

# Chapter 2

## Lab 1: Laboratory Equipment and Basic Resistive Devices

### Objectives:

- Become familiar with lab equipment: trainer, power supply, multimeters, and oscilloscope
- Become familiar with lab software: LabVIEW and Orcad
- Understand Series and Parallel Circuits
- Explore the uses of a resistor
  - current to voltage converter
  - voltage to current converter
  - current limiter
  - voltage divider
- Understand Input and Output Impedance
- Understand Potentiometers and their use as variable resistors and voltage dividers

## 2.1 Introduction

This laboratory begins by quickly leading you through the basics of the equipment that you will be using in the electronics lab. Then, basic concepts of series and parallel circuits are reviewed before proceeding to detailed investigations of voltage dividers, data acquisition with LabVIEW, and potentiometers. It is assumed that you have had an introduction to basic circuit analysis (ie. you are familiar with the concepts of current, voltage, resistance, Ohm's law, and series and parallel circuits).

At certain points in the lab, you will be required to demonstrate the successful completion of an activity to an instructor. **These points will be indicated by the use of this font.** Upon completion of this lab, you will be required to write a technical reference sheet for the potentiometer. More information on this is given at the end of this chapter and in Chapter 1. The technical report will be due at the beginning of your next lab period. If you are unable to finish all of the activities before the end of the lab, you can come back and finish in your spare time, as long as another lab is not being conducted in J-209 at that time.

## 2.2 Suggested Reading and Reference

### Horowitz and Hill

- 1.01 Voltage and Current
- 1.02 Relationship between voltage and current: resistors
- 1.03 Voltage Dividers
- 1.04 Voltage and Current Sources
- 1.05 Thevenin's Equivalent Circuit
- 1.10 Logic Levels
- 1.34 Variable Components (resistors)
- Appendix E: How to Draw Schematic Diagrams

### Hayes and Horowitz

- Chapter 1: Overview
  - Class 1: DC Circuits
  - Worked Examples: Resistors & Instruments
  - Lab 1: DC Circuits

## 2.3 Laboratory Activities

### 2.3.1 Introduction to laboratory tools and instruments.

We will start by introducing some basic functions of the main tools we'll be using in these labs. However, these instruments have so many functions that many of them will be introduced as we work through the labs. Because the spirit of this course is to provide practical experience with electronics and data acquisition, approximate monetary values are presented for the equipment introduced in this section.

**1. The Trainer Board.** Figure 2.3.1 shows the trainer (cost = \$360) on which you'll be constructing most of your circuits. The items you'll become most familiar with are the pin board (10), and the FUNCTION GENERATOR (5 THROUGH 8). The function generator can produce three waveforms: sinusoid, triangle, and square. Frequency is specified by first selecting a range (7), then finely adjusting the frequency with knob (5). The amplitude of the signal is adjusted using (6). The signal is a varying voltage difference between the OUTPUT and GND terminals. We will use the function generator and pin board in subsequent activities.

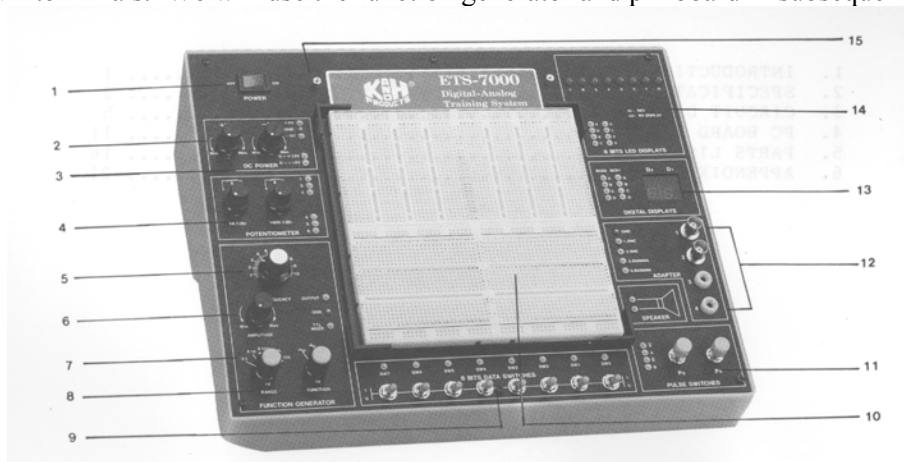


Figure 2.3.1: ETS-7000 Training and Prototyping Board (the *trainer*).

