Laboratory Manual

ME 6: Basic Electrical and Electronic Circuits

Department of Mechanical Engineering
University of California, Santa Barbara

$\sim$ 2014 $\sim$

Motivation

You may quickly notice that this lab manual is not prepared like a cookbook. It's not a shallow list of instructions that tell you exactly how to make something. Some follow cookbooks for years and never develop the confidence or ingenuity to create or deviate beyond what is laid out in front of them. Anyone can follow instructions; the purpose of this manual is to provide guidelines and occasionally opportunities for your own critical thinking to move you to the next step. Feel free to talk with people - your partner, surrounding groups, or with your TA, but don't sell yourself short!!

ME6 labs offer a lot to anyone that wants to learn. Three main deliverables are:

CONNECTING THEORY TO PRACTICE will help solidify concepts learned in class, and let you put them to use. Sometimes your measurements may not match your expected values, but this will evolve as you learn to account for real-world issues (equipment accuracy, non-ideal resistances, etc.), details that matter in design. Taking measurements alone is of limited value; your ability to effectively analyze your data is worth much more. Many lab report questions are designed to let you practice and to improve this capability.

TROUBLESHOOTING AND CRITICAL THINKING are at the heart of independent problem solving. These are among the most valuable and versatile skills that you'll develop. For circuits, most students initially think they'll have the circuit set up and done with no trouble. Well, the first attempt at even a basic circuit often isn't quite right - there may be one or more hard-to-spot problems. The key to solving the problems is to expect them! Then you can find what's wrong by (1) looking for likely culprits (no power, breadboarding mistake), or (2) checking front to back for behavior that does or doesn't make sense (correct resistances, voltages, etc.), or (3) by divide and conquer (keep chopping the circuit in half until you find what part is working vs not working). The point is: there is always a solution, stay calm and systematic and you will find it.

TECHNICAL WRITING is a major form of engineering communication. The value of an engineer's work will be limited by how well he or she can communicate it. It's very hard to respect a poorly written document. Good communication via technical writing is not something that can be turned on or off like a faucet. It takes practice, and report writing will help you improve it immensely.

Contents

Part I

LABORATORY INSTRUCTIONS

Laboratory 1

Introduction to Equipment

WHY AM I DOING THIS LAB? You need to know how to use the equipment! This lab will introduce you to the tools for supplying inputs and measuring outputs. These skills will be necessary for the rest of the labs and anytime you work with electronics in the future. You will examine the use of measuring tools like the multimeter and oscilloscope, as well as input tools like the DC power supply and function generator. You will also learn just how trustworthy your measurements will be by quantifying uncertainty. Paying close attention in this lab and understanding what you are doing will save you heaps of time and frustration in subsequent labs.

PRE-LAB ASSIGNMENT

This manual includes a chapter in the *supplementary information* section which shows what the different circuit components look like. This pre-lab is easy. Make yourself familiar with each component's name, appearance, and if you're really ambitious, its function as well. Since this pre-lab is merely a thought exercise, you have nothing to submit. This is the only exception; subsequent labs will require a submission as explained in the *Sample Lab and Report* chapter of the Supplementary Information section. Check it out!

Though this pre-lab will not be graded, skipping the few minutes it requires will probably slow you down.

LAB REPORT

For each lab, you must hand in a written report containing the answers and plots for all instructions labeled "**[LR]**". **Use the Sample Lab Report section in the Lab Manual as a guide**. Make sure to retain the numbering scheme used in the handout (For this lab there should be 12 items). Your Lab Report should clearly state your name, Lab Report number (lab 1 in this case), Lab date, your laboratory partner's name (if any), and your lab TA's name. Each lab report is due at the beginning of the following lab.

EQUIPMENT

BNC Adapter, Variety of Resistors, Capacitor, Diode.

PROCEDURE

Section 1: Digital Multimeter

The digital multimeter is one of the most convenient, versatile and simple tools. The ME 6 laboratory is equipped with a few different models all of which are able to measure voltage, current, and resistance. In addition, some can measure capacitance and inductance. The multimeter will soon be one of your best pals. Get to know it well, what it can do and perhaps, more importantly, what it can't. Multimeters can be very accurate if used properly but they are not perfect. Each measurement has some *uncertainty*. With the model number in hand, you can find this information in the *Accuracy of Equipment* section of the manual (or online). It will indicate the accuracy associated with each type of measurement (voltage, resistance, current, etc.). *You will need this information in this lab and future labs.*

1. To get started, make sure your probe cables are in the right spots. Colors help to make this more evident. Black is usually for negative terminals (or ground/common) and red is for the positive terminal. You will notice that there are two red terminals where you could put the red probe. One terminal will allow you to measure voltage and resistance and, possibly low currents. The other terminal is for high current measurements. There are markings on the terminals that should enable you to figure this out.

2. Your meters can measure both AC and DC components or circuit outputs. We will be measuring in DC, so choose appropriately.

3. Sort out the small circuit components that are in the plastic storage bin by your station. Try to guess which parts are resistors, capacitors and diodes.

4. When you have distinguished the components, use the color chart at your station to find resistances indicated on the resistors. **Record the values in ohms (Ω) and make sure to include the tolerances.** **[LR 1]**

   The main idea is that numbers from 0-9 are each represented by a single color. Reading these colors is like reading a number. You must read the colors from left to right. The right side is indicated by a single stripe spaced farther from the rest. The two left-most stripes (for 4-striped resistors) or three left-most stripes (for 5-striped resistors) each correspond to a digit. For example, 'yellow green' makes 45. The next stripe tells you how many times to multiply this number by ten. So, 'yellow green orange' makes $45 \times 10 \times 10 \times 10 =$

$45 \times 10^3 = 45\text{k}\Omega$. The right-most stripe indicates the tolerance. Tolerances indicate how accurate the label is. For example, $10\ \Omega \pm 10\%$ means that the actual resistance may deviate from the labeled value by up to 10 percent. *Note that the light blue resistors have 5 color bands while the beige resistors have four color bands.* **For really helpful information on this, look at the Resistor Color Code section.**

5. Let's see if we can measure these resistances and see how the labeled values compare. Make sure the appropriate resistance range is selected on the meter. For example, 200 corresponds to a maximum resistance of $200\ \Omega$, while 20M corresponds to a maximum of $20\ \text{M}\Omega$. If the display reads *OL* (overload), your resistance is above the selected range. **Record your resistance measurements with their associated uncertainty (found on the multimeter spec sheet or in the *Accuracy of Equipment* chapter of the Supplementary Information section).** **[LR 2]**

6. Your body is a resistor! How high is its resistance? Try measuring it from one hand to the other. Think of the consequences of this. If you hold a resistor in your hands while measuring, the meter will actually measure the equivalent resistance of your body in parallel with your resistor. If you must use your hand, be sure to only use one, so there is no complete circuit through your body. **Are your measurements more susceptible to this kind of handling error with resistors of high or low resistance? Why?** **[LR 3]**

7. Capacitors are components which can store charge and release charge. Your meter should be able to measure capacitance as well. Determine how to use it to do so and **measure your capacitors and record the values in Farads (F). Again, indicate the uncertainty associated with these measurements (also found on the *Accuracy of Equipment* section).** If you wonder what the strange capacitor codes refer to, take a look at the *How to Read Capacitor values* section. **[LR 4]**

8. Diodes let current flow in one direction but not the other. Let's test our diode. Try setting the meter dial to diode test position (indicated by an arrowhead against a vertical line, the diode symbol). **Test the diode in both directions and write down the voltage readings.** They should be quite different. **Try to describe what the meter may be doing that would give these very different results.** **[LR 5]**

9. Return all the circuit elements to the plastic bin and turn off your meter to conserve battery life.

Section 2: Digital Oscilloscope

If the multimeter is to be one of your best pals, the Tektronix TDS3012 Digital Phosphor Oscilloscope is sure to be like your favorite teacher—one that can be tricky to understand at first but who can give you a wealth of information if you know the right questions to ask. This is the device that you will use to learn about more interesting circuits—ones that have voltages changing in time, and ones that require you to look at two voltages at once! Please get familiar with this tool and its apparent complexities. Some students don't bother to do so and find this a stumbling block which hinders their progress. However, those that do learn its functions, find it to greatly accelerate their progress. Make your choice.

1. The oscilloscope has two channels to measure voltages with two different probes at the same time. There should be two oscilloscope probes at each station. Connect them to Channel 1 (CH1) and Channel 2 (CH2) BNC inputs. Each probe should have a hooked end for taking the input voltage signal and an alligator clip for defining ground (GND), $0\ \mathrm{V}$. Therefore, the signal at the hooked end of the probe will be measured relative to ground. When you want to measure an external voltage signal, you must connect the alligator (ground) clip to a suitable place in the circuit and then connect the hooked (input) probe to the point you want to investigate.

2. Let's turn on the oscilloscope! On the screen you should see two axis. The y-axis shows voltage and the x-axis shows time. This screen shows how the voltage across your probe changes in time. With no connection, your probes are just measuring noise, which is what appears on the screen.

3. Press the CH1, CH2 and OFF buttons to see how you can display or hide each channel. Make both channels appear at once. Channel 1 is in yellow and Channel 2 in blue. You can adjust the intensity of each signal display using the WAVEFORM INTENSITY knob.

4. Notice that the bottom left of the display shows the vertical scale (volts per division) for each channel. Turn the vertical SCALE knob and observe that you can change the voltage scale on the display. Press the yellow CH1 and blue CH2 buttons to adjust each channel respectively.

5. Set the vertical scale for both Channel 1 and Channel 2 to $5.00\ \mathrm{V}$ per division.

6. Notice that the bottom center of the display shows the horizontal scale (seconds per division) for each channel. Turn the horizontal SCALE knob and observe that you can change the time scale on the display. Note that the horizontal scale is the same for both channels. Try setting the time scale to $200\ \mu\mathrm{s}$ per division.

7. Before you use the probes to measure voltage signals, you should zero (center) the voltage reference of each channel. To do so, for each probe, connect its input end to its ground end. This is called *grounding the probe*. When you have done this, the probe is measuring $0\ \mathrm{V}$. Next, adjust the vertical POSITION knob until the corresponding arrow for each channel is at the mid-point of your display's voltage scale. Remember to press the CH1 and CH2 buttons to adjust each channel separately.

8. Now, turn off Channel 2, so only Channel 1 remains. **Use the oscilloscope button bearing the image of a printer to capture your display.** The capture function saves your display as an image file that you can use in your report. **[LR 6]**

9. Sometimes it's useful to make sure a probe is working. To do this, you can use the oscilloscope's internal $5.00\ \mathrm{V}$ square wave generator. You can access this square wave using the metal brackets to the immediate right of the Channel 2 plug. Connect the GND clip to the ground bracket on the oscilloscope (the symbol for ground on the oscilloscope looks like a three-pronged rake). Connect the input end of your Channel 1 probe to the square wave bracket on the oscilloscope. Now that you are connected to the square wave generator, you should see a moving square wave on the display. Do the period and amplitude look correct? If so, your probe is ok!

10. The oscilloscope displays repetitive signals. For the display on the screen to appear stable (stationary), the beam which draws the waveform must start on the left side of the screen at the identical point in the signal each time it sweeps across the screen. The *trigger controls* determine when the sweep starts. The controls allow the user to select at what voltage (level) the signal must be at the instant that the sweep starts. This voltage level is controlled by the TRIGGER LEVEL knob. The trigger level is shown by the arrow on the right side of the display graph. When the square wave appears stationary, it has been properly triggered. If it is drifting, it's poorly triggered. (Note that the trigger source must be set to the channel to which the voltage signal is connected. To set the trigger source, push the trigger MENU button to show the trigger menu on the screen. Select the appropriate channel using the buttons to the right of the screen.)

11. We can freeze the display by pressing the RUN/STOP button. Pressing it again, unfreezes the display.

12. The scope has a button for automatically optimizing the display settings (vertical scale, horizontal scale, and trigger level) for a given signal. However, be wary. This button is not always intelligent enough to capture what you want. It's never a match for your clear thinking. Try it out! Press the AUTOSET button. **Now look at the display and figure out the frequency of the square wave and record it. Use the screen capture to copy the square wave as measured by Channel 1.** **[LR 7]**

13. Adjust the vertical SCALE knob and the horizontal SCALE for fun to see how the signal display changes. Press AUTOSET to return to the optimized settings.

14. The scope also has built-in measurement functions so you don't have to measure manually. Let's try it out. Press the MEASURE button to select the parameters to measure and display. **Find frequency and amplitude and** **[LR 8]**

select them for display. Record these values. Of course, just like the multimeter, these measurements are not perfect. **Find the datasheet on this device to find the associated measurement accuracy. Record these uncertainties with your measurements.** You can press MENU OFF to remove the measurement menu.

15. You can now disconnect the Channel 1 probe from the test signal on the oscilloscope. Notice how the displayed signal is now almost a flat line since the probe is measuring the voltage through the air.

Section 3: Function Generator

So far you've examined tools for measuring components or circuit *outputs*. Now we will look at tools used to generate *inputs*. The first of these is the Tektronix CFG280 function generator. What kind of functions does it generate? Why, voltage as a function of time, of course. These functions have various shapes from sinusoidal waves, to square waves, to sawtooth waves and so on. This tool will be a common adjustable input for future circuits.

1. To start the function generator at minimum amplitude, find the AMPLITUDE knob and turn it left as far as you can, then turn on the function generator.

2. You are generating a periodic voltage which oscillates about some DC offset. For now, we would like the signal to oscillate about $0\ \mathrm{V}$ so, push in the DC OFFSET knob to set the DC offset to $0\ \mathrm{V}$.

3. Select the MAIN 0-2Vp-p setting by pressing in that button. This will limit your voltage output to $2\ \mathrm{V}$, peak-to-peak.

4. Now you are ready to examine the signal on the oscilloscope.

5. Generate a sine wave on your function generator. There should be a button that looks like the sine wave. Make sure that none of the other buttons on that row are pressed at the same time.

6. Try adjusting the amplitude until you can see the sine wave on your scope. You may want to use AUTOSET to make the signal more clear.

7. Let's try changing the frequency. This can be done by adjusting the MULTIPLIER buttons and FREQUENCY knob on the generator. Set the frequency to $1\ \mathrm{kHz}$ according to the display on the *function generator*. Then, try checking this output on the scope to see if it really is $1\ \mathrm{kHz}$. You may want to use the measure feature again. If it is not right on your target, gently adjust the frequency until it is. The fine adjustment knob might be helpful for this.

8. Increase the amplitude of your function generator output to the $2\ \mathrm{V}$ maxi- **[LR 9]**

mum and watch the scope display as you do so. Verify that the function generator output is a sine wave with peak-to-peak amplitude of approximately $2\ \mathrm{V}$ and a frequency of $1\ \mathrm{kHz}$. **Capture this screen.**

9. Minimize the amplitude of the sine wave and then select the OPEN CIRCUIT 0-20Vp-p setting by pressing out that button. This will extend your maximum voltage output to 20 volts, peak-to-peak. Then set the oscilloscope Channel 1 vertical scale to $2\ \mathrm{V}$ per division. Now adjust the amplitude on the function generator until the sine wave has a peak-to-peak amplitude of $4\ \mathrm{V}$. For the purposes of this exercise, **do not** press AUTOSET.

10. You can now add a DC offset to the generated signal by pulling out the DC OFFSET knob. Turn the knob until the bottom of the sine wave is moved to $0\ \mathrm{V}$ on the scope display. This is how you make sure that your waveform is within a certain voltage range (in this case, $0\ \mathrm{V}$ to $4\ \mathrm{V}$). For example, many logic circuits require that the voltage be within a $0\ \mathrm{V}$ to $5\ \mathrm{V}$ range. If you simply turned on the function generator without verifying the voltage output range, you could have damaged your circuit components. **Screen-capture your off-set sine wave.** **[LR 10]**

11. Disconnect the function generator and turn it off.

Section 4: DC Power Supply

The next tool for generating an input put is the Tektronix PS280 DC Power Supply. Unlike the function generator, this device supplies voltage (or current) which is *not* a function of time. In other words, it's constant. This will be your main power source for all your DC circuits. Today, we simply examine how to measure a fixed voltage from the supply.

1. Find the 5V FIXED 3A terminals on the power supply. This means that across the two terminals there is a $5\ \mathrm{V}$ potential difference. That is, between the positive (red) terminal and negative (black) terminal, we have $5\ \mathrm{V}$. If you take the negative terminal to be ground ($0\ \mathrm{V}$), the positive terminal will be $5\ \mathrm{V}$ above that ($+5\ \mathrm{V}$). If you take the positive terminal to be ground, the negative terminal will be $5\ \mathrm{V}$ below that ($-5\ \mathrm{V}$). These terminals are limited to supplying $3\ \mathrm{A}$ current.

2. Let's check the supply's output. We can use the multimeter or the oscilloscope. Try it with the meter first and then the oscilloscope, using the knowledge you gained in Section 1 and 2. Make the appropriate connections, then turn on your supply and meter/scope. **Record the measured voltage from each device and include the uncertainty as before.** *You will need banana plugs for the* 5V FIXED 3A *terminals*. You should also be careful when using the measurement tool on the scope. **Does it make sense to use the amplitude tool, or is another one more appropriate? Please explain.** **[LR 11]**

3. See what happens when you adjust the VOLTAGE and CURRENT knobs on the power supply. Nothing happened, right? This is because the output you are measuring is *fixed* at $5\ \mathrm{V}$. These knobs are linked to the other terminals which you will use in the next lab for getting other voltages of your desire.

4. Now you can disconnect these tools and turn them off.

Section 5: Test Your Skills

Generate a sine wave with a peak-to-peak amplitude of $8\ \mathrm{V}$, frequency of $10\ \mathrm{Hz}$ and $4\ \mathrm{V}$ DC offset. View this clearly on your oscilloscope using Channel 2. Don't forget to zero your vertical reference by grounding your probe each time you turn on the oscilloscope. **Screen-capture your output.** **[LR 12]**

CLEAN UP

Please make sure that your station appears exactly as it did at the beginning of the lab.

1. All circuit components should go in the cup at your station.
2. Lead cables should be returned to the racks.
3. BNC adapters should be put back in proper bins.
4. Scope probes should *not* be coiled, but draped over the bench.
5. All tools should be turned off - especially meters.

Laboratory 2

DC Circuits

WHY AM I DOING THIS LAB? In this laboratory you will learn how to use a Variable DC Power Supply. You will also learn how to build a circuit using a breadboard. Breadboards are indispensable tools for prototyping electronic circuits and for experimenting with circuit design. Because of this, we will use breadboards extensively throughout this course. You will also explore Ohm's law, and the types of devices for which it does, and does not, apply.

BACKGROUND

In the first lab, you learned how to take measurements and how to account for uncertainty in those measurements due to the limited accuracy of the tools. Sometimes it is necessary to make *important* comparisons or conclusions based on measured values. Doing this may involve making calculations with your measured values. For these types of calculations, it is necessary to carry the uncertainty through as well. For example, if it was crucial to calculate the current through an $8\ \Omega$ resistor with an applied 16 V source, you could not just report 2 A. You must consider uncertainty. Suppose the resistance measurement, with uncertainty, is $8.0 \pm 0.1\ \Omega$ and the voltage measurement is 16.00 ± 0.01 V. Then the $\pm 0.1\ \Omega$ and ± 0.01 V are the *absolute uncertainties* of the resistance and voltage measurements. The *relative uncertainty* is given by $0.1/8 \times 100\% = 1.25\%$ for the resistance and $0.01/16 \times 100\% = 0.0625\%$ for the voltage. So, how do measurement uncertainties propagate through our calculations? A simple way to calculate *worst case* uncertainty is to use the following rules:

1. **Addition and subtraction rule:** when adding or subtracting quantities, the *absolute uncertainty* of each quantity are summed. For example,

$$\begin{aligned}&(5.0 \pm 0.1\ \mathrm{V}) + (2.0 \pm 0.2\ \mathrm{V}) - (3.0 \pm 0.3\ \mathrm{V})\\ &= (5.0 + 2.0 - 3.0) \pm (0.1 + 0.2 + 0.3)\ \mathrm{V}\\ &= 4.0 \pm 0.6\ \mathrm{V}.\end{aligned}$$

2. **Product and quotient rule:** when multiplying or dividing quantities, the

relative uncertainty of each quantity are summed. For example,

$$\begin{aligned}\frac{(6 \pm 0.1\text{ V})^2}{3 \pm 0.2\ \Omega} &= \frac{(6 \pm 0.1\text{ V}) \times (6 \pm 0.1\text{ V})}{3 \pm 0.2\ \Omega} \\ &= \frac{6 \times 6}{3}\text{ W} \pm \left[\left(\frac{0.1}{6} + \frac{0.1}{6} + \frac{0.2}{3}\right) \times 100\right]\% \\ &= 12.0\text{ W} \pm 10\% \ \textbf{ or } \ 12.0 \pm 1.2\text{ W}.\end{aligned}$$

With these rules we can now say that the current from above should be 2.00 ± 0.03A. If the difference between observed and calculated current is within this range, we can be satisfied. If not, we have to attribute the error to something *other than* measurement error. This is an extremely important aspect of data analysis. As engineers applying theory in practice, you must be able to account for uncertainties and discrepancies.

