. In our case we will use two 9 Volt batteries to get a total range of 18 Volts for linear operation of the amplifiers. (The Op-Amps actually saturate at a reduced range from the source voltage, so we are more likely to get only a 15 or 16 Volt total linear range). Start by using a 9V battery to put 9V (on the PLUS) and ground (on the MINUS) on the left most secondary bus (Secondary Bus 1). Connect the grounds (MINUSES) of the Primary Bus and Secondary Bus 1. Use the second 9V battery to set up the next Secondary Bus (Secondary Bus 2), with 9V on the PLUS and ground on the MINUS. Now, **connect the PLUS of Secondary Bus 2 to the MINUS of the Primary Bus.** This should create 18V between the MINUS of Secondary Bus 2 and the PLUS of Secondary Bus 1.

**VERIFY THIS WITH YOUR DVM.** You should have +9V from the MINUS of the Primary Bus to the PLUS of Secondary Bus 1, this will be  $V_{CC}$ . You should have -9V from the MINUS of the Primary Bus to the MINUS of Secondary Bus 2; this will be  $V_{EE}$ . **If you don't have this correct, the rest of the lab won't work.**

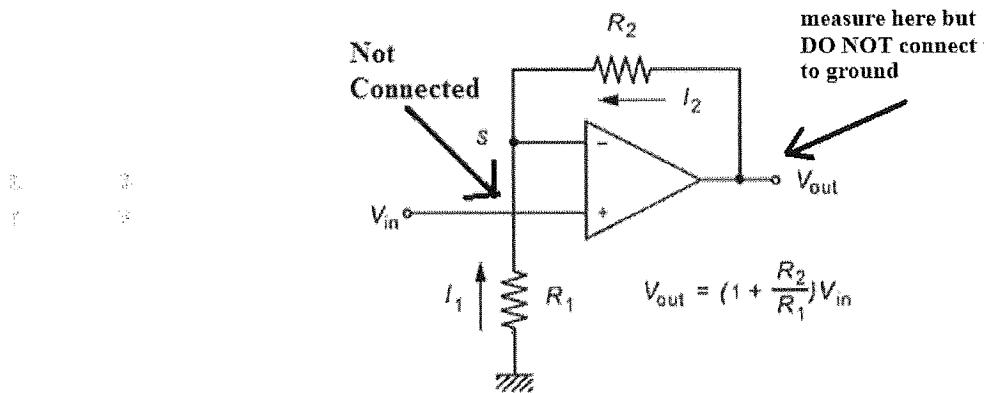
$V_{CC} =$  \_\_\_\_\_  $V_{EE} =$  \_\_\_\_\_

Once you have the above set up, place the operational amplifier in the breadboard so that it bridges the board gap between Secondary Buses 1 and 2. Make sure that the op-amp is properly oriented so that Pin 1 is in the upper left corner.

The DC Power Supply will be the input voltage,  $V_{in}$ , for these experiments. Start by connecting the pins on the breadboard to the Primary Bus to provide source voltage on the plus side and ground on the minus side. Then, connect the Variable DC Power Supply to the pins.

Have your instructor check your set-up before you complete the wiring of the circuit below.

### Basic Op-Amp Non-Inverting Amplifier Circuit



Use the attached Hitachi 741 spec sheets to setup the basic Op-Amp circuit above. Be very careful to use the correct pins. It may help to write the pin numbers on the diagram above. Remember to provide +9 Volts and -9 Volts to the Op-Amp as a power source even though it is not shown explicitly in the diagram above.

Wire this circuit using  $R_1=R_2=12\text{k}\Omega$

What are the true resistor values?  $R_1=$  \_\_\_\_\_  $R_2=$  \_\_\_\_\_

Calculate the gain of this circuit with the measured resistance values.  $G_{calculated} =$  \_\_\_\_\_

Vary the input voltage from the DC power source and record your data and calculations on the data sheet. Start with 0V input and then increase to 6V. For each data point, verify the input voltage with the DVM, and then measure the output voltage with the DVM. Then, switch the leads from the Variable DC Power Supply to create the negative input voltages.

Sketch  $V_{out}$  vs.  $V_{in}$  to the right of your table. What is the Gain as measured by the slope of the best fit line determined by the entire linear portion of your data?

$G_{measured} =$  \_\_\_\_\_

What are the maximum and minimum output voltages of the amplifier? Why?

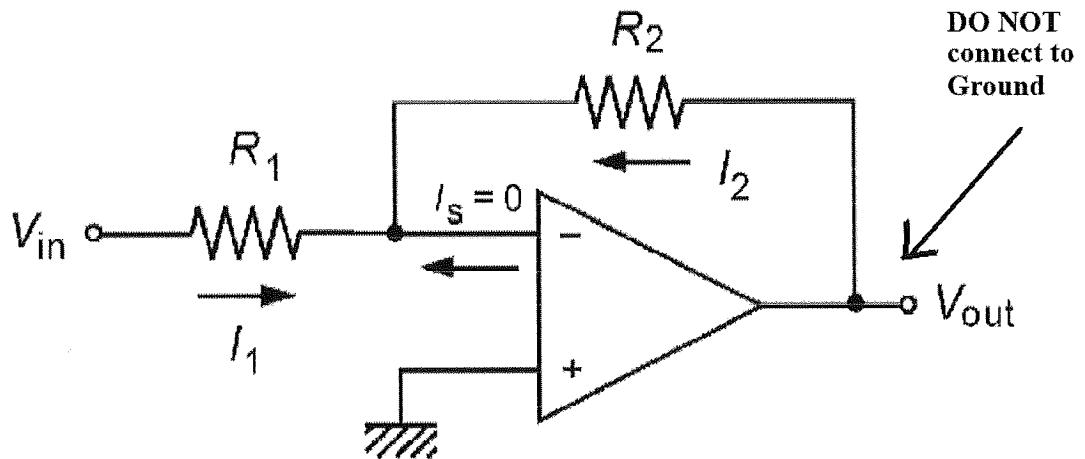
$+V_{sat}$  \_\_\_\_\_  $-V_{sat}$  \_\_\_\_\_

Table 1. Op-Amp Non-Inverting Amplifier

$V_{inTarget}$ (Volts)	$V_{inMeasured}$ (Volts)	$V_{out}$ (Volts)	<u>Sketch <math>V_{out}</math> vs. <math>V_{in}</math></u>
-6.0			
-5.0			
-4.0			
-3.0			
-2.0			
-1.2			
0.0			
1.2			
2.0			
3.0			
4.0			
5.0			
6.0			

Observation Notes:

## Basic Op-Amp Inverting Amplifier Circuit



Disconnect the Variable DC Power Supply before building the next circuit.

Use the attached Hitachi 741 spec sheets to setup the inverting amplifier circuit above. Be very careful to use the correct pins, writing them on the diagram will help. You will still need to provide +9 Volts and -9 Volts to the Op-Amp as a power source.

Wire this circuit using:

$R_1$ (feed-forward) = **12kΩ** resistor, what is the true resistor value?  $R_1 = \underline{\hspace{2cm}}$

$R_2$ (feed-back) = **1kΩ** resistor, what is the true resistor value?  $R_2 = \underline{\hspace{2cm}}$

What is the calculated gain based on the measured resistance values?

$$V_{out} = -\frac{R_2}{R_1} V_{in} \quad G_{calculated} = \underline{\hspace{2cm}}$$

Is the Op-Amp circuit an amplifier or an attenuator? Why?

Calibrate the relationship between  $V_{in}$  and  $V_{out}$  using the DC Variable Power Supply as you did for the non-inverting amplifier. Start with 0V input and then increase to 10V. For each data point, verify the source voltage with the DVM, and then measure the output voltage with the DVM. Record your data and calculations on the data sheet. Then, switch the leads from the Variable DC Power Supply to create the negative input voltages.

Sketch  $V_{out}$  vs.  $V_{in}$  to the right of your table. What is the Gain as measured by the slope of the best fit line determined by the linear portion of your data?

$$G_{measured} = \underline{\hspace{2cm}}$$

What are the maximum and minimum output voltages of the amplifier? Why?

Table 2. Op-Amp Inverting Amplifier

$V_{sat}$  \_\_\_\_\_  $-V_{sat}$  \_\_\_\_\_

$V_{inTarget}$ (Volts)	$V_{inMeasured}$ (Volts)	$V_{out}$ (Volts)
-10.0		
-8.0		
-6.0		
-4.0		
-2.0		
-1.2		
0.0		
1.2		
2.0		
4.0		
6.0		
8.0		
10.0		

Sketch  $V_{out}$  vs.  $V_{in}$

Observation Notes:

## **Electronics Laboratory 9 – Pre-Lab**

Read and complete the following Appendix  
Operational Amplifiers

# Electronics Laboratory 9

## Inverting Summers

**Materials:** two 741 Op-Amps, two 9V batteries, three  $1\text{k}\Omega$  resistors, one  $2\text{k}\Omega$  resistor, one  $1.2\text{k}\Omega$  resistor, one  $1\text{k}\Omega$  potentiometer

**FIRST, review the Hitachi Spec Sheet Data in the Appendix. IT IS IMPORTANT TO KNOW WHICH PIN OF THE OP-AMP IS WHICH.**

What are  $V_{cc}$  and  $V_{ee}$  as referred to in the sheets?

Which pins are they?

Which pin takes the inverting input,  $V_{in}(-)$ ? The non-inverting input,  $V_{in}(+)$ ?

Which pin is the output?

### Board Setup and Grounding

This is the same setup as in the first Op-Amp laboratory.

In this lab, you will be using multiple power supplies: two 9-Volt batteries, the DC Power Supply, and the Function Generator. All Power Sources will need to be connected to a Common Ground, so they share the same

0 Volt reference and measure from the same starting point.

The TWO 9-Volt batteries will be the power supply to drive your Op-Amp (Operational Amplifier). As an active circuit, these amplifiers need a separate power source. In our case we will use two 9 Volt batteries to get a total range of 18 Volts for linear operation of the amplifiers. (The Op-Amps actually saturate at reduced range from the source voltage, so we are more likely to get only a 15 or 16 Volt total linear range). Start by using a 9V battery to put 9V (on the PLUS) and ground (on the MINUS) on the left most secondary bus (Secondary Bus 1). Connect the grounds (MINUSES) of the Primary Bus and Secondary Bus 1. Use the second 9V battery to set up the next Secondary Bus (Secondary Bus 2), with 9V on the PLUS and ground on the MINUS. Now, connect the PLUS of Secondary Bus 2 to the MINUS of the Primary Bus. This should create 18V between the MINUS of Secondary Bus 2 and the PLUS of Secondary Bus 1.

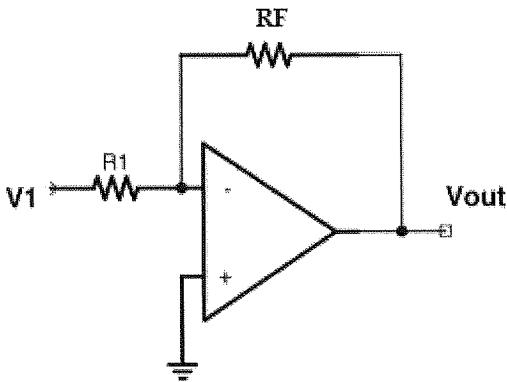
**VERIFY THIS WITH YOUR DVM.** You should have +9V from the MINUS of the Primary Bus to the PLUS of Secondary Bus 1, this will be  $V_{cc}$ . You should have -9V from the MINUS of the Primary Bus to the MINUS of Secondary Bus 2; this will be  $V_{ee}$ . If you don't have this correct, the rest of the lab won't work.

$V_{cc} =$  \_\_\_\_\_

$V_{ee} =$  \_\_\_\_\_

Once you have the above set up, place the operational amplifier in the breadboard so that it bridges the board gap between Secondary Buses 1 and 2. Make sure that the op-amp is properly oriented so that Pin 1 is in the upper left corner. Connect the appropriate pins to the appropriate powers to turn your op-amp on.

## Inverting Amplifier – Input 1



Setup the inverting amplifier circuit above. This is the same circuit that you did in Lab 8. Be very careful to use the correct pins, writing them on the diagram will help. DO NOT connect Vout to Ground. Wire this circuit using:

$R_1$ (feed-forward) = **1kΩ** resistor, what is the true resistor value?  $R_1$ =\_\_\_\_\_

$R_F$ (feed-back) = **2kΩ** resistor, what is the true resistor value?  $R_F$ =\_\_\_\_\_

What is the calculated gain based on the measured resistance values?

$$V_{out} = -\frac{R_F}{R_1} V_1 \quad G_{1\_calculated} = \text{_____}$$

Is the Op-Amp circuit an amplifier or an attenuator? Why?

Set up the DC power Supply to set  $V_1 = 1.5V$ . Use the Oscilloscope to verify the input  $V_1$  and measure the output  $V_{out}$ .

$V_1_{measured}$  \_\_\_\_\_  $V_{out\_measured}$  \_\_\_\_\_ \*\*\*\*\*

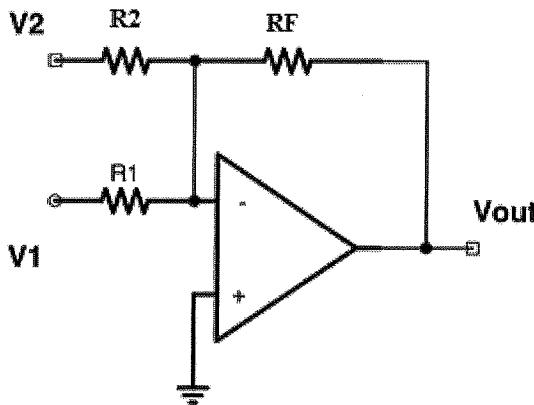
Compare the input and output signals and determine the gain.

$G_{1\_measured} = \text{_____}$

## Inverting Amplifier – Input 2

**Disconnect the DC power supply from the circuit, but do not turn it off or change the output.**

Now, modify the circuit to add the second input  $V_2$  as shown below.



Note that  $V_1$  and  $V_2$  are separate inputs and NOT CONNECTED. Use

$R_2$ (feed-forward) = **1kΩ** resistor, what is the true resistor value?  $R_2 = \underline{\hspace{2cm}}$   
 What is the calculated gain for the second input based on the measured resistance values?

$$V_{out} = -\frac{R_F}{R_2} V_2 \quad G_{2\_calculated} = \underline{\hspace{2cm}}$$

Will the circuit amplify or attenuate input  $V_2$ ?

Setup your function generator to output a 250Hz sine wave with Peak-to-Peak voltage of 1 volt.  
 Use the Oscilloscope to verify the input  $V_2$  and measure the output  $V_{out}$

$V_2\_{peak\_to\_peak\_measured} \underline{\hspace{2cm}}$

$V_{out}\_{measured} \underline{\hspace{2cm}}$

Compare the input and output signals and determine the gain.