**2. The Power Supply.** In the MAE 221 laboratory you will be using the Goodwill Model GPC-1850D (cost = \$500) power supply extensively. We will briefly introduce some of the operations that you will be conducting in this lab session.

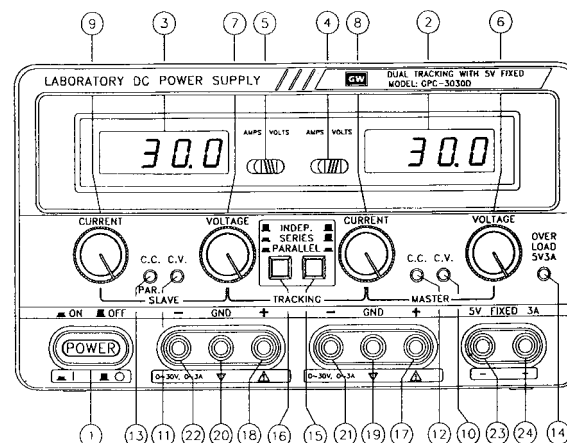


Figure 2.3.2: GPC-1850D dual DC power supply.

The GPC-1850D, shown in Figure 2.3.2, consists of two identical, independently adjustable DC power supplies. The left and right sides of the panel correspond roughly to each independent power supply, which include digital

displays (2 and 3). Switches 4 and 5 allow the displays to show either the current or voltage produced. Each power supply can operate in the constant current (C.C.) or constant voltage (C.V.) mode, as indicated by indicators 10 to 13.

In **constant voltage** mode (the normal mode of operation), the supply will produce the voltage specified by adjustment of the Voltage knob (6 or 7). To set either power supply to constant voltage mode, turn the Current knob (8 or 9) to its maximum (clockwise) position.

The **constant current** mode will not be used in this lab session but is discussed here for completeness. In constant current mode the current is specified by adjustment of the Current knob. To use constant current mode, the voltage knob should be turned fully clockwise. Think of the Current knob as *limiting* rather than *specifying* the current. If the attached circuit is such that the desired current cannot be achieved, the supply will default to constant voltage mode.

Setting the Tracking switches (15 and 16) specifies whether the two power supplies are operated independently, in series (for greater voltage), or in parallel (for greater current). For now, we will only use the power supplies independently.

**Power Supply exercise.** The purpose of this exercise is to become familiar with the power supply. We will simply set the power supply to produce an output of 6 volts.

- a) For now use only the right (master) side. Make sure that no wires are connected to the 8 terminals at the bottom of the front panel.
- b) With the power supply off, ensure that there is a *shunt* (a flat metal bar) connecting the negative (21) and ground (19) terminals. With the voltage knob (6), you can control the voltage **difference** between the positive and negative terminals. However, the reference from which these voltages are measured is arbitrary (we say that the power supply is *floating*). When the negative terminal is *shunted* to ground, it provides a reference. We can then say that the negative terminal is at 0 volts and the positive terminal is at the voltage specified on the voltage display, in reference to the negative terminal.
- c) Turn the voltage knob (6) fully counterclockwise to its minimum setting, then turn on the power supply.
- d) Make sure the tracking is in **independent** mode.
- e) Set the Current knob (8) to its maximum position – we will be operating in constant voltage mode.
- f) Ensure that the display switch (4) is set to Voltage, and adjust the voltage knob until the voltage display reads *approximately* 6 volts. Leave the power supply on for use in subsequent activities.

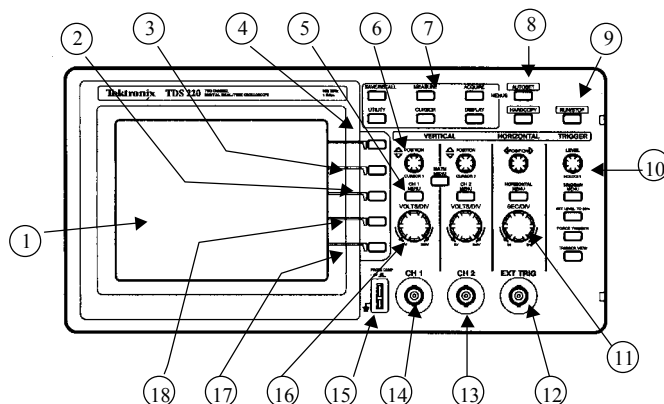
**3. The Hand-Held Multimeter (the Fluke).** The \$125 hand-held multimeter is a multi-function instrument for reading voltage, current, and resistance. Figure 2.3.3 contains a diagram of the front panel of the Fluke multimeter that you will be using in this laboratory.