Note: For important measurements, we will ask you to perform uncertainty calculations.

PRE-LAB ASSIGNMENT

From now on you must submit a *typed* pre-lab *at the start* of the corresponding lab. The TA will review, sign and return the pre-lab to you in the same session. To receive credit you must attach the *signed* copy to your final report. If you find errors in your pre-lab you may make corrections and attach the revision to your final lab report. The final lab report is a formal report and must be typed and well written. Report writing is explained in detail in the *Sample Lab and Report* chapter of the Supplemental Information section.

1. Read the sections in the lab manual titled *Breadboarding Basics*, *Tektronix PS280 DC Power Supply* and *Oscilloscope Probes*.

2. Consider two resistors in series, one with a resistance of $10.0 \pm 0.1\ \Omega$, and the other with a resistance of $4.9 \pm 0.1\ \Omega$. If the current through each resistor is 1.00 ± 0.05 A, what is the total voltage drop across both resistors?

3. For the voltage divider shown in Figure 2.4, use Kirchhoff's voltage law to determine V_{out} as a function of V_{in}.

4. If the voltage input, V_{in}, in Figure 2.4 is an AC voltage (i.e., a voltage that varies with time) do you expect the equation you found above to hold? Why or why not?

EQUIPMENT

Resistors: One: $1\ \mathrm{k}\Omega$, $20\ \mathrm{k}\Omega$, $100\ \mathrm{k}\Omega$ Potentiometer. Two: $10\ \mathrm{k}\Omega$

Other circuit elements: One: Incandescent Bulb. One: Diode IN4002.

PROCEDURE

Section 1: DC Power Supply

In Laboratory 1 you learned how to supply a $5\ \mathrm{V}$ DC voltage using the *5V FIXED 3A* output of the Tektronix PS280 DC Power Supply. In this section, you will learn how to supply DC voltages ranging from $-30\ \mathrm{V}$ to $+30\ \mathrm{V}$ using the DC Power Supply.

1. Find the "0–30V 2A" sockets on the DC power supply. There should be two sets of three sockets. Each set of three sockets acts as a separate variable DC power supply and is controlled by its corresponding *CURRENT* and *VOLTAGE* knobs. By utilizing both channels (i.e., both sets of three sockets) you can supply two different DC voltages. To ensure that the two channels are operating independently, set the *TRACKING* buttons to *INDEP* (independent).

2. Each channel can operate as a current supply or as a voltage supply. Set both channels as voltage supplies using the slider controls between the displays. With the channels operating as voltage supplies the *CURRENT* knob can be used to set a maximum on the amount of current that can be supplied by the DC power supply (from $0\ \mathrm{A}$ to $2\ \mathrm{A}$). The *VOLTAGE* knob is then used to set the voltage. In this lab, set the *CURRENT* knob to full as we are not worried about limiting the amount of current.

3. When the DC power supply is turned on and set to, say $10\ \mathrm{V}$, a voltage of $10\ \mathrm{V}$ is supplied between the $-$ve socket (black) and the $+$ve socket (red). We say that the power supply is floating, because the voltage at the $-$ve and $+$ve sockets can be anything so long as the difference between them remains at $10\ \mathrm{V}$. The middle socket (green) is *earth ground*. If we don't want the power supply to float we can run a cable from the $-$ve terminal to ground, thus fixing the $-$ve socket at $0\ \mathrm{V}$ and the $+$ve socket at $10\ \mathrm{V}$. In this lab we will just let the DC power supply float.

4. Pick one of the DC power supply channels. Connect a black banana cable to the $-$ve socket and a red banana cable to the $+$ve socket. We want to display the voltage supplied by the power supply on the oscilloscope. Connect the DC voltage supply (i.e., the two banana cables) to one of the channels of your oscilloscope by using the oscilloscope probe.

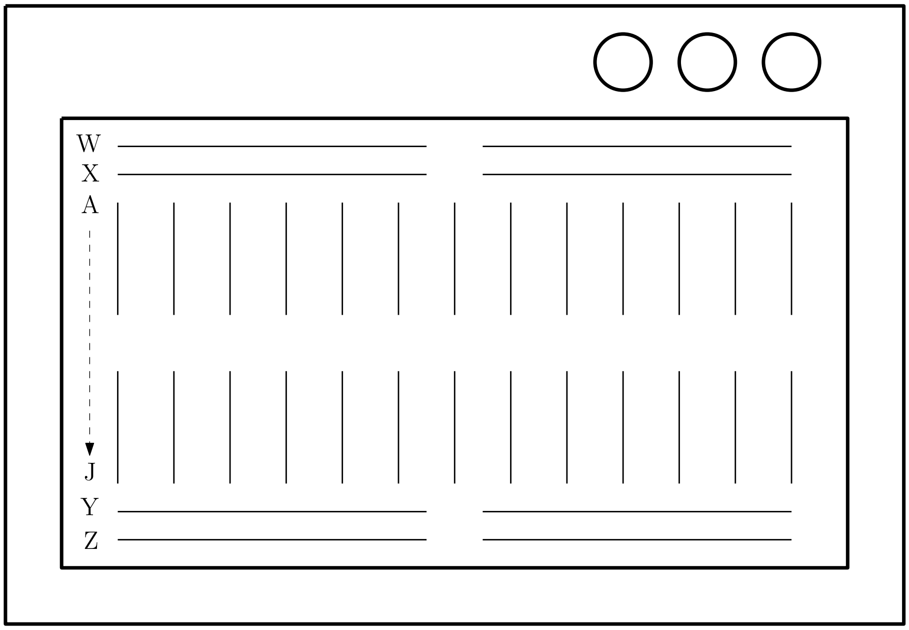


Figure 2.1: The breadboard. The lines represent how the holes on the breadboard are electrically connected.

5. Turn on the DC Power Supply. Use *AUTOSET* if necessary to view the resulting voltage signal on the oscilloscope. Verify that the output from the power supply agrees with what you see on the oscilloscope.
6. Adjust the *VOLTAGE* knob on the power supply and observe the output on the oscilloscope. Does the output make sense?
7. Disconnect the DC Power Supply from the oscilloscope.

Section 2: Introduction to the Breadboard

Figure 2.1 shows a schematic of the breadboard you will use to build circuits in this course. Of the three sockets on the board, two should be dark red and one should be black. We will use the *black* socket to supply the circuit with common ground (GND) and the dark red sockets to supply the circuit with power. The sockets accept banana cables. Wire with stripped ends are used to connect the sockets to the breadboard section. Various circuit elements can be easily assembled on the breadboard. Usually, power (voltage supply) and ground (GND) are provided to the W, X, Y, and Z rows (these are electrically connected horizontally, as shown in Figure 2.1), while circuit elements are assembled on rows A through J (these are electrically connected vertically, as shown in Figure 2.1).

As a habit, **always color-code your circuits.** For example, *black* wire should only be used for ground (GND) connections, while *red* wire should only be used for positive voltage supplies. Don't use *red* and *black* wire for anything else!

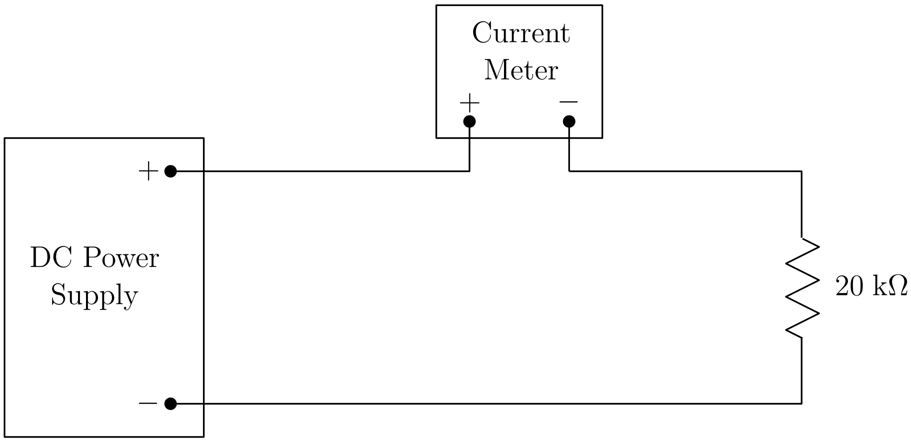


Figure 2.2: Circuit for verifying Ohm's law.

Section 3: Ohm's Law and Linear Devices

Ohm's law states that voltage V across a resistor is equal to current i multiplied by resistance R, $V = iR$. In this section we will verify that the resistor obeys Ohm's law. Although this may not seem very exciting, it will get us started setting up circuits on the breadboard and making measurements.

1. Setup the circuit shown in Figure 2.2. Make sure to test all resistors on the DMM first to ensure they are the correct resistances. Vary the voltage from 0 V to 20 V in increments of your choice and measure the corresponding current at each step. To use your digital multi-meter (DMM) as a current meter, you must put the *red* probe into the *yellow* mA jack and select the appropriate current (A) range using the dial of the DMM. You may have to use (appropriately colored) alligator connectors to connect your DMM probes to your circuit board.

2. In place of the 20 kΩ resistor put a 10 kΩ resistor and repeat the measurements. Plot your data for both resistor values on a graph with voltage values on the ordinate (y-axis) and current values on the abscissa (x-axis). Do the curves appear to be linear? If they are linear, what is the slope of each line? (Hint: You may want to use a spreadsheet program with graphing capabilities.) **[LR 1]**

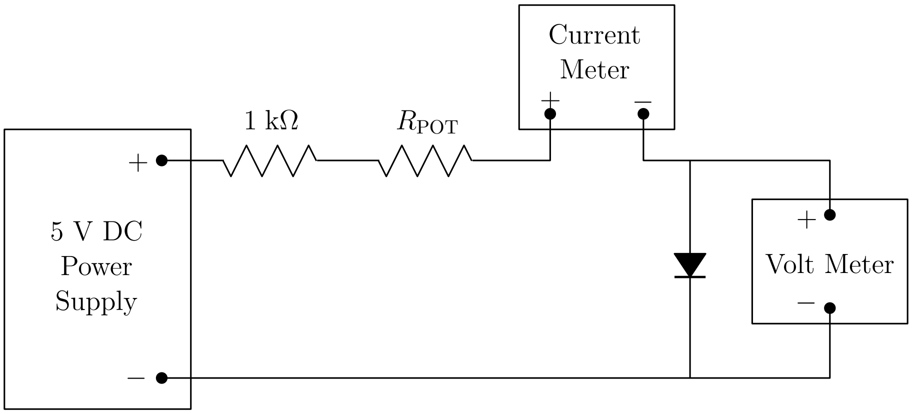


Figure 2.3: Circuit for measuring the voltage vs. current characteristics of the diode.

Section 4: Ohm's Law Defied—Nonlinear Devices

Incandescent lamps: Now perform the same experiment for the incandescent bulb. You will probably have to change the settings of the DMM in order to measure the higher currents present. **Do not exceed** $6.0\ \mathrm{V}$ **or you may burn out the lamp!**

1. Plot your results and provide a table. Get enough points to show how the lamp slightly diverges from resistor-like (linear) performance. **[LR 2]**

2. What is the "resistance" of the lamp? Is this a reasonable question? If the lamp's filament is made of material fundamentally like the material used in the resistors you tested earlier what accounts for the funny shape of the lamp's voltage vs. current curve? **[LR 3]**

Diodes: The diode is another nonlinear device that does not obey Ohm's law. We need to modify the test setup here because you cannot just apply a voltage across a diode, as you did for the resistor and lamp above. You'll see why after you've measured the diode's voltage vs. current characteristics. To do this, wire up the circuit shown in Figure 2.3. R_{POT} is a $100\ \mathrm{k}\Omega$ variable resistor (usually called a potentiometer or "pot") that can be manually set to any resistance between $0 - 100\ \mathrm{k}\Omega$. Connect the potentiometer using the center pin and one of the two outside pins. If your workstation does not have two DMM's you will have to get a second DMM from the front of the lab. Use one DMM as your current meter and the other as the volt meter.

Note that in this circuit you are applying a current (to the diode) and recording the resulting diode voltage. By varying the potentiometer you vary the amount of current being applied. This is somewhat different than before where you applied a voltage (to the resistor) and recorded the resulting current. The $1\ \mathrm{k}\Omega$ resistor limits the current to safe values.

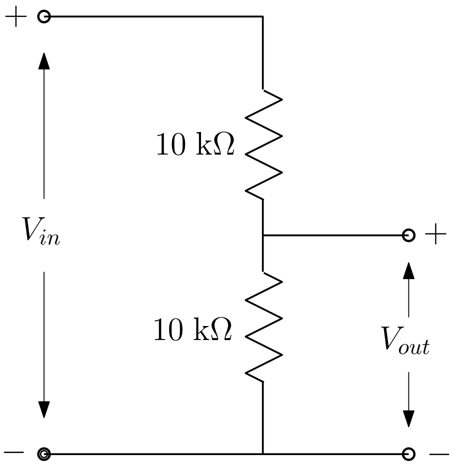


Figure 2.4: Voltage divider.

1. Vary R_{POT} starting at $100\ \text{k}\Omega$. As you vary R_{POT}, record values of voltage and current. Plot the voltage vs. current data on a graph making sure you've taken enough data points to determine the shape of the response. Next, plot this data on a semi-log graph, i.e., the current should be plotted on a log scale while the voltage scale remains linear. What can you say about the current voltage relationship for a diode from the semi-log plot? **[LR 4]**
2. See what happens when you reverse the direction of the diode. How would you summarize the voltage vs. current behavior of a diode? **[LR 5]**
3. Now explain what you think would happen if you were to put $+5\ \text{V}$ across the diode. (**Don't try it!**) Explain your answer. **[LR 6]**

Section 5: DC Voltage Divider

If we apply a voltage across two resistors, R_1 and R_2, connected in series the total voltage drop is split among the them. The voltage drop across each resistor can be determined from the relative magnitude of R_1 and R_2. This is known as voltage division.

1. Construct the voltage divider shown in Figure 2.4 and apply $V_{in} = 15\ \text{V}$ DC.
2. Use a DMM to measure V_{out}. What is the uncertainty on this measurement? Accounting for uncertainty, calculate an expected value for V_{out} and determine if this measured value agrees with your expected value. **[LR 7]**
3. The volt meter and current meter are used to take measurements of your circuit, but do they have any undesired effect upon it? The answer, as you can probably guess, is yes. Looking at how the meters are connected in the circuit, what would you want their *ideal* internal resistances to be? Can you **[LR 8]**

see how any deviation from this ideal value would cause the meters to affect the circuit? Explain the affect each meter has on the circuit.

Section 6: AC Voltage Divider

Using the same setup as in Figure 2.4, use the function generator to set V_{in} to a $1\ \mathrm{kHz}$ sine wave with a peak-to-peak amplitude of $15\ \mathrm{V}$ and zero DC offset.

1. Use the the oscilloscope and the probes to measure V_{out}. Was your answer to Question 4 of the pre-lab correct? This is an important conclusion so make sure you include uncertainty when presenting your measurement. **[LR 9]**
2. Apply a square wave and triangle wave of the same frequency and amplitude to the circuit. Is V_{out} still related to V_{in} by the equation you found in Question 3 of the pre-lab? Again, include uncertainty in this comparison. Save an image of the waveform and include in your report. Based these results, summarize the behavior of a voltage divider. Save an image of the waveform and include in your report. **[LR 10]**

CLEAN UP

1. **Make sure that all your equipment has been turned off**.
2. Please put components back into the cups and make sure that your station appears as it did at the beginning of the lab.

Laboratory 3

DC Circuit Analysis

WHY AM I DOING THIS LAB? This lab will enhance your ability to visualize circuits in their modular form— a composition of parallel and series sub-circuits. With this ability, you will be able to simplify complicated circuits with ease. Additionally, you will be able to use this perspective to spot voltage and current dividers, which are easily solved and will frequently appear in subsequent labs. You will gain a working understanding of how a linear circuit permits the use of the Superposition Principle— an effective tool for simplifying some circuits. You will also learn about sources of error and experimental uncertainty.

BACKGROUND

This lab, and many others, will be made much easier if you can understand, appreciate and recognize when to apply two very important rules, the *voltage divider* and *current divider*. The voltage divider rule (as expected) describes how voltage is divided among two resistors or equivalent resistance modules. This rule works for resistors or equivalent resistance modules that are in *series*. Some examples are shown in Figure 3.1. On the far left, there is a simple series circuit and the total voltage gets divided among the two resistors. Let's derive how this voltage is divided. Say we want to know the voltage across the resistor in the bottom circle,

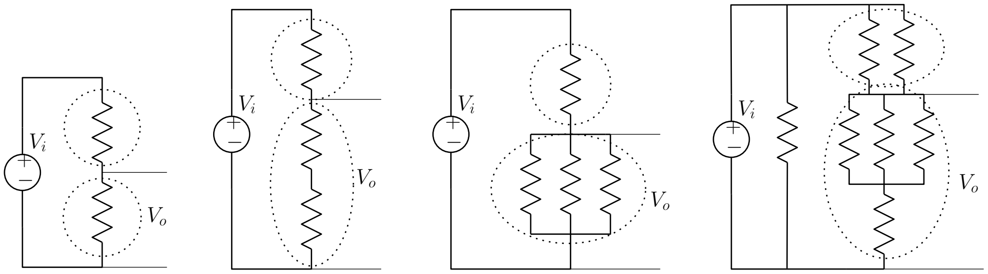


Figure 3.1: Various voltage dividers.

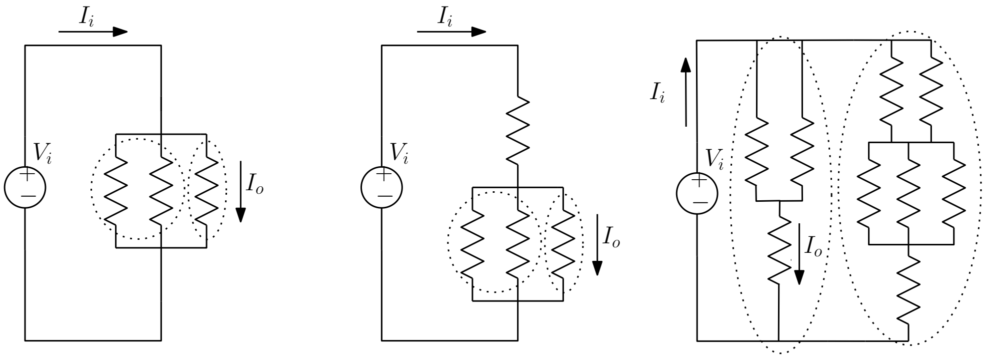


Figure 3.2: Various current dividers.

V_o. The voltage is given by

$$V_o = IR. \tag{3.1}$$

But we need to know I. This comes from

$$I = \frac{V_i}{R_{total}}. \tag{3.2}$$

Substituting Equation 3.2 into Equation 3.1 we get

$$V_o = V_i \frac{R}{R_{total}}.$$

This expression is the voltage divider rule and it shows that the voltage across one portion of series resistance is just a fraction of the total. This technique works on more complicated looking arrangements too, like the others depicted in Figure 3.1. The key is to spot two resistor modules that are in series. These are shown by the dotted circles.

The same idea applies to current dividers but with an important difference—current dividers apply to *parallel* resistance modules. Take a look at the various current dividers of Figure 3.2. We can find the expression as before but now we want to find current, I_o, through a branch in a parallel region with resistance R. We have

$$I_o = \frac{V}{R}. \tag{3.3}$$

But we need to know V. This comes from

$$V = I_i R_{total}. \tag{3.4}$$

Again, combining Equations 3.3 and 3.4 we obtain

$$I_o = I_i \frac{R_{total}}{R}.$$

This is the current divider rule.

PRE-LAB ASSIGNMENT

Section 1: Equivalent Resistance, Voltage Division and Current Division

1. The resistor network in Figure 3.3 is a rather complicated arrangement. Being able to spot parallel and series modules is the key to simplifying complex networks. Can you make the parallel and series sub-circuits stand out? Try redrawing the circuit to do so or try simply circling the sub-circuits.

2. Using this technique, find the equivalent resistance of this network across its terminals (R_{eq}). Take $R = 15\ \text{k}\Omega$.

3. Assume a voltage V has been applied across the terminals. What fraction of this total potential is lost across each resistor? Recall the parallel and series modules you spotted previously - they may help you apply the voltage divider rule with ease, making this question much more simple. Make sure you keep a record of this, and subsequent information as you will be using it during the lab.

4. If we apply a voltage V across the terminals, a current will flow through the network. Calculate the current passing through each resistor.

5. Let's see if your results make sense. This may be determined by simple "sanity checks". For example, at a junction, more current will pass through the less resistive path than the highly resistive path. Can you think of a similar sanity check for potential (voltage)? State your idea.

Section 2: Superposition in Action

1. Now you know how potentials and currents are distributed through a resistor network due to ONE source (in this case, a voltage source). How do you treat the circuit if there is MORE THAN ONE source? If the circuit is LINEAR you can employ the Superposition Principle. That is, you can solve the circuit for one independent source at a time, replacing all other independent sources with their resistive equivalents. The voltage and current across each resistor is then simply the sum of the contributions from each source. No matter how many independent sources are in a linear circuit, you can ALWAYS solve them with this simple tool.