$G_{2\_measured} = \underline{\hspace{2cm}}$

### Inverting Summer

The equation for an inverting summer with 2 inputs is  $V_{out} = -\frac{R_F}{R_1} V_1 - \frac{R_F}{R_2} V_2$ . The output is an inverted weighted average of the two inputs.

Reconnect the DC power supply as input  $V_1$  and use the Oscilloscope to measure the output  $V_{out}$ .

Press the Autoset button on the oscilloscope and then adjust the scale to see the whole signal on the screen. Press the cursor button, then the Type button and select Amplitude. Adjust the horizontal cursor line so that it corresponds to the  $V_{out}$  value when you had only Input 1, i.e. the value with the \*\*\*\*\* next to it. Make sure the oscilloscope is set to measure both Peak-to-Peak Amplitude and Mean.

Explain the output based on the results you saw from using each input independently and from the equation above.

Now, replace resistor  $R_1$  with the  $1k\Omega$  potentiometer so that you can vary the feed-forward resistance for input  $V_1$ . Vary the resistance of the potentiometer and watch what happens to the output on the oscilloscope screen.

What changes? Why?

Which Gain are you changing?

What happens when the potentiometer is at its maximum and minimum settings?

Remove the potentiometer and put resistor  $R_1$  back in place. Next replace resistor  $R_2$  with the  $1k\Omega$  potentiometer so that you can vary the feed-forward resistance for input  $V_2$ . Vary the resistance of the potentiometer and watch what happens to the output on the oscilloscope screen. What changes? Why?

Which Gain are you changing?

What happens when the potentiometer is at its maximum and minimum settings?

Remove the potentiometer and put resistor  $R_2$  back in place. Next, replace resistor  $R_F$  with the  $1k\Omega$  potentiometer so that you can vary the feed-back resistance. Vary the resistance of the potentiometer and watch what happens to the output on the oscilloscope screen.

What changes? Why?

Which Gain are you changing?

Does the signal ever reach saturation?

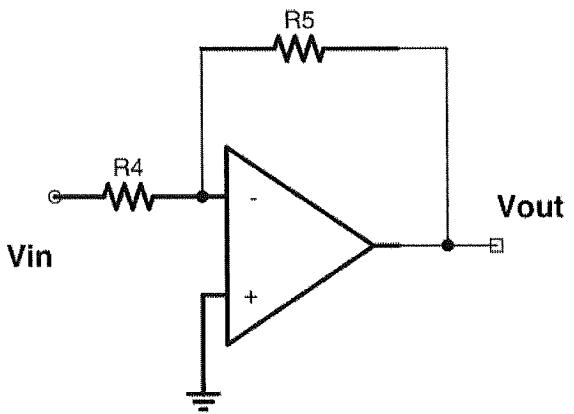
What happens when the potentiometer is at its maximum and minimum settings?

**Remove the potentiometer and put resistor  $R_F$  back in place.**

### **Using Two Op-amps to create a Non-Inverting Summer**

Disconnect the oscilloscope from the circuit. Leave both the DC Power Supply and the Function Generator ON.

Use a SECOND Op-amp to create the inverting amplifier below. Remember to power the op-amp. Be very careful to use the correct pins, writing them on the diagram will help. DO NOT Connect Vout to Ground.



Wire this circuit using:

$R_4$ (feed-forward) = **1.2kΩ** resistor, what is the true resistor value?  $R_4 = \underline{\hspace{2cm}}$

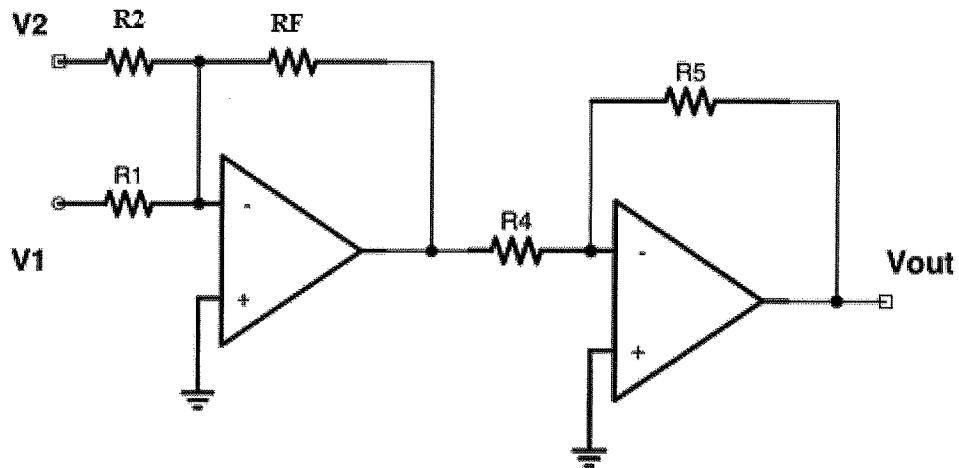
$R_5$ (feed-back) = **1kΩ** resistor, what is the true resistor value?  $R_5 = \underline{\hspace{2cm}}$

What is the calculated gain based on the measured resistance values?

$$V_{out} = -\frac{R_5}{R_4} V_{in} \quad G_{3\_calculated} = \underline{\hspace{2cm}}$$

Is the Op-Amp circuit an amplifier or an attenuator? Why?

Test your circuit by using the DC Power Supply for the input (without changing the input value) and measuring the output with the oscilloscope. When you are confident that the circuit works, disconnect the DC Power Supply and reconnect it to the first circuit as  $V_1$ , as you had it before. Next, connect the two circuits together as shown below by using the output of the first circuit as the input to the second circuit. Remember to properly connect  $V_1$  and  $V_2$  as you had before. DO NOT connect  $V_{out}$  to Ground.



Measure the output of the second circuit with the oscilloscope. What is the average of the signal?

Move the cursor to be the positive value of whatever you had before, i.e. if you had -2.5V move the cursor to +2.5V.

Does this make sense? Why?

Next, replace resistor  $R_5$  with the  $1k\Omega$  potentiometer so that you can vary the feed-back resistance. Vary the resistance of the potentiometer and watch what happens to the output on the oscilloscope screen.  
 What changes? Why?

Which Gain are you changing?

Does the signal ever reach saturation?

## **For extra credit**

Using any combination of resistors in your tool kit, design a circuit that takes these 2 inputs:  
2VDC and  $4\sin(400\pi t)$

And outputs a signal of the form (note the signs)

$$V_{out} = 6\sin(400\pi t) + 1$$

Build it and demonstrate that it works.

## **Electronics Laboratory 10 – Pre-Lab**

Read and complete the following Appendices

Pushbutton Switches

Relays

## Electronics Laboratory 10

### Logic Circuits and The Master Control Relay (CRM)

This Lab is an integrative experiment designed to put together pieces of what you have learned. You will start by examining the operation of each component. Then, you will put the complete system together and demonstrate a working Relay Logic Controller.

**Materials:** 5VDC coil relay, Two Momentary Pushbutton Switches, Red LED, Green LED, Two  $470\Omega$  Resistors

**The pushbutton switches** are your inputs to the circuit. When you press them you send a signal (on or off, i.e. a 1 or a 0 - it depends on how you have them wired) into your circuit/system.

**The Red and Green LEDs** are the outputs of the circuit. You will turn them on and off.

**The 5VDC coil relay** is the controller (the decision maker) for your circuit.

In this lab, you will go through the following 4 parts.

Part 1. Test Your Components:

- the Momentary Pushbutton Switches
- the LEDs (Light Emitting Diode)
- the Coil (& Contact) Relay

Part 2. Putting it All Together (Master Control Circuit)

**Caution:** It is important to take your time to carefully wire and check your boards. **Use your voltmeter to check** individual circuit parts as you wire them to verify that each section is working. This will save you time in the long run (by not having to go back, step-by-step to check for errors).

#### **Test Your Components**

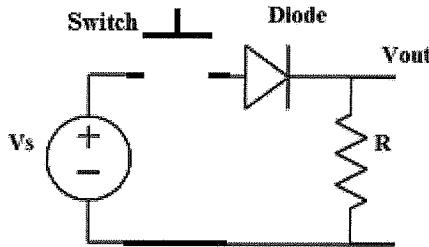
##### **Momentary Pushbutton**

Use your Ohm-meter to perform continuity tests on the various leads of the switch, and verify the appropriate changes when you push the button.

##### **LED**

Setup the RED LED as the output from one of your N.O. Pushbuttons as shown below. Wire the DC Power Supply to the input of the N.O. contact. Make sure the power supply is at its lowest voltage before you connect it to the circuit. Use a  $470\Omega$  resistor in SERIES with each LED; otherwise, they will burn out. **LEDs need very little current & voltage, so a resistor is needed in series to cut down both. Never connect a voltage source directly across the diode.**

Gradually increase the voltage to 5V. Now when you push the button, the LED should turn ON for as long as you hold the button down.



Now, move the circuit to the N.C. contact; when you push the button, the diode should turn OFF. Demonstrate for your Lab Instructor that you have TWO working diodes (RED and GREEN) and two working Momentary Pushbutton Switches

(Instructor Check-off) \_\_\_\_\_

### **The Coil (& Contact) Relay**

Now you will use the coil relay control component. The relay is like the push-button switch, but instead of manually pushing the button, the switching is electrically controlled.

Pins 1 and 16 of the relay are the “power” pins, with Pin 1 at a higher voltage than Pin 16. (Remember that the diagram in the Appendix is a BOTTOM VIEW.) When there is sufficient voltage across Pins 1 and 16 the relay will be activated. Use your DC voltage source and test your relay, i.e. hook up the DC voltage between Pins 1 and 16 (Ground on Pin 16 as Pin 1 must have the higher voltage). Start by turning the dial to minimum voltage and gradually increase the voltage until the relay is triggered. Check for continuity across the N.O. and N.C. connections when the relay is both activated and not activated.

### **Putting it All Together (Master Control Circuit):**

Because the circuit is complex, you will just build it first and then we will discuss its operation. Use the figures below and the demo model in the lab for reference. Note that the **RED LED should be on the RIGHT and the GREEN LED on the LEFT.**

First, disconnect any circuits that you currently have and make sure the power supply is turned off until the circuit is complete.

Start with the relay.

Connect relay Pins 4 and 13 to the source voltage from the DC Power Supply (shown as positive on the secondary buses in the figures above).

Connect Pin 16 to Ground (shown as negative on the secondary buses.)

Connect Pin 9 to a resistor in series with the **Green LED** to ground.

Connect Pin 11 to a second resistor in series with the **Red LED** to ground.

Now, add in the switches.

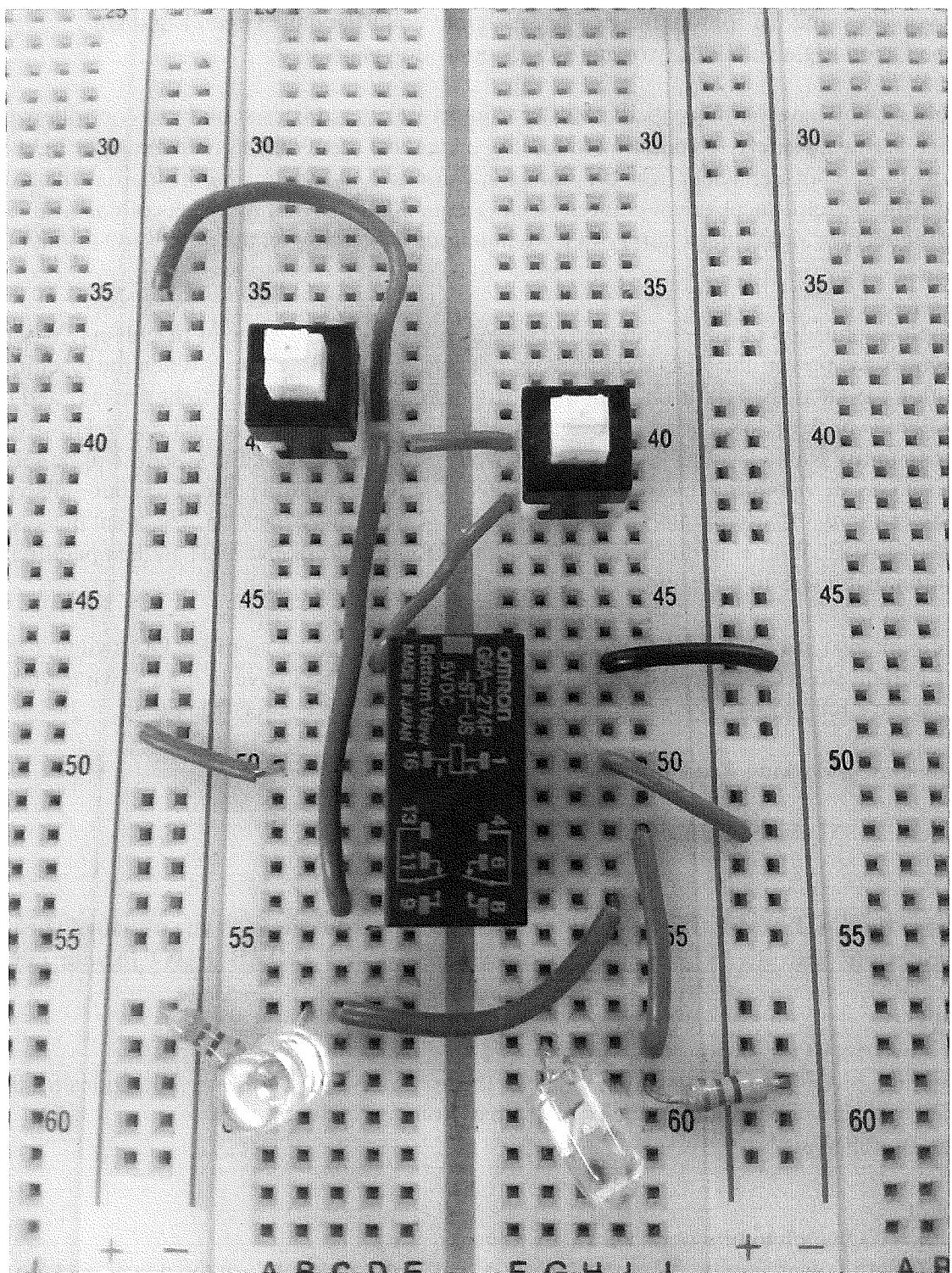
Connect Pin 5 of switch 1 to the Source voltage.

Connect Pin 4 of switch 1 to Pin 8 of the relay.

Connect Pin 4 of switch 1 to Pin 1 of switch 2.