#### **Hand-Held Multimeter exercises.**

**Voltage:** We will use the multimeter to measure the voltage level produced by the power supply. In order to read a DC voltage, you must turn the mode selector to the DC Voltage setting as shown in Figure 2.3.3. This multimeter is “auto ranging,” meaning that the internal circuitry determines and selects the range for your voltage input. Many other multimeters, however, require you to select the appropriate range. The Fluke multimeter can be put into manual ranging mode by pressing the yellow button on the mode selector dial. Ensure that the voltmeter leads are connected to the Voltage/Resistance (red) and Common (black) terminals. Touch the black lead to the negative terminal on the power supply, and the red lead to the positive terminal. The multimeter should display the same value as the power supply does (6V). Now reverse the leads (i.e. black on positive). Does the reading make sense? It should read negative 6V.



intimidating. However, this is one of the most useful tools in electronics and we suggest that you be persistent with this device, using it as often as possible because the hassles of thoroughly learning this tool now will pay dividends in the future. You can use your lab notebook to assist you in remembering buttons and sequences.



**Figure 2.3.5: Tektronix Oscilloscope.**

**Oscilloscope exercise.** Oscilloscopes measure voltage as a function of time, so a DC signal, which doesn't vary with time, will be displayed as a horizontal line. For this lesson we'll simply put a 5 VDC signal into channel 1. Set channel 1 VOLTS/DIV (16) to 2 volts. This setting will be displayed on the lower left of the LCD screen (1), and it means that the space between each horizontal line on the display represents an increment of two volts. Next set the power supply to 5 volts, and using a cable with a BNC connector on one side and two banana plugs on the other, connect the power supply to the Channel 1 input of the oscilloscope (14). You should see the horizontal line jump two and a half divisions. Turn the power supply voltage up and down and observe the moving line on the scope, and then set the power supply back to 5 volts, and change the VOLTS/DIV knob to 1 volts/div (observe the scope) and then 5 volts/div (observe the scope).

Next we'll put in two DC signals. Set the left side of the power supply to 3 VDC, and connect the left side of the power supply to the scope's Channel 2 (13). You should now have two DC signals displayed on the screen. Adjust the power supply voltages and the scope volt/div knobs and observe the display changes.

Select the SINE function on the Trainer's function generator and set the amplitude and frequency to the minimum values (turn the knobs counterclockwise). Now we will view the output of the function generator using the oscilloscope. To access the output of the function generator, use the wire strippers in your toolbox to cut two short (4-6") pieces of wire and remove about 1/4" of insulation from the ends. Insert one wire into the terminal labeled OUTPUT, and the other to the GND (ground) terminal. Plug an oscilloscope probe into Channel 1 on the oscilloscope and connect the probe clip to the function generator's OUTPUT wire and the reference clip (the wire on the side of the oscilloscope probe) to the GND wire. Now increase the amplitude of the sine wave to five volts, and increase the frequency knob until a sine wave appears on the scope. Continue to increase the frequency and adjust the scope so there are always two or three wavelengths showing on the scope. Stop at some point and determine the frequency by counting the time interval between wavelengths and inverting that time. Later we'll use the scope's math functions to do this (or those so motivated are welcome to attempt this now).

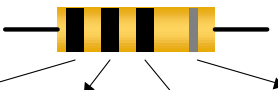
You should still have the DC signal on channel 2. Increase and decrease the power supply voltage and observe the the Channel 2 trace moving up and down with no effect on the sine wave.

**Because the oscilloscope is such an important and complicated tool, it is important that you have mastered the above exercise. Before continuing, show your sinusoidal and DC traces to an instructor.**

### 2.3.2 Basic Resistor Circuits.

We will now review some of the basic electronic circuits that most of you have had in Physics. We'll then use the tools of this chapter to supply voltages and currents to your circuits, and measure the outputs.