2. What is the resistive equivalent of
   (a) an independent current source?
   (b) an independent voltage source?
   (c) a dependent current source?

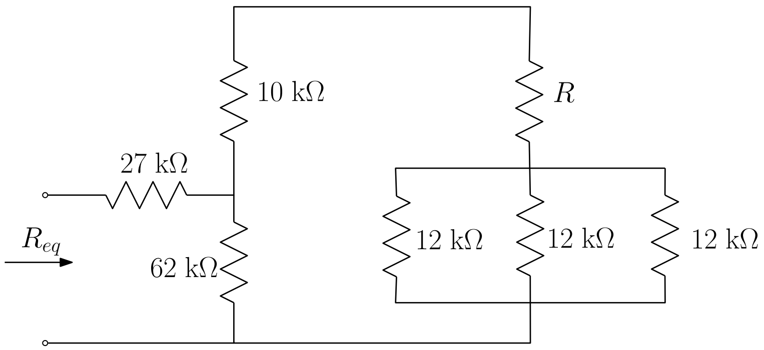


Figure 3.3: Equivalent resistance circuit.

   (d) a dependent voltage source?

3. Do any of the above NOT have an equivalent resistance? Which one(s)?

4. Try this new tool to solve the circuit: use superposition to find currents and voltages for the $1\ \text{k}\Omega$ and $3.9\ \text{k}\Omega$ resistors in Figure 3.4, taking the bottom source as V_1 and the upper source as V_2 (solve in terms of V_1 and V_2.)

EQUIPMENT

Resistors One: $1\ \text{k}\Omega$, $3.9\ \text{k}\Omega$, $5.6\ \text{k}\Omega$, $6.2\ \text{k}\Omega$, $10\ \text{k}\Omega$, $15\ \text{k}\Omega$, $27\ \text{k}\Omega$, $62\ \text{k}\Omega$. Three: $12\ \text{k}\Omega$.

PROCEDURE

Section 1: Equivalent Resistance, Voltage Division and Current Division

1. Build the circuit shown in Figure 3.3. Use a $15\ \text{k}\Omega$ resistor for R. Remember to use the color-code described in Lab 2. Otherwise, the T.A. will NOT assist you in debugging your circuit!

2. Experimentally determine the equivalent resistance of this network, R_{eq}. **[LR 1]** **Quantitatively**, how well does the experimental value agree with your calculated value? What sources of experimental error or uncertainty exist (resistor tolerances, multimeter error, etc.)? Quantify these error sources (for example, the basic accuracy of the Fluke 73/77 Series III Digital Multimeter is $\pm 0.5\%$). Note that human error sources such as miscalculation, incorrect resistor use, or incorrect measurement technique are NOT valid answers in this course. If attentive, you can catch and correct those errors in the laboratory.

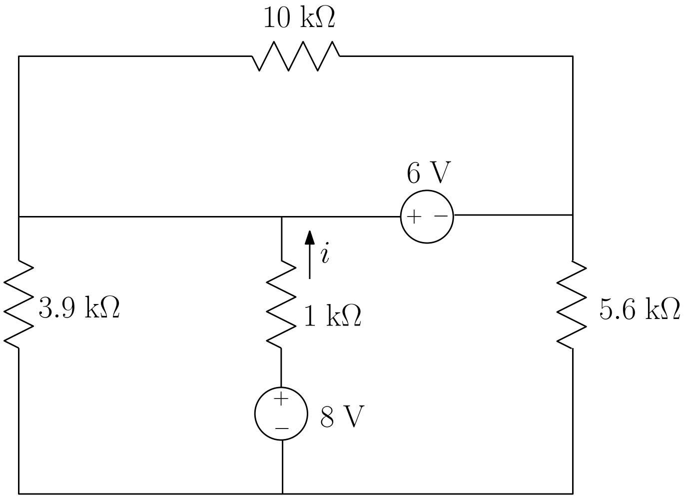


Figure 3.4: Superposition circuit with $V_1 = 8$ V and $V_2 = 6$ V.

3. In the pre-lab you found potential drops across, and current through, each resistor in Figure 3.3 for an arbitrary voltage. Do you expect these values to hold for an applied 10 V DC source? Experimentally, test your expectation for each resistor. As above, quantitatively show how well your experimental values agree with your calculated values. What sources of error exist? Quantify these error sources. It may be helpful to choose an organized method of collecting and presenting this data. **[LR 2]**

Section 2: Superposition in Action

1. With the power supply off, build the circuit shown in Figure 3.4.

2. In the pre-lab, you used the Superposition Principle to solve the current and voltage distribution in this circuit for arbitrary sources V_1 and V_2. Does this method work in practice with V_1 = 8 V and V_2 = 6 V? Check it out! Measure and record the currents and voltages across the 1 kΩ and 3.9 kΩ resistors. Quantitatively compare your experimental values with your calculated values and account for error sources as before. **[LR 3]**

3. Was the agreement above just a coincidence? The Superposition Principle says one can simply add up the voltage or current contributions from each individual independent source to get total current or voltage. You can verify this directly by setting V_1 = 8 V and V_2 to its equivalent resistance form, then take voltage and current measurements for the 1 kΩ and 3.9 kΩ resistors. Now do the same but activate the other source: Set V_2 = 6 V and V_1 to its equivalent resistance form. Take a look at the corresponding resistor currents and voltages from each set of measurements. Do they contribute to the total as expected? How close are they? Account for sources of error. What new error source might you need to consider? **[LR 4]**

4. Does this rule work for AC voltages as well? Let's make V_1 = 8 V and $V_2 = 6\sin(2000\pi t)$ V (6 is the amplitude in volts and 2000π is the frequency in radians per second). Now, what are your expected voltages across 1 kΩ and 3.9 kΩ resistors? Verify these values experimentally, using your function generator to generate the AC voltage, V_2. **[LR 5]**

   Hint: The oscilloscope and its **subtract** function may be of immense assistance in these measurements. Be mindful to only have ONE earth ground (the function generator ground is *always* earth ground and the oscilloscope ground is *always* earth ground too).

   Did these results turn out as expected? How similar or dissimilar are they from your expected values— quantitatively of course! What uncertainty do the function generator and oscilloscope introduce?

CLEAN UP

1. **Make sure that all your equipment has been turned off**.
2. Please put components back into the cups and make sure that your station appears as it did at the beginning of the lab.

Laboratory 4

DC Circuit Analysis II

WHY AM I DOING THIS LAB? This laboratory will give you a deeper understanding of capacitors; a linear circuit element that, unlike a resistor, can store energy. You will determine the equivalent capacitance of capacitors connected in parallel and capacitors connected in series. Using this, you will be able to simplify complicated capacitor networks. You will also see the relation between the current through a capacitor and the voltage applied. With this, you will gain a better feeling for the properties of capacitors.

BACKGROUND

Linear circuit elements: Consider a circuit element whose current is related to the voltage applied by $i = f(V)$, where f is some function. For example, in a resistor, the relation is given by $i = V/R$, and so $f(V) = V/R$. We say this circuit element is *linear* if the function f is a linear function. The function f is said to be linear if it satisfies the following two properties:

1. $f(V_1 + V_2) = f(V_1) + f(V_2)$, for any two voltages V_1 and V_2.
2. $f(\alpha V) = \alpha f(V)$, for any voltage V and constant α.

So, the output current due to the input $V_1 + V_2$, is the same as the output due to V_1 plus the output due to V_2 (this is the Superposition Principle!) Also, if we multiply the input by a constant, then the output is multiplied by the same constant.

Capacitors: A *capacitor* is a passive linear circuit element designed to store energy. A capacitor consists of two conducting plates separated by an insulator. When a voltage source V is connected to the capacitor, a charge $+q$ is deposited on one plate, and a charge $-q$ on the other. We then say that the capacitor is storing the charge q. The charge stored, q, is directly proportional to voltage V; the relation given by

$$q = CV, \tag{4.1}$$

where the constant C is known as the *capacitance* of the capacitor. By differentiating equation 4.1, and recalling that current is defined as $i = dq/dt$, we see that the current through the capacitor is given by

$$i = C\frac{dV}{dt}. \tag{4.2}$$

So, we can see that the current depends on the rate of change of the voltage applied.

PRE-LAB ASSIGNMENT

1. Read the section in the Lab Manual titled *How to Read Capacitor Values*.
2. Show that a capacitor, whose current-voltage relationship is given by equation 4.2, is a linear circuit element.
3. Looking at the circuit in Figure 4.1 with V set to 10 V DC, calculate the voltage across each capacitor. For the same DC power supply, calculate the energy stored in each capacitor.
4. Now, let's calculate the equivalent capacitance C_{eq} of the circuit shown in Figure 4.2. First, calculate the equivalent capacitance C_{AB} across terminals A and B (that is, the equivalent capacitance of the loop containing the 3.3 µF, 4.7 µF, and 1 µF capacitors). Also calculate the equivalent capacitance C_{DA} across terminals D and A. Finally, calculate the equivalent capacitance of C_{eq} of the entire circuit.
5. If a DC voltage V is applied across terminals D and E of Figure 4.2, what is the voltage across each capacitor? To calculate these voltages you will have to make use of C_{eq} and equation 4.1.

EQUIPMENT

Resistors: One: 1 kΩ, 3.3 kΩ, 6.2 kΩ.

Capacitors: One: 4.7 µF. Two: 1 µF, 3.3 µF

PROCEDURE

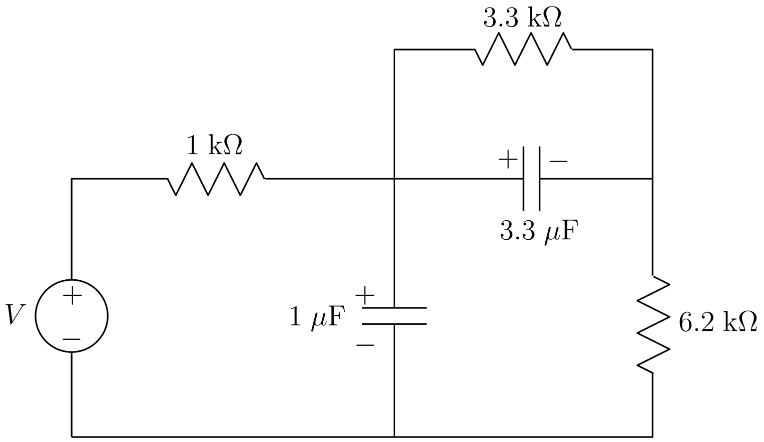


Figure 4.1: The capacitor voltage and current circuit.

Section 1: Capacitor Voltage and Current

1. Build the circuit shown in Figure 4.1 and, using your DC power supply, set V equal to 10 V. (Remember to color-code your circuit!)
2. Measure the voltage across each capacitor. How do the measured voltages agree with the values you calculated in the pre-lab? Try to determine the factors that could contribute to this discrepancy. If you can, quantify the contributions. **[LR 1]**
3. Measure the current through each capacitor. Referring to equation 4.2, do your measured values make sense? Explain why or why not. **[LR 2]**
4. Now let's explore the properties of capacitors using periodically switching DC voltages. Replace V with a square wave that has an amplitude of 10 V peak-to-peak, a DC offset of $+5$ V, and a frequency of 10 Hz.
5. The voltage across each of the capacitors should look like a square wave, but with "rounded edges." These rounded edges are referred to as the *transient response* of the capacitor. Measure the voltage across the 1 μF capacitor and adjust the time axis on the oscilloscope so that you can clearly see the transient response. Provide a screenshot of voltage across the 1 μF capacitor as a function of time, displaying the transient response. (Make sure to include units and label axes.) **[LR 3]**
6. Using equation 4.2 and the transient voltage data you captured above, sketch the shape of the curve describing the current through the 1 μF capacitor as a function of time. Label the axes of your sketch. **[LR 4]**

Section 2: Equivalent Capacitance

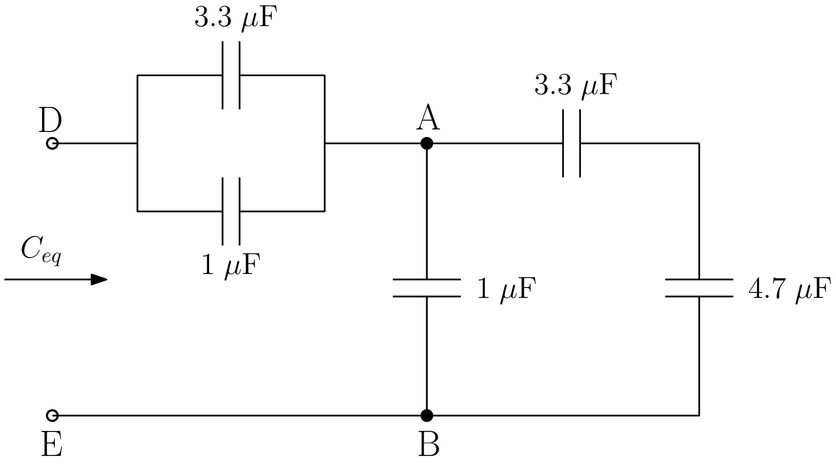


Figure 4.2: The equivalent capacitance circuit.

1. Components such as capacitors are built to be within a certain tolerance of their stated values. For example, you may find that a $1\ \mu\text{F}$ capacitor has a capacitance of only $0.98\ \mu\text{F}$. Measure the capacitance of each capacitor required for building the circuit in Figure 4.2. Record both the measured values and the stated values. **[LR 5]**

2. Now, use these capacitors to build the circuit in Figure 2. Measure C_{AB}, C_{DA}, and C_{eq}. Compare these measurements to the values you calculated in the pre-lab. Are the differences solely due to the capacitor tolerances? What else could be contributing to the difference in results? **[LR 6]**

3. Apply a DC voltage of $10\ \text{V}$ across terminals D and E and try to measure the voltage across any individual capacitor. What do you observe? Does this agree with your expectations from the pre-lab? Try to explain what is going on. *Hint: Consider the effect of the meter.* **[LR 7]**

CLEAN UP

1. **Make sure that all your equipment has been turned off**.

2. Please put components back into the cups and make sure that your station appears as it did at the beginning of the lab.

Laboratory 5

Op-Amp Circuits I

WHY AM I DOING THIS LAB? In this laboratory you will gain experience with one of the most useful and intuitive electronic devices, a basic op-amp. In addition to many other functions, op-amps can be used to make stable voltage sources, signal filters or to perform mathematical operations like addition, subtraction, multiplication, division, differentiation and integration. These are key components in electronic control systems. Any mechanical engineer interested in mechatronics will soon find op-amps indispensable.

BACKGROUND

All the circuits you have dealt with thus far, have been circuits whose output is controlled solely by the input. This is a subtle feature but when contrasted against the new type of circuits in this lab, you will notice the difference. The circuits of this lab control their output based on their input *and* output! That may sound strange at first, but you will soon appreciate this powerful idea known as *feedback*.

Feedback is essentially returning a portion of a system's output to its input. This is a fundamental idea in electronics and control systems. Positive feedback adds to the input. Negative feedback subtracts from the input. Though positive feedback systems are inherently unstable (think of what happens when a microphone collects sound from an amplifying speaker), negative feedback systems can offer a variety of benefits. Negative feedback will increase stability and frequency response of an amplifier and permit careful control of amplifier gain despite device parameters and external effects like changing temperature.

Operational amplifiers typically take advantage of this effect. Without feedback, an operational amplifier will exhibit tremendous gain, known as *open-loop gain* (i.e., a small input will cause an enormous output). By using a negative feedback loop, you can subtract from this gain to achieve a desired final gain. This modified gain is known as the *closed-loop gain*.

Figure 5.1 shows an example of a non-inverting amplifier with negative feedback. You may notice that the two resistors at the output comprise a voltage divider. That is, the ratio of the two determine what fraction of V_{out} is between the

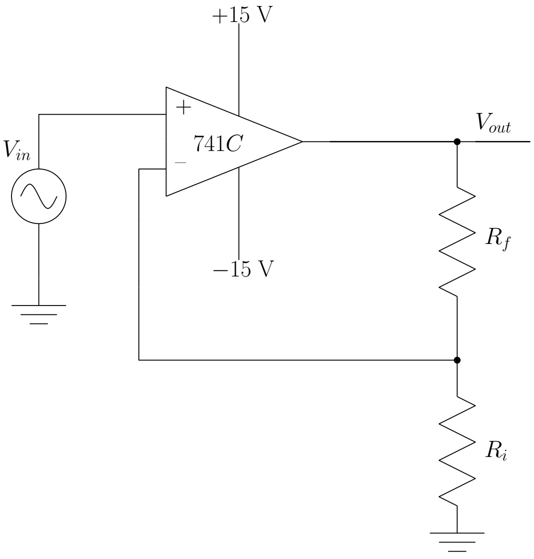


Figure 5.1: Basic Non-Inverting Amplifier.

two resistors. A wire connects this point to the negative terminal of the op-amp allowing it to sample a fraction of the output voltage at the input. This is negative feedback! The feedback fraction B is simply the voltage divider fraction:

$$B = \frac{R_i}{R_i + R_f} \tag{5.1}$$

The non-inverting closed-loop gain, $A_{cl(NI)}$, is taken as the reciprocal of B.

$$A_{cl(NI)} = \frac{1}{B} = \frac{R_i + R_f}{R_i} = 1 + \frac{R_f}{R_i} \tag{5.2}$$

This gain could have been found by employing the two golden rules for *ideal* op-amps connected in a negative feedback configuration:

- No current flows into the input terminals.
- The input voltages are equal and the op-amp will make V_{out} whatever voltage is necessary to make this true.

These two rules are possible under the ideal op-amp assumption. An ideal op-amp is shown in Figure 5.2. This shows what is going on inside the op-amp. The input impedance, Z_{IN}, is infinite but the output impedance, Z_{IN}, is zero. Due to the infinite input impendence, infinitesimal current, I_{IN} enters at the input terminals. Since infinitesimal current enters the input, the offset voltage, Z_{OS}, is infinitesimal. Finally, the dependent voltage source amplifies this infinitesimal voltage by an infinite gain, a. **Note: don't be scared of the term, impedance. It is simply a more general form of resistance which accounts for time-dependent resistances. You'll learn more in Lab 7.**

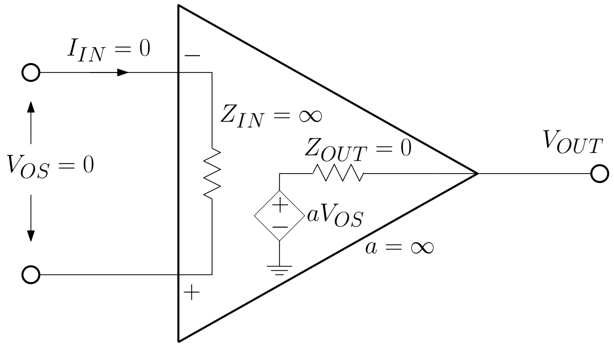


Figure 5.2: Ideal Op-Amp.

Dealing with indeterminate products (products of zero and infinity) is tricky, however, this model will help you to understand real, non-ideal op-amps.

PRE-LAB ASSIGNMENT

1. Using information from the background section, formulate an expression for the closed-loop gain of the inverting amplifier in Figure 5.4.
2. You may have noticed that these amplifiers are *active* devices and thus, require a supply voltage to operate. This is the ±15 V in the figures. If you were supplied two 15 V batteries, draw how you would connect these batteries to an op-amp to provide it with the necessary ±15 V. *This is possible because batteries act as* floating *voltage sources. The DC power supply in the lab can operate this way. Keep this in mind when it is time to assemble your circuits.*

EQUIPMENT

Resistors: Two: 1.0 kΩ. One: 10 kΩ One: 470 kΩ. One: 1 MΩ

Active Elements: One 741C op-amp

PROCEDURE

In the pre-lab, you explored expressions for the gain in two different amplifiers. Let's find out if we can use these ideal expressions to produce calculated results that match experimentally obtained results with non-ideal amplifiers.