Connect Pin 3 of switch 2 to Pin 1 of the relay.

Switch 1 will be considered your GO switch. Switch 2 will be considered your STOP switch.



**Operation should be as follows:**

1. When the board is powered up, the Red light should be on (Fail Safe - Stop is the Default).
2. If you Press the Stop Button, Nothing should happen.
3. If you Press the Go Button, the Red Light should go Off and the Green Light On.
4. The Green Light should stay on until the Stop Button is pressed.

The circuit you have built is called a Master Control Relay and is used on most controllers to provide hardware safety. In addition to turning on and off the red and green indicator lights, other N.O. and N.C. contacts from the same relay are used to turn on and off more demanding equipment. Electro-mechanical coil relays can frequently have 8 or more sets of contacts. PLCs (Programmable Logic Controllers) can have hundreds depending on the digital I/O (input/output) interface.

A simple application of this circuit could be supplying power to a piece of machinery. When the equipment has power, the RED light goes on indicating that it is "ready". When the machine is in operation the GREEN light is on. When you stop operation, the GREEN light goes off and the RED light is back on, indicating once again that the machine is powered and ready. However, the Master Control Relay is a Fail-Safe circuit as well. If the power fails, the system will be safe. Suppose you are operating the machine (GREEN light on) when the power goes out – disconnect the power supply. Everything is now off. When the power comes back on, you want the system to restart SAFELY – i.e. in the "ready" condition (RED light on), and NOT in the operating condition that it was in when power was lost.

This circuit also has a LATCH. This means that even when you let go of the GO switch, the GREEN light stays ON. The system is in operation and you don't have to hold down the button. Look at the circuit to see if you can figure out why. Hint: there is a feedback loop.

Demonstrate a working Red-Light, Green-Light circuit to your instructor.

(Instructor Check-off) \_\_\_\_\_

**Perform the FAIL SAFE TESTS below and answer the questions on the attached sheet.**  
**The questions with the \*\*\* next to them are CRITICALLY IMPORTANT to your next lab.**

**TEST1:**

Hold down the GO push button.

Depress the STOP push button slightly (as per the note above - this PB has 3 positions for the NC side - closed - open - and then closed again as you depress it all the way).

Which Button Wins?

**TEST2:**

Push the GO Switch to turn on the GREEN light.

Turn off your power supply. Wait until the light fades off, and then turn the power back on.

Which Button Wins?

## Questions:

- 1) What are the orientation features on the Coil Relay to know which is the top/bottom?
- 2) How many contacts (NO and NC) are available for the Coil Relay?
- 3) Why is Fail Safe operation important for a Master Control Relay?

**QUESTIONS 4-7 ARE CRITICALLY IMPORTANT TO YOUR NEXT LAB.**

**Answering the questions using descriptions rather than pin numbers will make next lab much easier.**

- 4) \*\*\*Identify the current path that includes the RED LED.
- 5) \*\*\*Identify the current path that includes the GREEN LED.
- 6) \*\*\*Why does pressing and holding the GO button energize the relay? Identify the current path that energizes the relay.

- 7) \*\*\*If the GO button is a momentary switch, why does the relay stay energized when the button is released? Identify the current path.

\*\*\*\*\*

**Be absolutely sure that you know the answers to the questions marked with the three asterisks \*\*\*. You will need this information for the next lab.**

\*\*\*\*\*

*Really, I mean it. I really really mean it. Get it?*

## Electronics Laboratory 11 – Pre-Lab

1. For Lab 11 on Ladder Logic & PLC's, each Lab Team will need to **bring in a Windows computer (a notebook or laptop)** in order to complete the lab. If you use a MAC, I cannot guarantee that the software will work. It might, but I haven't tested it, so you are on your own there. Give it a try AHEAD OF TIME.

**Prior to lab**, you should download, install, and test the PLC software from the TECO website as follows:

<http://teco.us.com/downloads.html>

From this page, you should select and download the **SG2 Quick Start Guide**. Then, select the **SG2 Programming Software**, and follow the prompts to install the software on your laptop. The Quick Start Guide will help you through this process.

Once you have the SG2 Client installed, launch the software and work your way through sections 3-5 of the Quick Start Guide. When you come to Lab 11, you should, at the very least, be able to open a new Ladder Diagram and understand how to place components.

### **Some Additional Ladder Logic Info & Tutorial eLinks:**

<http://www.electronicspal.com/plc/>

[http://en.wikibooks.org/wiki/Introductory\\_PLC\\_Programming](http://en.wikibooks.org/wiki/Introductory_PLC_Programming)

<http://www.thelearningpit.com/plc/psim/psim.html>

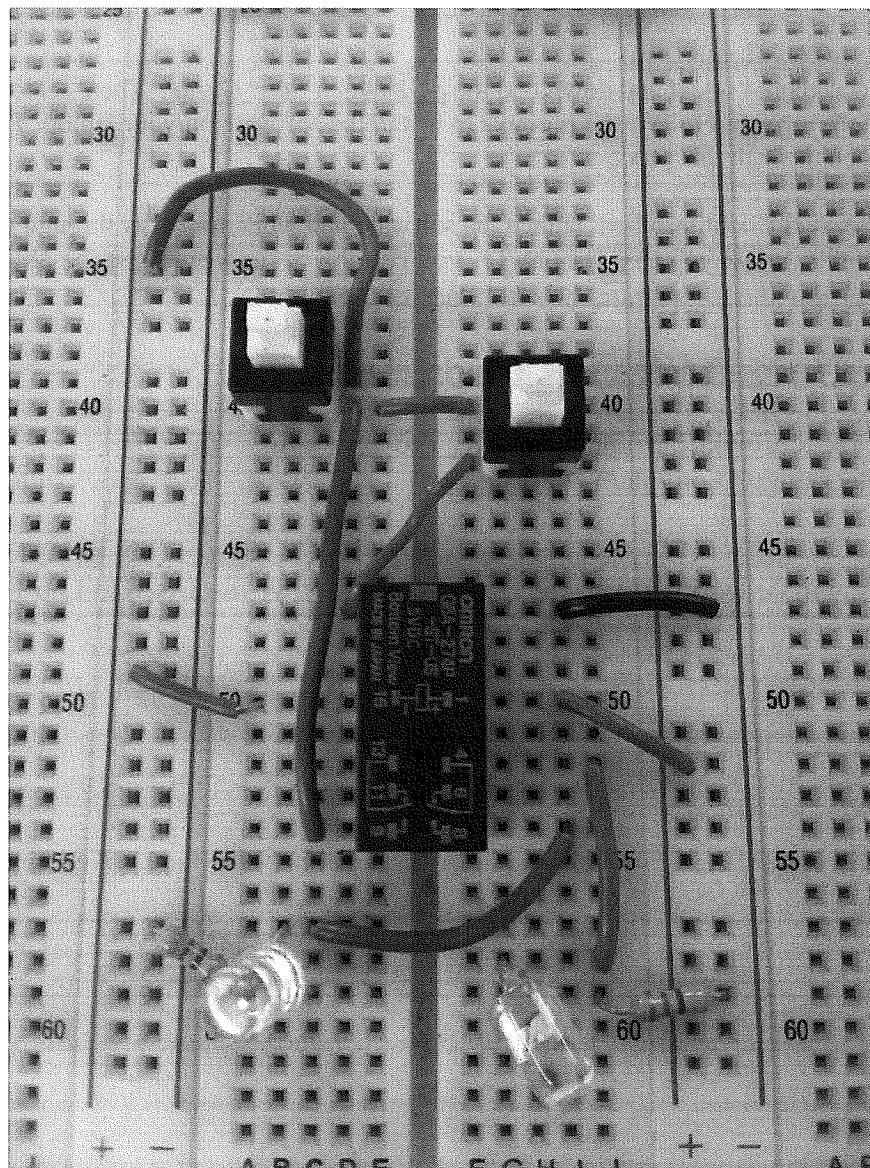
2. In Lab 11, we will be working with the current paths that we discussed in Lab 10. **REVIEW LAB 10!**

## Electronics Laboratory 11

### Programmable Logic Controllers (PLCs)

In the Previous Lab (Master Control Circuit), you used the following Hardware components to build a CRM (Master Control Relay) Logic Circuit:

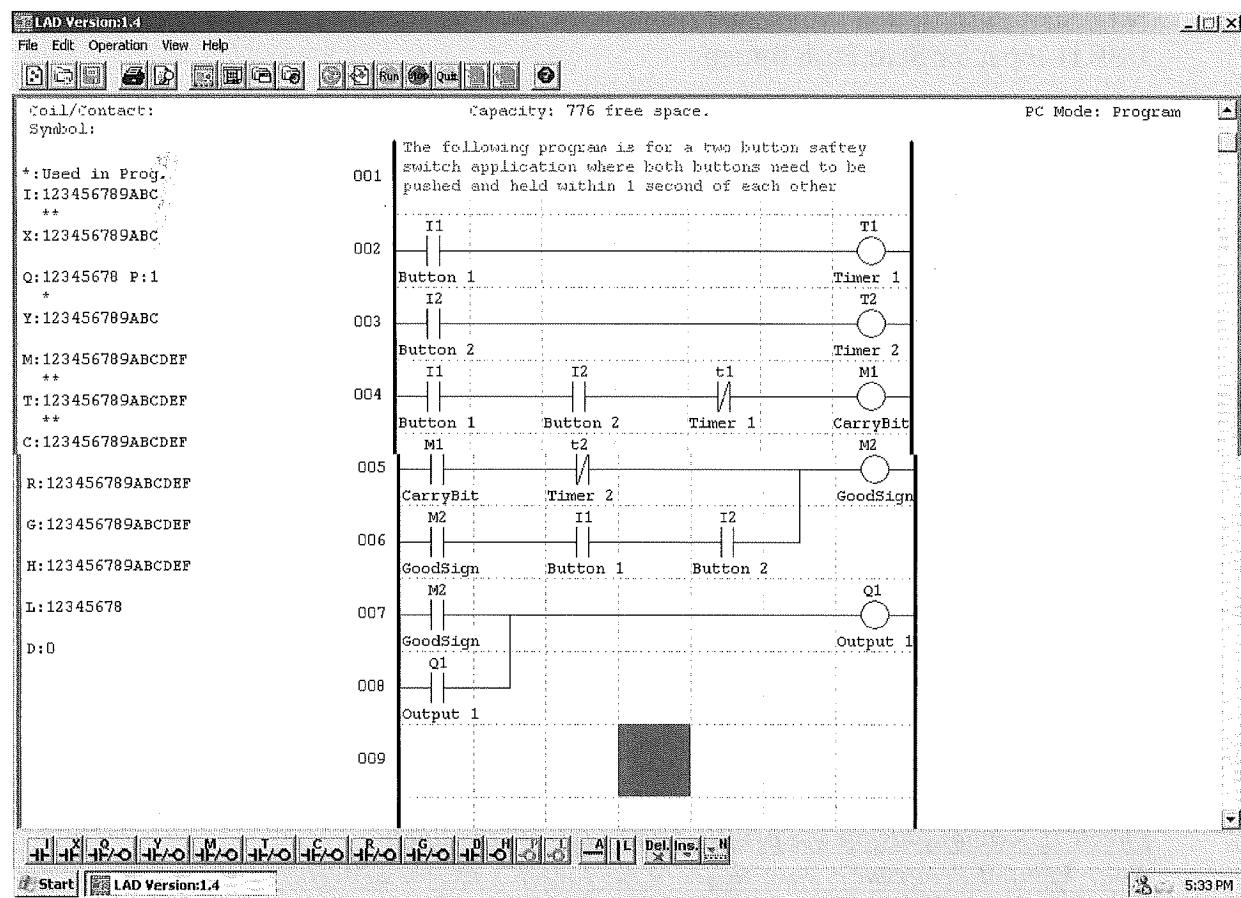
5VDC coil relay, Two Pushbutton Switches, Red LED, Green LED, Two 470 $\Omega$  Resistors



In this lab, we will replace the Hardware wiring with Software. The software is programmed using a Graphical Ladder Logic diagram and runs on a PLC. For this lab we are using the TECO-SG2 Series PLCs from B&B Electronics.

1. Your lab instructor will demonstrate the basic methodology and usage for PLC programming with the SG2 Client Software Package.
2. The SG2-20HR-A Series runs on 110 volt alternating current (110 VAC), and is capable of reading 12 simultaneous discrete inputs, and switching 8 simultaneous discrete outputs using the software code that runs internally on the unit microprocessor.

### SG2 Client Ladder Logic Programming Example Code:



## **Part I - PLC-CRM Relay Logic:**

Each team should have already downloaded and installed the software for the TECO SG2 Series PLCs.

Your job is to program and simulate the Ladder Logic for the Master Control Relay of the previous lab.

Use the following:

### **INPUTS from User to PLC**

N.O. Start Push Button (Momentary)

USE I1 Input for Start Push Button

N.C. Stop Push Button (Momentary)

USE I2 Input for Stop Push Button

### **OUTPUTS from PLC to User & System**

USE Q1 Output for Green Light

USE Q2 Output for Red Light

## **Part II – Create your own game**

Your job here is to create the ladder diagram for a pinball machine. The object of the game is to hit a certain number of specially placed but difficult to reach targets. You decide exactly what the player has to do to “be the hero.” You decide what happens when they do achieve “hero status.” You must use at least one timer, one relay, and one counter in your game. Create a Ladder Diagram for your pinball machine. Demonstrate to your lab instructor. Have fun!

## **Homework due at Lab 12**

1. Print out a copy of your Master Control Relay Logic Program. Be sure to label all of the physical components on the printout and indicate the latch.

Your Printed Program is your ticket to next week's exam. EACH STUDENT is responsible for their own ticket. If you do not bring your ticket, you will not be permitted to take the exam.

2. EMAIL a copy of your pinball machine ladder logic program to your instructor and include a brief explanation of how it works, i.e. what each of the inputs and outputs represent and how you “win” the game. Bonus points for the best game of the semester.

## Electronics Laboratory 12

### Comparators

**Read the entire lab before you set up the breadboard.**

**Do not tear down your circuit until your instructor has reviewed your work.**

#### **Deliverables**

At the end of the lab session, each team member should have their lab worksheet checked off by their instructor. There is also a homework assignment due next lab. Be sure to save all your lab handouts and worksheets for use in the final exam. Remember to clean-up your station.

Start by assembling the following components:

741 Op-Amp, two 9V batteries, one  $12\text{k}\Omega$  resistors, one  $100\text{k}\Omega$  resistor

**FIRST, review the Hitachi Spec Sheet Data in the appendix.** All the wiring diagram information you will need is spaced throughout the packet.