**Resistor color codes.** Figure 2.3.6 shows the resistance color code in  $\Omega$  (ohms). For resistances above  $1000\Omega$ , we state the value in kilo-ohms ( $k\Omega$ ). A common abbreviation is to simply use the letter k to designate  $k\Omega$  (ie. 2.1k).



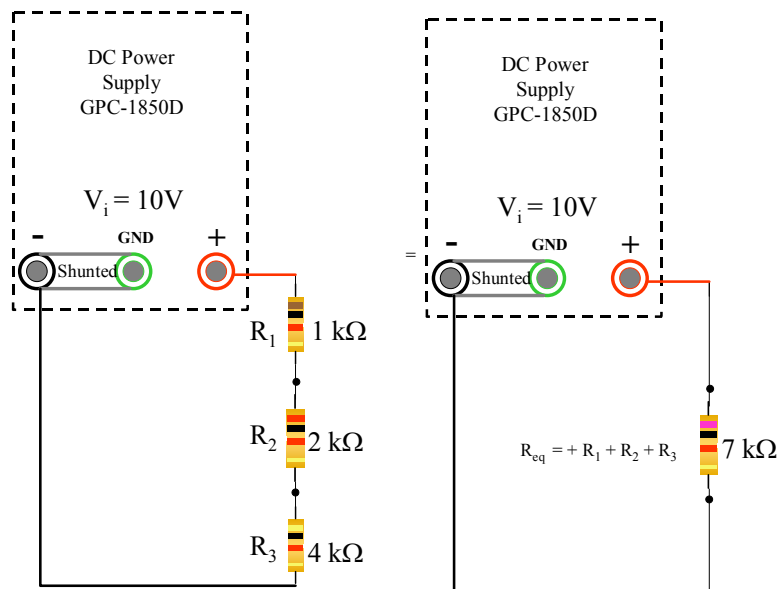
Black	0	0	x 1	Gold = +/- 5%
Brown	1	1	x 10	Silver = +/- 10%
Red	2	2	x 100	None = +/- 20%
Orange	3	3	x 1,000	
Yellow	4	4	x 10,000	
Green	5	5	x 100,000	
Blue	6	6	x 1,000,000	
Violet	7	7	x 10,000,000	
Gray	8	8	x 100,000,000	
White	9	9	-----	

**Figure 2.3.6: Resistor color codes.**

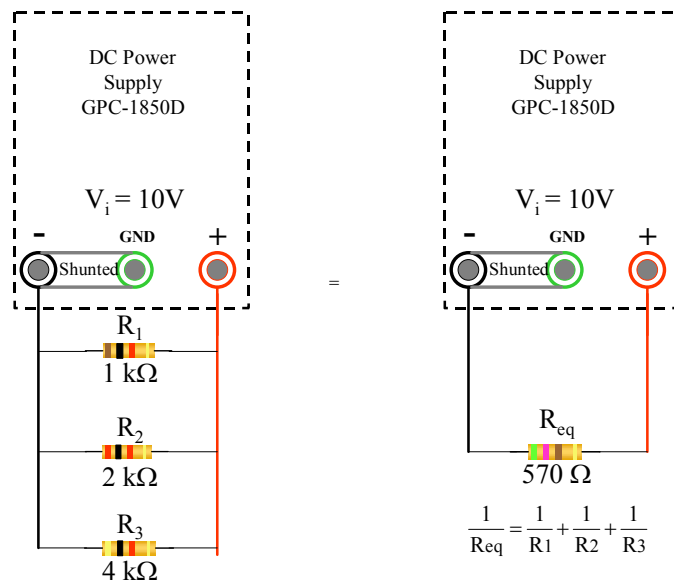
**Resistor basics:** Recall that resistances in series add directly, as in Equation (2.1) and Figure 2.3.7, and parallel resistances add inversely, as in Equation (2.2) and Figure 2.3.8.

$$R_{eq} = R_1 + R_2 + R_3 \quad (\text{resistors in series}) \quad (2.1)$$

$$\frac{1}{R_{eq}} = \frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} \quad (\text{resistors in parallel}) \quad (2.2)$$