1. Assemble the circuit shown in Figure 5.3 using $R_f = 10$ kΩ and $R_i = 1.0$ kΩ (Remember to color-code your circuit!). To implement your ±15 V rails, recall your method from the pre-lab. Also, make sure that these share the same ground as your input signal. To help with the 741C pin-outs, check the

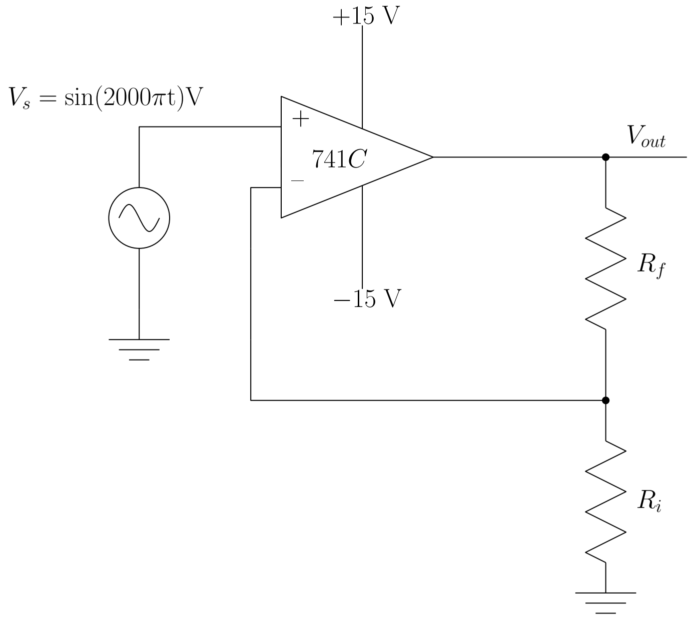


Figure 5.3: Basic Non-Inverting Amplifier.

schematic in the appendix. If you suspect that your 741C is misbehaving, test-circuit that should be found at the front of the classroom to determine if the device is faulty. Include a picture of your circuit in your lab report. **[LR 1]**

(a) Now that you have your resistors, you can use their measured values to calculate the closed-loop gain. Since you know the input voltage, V_{in}, you can use your calculated gain to find the expected V_{out}. This is an important calculation so don't forget to quantify the uncertainty on these calculations and their sources as you have in previous labs. **[LR 2]**

(b) Let's see how the non-ideal amplifier compares. Measure V_{out} and the feedback voltage. Now you can use these to compare your experimental results to your calculations. Are there any differences? If not or if so, explain why you arrived at this result. **[LR 3]**

(c) An ideal non-inverting amplifier has infinite input impedance. What is the input impedance of your non-ideal, 741C-based non-inverting amplifier? One way this could be measured is by placing a $1\ \mathrm{M}\Omega$ resistor in series with the power supply and finding the voltage drop across the resistor. It may be helpful to consider the voltage divider rule and to recognize this configuration as a combination of a module of known resistance and a module of unknown resistance. Why was a $1\ \mathrm{M}\Omega$ chosen? Would you have been successful with a $10\ \Omega$ resistor instead? **[LR 4]**

Note: You should account for the impedance of the multimeter ($10\ \mathrm{M}\Omega$) when solving for the non-inverting amplifier input impedance; Section 19 of the *Supplementary Information* chapter contains a description of signal source loading that will help you with this problem.

2. Now let's investigate an *inverting* amplifier. Assemble the circuit shown in Figure 5.4 using $R_f = 10\ \mathrm{k}\Omega$ and $R_i = 1.0\ \mathrm{k}\Omega$. Include a picture of your circuit in your lab report. **[LR 5]**
   (a) With your measured resistor values, calculate the closed-loop gain of this amplifier and use V_{in} and your calculated gain to find the expected V_{out}. Quantify the uncertainty on these calculations too! **[LR 6]**
   (b) Let's see how the non-ideal inverting amplifier compares. Measure V_{out} and the feedback voltage. In this case, the feedback voltage at Pin 2 is known as virtual ground. Now you can use these values for comparing your experimental results to your calculations. Again, explain any differences or lack-there-of. **[LR 7]**

3. The following are additional questions for your report and do not necessarily require laboratory practice. However, you may find experimental verification very useful.
   (a) Often gains are written in decibel form (dB). What advantage does this type of notation offer? Express your experimentally determined gains in decibels. **[LR 8]**
   (b) What result would you have expected for the inverting and non-inverting amplifier if $R_f = R_i = 10\ \mathrm{k}\Omega$ **[LR 9]**
   (c) For the non-inverting amplifier in Figure 5.3, what gain would you have obtained if R_f was zero and R_i is infinite? This type of arrangement is known as a *voltage follower*. Explain the function of this amplifier and what it might be useful for. **[LR 10]**

CLEAN UP

1. **Make sure that all your equipment has been turned off**.

2. Please put components back into the cups and make sure that your station appears as it did at the beginning of the lab.

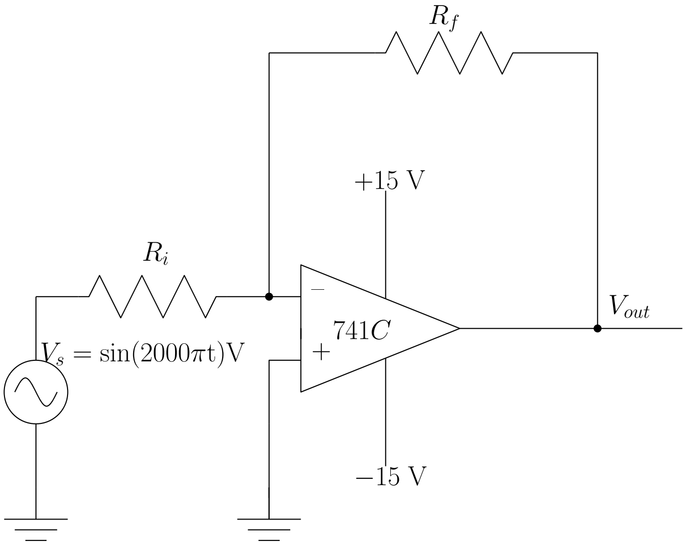


Figure 5.4: Basic Inverting Amplifier.

Laboratory 6

Op-Amp Circuits II

WHY AM I DOING THIS LAB? In this laboratory, you learn about several different uses of operational amplifiers. Complex circuits are often built in stages, which are then connected, or cascaded, together. When cascading two stages together there may be loading effects. This occurs when the input impedance of one stage is too low, and the result is that the overall circuit does not behave as expected. You will build a voltage follower (buffer) which has a very high input impedance and can be used as an intermediate stage to isolate one circuit from the other. You will also build a function generator that can be used to produce square and a triangle waveforms.

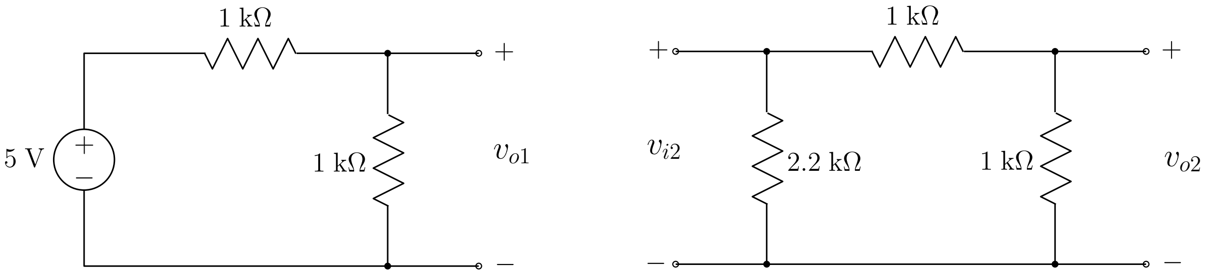
PRE-LAB ASSIGNMENT For the circuit shown in Figure 6.3, determine the resistor values for the five resistors, R_1 to R_5, such that the following conditions are met:

1. Select R_1 and R_2 such that the amplitude of V_{out} is $+5$ V or -5 V depending on whether V_{sq} is at $+15$ V or -15 V, respectively.
2. Select R_3 and R_4 such that the inverting amplifier has a gain of -1 .
3. Select R_5 such that the frequency of the triangle wave is 1 kHz when $C = 0.1\ \mu$F.
4. All resistor values must be selected so that the maximum currents are within the current capability of the LM741 op-amp (~ 25 mA).

EQUIPMENT

Op-Amps: Three: LM 741C

Resistors: Four: 1 kΩ. One: 2.2 kΩ, other resistors TBD.



(a) Stage 1. (b) Stage 2.

Figure 6.1: A two stage circuit.

PROCEDURE

Section 1: Voltage Follower (Buffer)

In this section we will see how a voltage follower (a unity gain non-inverting amplifier) can be used as a buffer between stages of a circuit. By buffer we mean, a module which will isolate the stages such that a change in second stage does not influence the performance of the first.

1. Let's build the first stage of our circuit, shown in Figure 6.1(a). Measure the output of the stage, v_{o1}.

2. Now you can build the second stage of our circuit, shown in Figure 6.1(b) and apply a voltage source to v_{i2} equal to the value you measured for v_{o1} above. Measure, v_{o2} the output of the second stage. **[LR 1]**

3. What happens if we combine your two stages? Connect v_{i2} to v_{o1} so that the two stages of the circuit are cascaded together to form a single circuit. Now, measure v_{o2}. Does this value equal that which you measured for the second stage of the circuit alone? What you are seeing is loading effects. Explain what's going on. **[LR 2]**

4. To prevent this loading from occurring we can build a voltage follower and use it to isolate the two stages of your circuit. To do this, build the circuit shown in Figure 6.2 and connect v_{o1} to v_{ib} and v_{ob} to v_{i2}. Measure v_{o1} and v_{o2} and explain how the voltage follower succeeds in eliminating (or at least reducing) the loading effects. What have you just made and why might this be useful? **[LR 3]**

5. Dismantle your circuit.

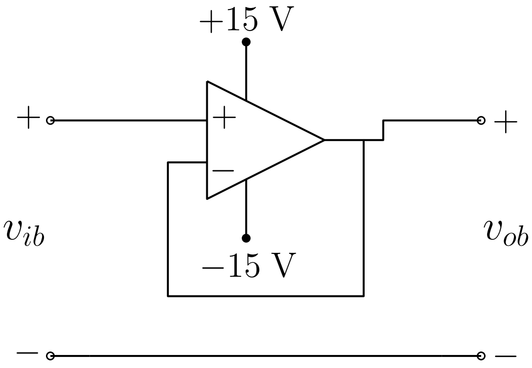


Figure 6.2: The voltage follower (buffer) stage.

Section 2: Function Generator

In this section you will build a function generator capable of producing square and triangular waveforms. The circuit shown in Figure 6.3 consists of three op-amp sub-circuits: (1) a comparator, (2) an inverting amplifier, and (3) an integrator.

1. Assemble the circuit shown in Figure 6.3, don't forget to apply ± 15 V to each op-amp.

   In the pre-lab assignment you were asked to assign values for the five resistors of the function generator circuit. If the lab does not have an exact match for your specified resistors you should use the next closest resistor value; be sure to record the *actual* values used. Using the relationships developed in the pre-lab, what is the *new* expected frequency of your waveforms? **[LR 4]**

2. Connect your oscilloscope to the circuit such that the square wave is shown on channel 1 (CH1) and the triangle wave on channel 2 (CH2). Provide a screen shot of both waveforms. Record the peak-to-peak voltage and frequency for both waveforms. Does this match the values you calculated previously? **[LR 5]**

Frequency Control

A function generator of fixed amplitude and frequency is of limited use. In the pre-lab you should have found that the frequency of the triangle wave (and hence square wave) is dependent on the resistors R_1 to R_5. You also should have found that the amplitude of the triangle wave is dependent on resistors R_1 to R_4 but independent of R_5. Thus, in order to achieve frequency control *independent* of amplitude control we can replace R_5 with a variable resistor[1].

1. Replace resistor R_5 with a $100\ \text{k}\Omega$ variable resistor.

2. Vary the $100\ \text{k}\Omega$ variable resistor and observe the response of both waveforms **[LR 6]**

---

[1] Variable resistors are also known as potentiometers, abbreviated as 'pot'.

on the oscilloscope. Does the response behave as expected? What happens when the resistance is increased? What happens when the resistance is decreased?

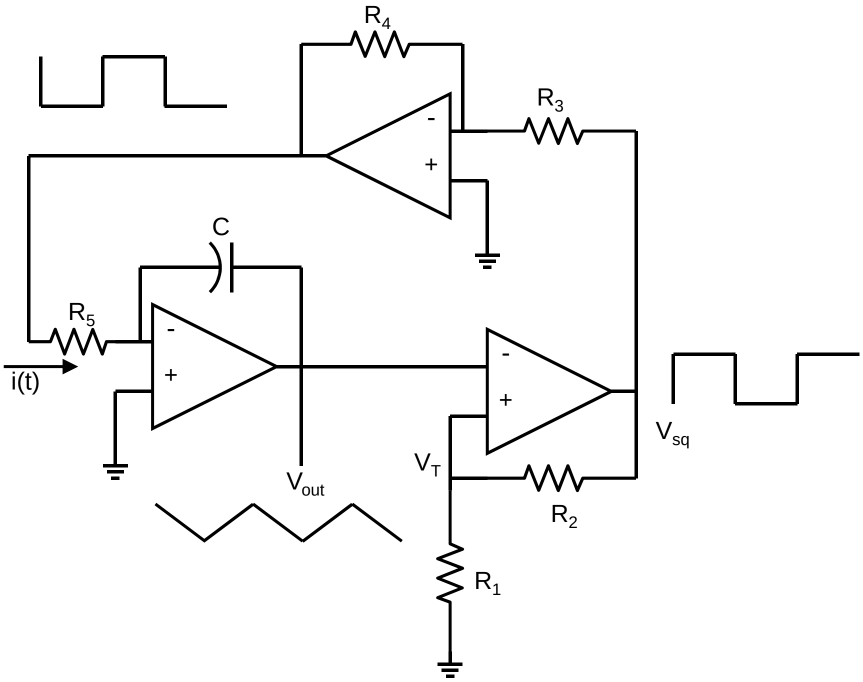


Figure 6.3: Circuitry used to develop a function generator capable of outputting square and triangle waveforms. Note: supply voltages have been left off for clarity, don't forget to supply *all* the op-amps with ± 15 V.

Section 3: Comparator

A comparator is an example of a op-amp used without negative feedback; the absence of negative feedback causes the op-amp to have infinite gain and run at saturation. An op-amp at saturation means the op-amp output is either at its most positive value, $+V_{sat}$, or its most negative value, $-V_{sat}$; where $+V_{sat}$ is slightly less than the positive supply voltage, and $-V_{sat}$ is slightly greater than the negative supply voltage. The sign of the output depends on the *comparison* between an input voltage to a reference voltage, see Figure 6.4. The output of a comparator is given by,

$$V' = \begin{cases} +V_{sat}, & V_{in} < V_{ref} \\ -V_{sat}, & V_{in} > V_{ref} \end{cases}$$

Recall, the output of a linear op-amp can be written $V_{output} = A\,(V_2 - V_1)$, where A is the gain. Since the output of the comparator is piecewise (i.e., the output is either $+V_{sat}$ or $-V_{sat}$), the comparator is a non-linear op-amp.

Note that in Figure 6.3, the resistors R_1 and R_2 form a voltage divider that generates V_T, or V_{ref}, for this comparator. Assuming the values of $\pm V_{sat}$ are equal to the positive and negative supply voltages, respectively, determine the two possible values of V_T. **[LR 7]**

Suppose we desire the triangle waveform to peak at the same voltages as the square waveform ($\sim \pm 15$ V). Describe how you could change the circuit in Figure 6.3 to achieve this. **[LR 8]**

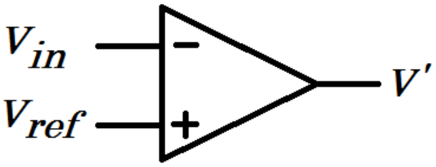


Figure 6.4: A comparator is an example of an op-amp used without negative feedback. The comparator runs at either positive or negative saturation depending on the values of V_{in} and V_{ref}.

Slew Rate

For an ideal comparator, switching between positive and negative saturation (or vise versa) occurs instantaneously; real op-amps, however, require a finite amount of time to adjust. The time rate of voltage change is known as the slew rate and defined as,

$$\text{SR} = \Delta V/\Delta t.$$

1. Turn CH2 off.
2. Adjust the variable resistor (added in the last section) until you are generating a square wave with a frequency of 1 kHz.
3. Using the oscilloscope's amplitude and time knobs, adjust your oscilloscope until you can view only one complete cycle of the square wave. Take a screen shot and calculate the slew rate, SR, for the LM741 chip. Make two sketches to demonstrate the affect of the slew rate on the square wave output. One sketch should be of a low-frequency square wave and the other of a high-frequency square wave. **[LR 9]**

For this lab we are forcing a multipurpose op-amp to act as a comparator. There are IC's designed to be used as a comparator. For these comparators, the switching speed is on the order of nano-seconds, enabling quicker and more accurate switching.

Do the following for the function generator circuit shown in Figure 6.5 : **[LR 10]**

1. Label each highlighted stage (e.g., comparator, integrator, voltage divider, inverting amplifier, buffer, switch).
2. *Briefly* describe the purpose, or function, of *each* stage.
3. The circuit makes use of two variable resistors. Describe the effect *each* variable resistors has on V_{out}^{GEN}.

CLEAN UP

1. **Make sure that all your equipment has been turned off**.
2. Please put components back into the cups and make sure that your station appears as it did at the beginning of the lab.

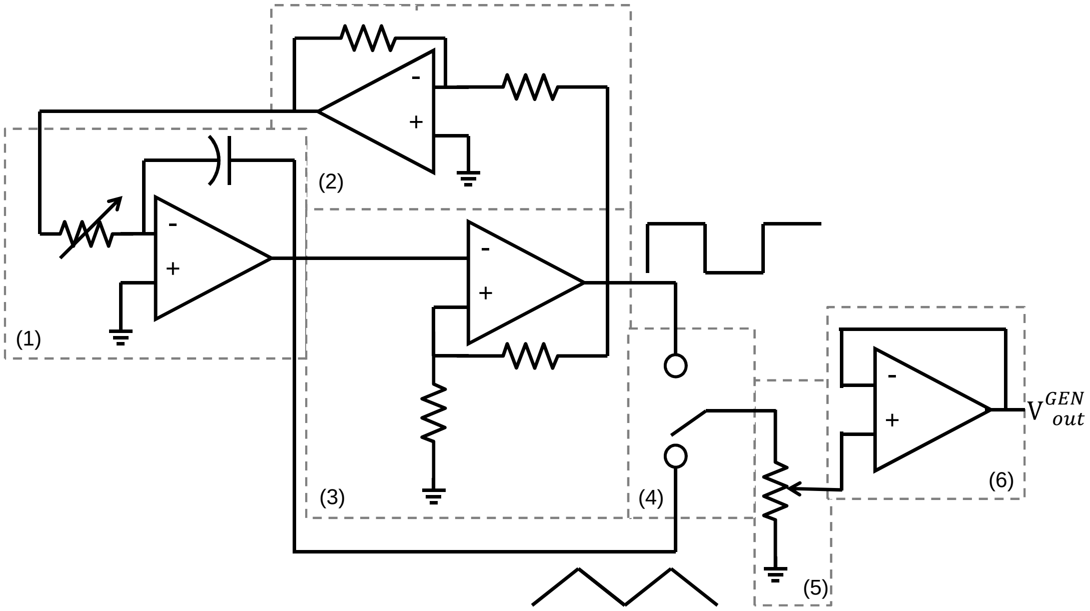


Figure 6.5: Print out or copy this image and include it in your lab report for reference. Label and describe the purpose of each of the five highlighted stages of the function generator circuit shown.

Laboratory 7

First-Order Circuits

WHY AM I DOING THIS LAB? As mechanical engineers, you have previously learned to build first- and second-order dynamical systems with masses, springs and dashpots. It turns out we can build electrical circuits out of resistors, capacitors, and inductors that are governed by the same equations! This lab will give you the chance to build and test first-order dynamical systems from this circuit elements. You will also examine the frequency response of these circuits. You will see that the frequency response could provide extremely useful signal selectivity — an indispensable aspect of a great deal of electronic devices.

BACKGROUND

Step Response of First-Order Circuits: In class you learned general solution to capacitor voltage in a first-order RC circuit:

$$V_C(t) = V_C(\infty) + [V_C(0) - V_C(\infty)]e^{(-t/\tau)}, \quad \text{where } \tau = RC. \tag{7.1}$$

This tells you how the capacitor behaves in the RC circuit with a sudden change in applied voltage (i.e., the step-response). What about the resistor? The resistor is simply the remaining component that satisfies Kirchhoff's voltage law. That is, at each instant in time, the voltage across the resistor and capacitor must add to equal the applied voltage at that instant.

If you'd prefer to think of it another way, the voltage across the resistor is proportional to the amount of current running through it. When the capacitor begins charging/discharging, charge migrates its fastest. Since current is the migration of charge, one would expect the voltage drop across the resistor to be highest during the early part of the discharge.

Frequency Response of First-Order Circuits The step-response analysis examines the circuit's response to a *discrete* change in voltage. What if the applied voltage is changed *continuously* as in AC? The previously considered circuits will exhibit a response based on their *impedance*. This requires a completely different treatment.

We have encountered three major circuit elements: resistors, capacitors and inductors. In DC, resistors have a resistance, R, capacitors have an infinite resistance and inductors have zero resistance. This treatment is actually a special case of the more *general* AC treatment.

In AC, resistance is generalized in a term known as *impedance*, which essentially has the same effect, with an added twist. Impedance, Z, is a *complex number*. That is, impedance is not restricted to the set of real numbers. It may include imaginary numbers as well. In fact, in AC, the impedances of capacitors and inductors are *pure imaginary numbers*. The impedance of a resistor is simply its resistance, which is a *pure real number*. As such, the equivalent impedance of the series capacitor and resistor in Figure 7.1 will be a complex number, the addition of a real contribution and an imaginary contribution, $Z_{eq} = Z_R + Z_C$.