**IT IS IMPORTANT TO KNOW WHICH PIN OF THE OP-AMP IS WHICH.**

***Mis-wiring will likely destroy the device!***

1. What are  $V_{cc}$  and  $V_{ee}$  as referred to in the sheets?
2. Which pins are they?
3. Which pin takes the inverting input,  $V_{in}(-)$ ? The non-inverting input,  $V_{in}(+)$ ?
4. Which pin is the output?

**Caution:** In this lab we are increasing the complexity of the setups. Your breadboard will use two different voltage systems: one to power the Op-Amp and one to generate the input signals.

It is important to take your time to carefully wire and check your boards. Loose connections will cause noise or open circuits.

## **Part 1. Board Setup and Grounding**

In this lab, you will be multiple supplies: two 9-Volt batteries and the DC Power Supply. All Power Sources will need to be connected to a Common Ground, so they share the same 0 Volt reference and measure from the same starting point.

The TWO 9-Volt batteries will be the power supply to drive your Op-Amp (Operational Amplifier). As an active circuit, these amplifiers need power sources to bias the transistor circuits within the op-amp to operate in the linear mode. In our case we will use two 9 Volt batteries to get a total range of 18 Volts for linear operation of the amplifiers. (The Op-Amps actually saturate at a reduced range from the source voltage, so we are more likely to get only a 15 or 16 Volt total linear range).

1. Start by using a 9V battery to put 9V (on the PLUS) and ground (on the MINUS) on the left most secondary bus (Secondary Bus 1). Connect the grounds (MINUSES) of the Primary Bus and Secondary Bus 1.
2. Use the second 9V battery to set up the next Secondary Bus (Secondary Bus 2), with 9V on the PLUS and ground on the MINUS. Now, connect the PLUS of Secondary Bus 2 to the MINUS of the Primary Bus. This should create 18V between the MINUS of Secondary Bus 2 and the PLUS of Secondary Bus 1.
3. **VERIFY THIS WITH YOUR DVM.** You should have +9V from the MINUS of the Primary Bus to the PLUS of Secondary Bus 1, this will be  $V_{CC}$ . You should have -9V from the MINUS of the Primary Bus to the MINUS of Secondary Bus 2; this will be  $V_{EE}$ . **If you don't have this correct, the rest of the lab won't work.**

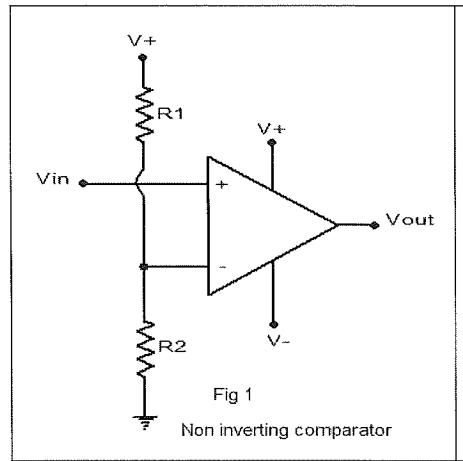
$$V_{CC} = \underline{\hspace{2cm}} \quad V_{EE} = \underline{\hspace{2cm}}$$

4. Once you have the above set up, place the operational amplifier in the breadboard so that it bridges (straddles) the board gap between Secondary Buses 1 and 2. Make sure that the op-amp is properly oriented so that Pin 1 is in the upper left corner.
5. The DC Power Supply will be the input voltage,  $V_{in}$ , for these experiments. Start by connecting the pins on the breadboard to the Primary Bus to provide source voltage on the plus side and ground on the minus side. Then, connect the Variable DC Power Supply to the pins.
6. **Have your instructor check your set-up before you complete the wiring of the circuit below.**

## Part 2. Basic Comparator Circuit – Non-Inverting

Use the attached Hitachi 741 spec sheets to setup the basic Op-Amp circuit above. Be very careful to use the correct pins. **It may help to write the pin numbers on the diagram above.**

1. Remember to provide +9 Volts and -9 Volts to the Op-Amp as a power source even though it is not shown explicitly in the diagram.
2. Using  $R_1 = 100 \text{ k}\Omega$  and  $R_2 = 12\text{k}\Omega$  for the circuit resistors. Measure the resistors before you wire them into your circuit. After measuring the resistors, wire this circuit.



What are the true resistor values?  $R_1 = \underline{\hspace{2cm}}$   $R_2 = \underline{\hspace{2cm}}$

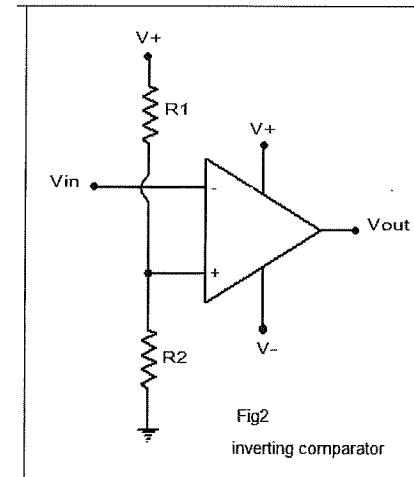
3. Calculate the gain of the voltage divider circuit formed by  $R_1$  and  $R_2$  with the measured resistance values.

**Gain<sub>calculated</sub>** =                   

4. Measure the voltage of the battery used for  $V_{cc}$
5. What is the calculated voltage at the nodal junction of  $R_1$  and  $R_2$ ?                     
This is now the reference voltage at the positive (+) input to the op-amp.
6. Connect the variable DC power supply to  $V_{in}$ . Vary the input voltage DC voltage source and record your data and calculations on the data sheet. Start with -4V input and then increase to +4V. For each data point, verify the input voltage with the DVM, and then measure the output voltage with the DVM.
7. **Neatly** sketch  $V_{out}$  vs  $V_{in}$  in the space provided below the data table for Non-Inverting Comparator
8. **Have your instructor check your results before you move onto the next section**

### **Part 3. Basic Comparator Circuit – Inverting**

1. Change the circuit input configuration to match the circuit shown – inverting amplifier.
2. Connect the variable DC power supply to Vin. Vary the input voltage DC voltage source and record your data and calculations on the data sheet. Start with -4V input and then increase to +4V. For each data point, verify the input voltage with the DVM, and then measure the output voltage with the DVM.
3. **Neatly** sketch Vout vs Vin in the space provided below the data table for Non-Inverting Comparator



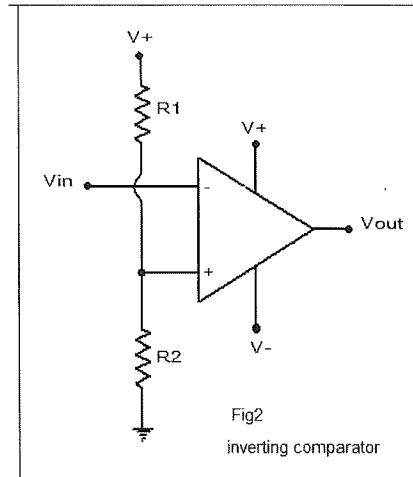
4. Comment on the differences between Vout vs. Vin plots for each of the comparator configurations.
5. **Have your instructor check your results before you move onto the next section.**

## **Part 4. Comparator Circuit – Rectangular Wave Generator**

In this section you will attempt to convert a sine wave to a rectangular wave. The sine wave peak voltage will be varied to produce a rectangular wave with varying pulse width and duty cycle. The frequency of the output will be unaffected by the peak voltage of the input voltage.

Definition:

$$\text{Duty Cycle} = \frac{\text{Pulse Width}}{\text{Period}}$$



1. Remove the variable DC power supply and replace it with a function generator. Set the function generator up to for a 1 kHz sine wave with a peak amplitude of 1 V.
2. Sketch the Vout, identifying the period and the pulse width of the output signal.

Pulse Width = \_\_\_\_\_

Period = \_\_\_\_\_

3. Vary the input voltage AC voltage source and record your data and calculations on the data sheet. Start with .5V input and then increase to +5V in .5 V increments. For each data point, verify the input voltage with the DVM, and then measure the output voltage with the DVM.
  4. Comment on the results for Vout as you varied the amplitude of Vin.
- 
5. **Have your instructor check your results before you tearing down your experiment.**

### Data Record Sheet: Non-Inverting Comparator

$V_{in\_target}$ (Volts)	$V_{in\_measured}$ (Volts)	$V_{out}$ (Volts)
-4.0		
-3.0		
-2.0		
-1.0		
0.0		
1.0		
2.0		
3.0		
4.0		

## Data Record Sheet: Inverting Comparator

$V_{in\_target}$ (Volts)	$V_{in\_measured}$ (Volts)	$V_{out}$ (Volts)
-4.0		
-3.0		
-2.0		
-1.0		
0.0		
1.0		
2.0		
3.0		
4.0		

## Data Record Sheet: Rectangular Wave Generator

$V_{in\_target}$ (Volts)	$V_{in\_measured}$ (Volts)	Pulse Width	Duty Cycle
.5			
1.0			
1.5			
2.0			
2.5			
3.0			
3.5			
4.0			
4.5			
5.0			

# Electronics Laboratory 13

## Comparator and Logic Design Project

**Read the entire lab before you set up the breadboard.**

**Do not tear down your circuit until your instructor has reviewed your work.**

### **Deliverables**

At the end of the lab session, each team member should have their lab worksheet checked off by their instructor. There is also a homework assignment due next lab. Be sure to save all your lab handouts and worksheets for use in the final exam. Remember to clean-up your station.

### **Part 1. Circuit Design**

1. Design a circuit using a comparator and an AND gate that will light an LED when a variable input voltage exceeds 2.5 VDC and a momentary push button switch is depressed.
  - a. Use a  $1\text{ k}\Omega$  potentiometer to help you vary the input DC voltage
  - b. Components at your disposal:
    - 741 op-amp
    - 1N4001 diodes
    - 2N2222 transistor
    - Green LED
    - Whatever value resistors you require to develop the proper voltages and currents
    - 9 V batteries
  - c. Do not include a variable DC supply or function generator in your design.
2. Draw the schematic of your circuit design below and have it approved by your instructor.

3. Assemble all the components for your design.
4. Measure all resistor values and note their values below. Make sure you associate the resistor reference number and the value for that resistor.

R1		R9	
R2		R10	
R3		R11	
R4		R12	
R5		R13	
R6		R14	
R7		R15	
R8		R16	

5. Create a layout for your circuit on the breadboard. **The simplicity and neatness of your layout will impact your grade for the next lab assignment.** Organize your breadboard so that your circuit occupies the middle of the portion of the breadboard. Establish busses for +9 VDC, -9 VDC, Ground and a bus for a variable input voltage.

**Review the Hitachi Spec Sheet Data in the appendix.** All the wiring diagram information you will need is spaced throughout the packet.

6. **Show your hand-drawn layout to your instructor before proceeding to the next step.**

## **Part 2. Board Setup and Grounding**

**IT IS IMPORTANT TO KNOW WHICH PIN OF THE OP-AMP IS WHICH.**  
***Miswiring will likely destroy the device!***

**Caution:** In this lab we are increasing the complexity of the setups. Your breadboard will use two different voltage systems: one to power the Op-Amp and one to generate the input signals.

It is important to take your time to carefully wire and check your boards. Loose connections will cause noise or open circuits. In this lab, you will be multiple supplies: two 9-Volt batteries and the DC Power Supply. All Power Sources will need to be connected to a Common Ground, so they share the same 0 Volt reference and measure from the same starting point.

The TWO 9-Volt batteries will be the power supply to drive your Op-Amp (Operational Amplifier). As an active circuit, these amplifiers need power sources to bias the transistor circuits within the op-amp to operate in the linear mode. In our case we will use two 9 Volt batteries to get a total range of 18 Volts for linear operation of the amplifiers. (The Op-Amps actually saturate at a reduced range from the source voltage, so we are more likely to get only a 15 or 16 Volt total linear range).

7. Start by using a 9V battery to put 9V (on the PLUS) and ground (on the MINUS) on the left most secondary bus (Secondary Bus 1). Connect the grounds (MINUSES) of the Primary Bus and Secondary Bus 1.
8. Use the second 9V battery to set up the next Secondary Bus (Secondary Bus 2), with 9V on the PLUS and ground on the MINUS. Now, **connect the PLUS of Secondary Bus 2 to the MINUS of the Primary Bus.** This should create 18V between the MINUS of Secondary Bus 2 and the PLUS of Secondary Bus 1.
9. **VERIFY THIS WITH YOUR DVM.** You should have +9V from the MINUS of the Primary Bus to the PLUS of Secondary Bus 1, this will be  $V_{CC}$ . You should have -9V from the MINUS of the Primary Bus to the MINUS of Secondary Bus 2; this will be  $V_{EE}$ .  
**If you don't have this correct, the rest of the lab won't work.**

$V_{CC} =$  \_\_\_\_\_

$V_{EE} =$  \_\_\_\_\_

10. **Once you have the above set up,** place the operational amplifier in the breadboard so that it bridges (straddles) the board gap between Secondary Buses 1 and 2. Make sure that the op-amp is properly oriented so that Pin 1 is in the upper left corner.
11. **Have your instructor check your set-up before you complete the wiring of the circuit below.**

### Part 3. Test For Functionality

9. Remember to provide +9 Volts and -9 Volts to the Op-Amp as a power source even though it is not shown explicitly in the diagram.
10. Adjust the 1 kΩ potentiometer so that the voltage to the + input to the op-amp is less approximately 0V.

Measure + input voltage \_\_\_\_\_

Is the LED on or off? \_\_\_\_\_

11. Adjust the 1 kΩ potentiometer so that the voltage to the + input to the op-amp is as large as you can make it.

Measure + input voltage \_\_\_\_\_

Is the LED on or off? \_\_\_\_\_

12. Depress the momentary switch. Does the LED go on when you depress the switch?

\_\_\_\_\_

13. Adjust the potentiometer slowly to where the output of the op-amp goes from a positive voltage to a negative voltage. Leave the potentiometer at that point then measure the voltage at the + input. This may take you a few attempts to get the exact point where this switch-over takes place. This is set-point voltage and should be approximately equal to the voltage at the – input to the op-amp.

Set point voltage \_\_\_\_\_

14. Fill in the table:

<b>Vin at + input</b>	<b>Switch Condition</b>	<b>LED Condition</b>
< set point voltage	open	
< set point voltage	open	
> set point voltage	closed	
> set point voltage	closed	

15. Modify the circuit so that the LED turn on when either  $V_{in}$  at the + input to the op-amp is greater than the set point voltage or the momentary switch is depressed.