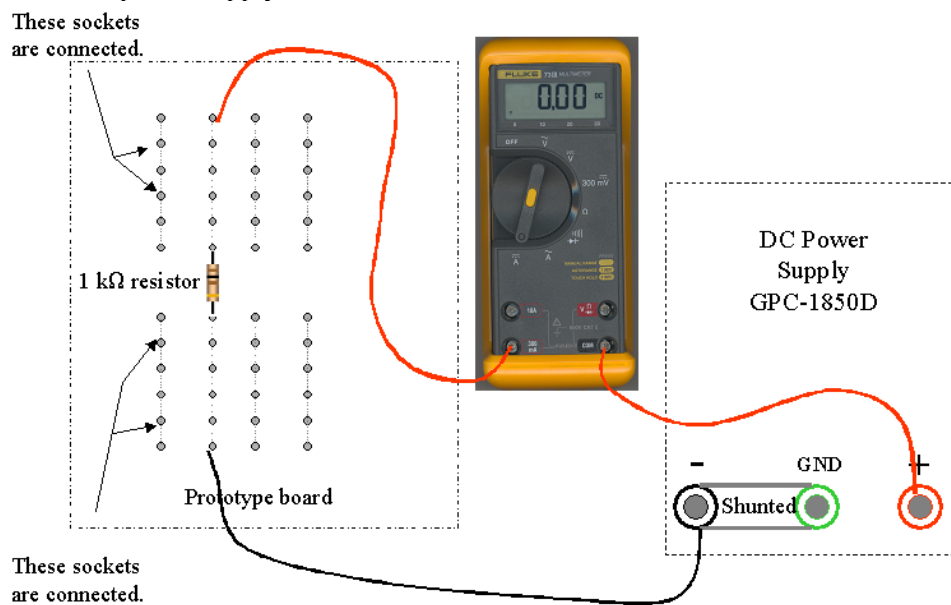


**Figure 2.3.7: Resistors in series.**



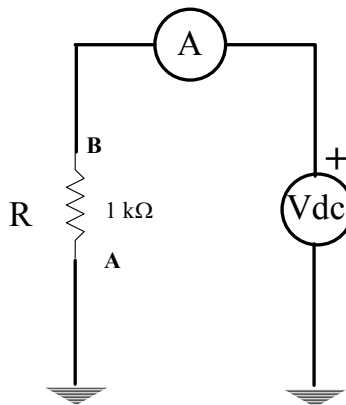
**Figure 2.3.8: Resistors in parallel.**

**Assembling a circuit.** The purpose of this exercise is to familiarize you with the handling of electronic components and construction of circuits on the trainer. Build the circuit shown in Figure 2.3.9 on your prototyping board, installing a 1k resistor onto the prototyping board, and connecting the Fluke between the prototyping board and the power supply as shown.



**Figure 2.3.9: Basic resistive circuit.**

The above sketch shows connections literally, which we hope is helpful in learning to use the equipment. However, this is not an efficient means of communication. Whenever possible, we will instead use schematic diagrams, such as the equivalent shown in Figure 2.3.10.



**Figure 2.3.10 Schematic of a basic resistive circuit.**

Before you connect the wires to the power supply and the multimeter, verify with the desktop multimeter that the resistance across the red and black wires is about  $1000\ \Omega$ . If you have made a mistake, the resistance that you read will probably either be very small (a short circuit) or effectively infinite (an open circuit). After verifying your connections, finish building the circuit. Make sure that the multimeter is configured to measure DC current (mA). Good engineering practice dictates that you verify that voltage and current knobs on the power supply are set fully counterclockwise (i.e. zero) before turning on the power. Turn the power supply on and adjust the voltage. The current displayed by the multimeter should change in proportion to the voltage for slow adjustment of the voltage. Keep this circuit assembled for use in the next exercise.

**A note on wire color coding:** Traditionally, red is used for the positive supply and black is used for negative. When you begin to build more complicated circuits that involve signals and multiple power supplies, you should introduce additional colors of wire for easy identification.