So if the impedance of the resistor is simply the resistance, what is the impedance of capacitors and inductors? These elements are a little more interesting. They are *frequency dependent*. Specifically: $Z_C = \frac{-j}{\omega C}$ and $Z_L = j\omega L$. Where ω is the angular frequency and $j = \sqrt{-1}$. As you can see, the impedance of a capacitor will increase towards infinity at low frequency but the impedance of an inductor will decrease to zero. How exciting! DC is the special case of AC where $\omega = 0$. This is why we treated capacitors as open circuits and inductors as short circuits in DC.

Now you're ready to examine the frequency response of the circuit in Figure 7.1.

PRE-LAB ASSIGNMENT

Section 1: Step Response of First-Order Circuits

1. Equation 7.1 consists of a *transient* portion and *steady-state* portion. This is a very important distinction. In terms of time-dependence, define transient and steady-state and indicate those terms in this equation.

2. Put these tools to the test. Assume voltage V_i in the RC circuit in Figure 7.1 has been at zero for a long time and suddenly steps up to $5\ \mathrm{V}$. That is, $V_i(t = 0^-) = 0\ \mathrm{V}$ and $V_i(t = 0^+) = 5\ \mathrm{V}$. Sketch the voltage across the capacitor and resistor as a function of time

3. Assume voltage V_i in the RC circuit in Figure 7.1 has been at $5\ \mathrm{V}$ for a long time and suddenly steps down to $0\ \mathrm{V}$. That is, $V_i(t = 0^-) = 5\ \mathrm{V}$ and $V_i(t = 0^+) = 0\ \mathrm{V}$. Sketch the voltage across the capacitor and resistor as a function of time.

4. Look at Figure 7.2. Assume the switch is at position A for a long time and suddenly is switched to position B at $t = 0$. In terms of V_1, V_2, R_1, R_2, R_3 and C write v_o as a function of time. (**Hint**: This may seem very difficult at first,

but the same rules apply. This is still a step-response. The only difference is your step does not start from 0 V.)

Section 2: Frequency Response of First-Order Circuits

1. Solve for V_o in Figure 7.1 using the values $R = 10\ \mathrm{k\Omega}$, $C = 0.1\ \mathrm{\mu F}$, and $V_i = \mathrm{A}\sin(\omega \mathrm{t})\ \mathrm{V}$.
2. For the circuit in Figure 7.1, Sketch the real part of V_o, $\Re\{V_o\}$, as a function of ω.
3. What happens if we take V_o across the resistor instead of the capacitor? Try the two previous steps where this time, $V_o = V_R$.
4. A low-pass filter attenuates high-frequencies while allowing low frequencies to pass with little attenuation. The opposite applies to a high-pass filter. A band-pass filter allows a select band of frequencies to pass with little attenuation. Indicate what type of filters you have in your sketches.

EQUIPMENT

Switches: One: 2-way.

Resistors: Two: $10\ \mathrm{k\Omega}$. One: $1\ \mathrm{M\Omega}$.

Capacitors: One: $0.1\ \mathrm{\mu F}$, $1\ \mathrm{\mu F}$.

PROCEDURE

Section 1: Step Response of First-Order Circuits

In the pre-lab, you obtained an expression for $V_o(t)$ in Figure 7.1 for unknown R and C. It would be interesting to observe this response in a circuit that *you* create and then compare theoretical predictions to experimental results.

1. Assemble the circuit shown in Figure 7.1 using $R = 10\ \mathrm{k\Omega}$ and $C = 0.1\ \mathrm{\mu F}$. (Remember to color-code your circuit!)
2. How might we implement a power supply for creating a sudden step in voltage? You may recall that the function generator can provide a square-wave output. This is essentially a DC source which periodically changes its voltage. You can specify how often the voltage is changed by adjusting the frequency of the wave. By adjusting the DC-offset and amplitude you can adjust what voltages the source switches between. To make it simple, adjust your supply so V_i switches between 0 V and 2 V.

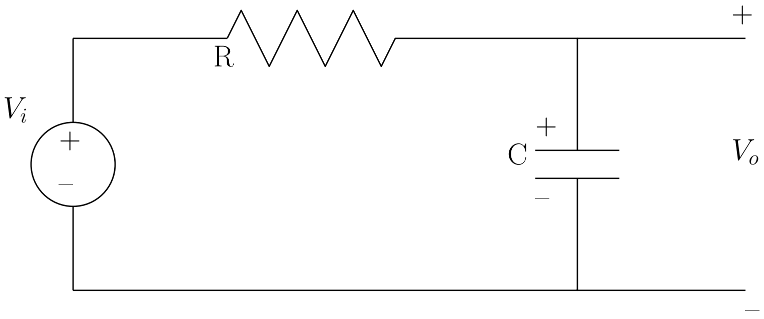


Figure 7.1: An RC First-order circuit.

3. But what about the period? Since it is of interest to see the whole response, that is, until steady state is reached, we should estimate how long to set the period. Recall from the pre-lab where you identified the steady-state and transient portions of $V_o(t)$. When the transient portion goes to zero, we are at steady state. Of course, you can see that this will take an infinite amount of time. It is generally accepted that after 5τ, the transient portion is negligible. Measure your capacitance and resistance and use these values to estimate τ and how long to set your period. **[LR 1]**

4. Try it out! Hook up your function generator to the circuit and observe the output, $V_o(t)$ on the oscilloscope. Remember, it's a good idea to view your input on the oscilloscope at the same time for assurance. Capture your output and label the steady-state and transient portions. **[LR 2]**

5. Did you notice that you are measuring the voltage across the capacitor? Indicate what the capacitor is doing at each portion of the curve on your screen capture (charging, discharging or staying the same). **[LR 3]**

6. Can you verify your time constant experimentally? **[LR 4]**
   (a) According to Equation 7.1, if $\Delta t = \tau$, by what percentage should the initial voltage change?
   (b) Measure the time constant τ by determining the time it takes the output to drop from 100% of the maximum to $x\%$ of the maximum, where x is the quantity you determined above (the cursor function on the oscilloscope may help with these measurements).
   (c) Do your two measured values agree with your calculated values? Since the time constant is often a critical design parameter, we should quantify the uncertainty in your calculations and measurements and indicate the sources (oscilloscope, capacitors, etc.).

7. Now, let's investigate what happens if we take the output of Figure 7.1 across the resistor instead of the capacitor. In the pre-lab you sketched your prediction of the time-dependent voltage across the resistor. Can you vindicate your **[LR 5]**

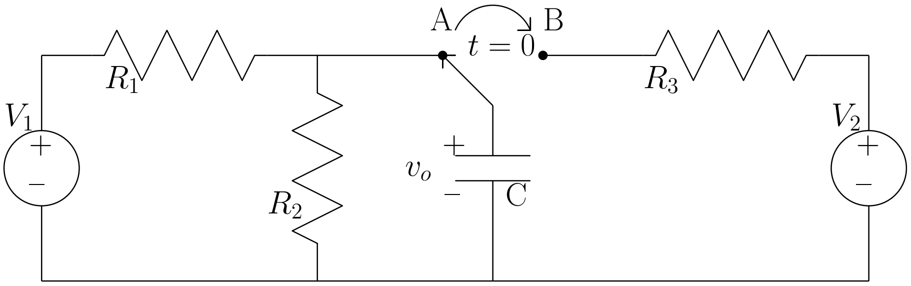


Figure 7.2: A more complicated first-order circuit.

claim? Supply $V_i(t)$ with period of 10τ switching between 0 V and 2 V and examine the output across the resistor. Do you see any differences? If so, why might they exist? Capture this result, label the steady-state and transient portions, and describe what is happening.

8. Let's examine a more interesting circuit that responds to a voltage step elicited by the flip of a switch. You may want to check the components section of the manual to find out how the switch works. Build the circuit shown in Figure 7.2 using $R_3 = 1$ MΩ, $R_1 = R_2 = 10$ kΩ, $C = 1$ μF, $V_1 = 5$ V and $V_2 = 10$ V. It will be helpful to view the response on the oscilloscope. Estimating the time constant may help you to easily capture the output.

9. When you are ready, flip the switch from position A to position B and observe the result. You may want to use the RUN/STOP button to freeze your oscilloscope display.

10. In the pre-lab, you generated a general formula for v_o. Let's see if it works! **[LR 6]** Input your values for V_1, V_2, R_1, R_2, R_3 and C and calculate v_o at $(t = 1\text{ s})$ and $(t = 4\text{ s})$. Do your these calculations match the corresponding measured values? You may want to include a screen-shot to augment your claim in your report. Being able to reliably predict voltage at points in time can be very important, so again, we can't forget to cite uncertainty and possible sources.

Section 2: Frequency Response

Previously we applied periodically changing DC voltages. Now we will apply continuously changing, AC voltages to the circuit in Figure 7.1. Set V_i as a sine wave with peak-to-peak amplitude 5 V and zero DC offset.

A common way to show the frequency response of a *filter* is to plot its gain in decibels against frequency where gain is defined as

$$G_{dB}(f) = 20\log\left(G_V(f)\right) = 20\log\left|\frac{V_o(f)}{V_i(f)}\right| = 10\log\frac{P_o(f)}{P_i(f)}. \tag{7.2}$$

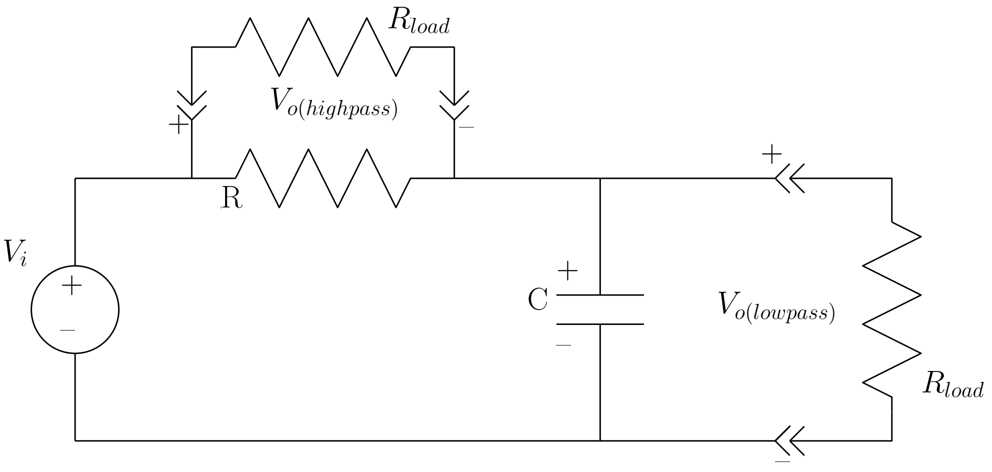


Figure 7.3: Hi-pass and Low-pass filter loading.

1. In the pre-lab, you showed that by taking the output across the capacitor or resistor of Figure 7.1, you can have a high-pass or low-pass filter. Let's assess the frequency response of the low-pass filter by varying frequency and finding corresponding gains. Plot the data to show the trend and make sure you have enough points to do this sufficiently. It may be helpful to note that conventionally, these plots have logarithmic axes. Keep this in mind when determining the range of frequencies to examine. **[LR 7]**

2. The frequency at which half-power is obtained is known as the cutoff frequency. This is an important number for any filter. Indicate the cutoff frequency on your plot. To better understand where the power is being dissipated, consider Figure 7.3 which shows the same circuit as in Figure 7.1 but with load resistors added to the low-pass output and high-pass output. It is across these resistors that the power is dissipated. **[LR 8]**

3. Repeat the above two steps for the high-pass filter. **[LR 9]**

4. The cutoff frequency is also known as the -3 dB point. The cutoff frequencies you just found should correspond to a gain that is 3 dB less than the maximum. Recalling that cut-off frequency is the point of half-power, prove that the -3 dB frequency is really just the same thing. **[LR 10]**

5. Your plots will have characteristic shapes. Think about what these shapes mean and suggest a practical use for both types of circuits. **[LR 11]**

CLEAN UP

1. **Make sure that all your equipment has been turned off**.

2. Please put components back into the cups and make sure that your station appears as it did at the beginning of the lab.

Laboratory 8

Second-Order Circuits

WHY AM I DOING THIS LAB? In the previous laboratory we looked at circuits with a single storage element, a capacitor. In this laboratory we will investigate circuits which contain two storage elements. Such circuits are known as second-order circuits. We will look at a series RLC circuit which contains a resistor an inductor and a capacitor in series. We will see how, by varying the values of these three components, we can obtain three distinct voltage responses; underdamped, critically damped, and overdamped.

BACKGROUND

Second order circuits are circuits whose responses are described by differential equations which contain second derivatives. For this laboratory we will derive the necessary background in the Pre-lab section.

PRE-LAB ASSIGNMENT

1. Let's derive the differential equation for the series RLC circuit shown in Figure 8.2.
    (a) Since Figure 8.2 is a series circuit, the current is the same through all three elements. Call this current i. Express the voltage across the resistor and the voltage across the inductor in terms of this current i.
    (b) Now, apply Kirchhoff's voltage law around the loop, using two expressions you found in Question (a) and denoting the voltage across the capacitor as v_C.
    (c) Recall that the current, i, is related to the capacitor voltage, v_C by the expression

$$i = C\frac{dv_C}{dt}. \tag{8.1}$$

Differentiating this we get

$$\frac{di}{dt} = C\frac{d^2 v_C}{dt^2}. \tag{8.2}$$

Substituting Equations 8.1 and 8.2 into the expression you found in Question (b), obtain a second order differential equation in terms of v_C.

(d) Rearrange the differential equation so that it is in the form

$$\frac{d^2 v_C}{dt^2} + 2\alpha\frac{dv_C}{dt} + v_C\omega_0^2 = A,$$

and determine the constants ω_0 and α in terms of R, L, and C. The constant A should also depend on the input voltage V. The constant ω_0 is called the *natural frequency*, and α is called the *neper frequency*.

2. The natural and step responses of this RLC circuit can be split into three categories based on the values of ω_0 and α: overdamped, critically damped, and underdamped. Using your textbook if necessary, give the criteria for these three categories in terms of ω_0 and α and sketch their step responses.

EQUIPMENT

Resistors: One: $1\ \mathrm{k\Omega}$. Two: $10\ \Omega$. One: $1\ \mathrm{k\Omega}$ potentiometer.

Inductors: One: $0.33\ \mathrm{mH}$.

Capacitors: Two: $1\ \mu\mathrm{F}$. One: $0.1\ \mu\mathrm{F}$.

Op-Amps: One: LM 741C.

PROCEDURE

Section 1: Voltage Follower

Figure 8.2 shows the series RLC circuit which will be the focus of this laboratory. We will use the function generator to provide the voltage V to the circuit. However, this circuit has a very low input impedance. As we've seen before, when cascading circuits together this low input impedance can introduce loading effects. To buffer the function generator from the RLC circuit, we will build a voltage follower.

1. Build the voltage follower shown in Figure 8.1. The $-$ line is simply the ground. The capacitors running from the $+15$ V and -15 V lines to ground are called *coupling capacitors*. They filter out any AC component of the signal, and ensure that the voltage supplied to op-amp is steady.

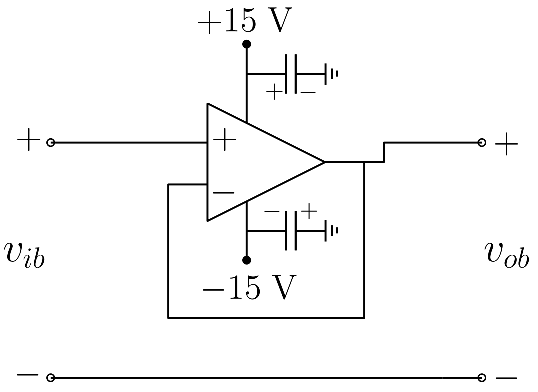


Figure 8.1: The voltage follower.

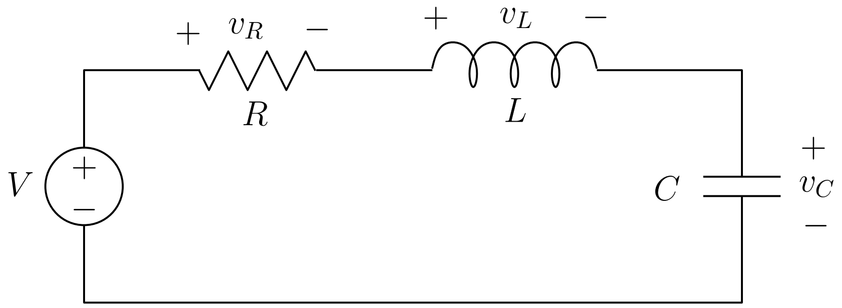


Figure 8.2: Series RLC circuit.

2. Connect the function generator to v_{ib} and verify that the output at v_{ob} is as expected.
3. Leave this circuit assembled.

Section 2: Series RLC Circuit Response

1. Build the series RLC circuit shown in Figure 8.2 using the $1\ \mathrm{k\Omega}$ potentiometer for R, $L = 0.33\ \mathrm{mH}$, and $C = 0.1\ \mu\mathrm{F}$. Use the output of the voltage follower, v_{ob} as V.
2. Set the function generator to a square wave with peak-to-peak amplitude of $2\ \mathrm{V}$ and zero DC offset and set $R = 1\ \mathrm{k\Omega}$. Use both channels of the oscilloscope, displaying the square wave on one channel, and $v_C(t)$ on the other. Adjust the frequency of the square wave so that the step response of the circuit is clearly visible.
3. Is your system overdamped, underdamped, or critically damped? Take a screenshot of the response. **[LR 1]**
4. What are the natural frequency ω_0, and neper frequency α for this circuit? **[LR 2]**

5. Calculate the resistance, R_{crit}, required to make the system critically damped. Next, vary the potentiometer until the response appears critically damped. Compare this resistance with the value you calculated for R_{crit}. Comment on any differences between the two values. **[LR 3]**

6. Experiment with different values of R. Notice how the step response changes as the resistance is varied. For what values of R is the system underdamped? Overdamped? Take screenshots of these two responses. **[LR 4]**

7. Now, switch the resistor and capacitor so that you can measure $v_R(t)$, and set R so that the system is underdamped. (The switch is required so that you don't ground the circuit at the $-$ve terminal of the resistor with your oscilloscope probe.) Take a screenshot of the response. Does this response make sense? Explain why or why not? **[LR 5]**

8. Now rearrange the circuit so that you can measure the inductor voltage $v_L(t)$ and set R so that the system is overdamped. Take a screenshot of the response. Does this response make sense? Explain why or why not? **[LR 6]**

CLEAN UP

1. **Make sure that all your equipment has been turned off**.

2. Please put components back into the cups and make sure that your station appears as it did at the beginning of the lab.

Part II

REPORT EXPECTATIONS

Section 9

Grading Rubric

Since excellence in technical writing is critical in engineering careers, we will develop this skill in this course. In this part you will find a sample laboratory and the associated report to demonstrate the expected level of quality in analysis, writing, and presentation. The report illustrates what is expected for your lab reports. The numbers on the right highlight the strong points and are explained on the following page. For those that like to learn from mistakes, an example *bad* report is provided subsequently with a list common errors and those specific to the report.

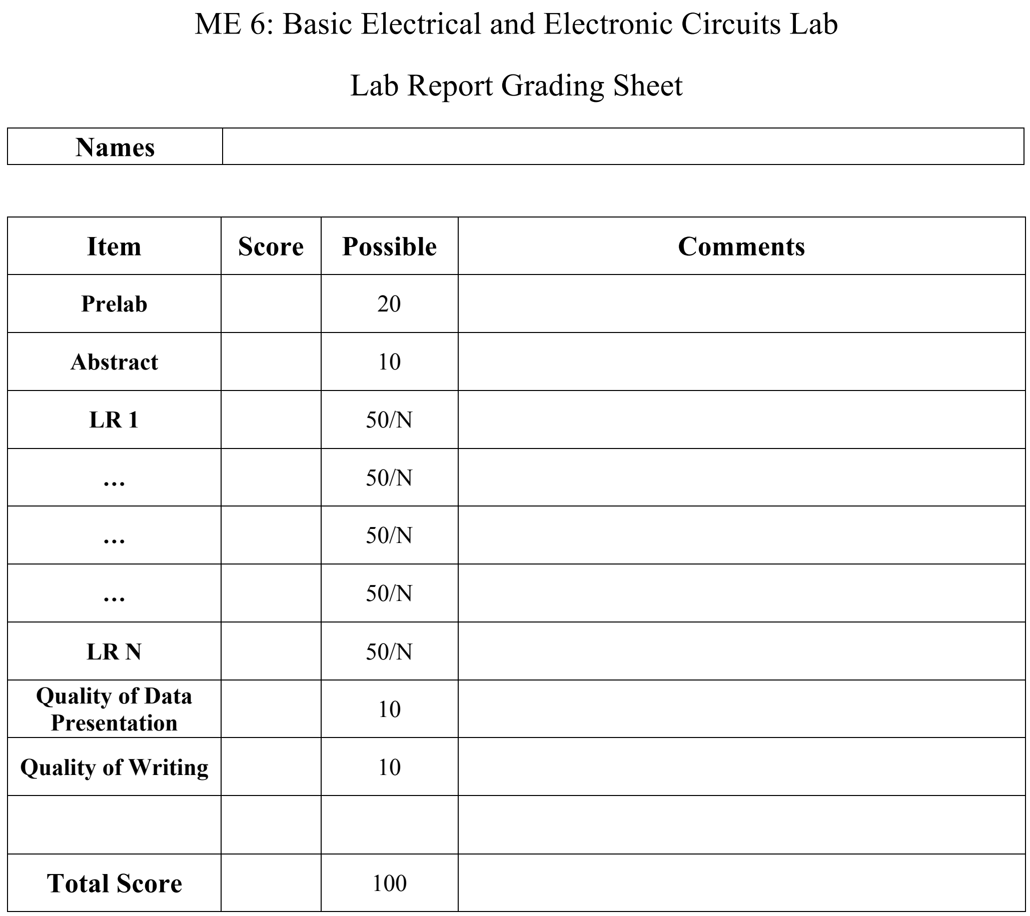
You should organize your report in the following manner.