16. Complete the table below with the OR condition.

<b><math>V_{in}</math> at + input</b>	<b>Switch Condition</b>	<b>LED Condition</b>
< set point voltage	open	
< set point voltage	open	
> set point voltage	closed	
> set point voltage	closed	

17. **Have your instructor check your results before you tearing down your breadboard**

## Electronics Laboratory 14

### Third-Order Filters and Op-Amp Isolation

**Materials:** 741 Op-Amps, two 9V batteries, three  $12k\Omega$  resistors, three  $0.1 \mu F$  capacitors

**FIRST, review the Hitachi Spec Sheet Data in the Appendix. IT IS IMPORTANT TO KNOW WHICH PIN OF THE OP-AMP IS WHICH.**

What are  $V_{cc}$  and  $V_{ee}$  as referred to in the sheets?

Which pins are they?

Which pin takes the inverting input,  $V_{in}(-)$ ? The non-inverting input,  $V_{in}(+)$ ?

Which pin is the output?

The lab will go through the following 4 main parts:

**Part 1. Board Setup and Grounding**

**Part 2. Op-Amp Isolation Amplifier (Voltage Follower)**

**Part 3. 3rd Order Hi-Pass RC Filter**

**Part 4. 3rd Order Hi-Pass RC Filter WITH Op-Amp ISOLATORS**

#### **Board Setup and Grounding**

This is the same setup as in the first Op-Amp laboratory.

In this lab, you will be using multiple power supplies: two 9-Volt batteries, the DC Power Supply, and the Function Generator (although you will not be using them all at the same time.) All Power Sources will need to be connected to a Common Ground, so they share the same 0 Volt reference and measure from the same starting point.

The TWO 9-Volt batteries will be the power supply to drive your Op-Amp (Operational Amplifier). As an active circuit, these amplifiers need a separate power source. In our case we will use two 9 Volt batteries to get a total range of 18 Volts for linear operation of the amplifiers. (The Op-Amps actually saturate at reduced range from the source voltage, so we are more likely to get only a 15 or 16 Volt total linear range). Start by using a 9V battery to put 9V (on the PLUS) and ground (on the MINUS) on the left most secondary bus (Secondary Bus 1). Connect the grounds (MINUSES) of the Primary Bus and Secondary Bus 1. Use the second 9V battery to set up the next Secondary Bus (Secondary Bus 2), with 9V on the PLUS and ground on the MINUS. Now, connect the PLUS of Secondary Bus 2 to the MINUS of the Primary Bus. This should create 18V between the MINUS of Secondary Bus 2 and the PLUS of Secondary Bus 1.

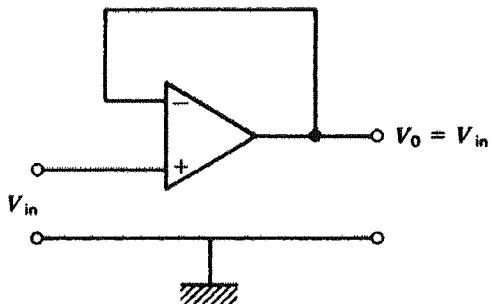
**VERIFY THIS WITH YOUR DVM.** You should have +9V from the MINUS of the Primary Bus to the PLUS of Secondary Bus 1, this will be  $V_{cc}$ . You should have -9V from the MINUS of the Primary Bus to the MINUS of Secondary Bus 2; this will be  $V_{ee}$ . If you don't have this correct, the rest of the lab won't work.

$V_{cc} =$  \_\_\_\_\_  $V_{ee} =$  \_\_\_\_\_

Once you have the above set up, place the operational amplifier in the breadboard so that it bridges the board gap between Secondary Buses 1 and 2. Make sure that the op-amp is properly

oriented so that Pin 1 is in the upper left corner. Connect the appropriate pins to the appropriate powers.

### Op-Amp Isolation Amplifier (Voltage Follower)



In this experiment, you will setup a high impedance Op-Amp circuit to ISOLATE an input signal from an output signal. This high impedance, simple circuit will transmit the voltage signal information through the Op-Amp, while not introducing any load (not drawing any significant current). These Isolators (Voltage Followers) are common in signal processing applications.

1. Use the attached Hitachi 741 spec sheets to setup the basic Op-Amp circuit above. Be very careful which pins you use. It may help to write the pin numbers on the circuit diagram above.
2. Remember to provide +9 Volts and -9 Volts to the Op-Amp as a power source even though it is not shown explicitly in the diagram above. Check the spec sheet to make sure you have these on the right pins.
3. Use your Variable DC Power Supply to input positive voltage signals to the Op-Amp and record the output in the data table. Start by connecting the pins on the breadboard to the Primary Bus to provide source voltage on the plus side and ground on the minus side. Then, connect the Variable DC Power Supply to the pins. What would happen if your  $V_{in}$  is 13V?
4. Reverse the Polarity of the Variable DC Power Supply leads to get negative voltage inputs. What happens if your  $V_{in}$  is -13V?
5. Are your  $\pm$ saturation voltages the same? Why or why not? Record your data and calculations below.

### Op-Amp Isolator (Voltage Follower)

$V_{CC}$  \_\_\_\_\_  
 $+V_{sat}$  \_\_\_\_\_

$V_{EE}$  \_\_\_\_\_  
 $-V_{sat}$  \_\_\_\_\_

$V_{inTarget}$ (Volts)	$V_{inMeasured}$ (Volts)	$V_{out}$ (Volts)
-10.0		
-8.0		
-6.0		
-4.0		
-2.0		
-1.2		
0.0		
1.2		
2.0		
4.0		
6.0		
8.0		
10.0		

Sketch  $V_{out}$  vs.  $V_{in}$

Observation Notes:

### Third Order Hi-Pass RC Filter:

BEFORE DOING THIS SECTION, REMOVE THE WIRE THAT CONNECTS THE INPUT VOLTAGE TO THE NON-INVERTING INPUT OF THE OP-AMP. LEAVE THE REST OF THE OP-AMP CIRCUIT AND ALL THE WIRING AS IS AND CHOOSE ANOTHER AREA ON THE BREADBOARD TO MAKE THE NEXT CIRCUIT.

Remove the DC power supply from the pins.

Using three  $12\text{ k}\Omega$  resistors and three  $0.1\text{ }\mu\text{F}$  capacitors, setup the hi-pass filter below on a different part of your breadboard. Measure the actual resistances and the actual capacitances and calculate the averages.

$R_1$ \_\_\_\_\_

$R_2$ \_\_\_\_\_

$R_3$ \_\_\_\_\_

$R_{ave}$ \_\_\_\_\_

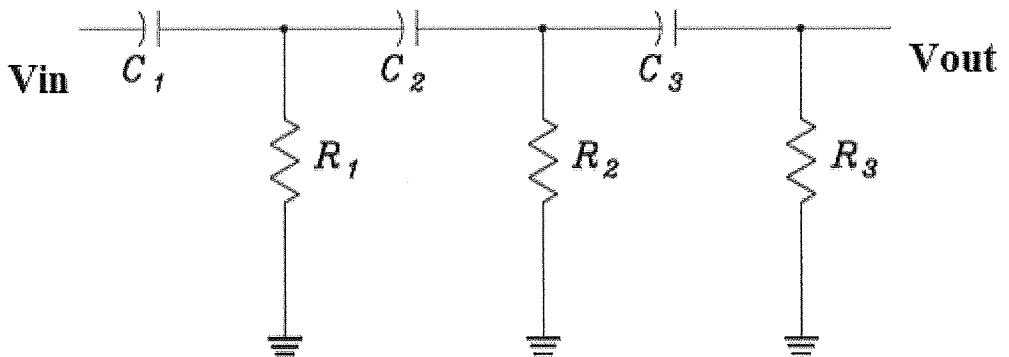
$C_1$ \_\_\_\_\_

$C_2$ \_\_\_\_\_

$C_3$ \_\_\_\_\_

$C_{ave}$ \_\_\_\_\_

Calculate the corner frequency for this filter using  $R_{ave}$  and  $C_{ave}$ .  $f_c = \frac{1}{2\pi R_{ave} C_{ave}}$



Setup your function generator to output a fixed sine wave at the corner frequency that you just calculated, with Peak-to-Peak voltage of 4 volts, similar to what you did in the lab for the first order filters. Bring the signal from the Function Generator (instead of the Variable DC Power Supply) onto the pins of the breadboard. Once you have set up the input sine wave, **LEAVE the AMPLITUDE of the input wave FIXED at 4 Volts Peak-to-Peak.**

Now, move the oscilloscope probe to measure the output signal of the THIRD High Pass Filter, i.e. measure the voltage across Resistor 3. Vary the input frequency around the corner frequency and measure the output signal as you did in the filters lab. Record the data in the table. Start at the corner frequency and increase the frequency until you complete the bottom of the table. Then, return to the corner frequency and decrease the frequency for the remaining data. Note that you may get some unstable readings.

### Third Order High Pass Filter Data

$R_{ave}$  = \_\_\_\_\_

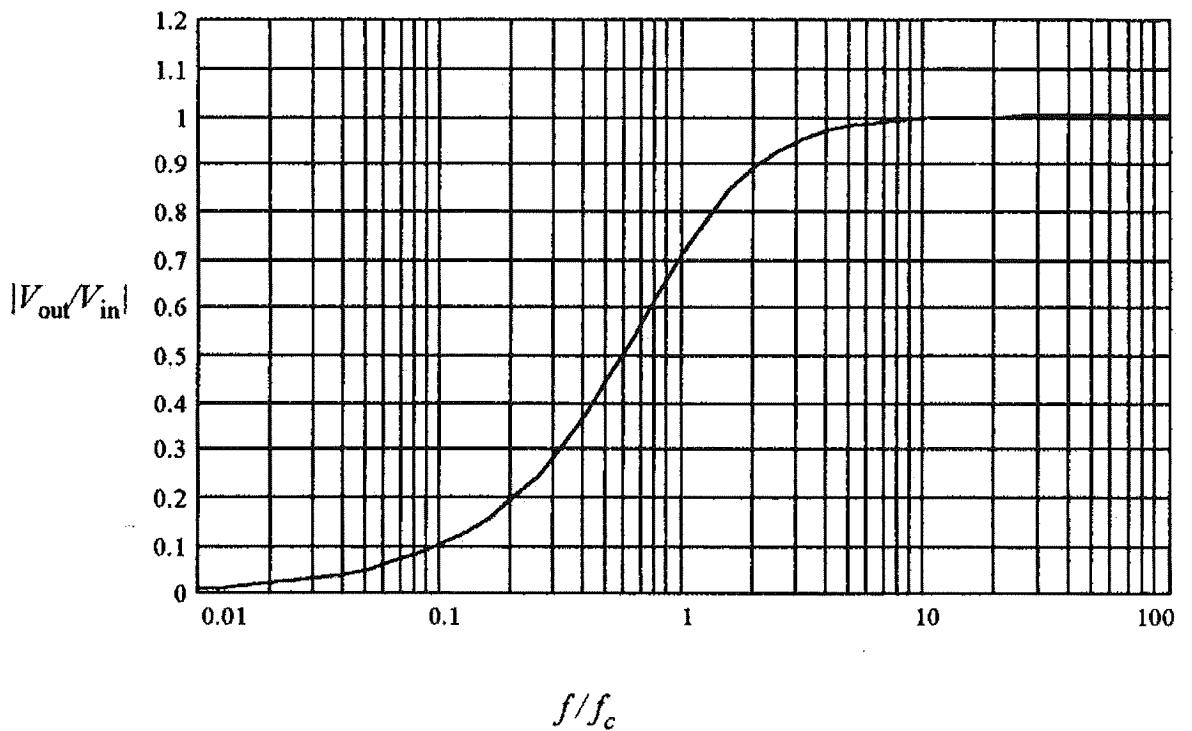
$C_{ave}$  = \_\_\_\_\_

Calculated corner frequency;  $f_c$  = \_\_\_\_\_

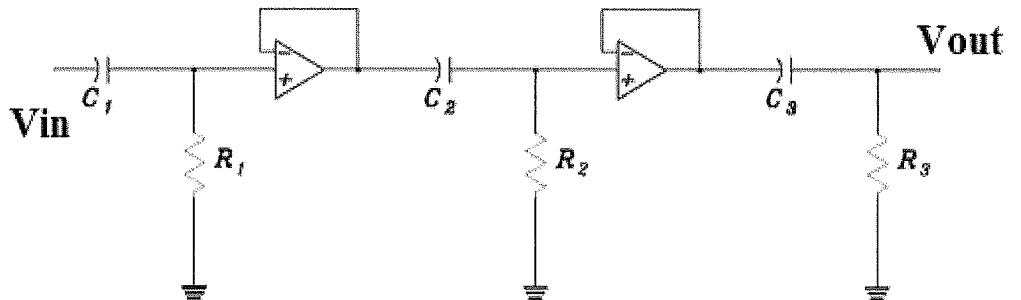
$$fc = \left( \frac{1}{2\pi R_{ave} C_{ave}} \right)$$

$V_{in}=V_s$ (volts, P-to-P)	Freq Ratio $F_R = f/f_c$	Target Freq $f$ (Hz)	Actual Freq $f$ (Hz)	Vout (volts) Peak-to- Peak	Mag Ratio = gain = $\left  \frac{V_{out}}{V_{in}} \right $
4	0.1				
4	0.5				
4	0.8				
4	0.9				
4	1.0				
4	2				
4	3				
4	5				
4	7				
4	10				
4	20				
4	30				

The figure below shows the ideal curve of a FIRST-order high pass filter. Make a rough sketch of your data on the plot below. Note that you are plotting frequency ratio on the x-axis with a LOG scale and GAIN on the y-axis. How well does your THIRD-order filter match up to the ideal curve for a FIRST-order filter? What is different? Why?



### Third Order Hi-Pass RC Filter WITH Op-Amp ISOLATION



**THIS IS A COMPLEX CIRCUIT.** Be careful setting it up.

Your job here is to insert an isolator BETWEEN each of the high pass filters as shown above. Put the isolators between Secondary Busses 1 & 2 and remember to power the op-amps. Use wires to make the connections. The output of the 1<sup>st</sup> filter should be connected to the non-inverting input to the isolator. The output of the isolator should then become the input to the second filter. The output of the 2<sup>nd</sup> filter should be connected to the non-inverting input of the next isolator. The output of the 2<sup>nd</sup> isolator should then become the input to the third filter. This creates a no-load filter circuit i.e. the second filter does not load the first because it is isolated by the op-amp isolator circuit; likewise for the third stage not loading the second.

**HAVE YOUR INSTRUCTOR CHECK YOUR CIRCUIT BEFORE YOU BEGIN TO TAKE DATA.**

Once again, make sure your function generator will output a fixed sine wave with Peak-to-Peak voltage of 4 Volts, and verify that at various frequencies the peak-to-peak amplitude stays constant. Complete the data table as you did before. Start at the corner frequency and increase the frequency until you complete the bottom of the table. Then, return to the corner frequency and decrease the frequency for the remaining data. Note that you may get some unstable readings.