**Note these circuit assembly tips:**

- Currents in the pin board should not exceed 100 mA. High currents generate heat, causing damage.
- Although it is easier to construct a circuit using excessive wire lengths, wires should be kept as short as possible. This results in a neater circuit that is easier to understand and diagnose, and reduces noise in the circuit. In addition, short, well-placed wires are less likely to be accidentally yanked out of sockets.
- Always insert a separate wire into the board for testing with an oscilloscope or multimeter probe – never attach the probe to the lead of a component. The weight of the probe can pull out the component.
- If the end cover of the oscilloscope probe is removed, a pin is revealed. Do not insert this into the board. It will damage the probe.
- Leads and jumpers:
  - Make sure the bared end of the wire is straight before inserting into the pin board
  - Bare just enough wire to reach the bottom of the socket. There should be no bare wire visible when the wire is inserted into the socket

### 2.3.3 The Functions of a Resistor.

Resistors slow the flow of electrons, and as a result they have three basic functions:

- 1) Connecting a resistor to the positive and negative terminals of a power supply will result in a unique current flowing through the resistor, given by Ohm's law, so resistors **convert voltage to current**.
- 2) When current flows through a resistor there is a measurable voltage drop between the positive and negative sides so it can be used to **convert current to voltage**. The relationship between current and voltage is uniquely determined by Ohm's law. As we will see, this is the basis for an essential diagnostic tool in circuit analysis.



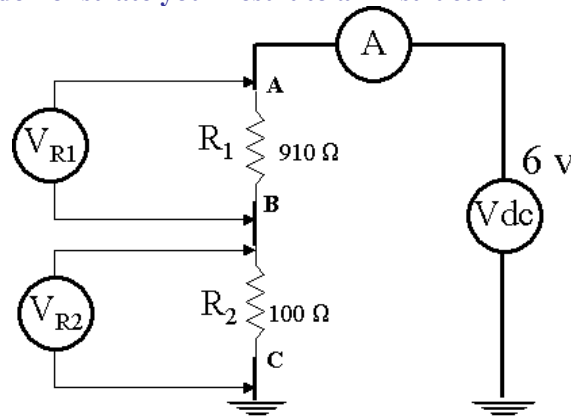
- 3) And because resistors slow the flow of electrons, in more complicated circuits resistors **limit the current through a circuit**. Because of this behavior, resistors are often used to *protect* other components in a circuit.

**The Resistor as a Voltage to Current Converter.** If a known voltage is applied across a given resistance, this will generate a unique current. Start by calculating the expected current across the 1k resistor of Figure 2.3.10 (with 6V applied), using Ohm's law:

$$V = RI \quad (2.3)$$

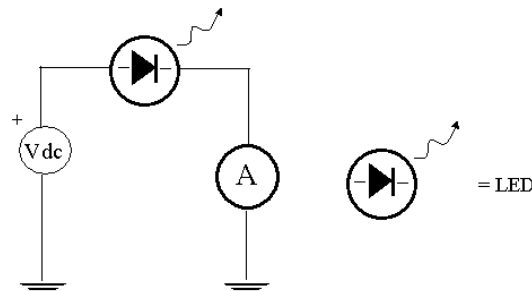
Turn on the power supply, and set it to 6V. Read the current on the Fluke and verify that this is approximately the same as your calculation (within 10%). If there is a large deviation, make sure you resolve this before proceeding. The current should read approximately 6 mA.

**The Resistor as a Current to Voltage Converter.** Replace the 1k resistor of your circuit with two resistors - a 910  $\Omega$  (900 $\Omega$  resistors are rare) and a 100  $\Omega$ , as shown in Figure 2.3.11, and then use the Keithley multimeter to measure the voltage across each resistor. Notice that the ammeter stills reads 6 mA, but there are two distinct voltages across  $R_1$  and  $R_2$  (5.4 V and 0.6 V, respectively), and both of these voltages can be used as signals to other devices. Use the measured voltages and the values of the resistors to calculate (in your notebook) the current flowing through each resistor. This should match the current displayed on the ammeter. Most data acquisition devices (oscilloscope, computer data acquisition cards) measure only voltage, not current. When using one of these devices, this is the only way that current can be measured. **This is an essential concept. Before continuing, be sure to demonstrate your result to an instructor.**



**Figure 2.3.11: Voltage measurement across resistors.**

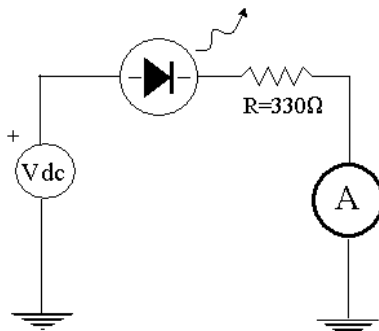
**The Resistor as a Current Limiter.** An important function of resistors is the protection of circuit elements that can be damaged by excess current. This is because resistors limit the current through a circuit for a given voltage, as imposed by Ohm's law. A classic use of this current-limiting effect is protecting an LED (light emitting diode). Using the smallest LED you can find, build the circuit in Figure 2.1.12, verifying that the power supply voltage knob is set to minimum.