ABSTRACT You've just conquered the lab; now give the highlights! This section briefly summarizes what you did and what you learned. It comes from your own mind; it should be in your own words. The more comfortable you are with your work, the more easily you'll be able to write this.

PRE-LAB The pre-lab is essential since it ties theory to your experimental findings; it gives you educated expectations. There are **three** steps. **First**, *before* the lab begins, complete a *typed* pre-lab that answers all the pre-lab questions in complete sentences. Calculations may be handwritten and attached if done *neatly*. **Second**, give this to your TA at the start of the lab. The TA will review it, sign it, and return it to you in the same session. To get credit for the pre-lab, attach the signed copy to your lab report. **Third**, while preparing your lab report, make sure your pre-lab (especially anything calculated) is in agreement with your data. If it is not, *find the source(s) of the error(s)*. If it's in your pre-lab, you may make corrections and *attach the revised pre-lab to your report as well*. If it's in your data, you will need to *explain the disagreement in your report*. Note: some disagreements are expected and you ought to be able to justify them.

DISCUSSION In this section you should answer the **[LR]** questions. The point here isn't just to show your data; there is little value in that alone. This is your chance to show that you've understood the concept in question, if you can connect theory to practice and how you can account for disparity. Retain the **[LR]** numbering scheme used in the laboratory instructions. Again, your answers can be brief, but they should be in full sentences. All plots and tables must have appropriate labels and captions.

The following grading rubric will be used to evaluate your reports.



ME 6: Basic Electrical and Electronic Circuits Lab

Lab Report Grading Sheet

Names	

Item	**Score**	**Possible**	**Comments**
Prelab		20	
Abstract		10	
LR 1		50/N	
…		50/N	
…		50/N	
…		50/N	
LR N		50/N	
Quality of Data Presentation		10	
Quality of Writing		10	
Total Score		100	

Figure 9.1: ME 6 Sample Grading Rubric. Note that *N* indicates the number of LRs in the given lab

Section 10

Sample Laboratory

WHY AM I DOING THIS LAB? Luckily, you're not! However, you should read the lab and use the sample lab report that has been written for this lab as a guide for your reports.

BACKGROUND

Electric charge is a characteristic property of some subatomic particles, and it comes in two types, positive and negative. For example, an electron has negative charge, and a proton has positive charge. Now, negative charge attracts positive charge. So, we must expend energy when we separate a negative charge from a positive one. Or, to think of it differently, in separating charge, we are storing energy. *Voltage* (v) is the energy (E) per unit charge (q) stored in the separation. We write this as

$$v = \frac{dE}{dq}.$$

When energy is measured in joules, and charge in coulombs, the voltage has units of volts (V).

When we start moving charges around, we generate electromagnetic fields—intertwined electric and magnetic fields which are the basis for wonderful things such as induction motors and transformers. To quantify the motion of charge, we introduce *electric current* (i), which is the rate of charge flow

$$i = \frac{dq}{dt}.$$

When charge is measured in coulombs, and time in seconds, current has units of amperes (A). Now, if we are given a resistor—a circuit element which impedes current—with resistance R (measured in ohms, Ω), the voltage across, and current in, the resistor are related by:

$$v = iR.$$

This is expression is known as *Ohm's law*.

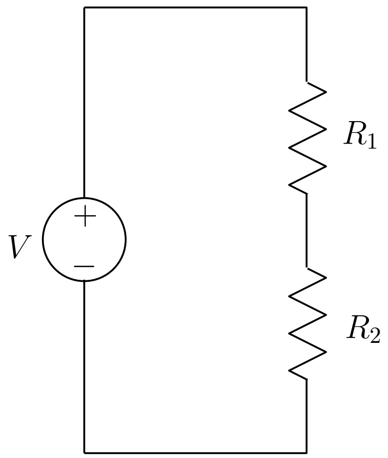


Figure 10.1: Voltage divider.

PRE-LAB ASSIGNMENT

1. Using Ohm's law, calculate expressions for the current in, and voltage across, the resistor R_2 in Figure 10.1.
2. If the current you calculated flows through the resistor for 10 seconds, how much charge has passed through the resistor.

EQUIPMENT

Resistors: One: $1\ \mathrm{k\Omega}$, $6.2\ \mathrm{k\Omega}$.

PROCEDURE

1. Build the circuit shown in Figure 10.1. Set R_1 to $10\ \mathrm{k\Omega}$, R_2 to $100\ \mathrm{k\Omega}$, and using your DC power supply set V to $10\ \mathrm{V}$. (Remember to color-code your circuit!)
2. Measure the voltage across, and current in, the resistor R_2. How do these measured quantities agree with the values you calculated in the pre-lab? Try to determine the factors that could contribute to this discrepancy. **[LR 1]**
3. Now set V to a triangle wave with a frequency of $2\ \mathrm{kHz}$, a peak-to-peak amplitude of $5\ \mathrm{V}$ and zero DC offset.
4. Measure the voltage across R_1 using the oscilloscope. Take a screen shot of the voltage as a function of time and include it in your report. Measure the period of the waveform on your oscilloscope and compare it to that of the input. **[LR 2]**

CLEAN UP

1. **Make sure that all your equipment has been turned off**.
2. Please put components back into the cups and make sure that your station appears as it did at the beginning of the lab.

Section 11

Example of a *Good* Report

By: Siméon-Denis Poisson
Partner: Pierre-Simon Laplace
Course: ME 6, Lab Section 3 **[1]**
TA: Joseph-Louis Lagrange
Date: December 13, 2013

ABSTRACT In this lab, AC and DC signals were applied to a voltage divider. Observed AC results were shown to match expected values. Under experimental uncertainty analysis, DC results were shown to fall within experimental uncertainty of expected values. **[2]**

PRE-LAB

1. In order to find expressions for current through, and voltage across, R_2, we need the equivalent resistance R_1 and R_2: $R_{eq} = R_1 + R_2$. Recognizing this arrangement as a voltage divider, the voltage across R_2 is

   $$V_2 = V\frac{R_2}{R_{eq}} = V\frac{R_2}{R_1 + R_2}.$$

   The current is the same through both resistors and is simply found with the equivalent resistance and Ohm's law:

   $$I = \frac{V}{R_{eq}} = \frac{V}{R_1 + R_2}.$$

2. The total charge Q passing through R_2 in 10 seconds can be found by integrating the current across that span of time. Since the current is constant, we have

   $$Q = \int_{t=0}^{t=10} I dt = 10I = 10\frac{V}{R_{eq}}.$$

DISCUSSION

1. The measurements made with the TX3 are shown in Table 1-1.

Table 1-1: Measurements made with TX3.

	Measured	Units	Absolute Uncertainty (±)	Relative Uncertainty (%)
V	10.06	V	0.015	0.15
R_1	9.96	kΩ	0.03	0.30
R_2	99.8	kΩ	0.3	0.30
V_2	9.14	V	0.015	0.16
I	91.7	µA	0.2	0.24

[3]

Note that absolute uncertainty was found with the help of the *Accuracy of Equipment* section of the Lab Manual. The relative uncertainty was found by writing the ratio of the absolute uncertainty to the measured value as a percentage.

Now the formulae from the pre-lab can be used to find the expected V_2 and I. In these calculations $RU(x)$ is the relative uncertainty of x and $AU(x)$ is the absolute uncertainty of x. For the equivalent resistance we have **[4]**

$$\begin{aligned} R_{eq} &= (R_1 + R_2) \pm (AU(R_1) + AU(R_2)) \\ &= (9.96 + 99.8) \pm (0.03 + 0.3) \\ &= 109.8 \pm 0.3 \text{ k}\Omega. \end{aligned}$$

For the voltage across R_2 we get **[5]**

$$\begin{aligned} V_2 &= V\frac{R_2}{R_{eq}} \pm \left((RU(V) + RU(R_2) + RU(R_{eq})) \times V\frac{R_2}{R_{eq}}\right) \\ &= 10.06\frac{99.8}{109.8} \pm \left((0.15\% + 0.30\% + 0.30\%) \times 10.06\frac{99.8}{109.8}\right) \\ &= 9.15 \pm 0.07 \text{ V}. \end{aligned}$$

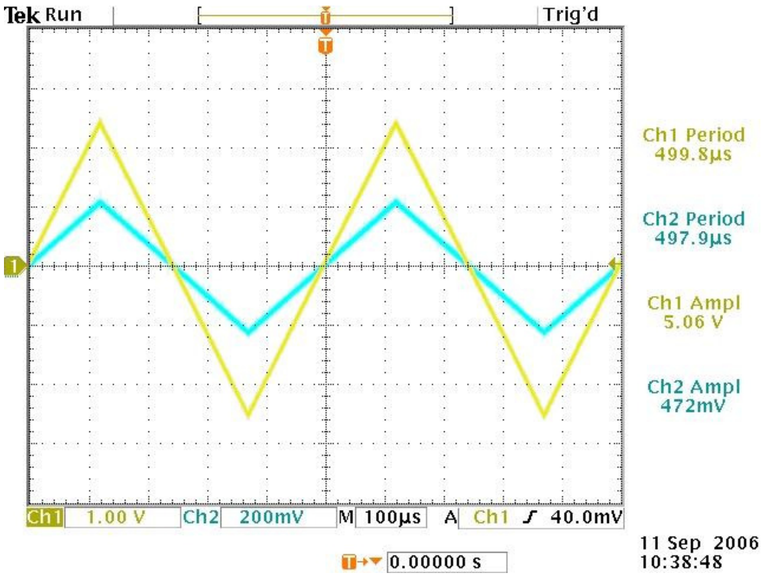
Finally, the current is

$$\begin{aligned} I &= \frac{V}{R_{eq}} \pm \left((RU(V) + RU(R_{eq})) \times \frac{V}{R_{eq}}\right) \\ &= \frac{10.06}{109.76} \pm \left((0.15\% + 0.30\%) \times \frac{10.06}{109.76}\right) \\ &= 91.7 \pm 0.4 \ \mu\text{A}. \end{aligned}$$

We can see that expected values of V_2 and I fall within experimental uncertainty. Had they been a little off, it may have been due to the internal resistance of the multimeter and the power supply, providing unaccounted for resistances in the voltage divider.

2. The input voltage, V, and the voltage, V_1 are shown in Figure 1-1. The input

amplitude and period are 5.06 V and 499.8 µs respectively. Across R_1 the amplitude and period are 472 mV and 497.9 µs respectively. Here it can be seen that though the amplitudes differ by nearly a factor of 11 as expected by the voltage divider, the period remains relatively unchanged at approximately 500 µs.



[6]

Figure 1-1: Ch1 shows the input triangle waveform and Ch2 the V_2 waveform.

EXPECTATIONS FOR LABORATORY REPORTS

The items in the following list describe the highlighted features of the sample lab report.

[1] The report is typed, titled (normally it would contain the lab number as well) and labeled with

- your name,
- name(s) of your lab partner(s),
- course number and lab section,
- your TA's name,
- date submitted.

[2] The abstract provides a very quick (1 paragraph) overview of what you did in the lab and what you learned. The abstract should summarize the following:

- What was done?
- What was observed?
- What were the conclusions?
- What did you learn from the observations and conclusions?

The abstract should be written after you conduct your experiment and in past tense.

[3] Tables are numbered with the lab report number and the table number, separated by a hyphen. Units are indicated for all values.

[4] New variables are defined as they are used.

[5] Equations are shown for calculations and values have units.

[6] Plots are scaled to clearly display the data of interest. Axes are clearly identified with labels, scales, and units. Figures are numbered with the lab report number and figure number, separated by a hyphen.

Section 12

Example of a *Bad* Report

By: Siméon-Denis Poisson
Partner: Pierre-Simon Laplace
Course: ME 6, Lab Section 3
TA: Joseph-Louis Lagrange
Date: December 13, 2013

ABSTRACT In this lab, we familiarized ourselves with the lab equipments. **[1]**

DISCUSSION

LR 1 **[2]**

Resistor 1: green, blue, yellow, gold = $560\ \Omega \pm 5\ \%$ **[3]**
Resistor 2: red, orange, violet, black, brown= $237\ \Omega \pm 2.37\ \%$ **[4]**

LR 2

Using the multimeter (BK Tool Kit 2704B), we measured the resistance of resis-
tors 1 and 2 from LR1 and found that resistor 1 is 558 and resistor 2 is 238 Ω. **[5]**
According to the lab manual (p.52), the BK Tool Kit multimeter has a 3 % uncer-
tainty in measuring the resistance. This relative uncertainty (RU) is translated **[6]**
to absolute uncertainty (AU) by equation 1.

AU = RU * Measured Resistance (1)

Using equation (1), the measured resistances of resistors 1 and 2, including the AU, are 558 ± 17 Ω and 238 ± 7 Ω, respectively. Furthermore, after converting RU to AU, the labeled resistances of resistors 1 and 2 are 560 ± 28 Ω and 237 ± 2 Ω. Therefore, it can be seen that the estimated resistances from deciphering the color bands and the measured resistances from multimeter reading converge to each other, when the AU is considered. **[7]**

LR 3

The resistance from my body was 100 kΩ. Our measurements are more suscepti- **[8]**

ble with this kind of handling error with low resistance. [9]

LR 4

99.8 nF [10]

LR 5

one way: 0.542 V; the other way: O.L. The multimeter is trying to measure the voltage across the diode. [11]

LR 6

In working with oscilloscope probes, it is important to ground the probes first. This ensures that the voltage reference is zero prior to making measurements. We did so by connecting the input clip to the ground clip, and adjusting the voltage at the mid-point on the oscilloscope screen. Furthermore, we had adjusted the vertical (voltage) and horizontal (time) scales to 5 V and 200 μs per division. The signal from the grounded probe and adjusted scales displayed in channel 1 of the oscilloscope is shown in Figure 1-1.

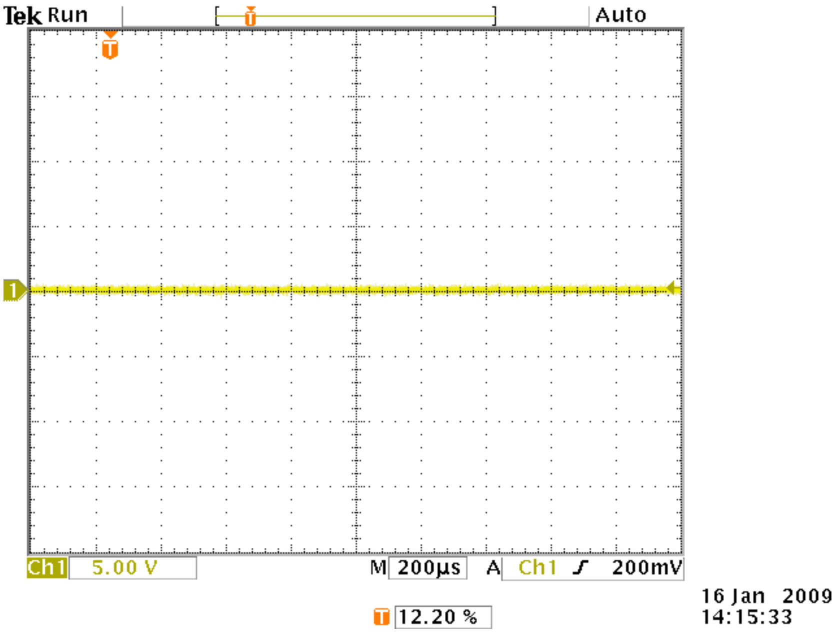
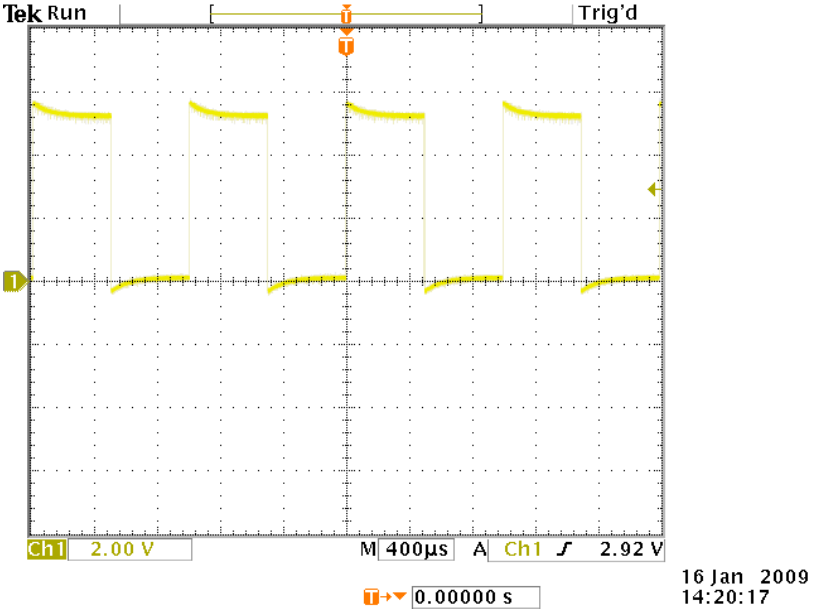


Figure 1-1. The oscilloscope signal of a grounded probe in adjusted scales displayed in channel 1.

LR 7

[12-13]

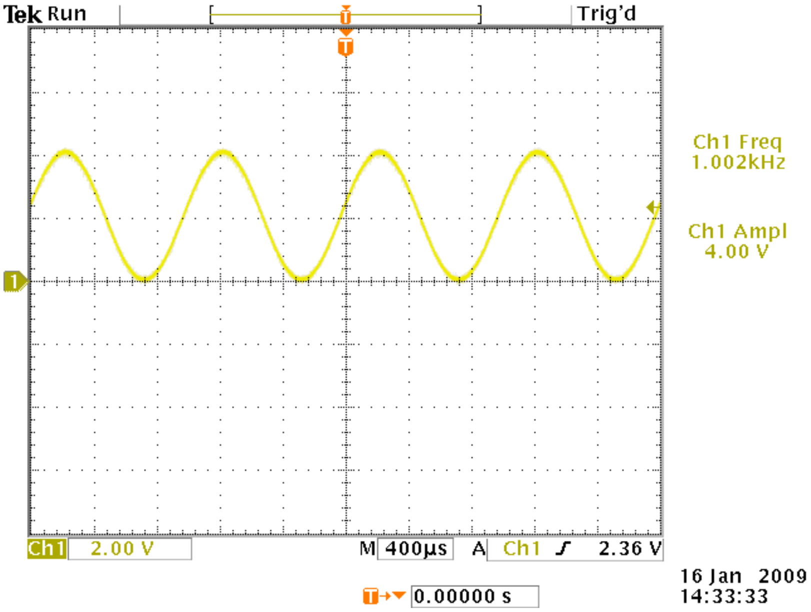


LR 8

The frequency and the amplitude are 1.006 kHz and 5.20 V. **[14]**

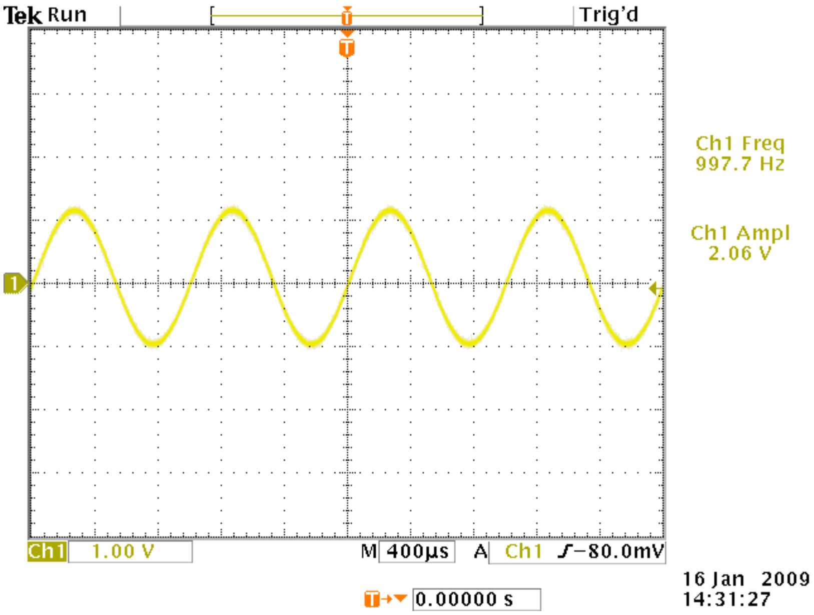
LR 9

[12-13]



LR 10

[12-13]



LR 11

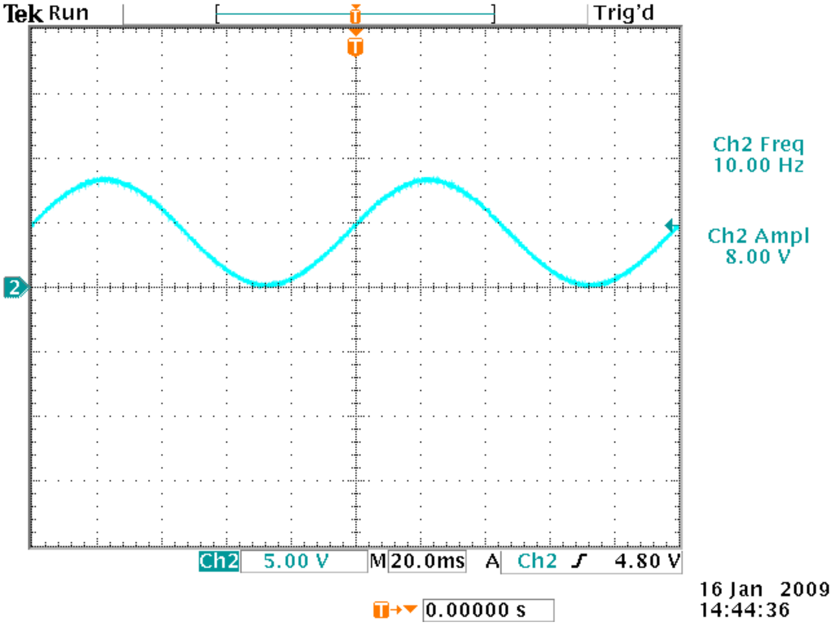
Multimeter: 5.02 V.