### **Third Order High Pass Filter with Op-Amp ISOLATOR Data**

$R_{ave}$  = \_\_\_\_\_

$C_{ave}$  = \_\_\_\_\_

Calculated corner frequency;  $f_c$  = \_\_\_\_\_ 
$$f_c = \left( \frac{1}{2\pi R_{ave} C_{ave}} \right)$$

$V_{in}=V_s$ (volts, P-to-P)	Freq Ratio $F_R = f/f_c$	Target Freq $f$ (Hz)	Actual Freq $f$ (Hz)	Vout (volts) Peak-to- Peak	Mag Ratio = gain = $\left  \frac{V_{out}}{V_{in}} \right $
4	0.1				
4	0.5				
4	0.8				
4	0.9				
4	1				
4	2				
4	3				
4	5				
4	7				
4	10				
4	20				
4	30				

Make a rough sketch of your data on the SAME figure that you plotted your data on before. Be sure to label the two curves that you plot.

How well does your THIRD-order filter WITH op-amp isolation match up to the ideal curve for a FIRST-order filter? What is different?

## APPENDICES

## **Breadboards**

In Electronics Lab, you will use breadboards to make your circuits.

Watch the following you-tube videos to refresh your memory on how breadboards work.

Introduction to Breadboards by electroninstructor, Parts 1 and 2

<http://www.youtube.com/watch?v=oiqNaSPTI7w>

<http://www.youtube.com/watch?v=Mq9XMNsoAd8&feature=fvwrel>

Electronics – Breadboards by Science Online

<https://youtu.be/k9jcHB9tWko>

Define An Electrical Busbar .

## Capacitors

Since you have passed Engine Physics II, you should be familiar with capacitors and be able to answer the questions below.

A capacitor is a passive electronic component. What is it used for?

What is the unit of measure for capacitance? Is ONE of these units a lot of capacitance or only a little?

In general, capacitors usually have a larger tolerance than resistors, meaning that the actual value can have a wide range around the nominal value. The measured value should be compared to the nominal value using the percent difference equation below.

$$\% \text{ Difference} = \frac{\text{measured} - \text{nominal}}{\text{nominal}} \times 100\%$$

Write the equations for two capacitors in series.

Write the equation for two capacitors in parallel.

What is the governing equation that relates the current through and voltage across a capacitor?

Note that the only variables should be I, V, and C. Hint: The equation has a time derivative.

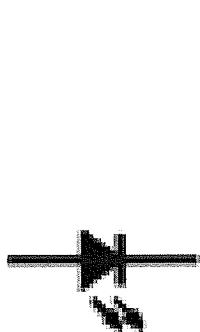
## Diodes and LEDs

A diode is an electrical component that only allows current to flow through it in one direction. It can be viewed essentially as an electronic check valve. The symbol below for a diode gives a hint as to which direction the current can flow. Flow is allowed in the direction of the arrow, i.e. from the side of the triangle to the point of the triangle. Flow is blocked when tried in the opposite direction by the “wall” that is encountered. Diodes are usually designed to have a fixed voltage drop across them regardless of the current going through them and are most often used for circuits with low power. For circuits with higher power a rectifier circuit is used. Physically, diodes often resemble resistors, so be careful when digging through a box of components to not get them confused. Instead of color bands, the positive and negative sides of the diode should be indicated by + and – symbols, respectively. For current to flow through the diode, the positive side should have the higher voltage.

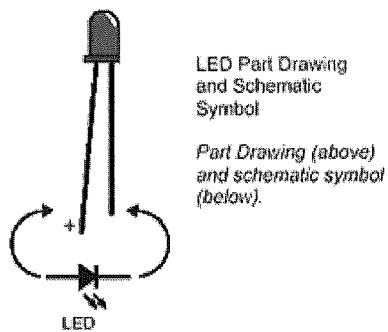


Diode

A very common component in electronics is the Light Emitting Diode or LED which can be thought of as a tiny light bulb. *Functionally* diodes and LEDs are the same, and the difference is only that when the current flows through the LED, the light goes on. (The chemistry inside LEDs can be quite different from that inside a standard diode.) If an LED is ON, then there is current flowing through it. If the LED is OFF, then there is no current flowing through it. The electrical symbol for an LED is shown below – the two small arrows indicating that light is being emitted. The leads on an LED are usually different lengths and for the current to flow through the LED, the longer lead should be at the higher voltage (positive side). Some LEDs also have a flat side that represents the side that should have the lower voltage (negative side).



Light Emitting Diode (LED)



In this course, we will use LEDs for all diode applications, and may take the shortcut of just using the diode symbol instead of the LED symbol.

LEDs come in all different colors depending on the chemical properties of the materials used to make them. Check out this website:

[www.allaboutcircuits.com](http://www.allaboutcircuits.com)

Education -> Volume III Semiconductors, Chapter 3 Diodes and Rectifiers -> Special Purpose Diodes

## Figures and Tables

Periodically through this course, you will be asked to create figures and tables of the data that you collect. Below are the guidelines for how to display this in your homework. The rules are the same for ALL technical writing, so you can use this information when you write lab reports for Strength Lab or Thermo/Fluids Lab too. When writing reports you only have 3 options for “stuff” that goes in the report – text, tables, and figures. Text is self-explanatory and includes not only the body of the report, but headings and the like. Tables are exactly that, tables of data that were taken during the experiment. Everything else is a figure, period. Charts, graphs, pictures, etc., anything that is not text and not a table is a figure, period.

### Figures

All figures have captions that go directly BELOW the figure. A *caption* is a brief phrase that states what the figure is. For example:

Figure 1. Circuit diagram of inverting amplifier.

Figure 2. Capacitor Voltage vs. Time.

The caption starts with the word Figure and the number of the figure, followed by a period. The phrase is capitalized like a sentence and is followed by a period. If plotted variables are included in the phrase, they should be capitalized. No other punctuation is necessary. The captions do not need to be long and complicated, they just provide a statement about what the figure is. Each figure caption phrase should be unique. Figures are numbered in the order in which they appear in the document. **FIGURE CAPTIONS GO BELOW THE FIGURE.**

The axes of a figure should be labeled with BOTH the variable name and the units, for example “Time (seconds)”. The data should be plotted as BOTH points and connect-the-dots, unless otherwise specified. If you use Excel, this means you should use the xy-scatter plot. When plotting *This* vs. *That*, *This* goes on the *y-axis*, and *That* goes on the *x-axis*.

### Tables

All tables have titles that go directly ABOVE the table. A *title* is a brief phrase that states what the table is. For example:

Table 1. Capacitor Charging Data.

The title starts with the word Table and the number of the table, followed by a period. The phrase is capitalized and is followed by a period. No other punctuation is necessary. The titles do not need to be long and complicated, just provide a statement about what the table is. Each table title should be unique. Tables are numbered in the order in which they appear in the document. Table numbering and Figure numbering are independent of each other. **TABLE TITLES GO ABOVE THE TABLE.**

The column headings of a table should be labeled with BOTH the variable name and the units, for example “Time (seconds)”. Only the data numerical values themselves should be in each datum cell. The data should be given with a reasonable number of significant digits, for this course, no more than 4, but often less than 4 will be used. Try to keep the same number of significant digits in the whole data column, but not necessarily in the whole data table.

**NEVER SPLIT A DATA TABLE OVER TWO PAGES.**

**DO NOT** put figures and tables side-by-side on a page, place them one after the other. The first one mentioned in the text is the one that should appear first.

## Grading

I tend to be a stickler for formatting of figure and tables. **FORMAT MATTERS. DETAILS MATTER. CARELESSNESS WILL COST YOU.**

For each figure, deductions are as follows

- Each variable or unit that is not given – 1 point. Thus, a figure with no axes labels will lose 4 points.
- Each figure caption that is missing – 2 points.
- If the figure caption is included, but incomplete or incorrect – 1 point.

Think about this. It could prove to be VERY COSTLY if there are several figures displayed incorrectly or if the homework is not worth a lot of points.

For each table, deductions are as follows

- Each variable or unit that is not given – 1 point. Thus, EACH column in the table with no heading will lose 2 points.
- Each table heading that is missing – 2 points
- If the table title is included, but incomplete or incorrect – 1 point.

Think about this. It could prove to be VERY COSTLY if there are several tables displayed incorrectly. If there is a table with several columns, it could be VERY COSTLY to not have the columns right.

\*\*\*\*\*

**THEORETICALLY, IT IS POSSIBLE TO LOSE ALL THE HOMEWORK POINTS FOR NOT HAVING THE FIGURE/TABLE FORMATS CORRECT. PAY ATTENTION!**

\*\*\*\*\*

# Laboratory Equipment

## Multimeter (or Digital Volt Meter - DVM)

A multimeter is used to make various electrical measurements, such as AC and DC voltage, AC and DC current, and resistance. It is called a multimeter because it combines the functions of a voltmeter, ammeter, and ohmmeter. Multimeters may also have other functions, such as diode and continuity tests.

**Important note:** The most common mistake when using a multimeter is not switching the test leads when switching between current sensing and any other type (i.e., voltage or resistance). It is critical that the test leads be in the proper jacks for the measurement you are making.

### **Safety Information**

Be sure the test leads and rotary switch are in the correct position for the desired measurement. Never use the meter if the meter or the test leads look damaged.

Never measure resistance in a circuit when power is applied.

Never touch the probes to a voltage source when a test lead is plugged into the 10 A or 300 mA input jack.

To avoid damage or injury, never use the meter on circuits that exceed 4800 Watts.

Never apply more than the rated voltage between any input jack and earth ground (600 V).

Be careful when working with voltages above 60 V DC or 30 V AC rms. Such voltages pose a shock hazard.

Keep your fingers behind the finger guards on the test probes when making measurements.

To avoid false readings, which could lead to possible electric shock or personal injury, replace the battery as soon as the battery indicator appears.

### **Input Jacks**

The black lead is always plugged into the common terminal. In Electronics Black means ground and Red means Hot. In home wiring, Black means Hot, White means neutral, and Green is the “extra” ground. KNOW THIS! Mistaking Hot for Ground is usually “bad.”

## DC Power Supply

Both the DC Power Supply and the Function Generator are capable of supplying voltage and power to the circuits you build. The DC Supply has a variable voltage that ranges from approximately 1.1 – 30 volts. Note that even when the dial is turned down as far as it will go, there is still some voltage being supplied.

## Function Generator Supply

The Function Generator can supply a variety of periodic wave-forms over a range of Frequencies and Voltage Amplitudes. The function generator is capable creating 3 different signal waveforms – sine waves, square waves, and triangle waves. Both the frequency and the amplitude of the signal can be adjusted. Make sure you are using the right cable, i.e. the one with the red and black alligator clips. Consider the following definitions below:

**Signal Period:** the period ( $T$ ) of a periodic signal  $x(t)$  is the smallest time interval  $T$  for which  $x(t+T)=x(t)$  holds for any time  $t$ . In other words, the time it takes for one complete fundamental

unit of the periodic signal. A fundamental segment of a periodic signal's waveform spans one period of the signal's waveform.

**Frequency:** the frequency ( $f$ ) of a periodic signal is the number of periods (cycles or fundamental units) through which the signal goes per second. The frequency (cycles/second or Hz) is the inverse of the period (seconds/one cycle).  $f = \frac{1}{T}$

**Amplitude:** In electronics, there are actually THREE different amplitudes to keep track of and it is important to know which one you are talking about. When the term "amplitude" is used, it usually refers to standard amplitude, i.e. the distance between the average value and the peak value of a periodic signal. The "peak-to-peak amplitude" is the distance between the peak and the trough. For a sine wave, the peak-to-peak voltage ( $V_{PP}$ ) is twice the voltage amplitude. The third amplitude is the "RMS amplitude." RMS stands for Root Mean Square and refers to how the amplitude would actually be calculated (essentially, the procedure consists of squaring all the positive and negative points on a waveform graph, averaging those squared values, then taking the square root of that average to obtain the final answer). **RMS is NOT AN AVERAGE value of the signal.**

Consider this, the average value of a sine wave is zero, but the RMS value is not zero. The importance of the RMS amplitude is that it is a measure of the amount of power in a varying signal. One way to think about it is this: The RMS Amplitude represents the value of a constant DC signal that contains ***the same amount of power*** as the varying signal. Read that sentence again and think about it! IF AND ONLY IF you are working with a pure sine wave, there is a simple mathematical relationship between the RMS amplitude and the standard amplitude. If you have anything other than a pure sine wave, you need to do the root mean square calculation to get the right value of the RMS amplitude. For more on this, checkout the Measurement of AC Magnitude portion of the textbook at [www.allaboutcircuits.com](http://www.allaboutcircuits.com)

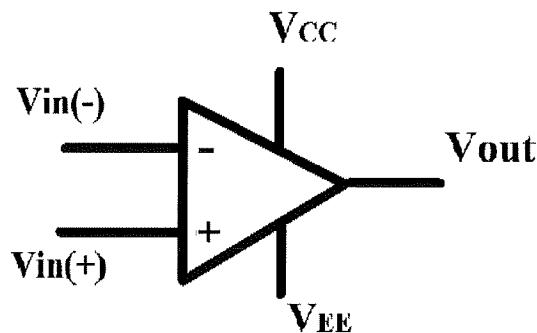
**DC Offset of a periodic signal:** the quantity representing the average signal value calculated over one period of the signal (constant voltage component of a voltage waveform).

## Oscilloscope

This laboratory uses a digital oscilloscope which can not only display the measured signal, but also provides information about the signal such as frequency, period, and amplitude. Also, the AUTOSET button will enable the oscilloscope to automatically adjust the ranges of the X and Y scales to show the measured signal. Remember, the Oscilloscope only measures the signal. You cannot change the amplitude or frequency of a signal using the oscilloscope; you can only look at the signal. Before using the oscilloscope, you should make sure that it is properly compensated.

## Operational Amplifiers

The operational amplifier (op-amp) is arguably the most useful single device in analog electronic circuitry. With only a handful of other components, the operational amplifier can be used in a wide variety of analog signal processing circuits such as amplifiers, attenuators, integrators, differentiators, and summers. Op-amps are considered active components because you must turn them on. Passive components such as resistors and capacitors react passively to the signals that are applied to them, but the operational amplifier only works if you supply power to it, i.e. you turn it on. The symbol used to designate an operational amplifier is shown below. Power is supplied to the op-amp with the terminals labelled  $V_{CC}$  and  $V_{EE}$ . Many operational amplifiers use balanced (equal positive and negative) supply voltages and sometimes the power terminals are called  $+V_s$  and  $-V_s$ . Note that the op-amp has TWO input terminals, the inverting and the non-inverting input terminals, labeled with - and + respectively. Don't worry too much about the significance of the names just yet. The op-amp has only ONE output terminal.