**Figure 2.3.12: Unprotected LED circuit.**

Note that an LED is a true diode, and an aspect of all diodes is they conduct in only one direction. Current flows from long lead to short, but we all inevitably forget this, so if the LED doesn't work, just turn it around. When

connected, turn on the power supply and slowly increase the voltage. The LED will get very bright and change color, and will eventually burn out. In the next lab, we will see that the LED burns out because the resistance actually drops with increasing voltage, and therefore the current rapidly increases. The LED can typically only handle about 20 mA. Note the current at the time the LED burns out. Throw out the fried LED (DON'T PUT IT BACK IN THE CONTAINER!!).

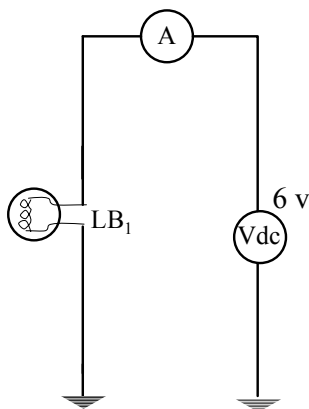


**Figure 2.3.13: Protect LED circuit.**

Now put a 330  $\Omega$  resistor in series with the led, as shown in Figure 2.3.13, and run the experiment again. Since the resistor and LED are in series, the same current passes through both. The resistor limits the current, and therefore the LED won't burn out. To determine the maximum current a particular component can withstand, either look at the spec sheet that came with the component, or the catalog through which the component is sold (like the Digikey or the Radio Shack catalog).

#### 2.3.4 Series and Parallel Circuits.

As series circuits have the same current through each component, **parallel circuits have the same voltage drop across each parallel component.** We'll motivate the purpose of the parallel circuit by first building another series circuit.



**Figure 2.3.14. Basic light bulb circuit.**

**Light bulb series circuits.** Build the light bulb circuit of Figure 2.3.14. Note the current and the bulb brightness and then, one at a time, wire in two more bulbs, as in Figure 2.3.15. You should have noticed that as each light bulb was added, the resistance of the circuit increases so the current must decrease, and therefore the bulbs become dimmer with each additional bulb. Also note that if one light bulb were to burn out, the circuit is broken and all of the bulbs will turn off.

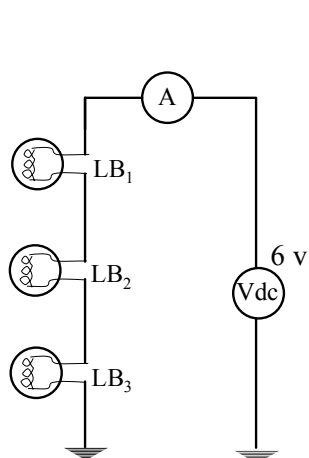


Figure 2.3.15. Series light bulb circuit.

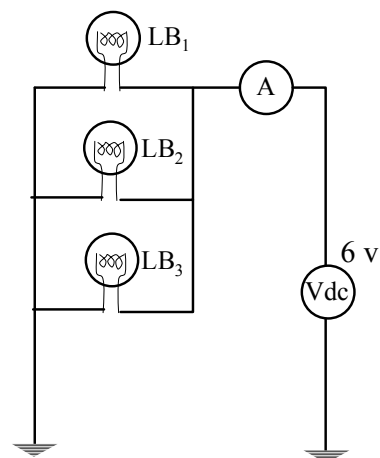


Figure 2.3.16. Light bulbs in parallel.

**Light bulb parallel circuits.** Now wire the bulbs in parallel, as in Figure 2.3.16, again installing each light bulb one at a time. Notice that the addition of each bulb has no effect on the brightness of the other bulbs. Also notice that as you add each bulb, you are decreasing the resistance in the circuit, and therefore increasing the sum of the currents through all of the branches. Historically, the first city electrical systems were simply lights wired in series. Thomas Edison quickly recognized this problem with series circuits, and designed the parallel circuit - an ingenious innovation for the time.