Oscilloscope: 5.14 V. **[10]**

No, it does not make sense to use the amplitude tool. **[16]**

LR 12

[12-13]


Tek Run
Trig'd
T
2
Ch2 Freq
10.00 Hz
Ch2 Ampl
8.00 V
Ch2 5.00 V
M 20.0ms
A Ch2 ∫ 4.80 V
16 Jan 2009
14:44:36
0.00000 s

COMMON ERRORS IN LABORATORY REPORTS

The items in the following list are the most common mistakes made by students in submitting lab reports since 2006. You can learn from their errors.

- Abstracts miss the point; they do not mention what was observed and learned.
- Pre-lab calculations do not match data yet the difference is not even acknowledged or no explanation is given to account for the difference.
- Equipment inaccuracy is used to explain a difference far greater than it could possibly account for.
- LR answers are given in incomplete sentences.
- LR answers are given without LR context (without indicating the question or without indicating experimental procedures that produced the data).
- LR answers do not convey thought; a value, figure, or table alone does not convey thought. Show your work. Demonstrate clear thinking.
- Formatting: headlines are not correct; plots and tables are not labeled and captioned.
- Data which should be presented in a table is written as a list, making it difficult to read and compare.

SPECIFIC ERRORS IN SAMPLE REPORT

[1] This abstract is too brief. Remember, the abstract should cover four things: (1) what was done, (2) what was observed, (3) what were the conclusions, and (4) what were the lessons from the observations and conclusions. In the case of lab 1, some of the main points include working with the lab components and equipments and knowing their uncertainties. Knowing these objectives, what experiments were done to help achieve these learning objectives? Use your own words.

[2] A proper table can be created to help present data clearly. According to the manual, a proper table is numbered with the lab report number and the table number, separated by a hyphen. Also, units are indicated for all values.

[3] A report should be written so that someone who has no knowledge of the experiments can understand what happened. So, please elaborate on what was done (e.g. you determined the resistance from reading the color bands on the resistors, then you used a conversion chart to figure out the estimated resistance and tolerance). In this case, it would be a good idea to include a formula to explain how the color bands are translated to resistance values.

[4] Here, the relative uncertainty was converted to absolute uncertainty without showing work. While extra work was done, it was not done properly. You should always show work. In this case, all that is needed is equation 1 and a short description that equation 1 was used to convert relative uncertainty to absolute uncertainty.

[5] Although it was not the case here, you can make a proper table if you find it better presents the data.

[6] The information source should always be cited. In this case, a short sentence would do the trick.

[7] This is the proper way to ”compare” the labeled values and the measured values. Do they converge when the uncertainties are considered? If not, why not? Overall, this answer to LR2 shows what the expected answers look like.

[8] There is no description of what was done to reach the observation that the body resistance was 100 kΩ. Also, there is no calculation.

[9] An explanation is expected for a question like this.

[10] An answer with only a value is not a good answer for a proper report. A proper answer is written in complete sentences and it describes what was done and what was observed. Also, the answer did not include the uncertainty as the manual instructed.

[11] The first answer here was not in complete sentences. The second answer, while written in a complete sentence, does not answer the question at an analytical level as expected.

[12] In the case where a figure is displayed, the figure must be explained with context. The LR question must also be answered explicitly.

[13] In addition to the explanation, the figure should be labeled and captioned.

[15] The answer did not include uncertainties. The uncertainties of the different equipments can be found in the manual.

[16] This is an improper way to compare values. How far off is too far off? Do they match when the uncertainties are considered? If not, why not?

Part III

SUPPLEMENTARY INFORMATION

Section 13

Accuracy of Equipment

The accuracy of each piece of equipment is summarized in Table 13.1.

Device	**Measurement**	**Percent Error**	**Offset**
Tektronix TX3	DC Voltage	0.05%	1
	AC Voltage	0.4%	2
	DC Current	0.02%	2
	AC Current	0.06%	2
	Resistance	0.1%	2
	Capacitance	1%	3
	Frequency	0.002%	1
Fluke 37	DC Voltage	0.1%	1
	AC Voltage	2%	3
	Resistance	0.2%	1
BK Tool Kit 2704B	DC Voltage	0.5%	1
	AC Voltage	1.5%	4
	DC Current	1%	1
	AC Current	1.2%	4
	Resistance	3%	1
	Capacitance	3%	10
	Frequency	0.1%	2
Tektronix TDS 3012	Voltage	2%	
	Time	0.02%	

Table 13.1: Measurement accuracies of laboratory equipment.

The total accuracy of each device is determined by the combination of the percent error and offset. Percent error indicates gives the maximum amount that your measured value can deviate from the true value, as a percentage of the measured value. For example, a reading of $3.200\ \mathrm{k\Omega}$ on the Tektronix TX3 multimeter has a percent error of 0.1%. This means that due to this quantity, your measurement with error would be

$$3.200 \pm (0.001 \times 3.200)\ \mathrm{k\Omega} = 3.200 \pm 0.0032\ \mathrm{k\Omega} \simeq 3.200 \pm 0.003\ \mathrm{k\Omega}.$$

The offset column gives how maximum deviation of the least significant digit of your measurement. For example, a reading of $22.01\ \mathrm{mV}$ might be made with the TX3. It would have an offset of 2. This means the least significant digit (i.e., the right-most digit, $22.0\mathbf{1}\ \mathrm{mV}$) can be off by as much a 2. In other words, the offset error would be $22.01 \pm 0.02\ \mathrm{mV}$.

Total accuracy can then be found by combining the percent error and offset. They can be combined easily like so:

$$\text{total accuracy} = (\text{measurement}) \times (\text{percent error}) + \text{offset}.$$

As an example, say the TX3 measures $12.23\ \mathrm{k}\Omega$. Its total accuracy is

$$(12.23)(0.001) + 0.02 = 0.03223 \simeq 0.03.$$

So, the measurement should be reported as $12.23 \pm 0.03\ \mathrm{k}\Omega$.

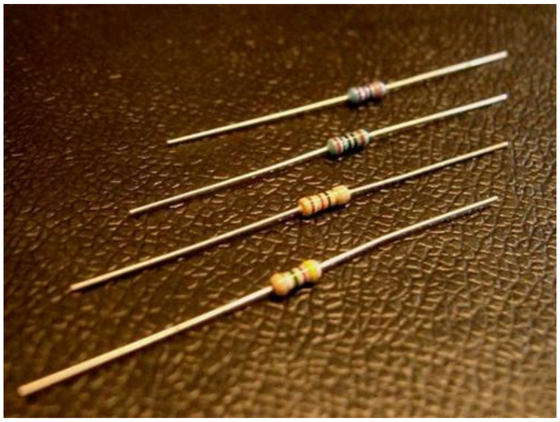
Note that we do not report error beyond the least significant digit. It doesn't make sense to report a measurement like $12.23 \pm 0.03223\ \mathrm{k}\Omega$ since our uncertainty portion has digits beyond the least significant digit of our measurement itself. So, for that reason we report, $12.23 \pm 0.03\ \mathrm{k}\Omega$ instead.

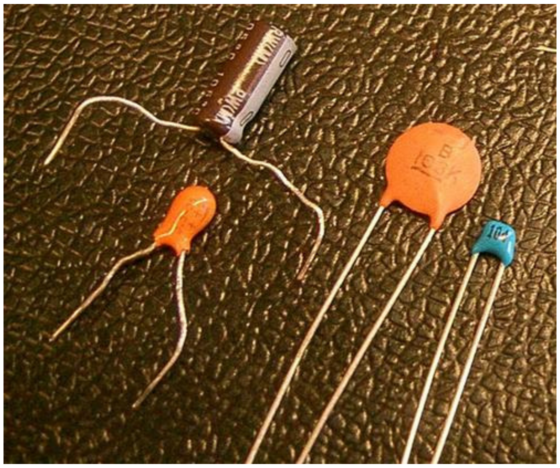
Section 14

Laboratory Components

COMPONENTS TO RECOGNIZE The labs will introduce you to components for constructing circuits. The following is a summary of the components and the lab, and what they look like. Familiarizing yourself with this set could improve the efficiency of your laboratory execution.

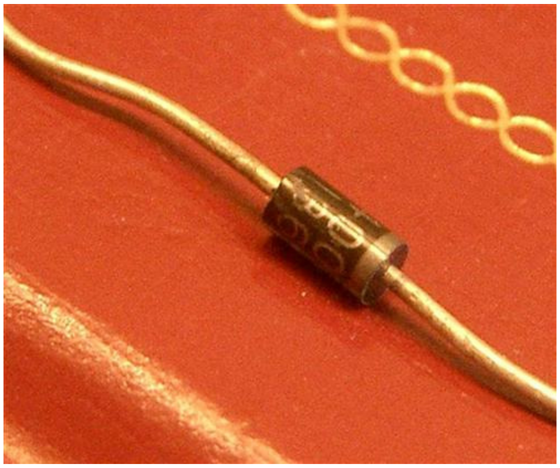
Resistors. Note the color bands on these. Some have 4 bands others have 5. These bands indicate the resistance and the tolerance. These resistors are able to dissipate 0.25 W. For high-power applications, more robust resistors are required.





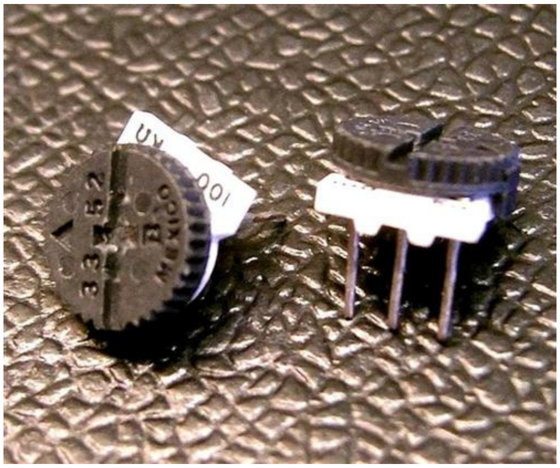
Capacitors. Some types (like the one on the upper left) indicate which side is the negative terminal. Hooking this type up incorrectly can ruin your circuit.

Diodes. The diode permits current to exit through the silver-banded end only.



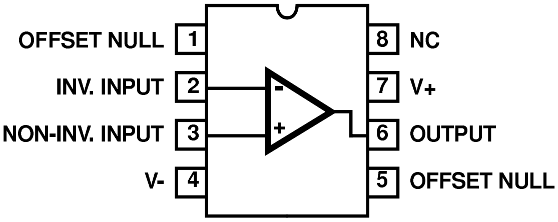
BNC cables. These cables are used for connecting devices such as oscilloscopes and function generators.

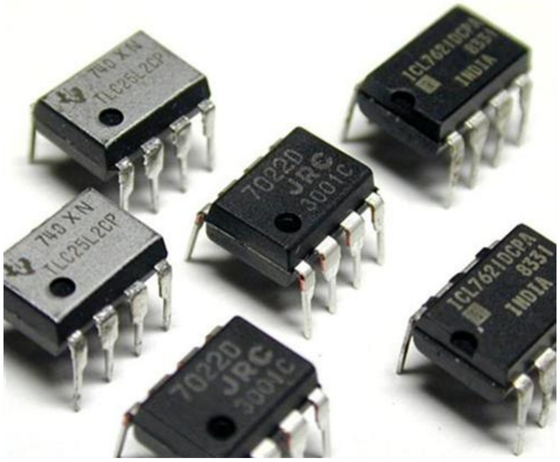
Banana cables. These cables are used for connecting to devices with the appropriate plugs. (For example, multimeters.)

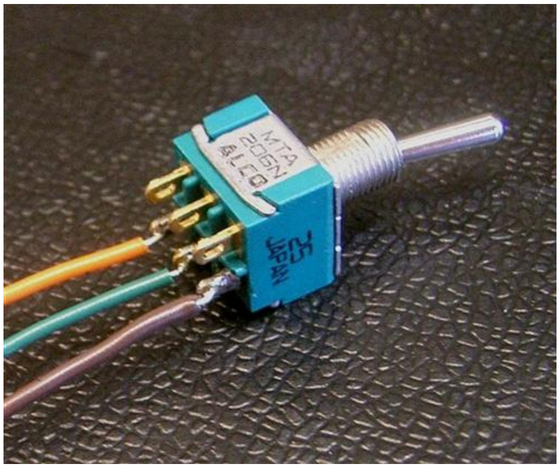


Potentiometers. A potentiometer ("pot") has a fixed resistance between the left and right pins. The center pin is connected to the wheel such that the resistance between left and center pin can be adjusted from zero to the maximum. The resistance between the center and right pins is simply the remaining portion of the resistance.

Op-Amps. Typical op-amps chips, and the pin diagram for the LM741C.





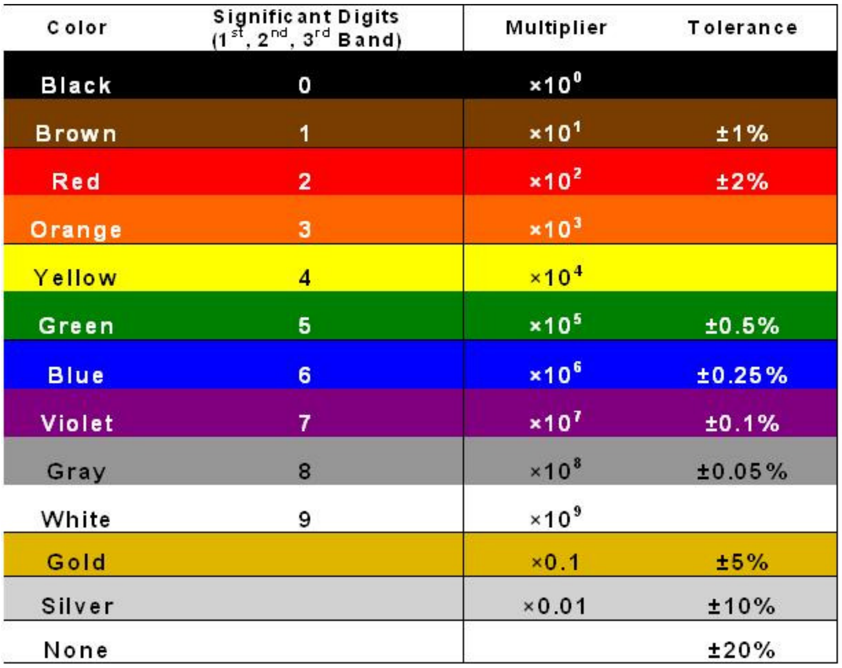


Toggle switch. The toggle switch connects the green wire in the middle to the orange wire on the left (when toggled to the right) or to the brown wire on the right (when toggled to the left).

Section 15

How to Read Resistor Color Codes

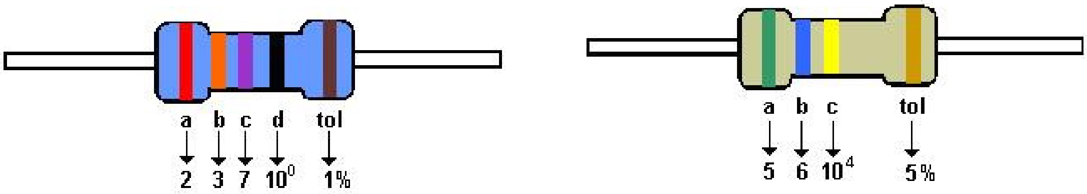
The most common electrical component found in almost every electrical circuit is the resistor. A resistor's value and tolerance are usually coded with colored bands on the resistor body. The type we will use in the lab is the -watt axial-lead resistor. Your "garden variety" 5% general purpose types will have a four-band resistance code. Resistors with a five-band resistance code usually have higher precision (0.1%, 0.25%, 0.5%, 1%, or 2%). The colors used for the bands are listed with their respective values in the color code chart in Table15.1.



Color	Significant Digits (1st, 2nd, 3rd Band)	Multiplier	Tolerance
Black	0	$\times 10^0$	
Brown	1	$\times 10^1$	±1%
Red	2	$\times 10^2$	±2%
Orange	3	$\times 10^3$	
Yellow	4	$\times 10^4$	
Green	5	$\times 10^5$	±0.5%
Blue	6	$\times 10^6$	±0.25%
Violet	7	$\times 10^7$	±0.1%
Gray	8	$\times 10^8$	±0.05%
White	9	$\times 10^9$	
Gold		×0.1	±5%
Silver		×0.01	±10%
None			±20%

Table 15.1: Resistor color code.

The tolerance band is the last band on the resistor and is usually spaced a little further away from the other bands. (Admittedly, sometimes it's difficult to determine which end the tolerance band is on.) To determine the value of a resistor,



(a) Five-band resistor.

(b) Four-band resistor.

Figure 15.1: Five-band and four-band resistors.

first identify the tolerance band and hold the resistor such that the tolerance band is to the right. Then, read the color bands from left to right. The first two bands of the four-band convention and the first three bands of the five-band convention are the significant digits of the resistor value. The second to last band indicates the multiplier.

For the five-band resistor shown in Figure 15.1(a), the resistor's value and tolerance are expressed as

$$\begin{aligned} R &= \mathbf{abc} \times \mathbf{d}\ \Omega \pm \mathrm{tol}\% \\ &= 237 \times 10^0\ \Omega \pm 1\% \\ &= 237\ \Omega \pm 1\%. \end{aligned}$$

A four-band resistor is read in the same way, except that there are only two significant digits instead of three. Thus, the value of the resistor in Figure 15.1(b) is

$$\begin{aligned} R &= \mathbf{ab} \times \mathbf{c}\ \Omega \pm \mathrm{tol}\% \\ &= 56 \times 10^4\ \Omega \pm 5\% \\ &= 560\ \mathrm{k}\Omega \pm 5\%. \end{aligned}$$

Section 16

How to Read Capacitor Values

Reading capacitor values is a little more cryptic than reading resistor values as there are a few different conventions for marking capacitor values. Here, we will explain the most commonly seen marking schemes.

On large capacitors, the value is printed plainly on them, such as $10\ \mu\text{F}$ (ten microfarads). Smaller capacitors normally use a two- or three-digit code. In either case, the unit is the picofarad (pF), or $10^{-12}\ \text{F}$. In the two-digit marking scheme, the two digits simply indicate the capacitor value to two significant digits. So, a capacitor marked '47' indicates a value of $47\ \text{pF}$. The three-digit convention is somewhat similar to the resistor coding scheme. The first two numbers are the first and second significant digits and the third is a multiplier code. Generally, the third digit tells you how many zeros to write after the first two digits but there are a few exceptions. Table 16.0(a) specifies the multiplier corresponding to each number. For example, a capacitor marked 104 has a value of $10 \times 10,000\ \text{pF}$ or $100,000\ \text{pF}$ (or $0.1\ \mu\text{F}$). Sometimes, following the number code is a capital letter

(a) Multiplier code for three-digit convention.

Third Digit	**Multiplier**
0	1
1	10
2	100
3	1,000
4	10,000
5	100,000
6	not used
7	not used
8	0.1
9	0.01

(b) Capacitor tolerance code.