The last 2 pages of this section show the spec sheet of the 741 Operational Amplifier that we will use in Lab. This is the most common op-amp on the market. The first page of the spec sheet shows the pin connections, i.e. the pin numbers that correspond to the terminals. Remember that the numbering convention in electronics is to start in the upper left corner and work down and around in a counter-clockwise U-shape (see the Section on Pushbutton Switches). The op-amp will either have dot on a small cut-out to determine which way is "up." The spec sheet shows the cut-out.

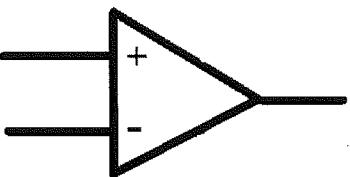
Although the op-amp physically has 8 pins for this course we will only use 5. Pins 1 and 5 are used to eliminate any offset that may occur due to small inaccuracies in manufacturing. Ideally when the inverting and non-inverting inputs are the same, the output should be zero. If they are not, a small variable resistor is connected between pins 1 and 5 and tuned until the output is in fact null. This is not a complicated process, but in the interest of time, we skip it in our experiments. Pin 8, although physically present on the part, is not actually connected to

anything. This is essentially the result of the manufacturing process. It is simply easier to put the same number of pins on both sides rather than 4 on one side and 3 on the other.

Using the spec sheet, label the diagram above with the appropriate pin numbers. (Look at the table on the second page of the spec sheet for information about the power-supply voltage.)

Since even those with a nominal amount of experience with electronics understand that an op-amp must be powered in order to work, usually the power terminals are omitted from a circuit diagram. The op-amps still **MUST** be powered, and the omission is simply to reduce clutter in the wiring diagram. Sometimes also the op-amp is drawn with the non-inverting input terminal on the top and the inverting input terminal on the bottom, so look carefully at the symbol.

Label each terminal of the symbol below with **BOTH** pin number AND function.



The internal circuitry of the op-amp is shown by the diagram on page 2 of the spec sheet. We will not analyze this circuit or follow the current loops through it, we will simply connect the op-amp the way we want to and concern ourselves more about how it can be used. However, take a few moments to look at the circuit. At this point there should not be any part of the circuit that you are not familiar with.

How many of each of the following do you see?

Resistors \_\_\_\_\_

Current Sources \_\_\_\_\_

Capacitors \_\_\_\_\_

Transistors \_\_\_\_\_

Diodes \_\_\_\_\_

One key to the usefulness of these little circuits is in the engineering principle of feedback, particularly *negative* feedback, which constitutes the foundation of almost all automatic control processes. Most, but not all, circuits made with op-amps have negative feedback. For us this means that feedback will go from the output terminal to the non-inverting (negative) input terminal.

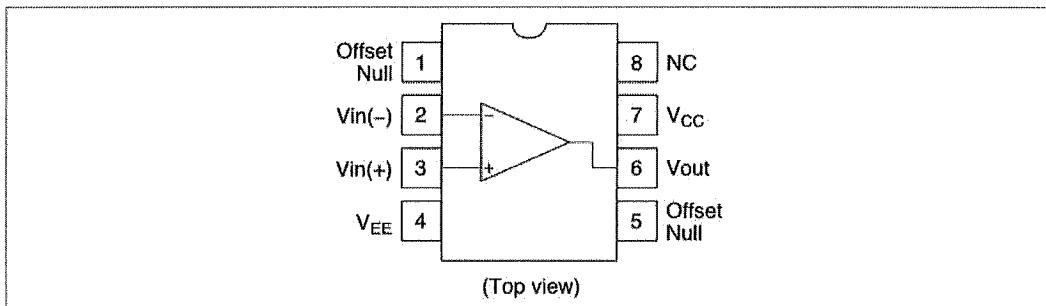
## Hitachi HA17741/PS (741 Op-Amp)

**General-Purpose Operational Amplifier:** The HA17741/PS is an internal phase compensation high-performance operational amplifier that is appropriate for use in a wide range of applications in the test and control fields.

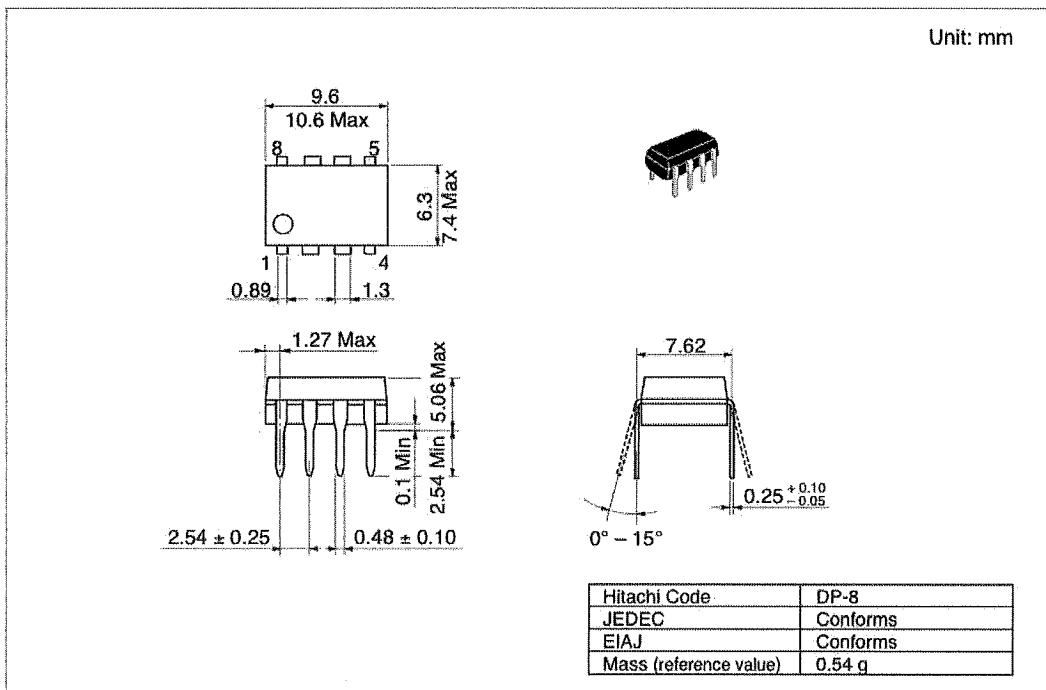
### Features

- High voltage gain : 106 dB (Typ)
- Wide output amplitude :  $\pm 13$  V (Typ) (at  $R_L \geq 2$  k $\Omega$ )
- Shorted output protection
- Adjustable offset voltage
- Internal phase compensation

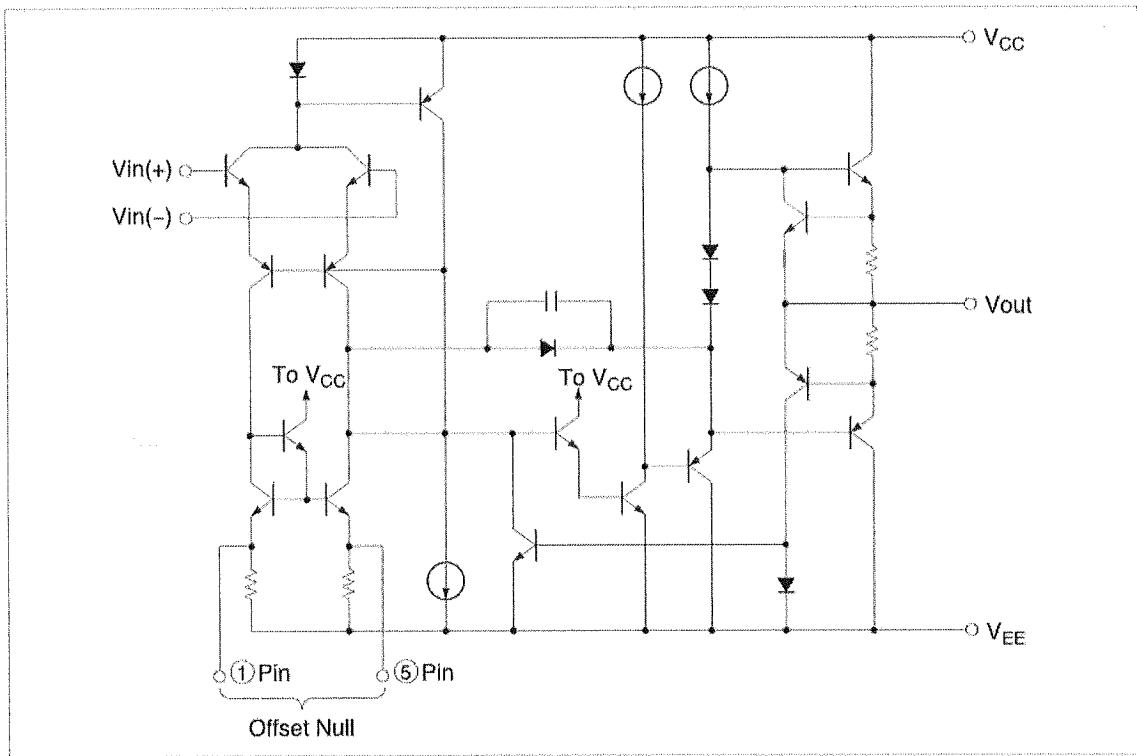
### Pin Arrangement



### Package Dimensions



## Circuit Structure



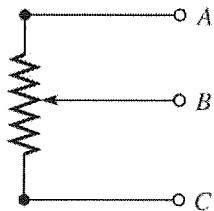
## Absolute Maximum Ratings (Ta = 25°C)

Item	Symbol	Ratings		Unit
		HA17741PS	HA17741	
Power-supply voltage	V <sub>cc</sub>	+18	+18	V
	V <sub>ee</sub>	-18	-18	V
Input voltage	V <sub>in</sub>	±15	±15	V
Differential input voltage	V <sub>in</sub> (diff)	±30	±30	V
Allowable power dissipation	P <sub>T</sub>	670 *	670 *	mW
Operating temperature	T <sub>opr</sub>	-20 to +75	-20 to +75	°C
Storage temperature	T <sub>stg</sub>	-55 to +125	-55 to +125	°C

Note: These are the allowable values up to Ta = 45°C. Derate by 8.3 mW/°C above that temperature.

## Potentiometers

A potentiometer is a variable resistor that has 3 leads as shown below.

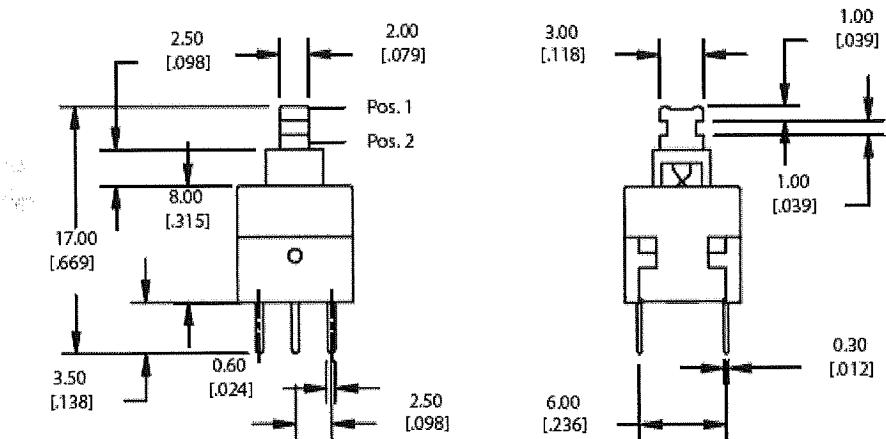


The total resistance between leads A and C is always constant. The position of lead B determines how the total resistance is broken up into 2 pieces, i.e.  $R_{AB}$  and  $R_{BC}$ . Lead B is called the wiper and, by turning a knob or screw, can be moved to any location between A and C. Thus, the total resistance  $R_{AC}$  should be the sum of each partial resistance  $R_{AB}$  and  $R_{BC}$ .

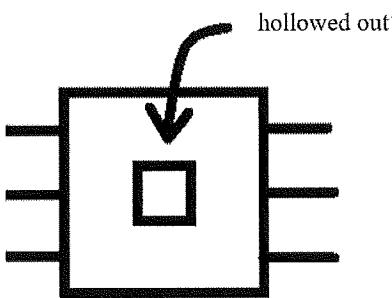
In Electronics Lab, we will actually use only two leads of the potentiometer for our variable resistor, i.e. A and B in the figure above.

## Pushbutton Switches

The pushbutton switch shown below is a momentary switch with a total of 6 pins. The stem of the plunger has one hollowed out side. When placed in the breadboard, that should be pointed "up" or facing the breadboard Power Pins. Convention in electronics is to number the pins of a component by starting in the upper left and going down, across, and then up the right making a U shape by going counter-clockwise. Thus, Pin 1 is at the upper left; Pin 3 is at the lower left; Pin 4 is at the lower right; and Pin 6 is at the upper right.



Label the pin numbers in the top view diagram below.



The switch has two sets of independent contacts, one set on the left side and one set on the right side. If the switch were large enough to fit over the breadboard gap, the two sides of the switch could operate independently. Unfortunately, this is not the case, so we will focus only on the left side of the switch.

Pin 3 at the bottom acts as the **common** for the left side of the switch. A Common pin must **always** be part of your circuit. Pin 2 is a Normally Open contact, meaning that when the switch is in its normal condition (you are NOT pressing down on the button) that Pin 2 forms an Open connection with the Common. Pin 1 is a Normally Closed contact, meaning that when the switch is in its normal condition (you are NOT pressing down on the button) that Pin 1 forms a Closed connection with the Common. Thus, there IS NOT continuity between pins 2 and 3 when the switch is in its normal condition, but there IS continuity between pins 1 and 3. When you depress the switch, the open and closed connections change, i.e. pins 1 and 3 were closed, but when you

press the button they become open. Pins 2 and 3 were open, but when you press the button they become closed.

The switch is symmetrical, so on the right side of the switch Pin 4 is the Common, Pin 5 is Normally Open with the Common, and Pin 6 is Normally Closed with the Common. Thus, IF the switch fit over the breadboard gap, FOUR different connections could be controlled by this single switch.

Label the figure above with the function of each pin.

This switch has a 0.1A rating, which should be more than sufficient for the experiments that you will perform.

**IMPORTANT NOTE:** *These particular switches can be a bit finicky and are of relatively poor quality. So, you will need to SLOWLY depress the button until you see the change. In some of them, if you push the plunger all the way, it changes and then changes back. Welcome to the real world.*

These Pushbutton switches are also momentary switches. That means that once you release the button, it goes back to the original condition. So, if you want the condition to stay switched, you have to keep holding the button down.

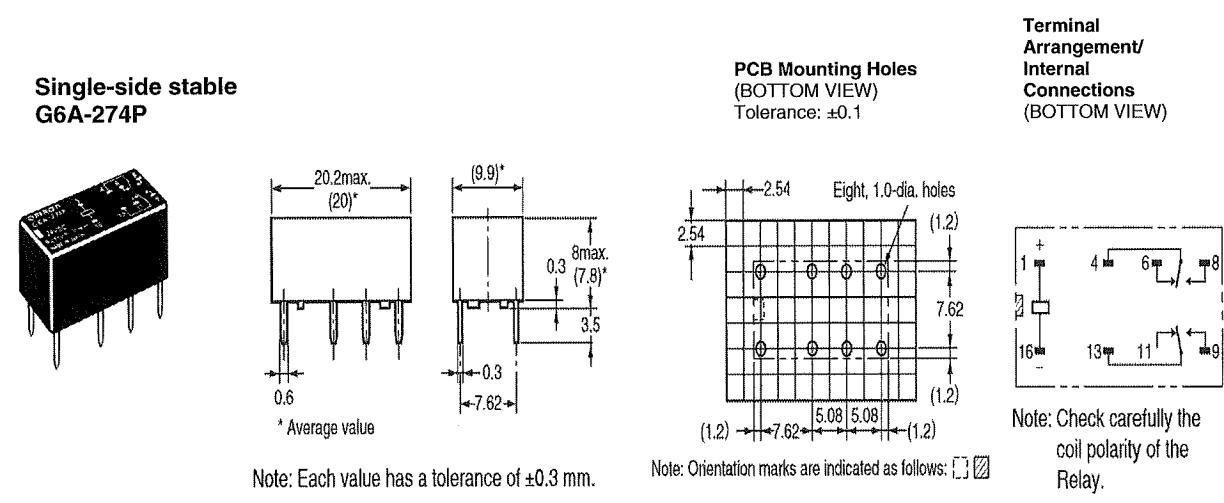
## Relays

A relay is like a push-button switch, but instead of manually pushing the button, the switching is electrically controlled. The coil relay that we will use is controlled by a 5V signal. When the controlling voltage is NOT applied, the internal switches of the relay are in their Normal Condition, and when the controlling voltage IS applied, the internal switches of the relay are NOT in their Normal Condition.

Specifications for the relay are shown in the figure below.

The pin numbers on a coil relay follow the same convention of all electronics, i.e. starting in the upper left-hand corner and going counter-clockwise. However, there are two things to note about the diagram below. First, the diagram shows a BOTTOM view of the pin connections of the relay, i.e. if the relay were upside-down with the pins sticking up at you. The diagram on the topside of the relay itself is also a BOTTOM view. Welcome to the real world. Second, note that a few of the numbers are missing. This is because there is no physical pin in that space.

When the relay is placed in the breadboard, pin 1 is in the upper left corner following the standard numbering convention.



The relay can be divided into two parts: the control part and the switching part. Pins 1 and 16 of the relay are the control pins or power pins. When there is a sufficient voltage difference across Pins 1 and 16 (with Pin 1 at the higher voltage) the relay will be activated. For this relay nominally the value is 5V, but it will actually switch at about 3V.

The switching part of the relay operates in the same way as the pushbutton switch. On each side of the relay there is a Common, a Normally Open connection, and a Normally Closed connection. According to the diagram, on the left side of the relay Pin 4 is the Common, Pin 6 is the Normally Closed connection, and Pin 8 is the Normally Open connection. Remember that the diagram above is a Bottom View. On the right side of the relay Pin 13 is the Common, Pin 11 is the Normally Closed connection, and Pin 9 is the Normally Open connection.

Sketch a **TOP VIEW** of the relay, label the pin numbers and the pin functions. Orient your diagram so that pin 1 is in the upper left-hand corner and pin 16 is in the upper right-hand corner.

## Resistors

Since you have passed Engine Physics II, you should be familiar with resistors, the color code, and how the color bands indicate the nominal resistance. You should consider memorizing the resistor color code values in the table below. Notice that ROY G. BIV is in the table and that Black and White are as far away as possible. Also, whenever working with electronics or computers, when counting, always start with ZERO. All the resistors that you will use in lab have 4 bands. The 4<sup>th</sup> band is used to determine the stated tolerance, where gold means within 5% and silver means within 10%. In the real world there are some resistors with only 3 bands (tolerance of 20%) and some resistors with 5 bands (tolerance less than 5% and determined by the color of the last band.)

Table 2-1: Resistor Color Code Values	
Digit	Color
0	Black
1	Brown
2	Red
3	Orange
4	Yellow
5	Green
6	Blue
7	Violet
8	Gray
9	White

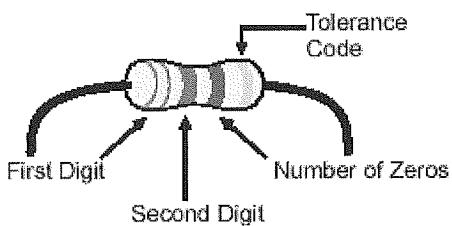


Figure 2-3  
Resistor Color  
Codes

When using resistors, it is important to make sure that the resistance is within the indicated tolerance given by the tolerance code band. To do this, the measured value should be compared to the nominal value using the percent difference equation below.

$$\% \text{ Difference} = \frac{\text{measured} - \text{nominal}}{\text{nominal}} \times 100\%$$

If a resistor has the following band colors, what is the range of possible values of resistance?

- a. BROWN, RED, ORANGE, SILVER
- b. WHITE, BROWN, ORANGE, GOLD

Write the equation for two resistors in series.

Write the equation for two resistors in parallel.

Write the fundamental equation for a resistor, i.e. the equation that relates the current through and the voltage across the resistor. **Note that the only variables should be  $I$ ,  $V$ , and  $R$ .**

## Soldering

In Electronics Lab instead of soldering, we will use the breadboard to make electrical connections. However, in the “real world” you may need to solder a connection that has come loose, so we will do it once.

“Soldering is the process of using a filler material (solder) to join pieces of metal together. Soldering occurs at relatively low temperatures (around 400°F) as compared to brazing and welding, which actually melt and fuse the materials themselves at higher temperatures. In soldering the filler material becomes liquid, coats the pieces it is brought into contact with, and is then allowed to cool. As the solder cools it hardens, and the two materials are joined. Soldering is a quick way to join many types of materials, from copper pipe to stained glass. It creates an electrically conductive strong bond between components that can be re-heated (desoldered) if you should ever want to disconnect two items joined together. It's great for joining electrical components and wires and is used in just about everything electronic.” “Soldering differs from welding in that soldering does not involve melting the work pieces.”

“Most solder is made from a combination of tin and lead - it's about a 60% tin, 40% lead mix depending on what solder you're using.” Some solders have a small amount of silver in them, and there are lead-free solders as well. Solder comes in a variety of thicknesses depending on the needs of the application. The solder used in lab does have lead in it, so be sure to wash your hands when you are done. Also, as the solder melts there is often some smoke that plumes up from the workpieces. It is generally considered “bad” to breathe the fumes, so it is a good idea to solder in a ventilated area and simply blow the smoke away from you.

In electrical applications, the best way to solder is to heat the wires and melt the solder by touching it to the hot wires, not the soldering iron. I prefer to heat from below and melt the solder from above.

“It's tempting to just touch the solder to the tip of the iron and melt it right away, but don't! You will end up making what's called a *cold solder joint*. This occurs when you melt the solder around the joint, but you aren't melting the solder *into* your joint or onto your components to make a good connection. It's much better to wait the few seconds and melt the solder onto the hot wire itself.”

If you make a mistake or use “too much solder or position something incorrectly you can usually reheat your joint, melt the solder, and then reposition your component as necessary. Solder can be heated and cooled as many times as you need to get your joint the way you want it.” If you are working with something very small or very delicate, you can use solder wick to suck up the unwanted solder.

A few things to remember when soldering:

1. Use a wet sponge to clean the tip of the soldering iron.
2. Always store the iron in its cage, never lay it down on the bench.
3. Use only a small amount of solder. It doesn't take much to make a good connection and excess solder can lead to short circuits.
4. Sometimes you might need a *tiny* bit of solder on the tip of the iron to help conduct the heat into the joint faster.

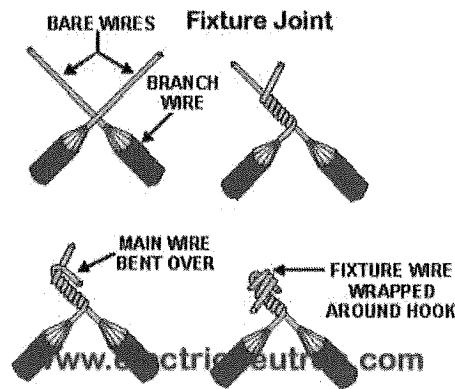
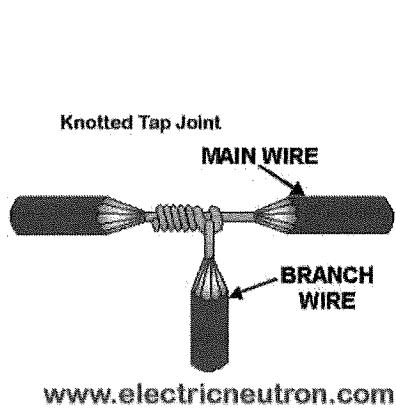
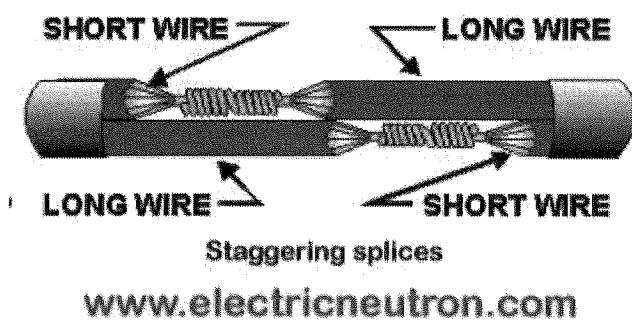
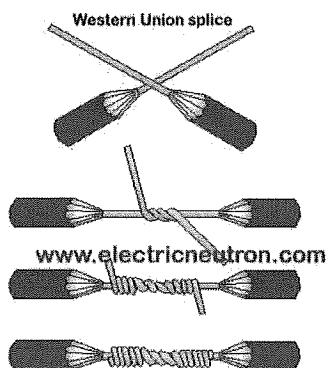
5. When soldering components, use as little heat for as short a time as possible to avoid damage to the components.
6. Do not touch the soldered components until you are sure that they have cooled.
7. Do not breathe the fumes. Work in a ventilated area and gently blow the fumes away from you.
8. Wash your hands when you are done as many solders contain lead.

Check out this video:

How to Solder – Intro/Joining Stranded Wires –by NJprepper321

<https://youtu.be/Q9G9gaokqvM>

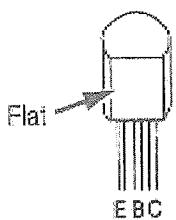
There are many different types of soldering splices for a variety of applications. A few are shown below, but you should do a google search on “wire splices and joints” to find some great images and you-tube videos.



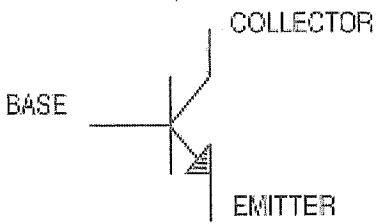
## Transistors

A transistor is an electrical component used to amplify current or switch electrical signals. There are many different types of transistors but the one you will use in lab is the NPN Bipolar Junction Transistor or BJT. It has 3 connection points: the Emitter, the Base, and the Collector. The symbol for an NPN transistor is shown below.

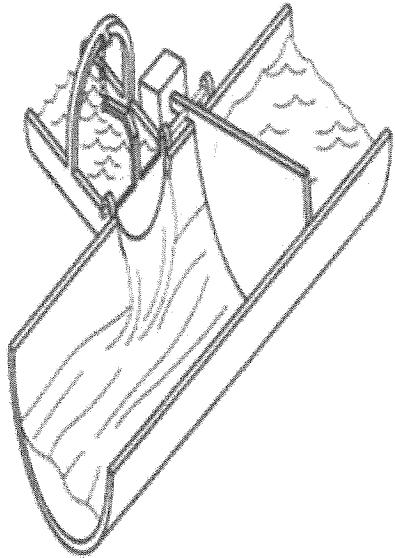
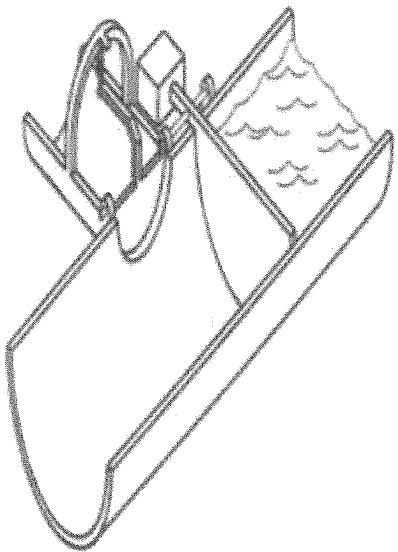
TRANSISTOR



Symbol for NPN TRANSISTOR



Basically, voltage applied to one pair of the transistor's terminals changes the current through another pair of terminals. Sometimes the applied voltage is viewed as a current that travels between the Base and the Emitter. The current (or voltage) is considered the input to the transistor, and the output is the current between the Collector and the Emitter. Thus, it operates as a switch. One way to think about them is using the pipe flow diagrams below. The transistor operates like a check valve that is connected to a lever arm that is essentially a gate for the larger pipe. When there is no water in the small pipe on the left, there is no force pushing on the small semicircular gate. In this case, the larger gate remains as shown in the left diagram preventing any water from flowing in the larger pipe. However, when there is water in the smaller pipe, it forces the small gate to open which in turn causes the larger gate to pivot open allowing flow in the larger pipe. In the analogy, the transistor is the combination gate. When there is enough current that enters the transistor at the Base, the switch opens and allows current to flow from the Collector to the Emitter. The total current exiting the transistor at the Emitter is the sum of the currents into the Base and into the Collector. The required voltage to make this happen is usually quite small, approximately 0.7V depending on the exact transistor that you use.



Because the output power can be higher than the input power, a transistor is said to amplify a signal. There are actually several kinds of transistors depending on how they are physically made.

You should look on the internet to learn more about transistors. Here is a good website:  
[www.allaboutcircuits.com](http://www.allaboutcircuits.com) >Education > Volume III Semiconductors, Chapters 2 and 4