Letter Symbol	**Tolerance of Capacitor**
D	$\pm 0.5\ \text{pF}$
F	$\pm 1\%$
G	$\pm 2\%$
H	$\pm 3\%$
J	$\pm 5\%$
K	$\pm 10\%$
M	$\pm 20\%$
P	$+100\%, -0\%$
Z	$+80\%, -20\%$

Table 16.1: Capacitor multiplier and tolerance codes.

that specifies the tolerance. The tolerance code is given in Table 16.0(b). So, a

103J capacitor has a value of $10,000$ pF with $\pm 5\%$ tolerance.

Section 17

Breadboarding Basics

INTRODUCTION

You could build a circuit by connecting resistors and other components with alligator clips that run to the power supply and meters. However, your circuit would become a precarious, tangled, and confusing mess of wires hanging in mid-air or lying on the bench. A more robust and orderly way to construct a circuit is to build it on a breadboard. The breadboard contains sockets for inserting components and wires. This allows you to build circuits without having to spend time soldering components together and, hence, also allows flexibility in changing and moving components and wire connections. There are some basic guidelines for building circuits on a breadboard and handling components. These rules were developed over time to help you troubleshoot your circuits quickly, you work faster and more efficiently, and avoid common problems and incorrect connections that may lead to equipment damage. As you build your circuit, aim for:

1. Neatness – This facilitates troubleshooting and reduces errors due to incorrect connections

2. Robustness – This reduces accidental disconnects.

BASIC RULES

The basic rules you should always be careful to follow are:

1. **Verify value of components (resistors, capacitors, etc.)**. They may have been placed in the wrong bins.

2. **Organize your components before building your circuit**. Create a sheet with labels for identifying and arranging components, as is shown in Figure 17.1. You will be able to build your circuit a lot faster if you have your components identified and laid out ahead of time.

3. **Build your circuit to resemble the circuit diagram**. This makes it a lot easier to trace through your circuit and troubleshoot it. Mark off the components on the diagram as you insert them into your circuit.

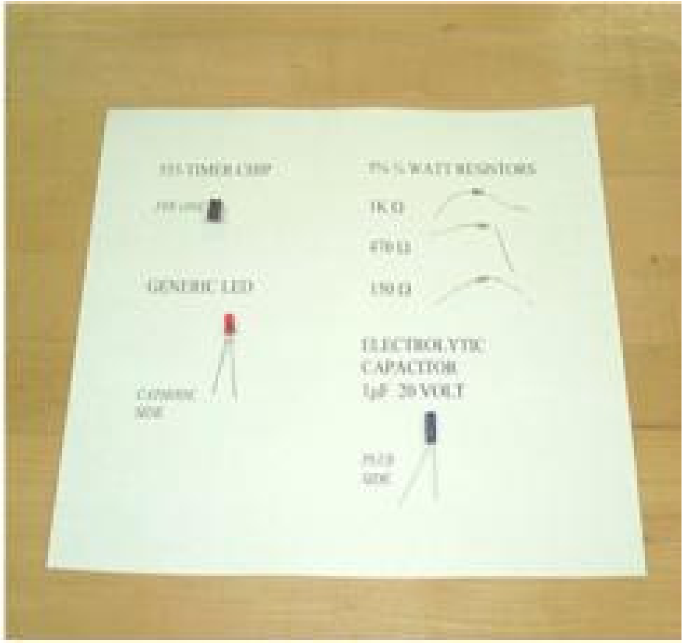


Figure 17.1: Organize your components.

4. **Keep your work neat**. You will often have to locate a particular point or component in the circuit, to make a measurement or try a component of a different value. It's much easier to do this if your work is neat and easy to trace. Neat circuits are also easier to troubleshoot.

5. **Use a color-coding scheme for wire connections**. Wires come in different colors and using a color-coding scheme helps you to easily trace through your circuit when troubleshooting. For the power supply to your circuit, always use the following color-coding scheme:

   RED = + V, positive power supply.

   BLACK = 0 V, ground.

   PURPLE = − V, negative power supply.

   Don't use these colors for anything else; and never, switch them! *Note: For all electronics courses in ME at UCSB, this wire color coding scheme will be strictly enforced! This helps your TA to help you troubleshoot. Your TA reserves the right NOT to help you troubleshoot your circuit if you do not follow this color scheme.* Beyond that, the choice is yours. For example, you could choose to identify all outputs with yellow wire. That way, you can find and check all of your output voltages with ease.

6. **Create power and ground busses at the top and bottom of your breadboard**. Frequently, you will need to provide power to more than one place in your circuit. Using the top and bottom rows of the breadboard as power

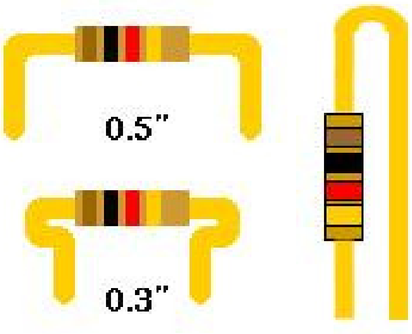


Figure 17.2: Bend resistor leads a small distance away from the resistor body.

busses makes your wiring neater and makes it easy to see power and ground connections. Note that the busses have a break in the very middle! If you want a power or ground bus to run the length of the breadboard, you must insert a jumper in the middle of the row to join the two halves together.

7. **Use "pigtails" at binding posts**. This provides a convenient place for clipping alligator clips, especially useful for grounding and power signals.

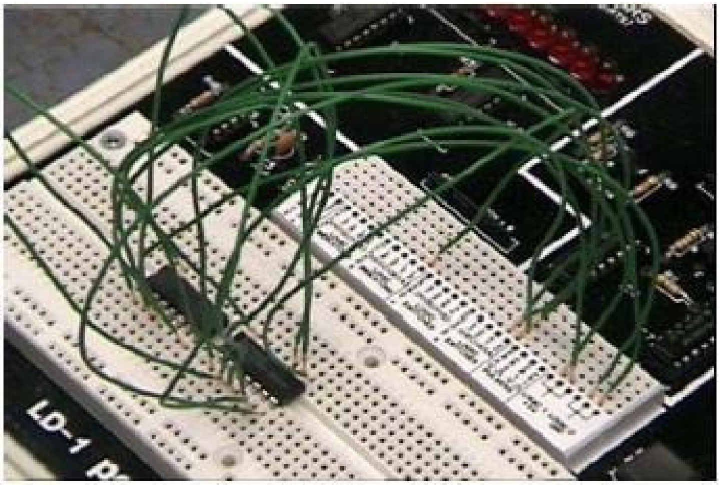
8. **Keep your component leads short**. Component leads are not insulated. If you keep them long, they'll hit one another and cause short circuits. Long leads also get messy and make it hard to check your circuit.

9. **Never bend a lead at the body of the component**. If you bend a component lead right at the component body, you risk damaging it or even tearing the lead off. Bend the leads a small distance away from the component body. Figure 17.2 shows examples of how resistor leads could be bent to fit snugly to the breadboard without damaging the component.

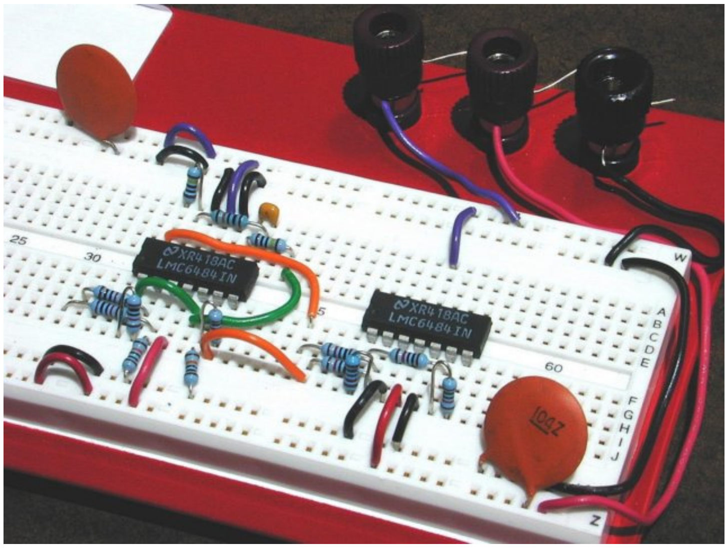
10. **Route wires around IC chips instead of over them**. Sometimes your IC chip may be bad and you'll need to remove it and insert another one.

11. **Never force large wires or components into the breadboard socket contacts**. Use only the wires provided in the lab, either AWG (American Wire Gauge), #22 or #24 solid hookup wire. Numbers higher than 24 indicate wire that may be too thin to provide reliable connections, while numbers lower than 22 indicate wire that is thick enough to damage the breadboard socket contacts. (Note: large AWG numbers indicate thinner wire while small AWG numbers indicate thicker wires). Figure 17.3 shows examples of good breadboard wiring and bad breadboard wiring. In Figure 17.3(a), there is no color-coding, the wires are long and messy, the wires cross over IC chips, and the power busses not used. On the other hand, in Figure 17.3(b), the wiring is color-coded, wires and components are snug to the breadboard,

wires are routed around IC chips, power busses are established, and pigtails have been used at binding posts.



(a) Poor breadboard wiring.



(b) Good breadboard wiring.

Figure 17.3: Good breadboarding versus bad breadboarding.

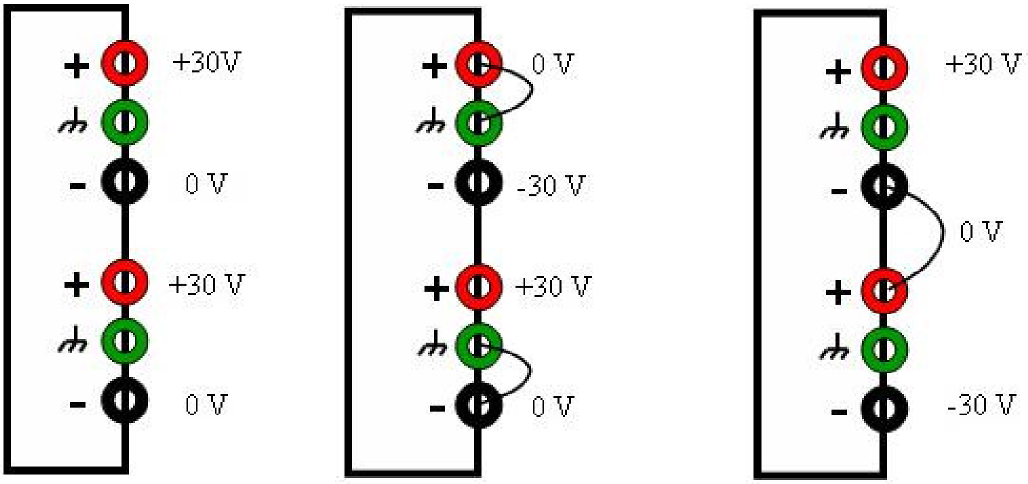
Section 18

Tektronix PS280 DC Power Supply

The two variable supplies on your PS280 DC power supply can be can be operated independently of each other, or the slave supply can track the master supply. The test mode is set using the TRACKING buttons in between the VOLTAGE/CURRENT control knobs. Following are explanations of the independent, series, and parallel tracking modes.

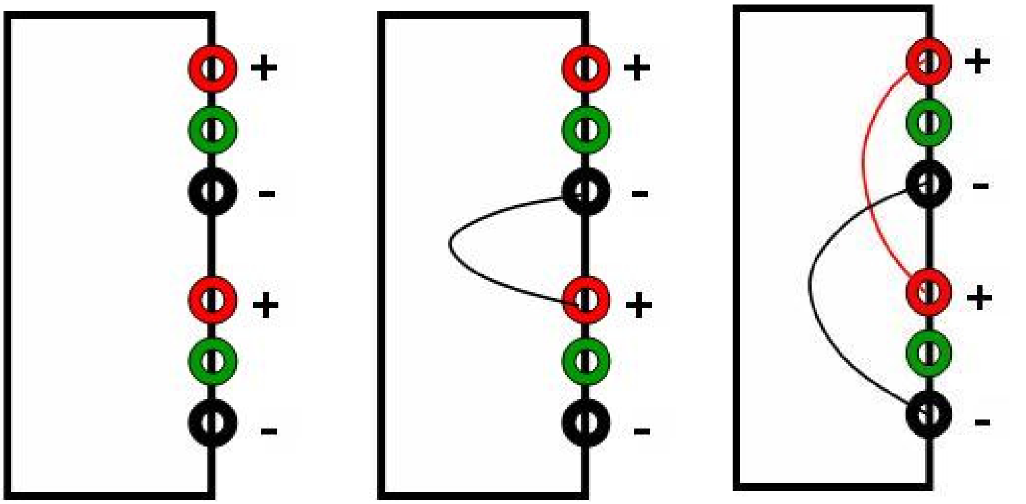
INDEPENDENT MODE

In independent mode, the variable output terminals are independent of each other and the variable supplies are independently controlled by the front panel VOLTAGE and CURRENT control knobs. There are no internal connections between the two variable supplies. Each variable supply can be set to deliver a different voltage and current. Using external connections, you can operate the power supply in three different independent modes. These are shown in Figure 18.1.



(a) Floating mode. (b) Ground-referenced mode. (c) Stacked mode.

Figure 18.1: Internal wiring for series and parallel tracking modes.



(a) Independent mode. (b) Series tracking mode. (c) Parallel tracking mode.

Figure 18.2: Internal wiring for series and parallel tracking modes.

Floating mode The power supply is not referenced with respect to earth ground (the green terminal labeled by the rake symbol). Each variable supply basically acts like a battery.

Ground-referenced mode One of the output terminals is grounded to the earth using the green terminal, providing a fixed reference point for your measurement. In this course, the green terminal and ground-referenced mode is NOT used.

Stacked mode The negative output terminal of one variable power supply is connected to the positive output terminal of the other. The stacked configuration allows you to test a circuit requiring a voltage between -30 V and $+30$ V at 0 to 2 A. A stacked configuration can be either floating or ground-referenced.

TRACKING MODE

There are two tracking modes in which you can operate the power supply; series and parallel.

Series: In series mode, the positive output terminal of the master variable power supply is internally connected to the negative output terminal of the slave power supply. This connection allows the power supply to produce a maximum voltage difference of 60 V (i.e., 0 to $+60$ V, 0 to -60 V, or $+30$ V to $+30$ V) at 0 to 2 A.

When you place the power supply in series mode, the output terminals are hooked together internally as shown in Figure 18.2(b). The voltage knob for the master variable power supply controls the voltage for both variable power supplies. Using the master voltage control, the slave supply voltage is automatically tracked to the same value as the master supply.

Parallel: In parallel tracking mode, the positive output terminals of both variable power supplies are internally connected, and the negative output terminals of both variable power supplies are internally connected. These connections allow the power supply to produce a maximum voltage difference of $30\ \mathrm{V}$ at 0 to $4\ \mathrm{A}$.

When you place the power supply in parallel mode, the output terminals are internally connected as shown in Figure 18.2(c).

Section 19

Oscilloscope Probes

SIGNAL SOURCE LOADING

Any external device, such as a probe, that's attached to your circuit appears as an additional *load* to your circuit. That is, it draws current from your signal. This resistive loading, or signal current draw, changes the operation of your circuitry. To illustrate this, consider the measurement of a simple DC voltage divider circuit shown in Figure 19.1. Before a probe is attached, the voltage is divided across the resistors. For the values given in the Figure 19.1, The output voltage V_0, is

$$V_0 = V\frac{R_2}{R_1 + R_2} = 15 \text{ V}.$$

Now, in Figure 19.2, a probe (load) has been attached to the circuit, placing the probe resistance R_P in parallel with R_2. If R_P is equal to R_2, half of the current will be diverted from R_2 and will be drawn through the probe. Now, what is the output voltage V_0? First, we determine the equivalent resistance of R_P and R_2 in parallel, which we'll call $R_{2\|P}$. We have $R_{2\|P} = (10 \times 10)/(10 + 10)\text{k}\Omega = 5 \text{ k}\Omega$. Thus, using the same calculation as above, we see that the output voltage is $V_0 = 10$ V.

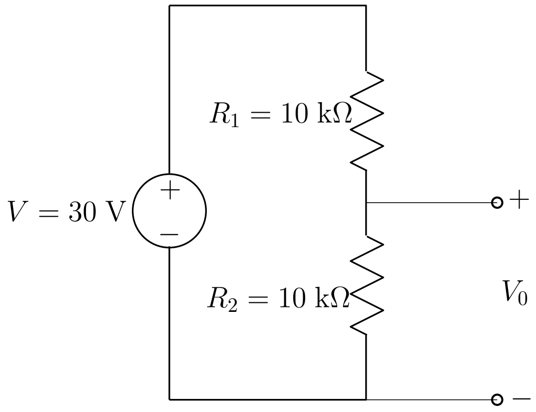


Figure 19.1: DC voltage divider.

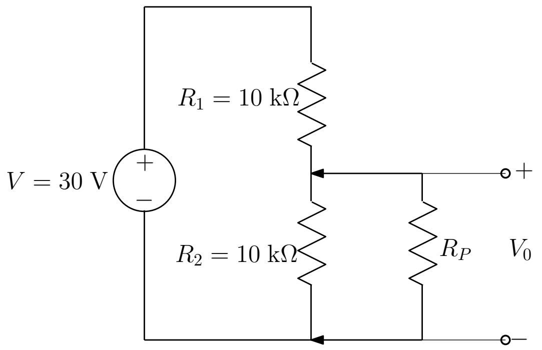


Figure 19.2: DC voltage divider with probe load added in parallel.

Thus, the presence of the probe changed the measured voltage from $15\ \mathrm{V}$ to $10\ \mathrm{V}$. This example illustrates how the testing equipment, the probe in this case, can affect the measurement.

Looking at Figure 19.2, it can be seen that if R_P is much larger than R_2, the probe will draw far less current than the resistor R_2. This will minimize the probes effect on the circuit. So, the trick in minimizing resistive loading is to ensure that the probe resistance R_P is much greater than R_2. An ideal probe causes zero signal source loading. In other words, it doesn't draw any current from the circuit. This means that, for zero current draw, the probe must have infinite resistance (or, in more general terms, infinite impedance), essentially presenting an open circuit to the test point.

ATTENUATING PROBES

In practice, a probe with zero signal source loading cannot be achieved. This is because a probe must draw some small amount of signal current in order to develop a signal voltage at the oscilloscope input. Consequently, some signal source loading is to be expected when using a probe. The goal, however, should always be to minimize the amount of loading through appropriate probe selection.

To minimize resistive loading, attenuating probes are used. Attenuating probes contain circuitry that reduces circuit loading. **Regular test leads and BNC cables do not contain this circuitry**. Attenuating probes are called as such because the circuitry that reduces circuit loading also attenuates (reduces the strength of) the signal. The attenuation factor indicates how much the signal is attenuated (reduced in strength.) For example, an attenuation factor of 10X reduces the signal amplitude by a factor of 10 and appears at the oscilloscope display 10 times less than it should be. The oscilloscope's probe setup can be configured to compensate for this and display the actual voltage.

In addition to minimizing resistive loading, attenuating probes also reduces capacitive loading and increases the range of the oscilloscope. However, we will not be concerned with these issues in this course.

Now that you understand the reason for using attenuating scope probes with oscilloscopes, we give you the rules of thumb for viewing waveforms on the oscilloscope:

1. When viewing measurements from a test circuit, such as your breadboard circuits, always use a scope probe with the oscilloscope, not BNC cables or test leads.
2. For most general purpose use, the 10X attenuation factor should be used. The 1X probe should be used when measuring voltages less than $10\ \mathrm{mV}$ peak-to-peak.
3. However, when viewing signals from the function generator on the oscilloscope, BNC cables can be used to connect the function generator to the oscilloscope because the output impedance of the function generator is very low compared to the impedance of the oscilloscope.
4. Scope probes should never be used to connect the function generator to your test circuit as it will attenuate the signal from the function generator.

PROBE SETUP

In the lab, several different probes are available. Scope probes made by Tektronix have a readout pin that tells the oscilloscope the attenuation factor of the probe. So, the attenuation factor is automatically set. Some Tektronix probes have a slide switch on the probe handle that allows you to choose the attenuation factor (1X or 10X). Unless you measuring voltages less than 10 mV, choose the 10X setting.

Probes made by other manufacturers do not have the readout pin and the probe setting must be done manually. First, choose the attenuation factor of the probe using the slide switch on the probe. Then, manually set the attenuation factor on the oscilloscope. To do this:

1. Press the button corresponding to the channel that the probe is connected to, Ch1 or Ch2.
2. Press the MENU button (next to the white REF button) in the panel labeled VERTICAL.
3. Press the button along the bottom of the display screen beneath Probe Setup to select Probe Setup.
4. Set the probe attenuation factor by using the knob in the top left corner, above the VERTICAL panel.
5. Press the MENU OFF button.

To verify that the attenuation factor is correctly set, use the probe to measure a known voltage, such as the internal 5V square wave generated by the oscilloscope. You can access this square wave by connecting the probe to the brackets to the right of the Ch2 jack. The brackets are labeled PROBE COMP.