**Parallel exercise.** Replace the light bulbs with three resistors of your choice (greater than  $500\Omega$ ), as in Figure 2.3.17a, and then using equation 2.2, determine the equivalent resistance  $R_{eq}$ , and predict the current measured by the ammeter A of Figure 2.3.17b. Verify that theory and practice match, and record your results in your notebook.

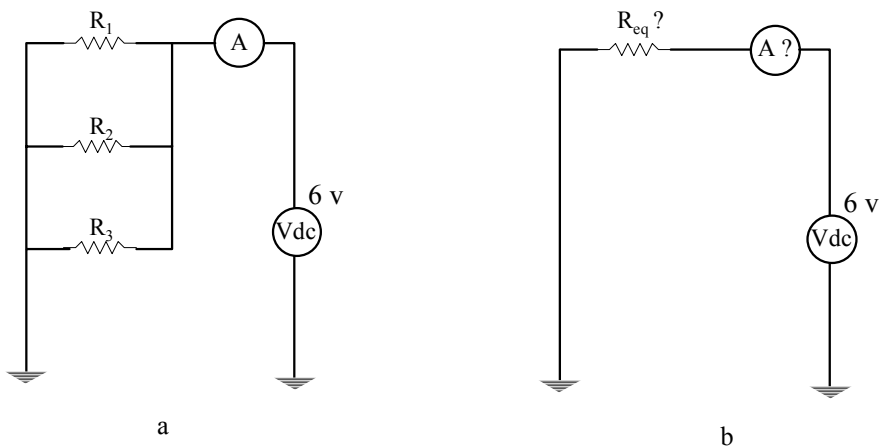


Figure 2.3.17. Parallel resistive circuit and resistive equivalent.

### 2.3.5 Voltage Division and Regulation.

In this exercise we will investigate two methods of reducing a voltage level: the voltage divider and the voltage regulator. You will see that the resistive series circuit that we built previously is simply a voltage divider so the concept will be somewhat familiar. On the other hand, the voltage regulator is entirely new. We will deal with each one briefly.

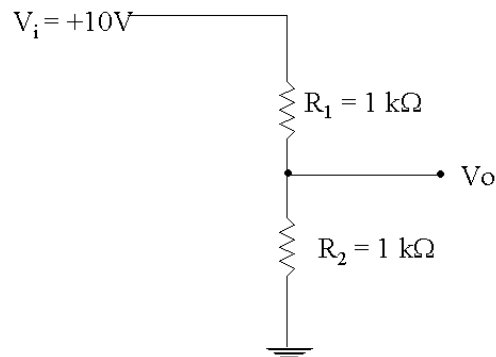
**Voltage dividers.** Figure 2.3.19 contains a voltage divider circuit. The transfer function for this circuit is:

$$\frac{V_o}{V_i} = \frac{R_2}{R_1 + R_2}. \quad (2.4)$$

An essential assumption in the derivation of this equation (which you can easily derive) is that the output ( $V_o$ ) draws no current. In reality, all loads draw some current. When designing circuits you must be aware of this current loading effect, which we will address in the next section.

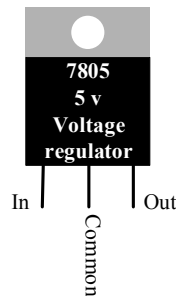
A typical voltage divider is shown in Figure 2.3.18. Construct this circuit and measure  $V_o/V_i$ . This should agree with Equation 2.4. Now, construct a voltage divider that divides by 6 by selecting the appropriate resistors.

**Demonstrate this to an instructor before continuing.**



**Figure 2.3.18: A Voltage Divider**

**Voltage Regulators.** With series resistive circuits we saw that we can get different voltages by selecting appropriate resistors. A problem with voltage dividers is that they generally cannot deliver sufficient current to power a device such as a motor. Voltage regulators have the advantage that they can deliver high current while maintaining a constant voltage. An example is the 7805, shown in Figure 2.3.19. These widely used integrated circuits (IC's) take in a range of DC voltages and put out a fixed DC voltage.



**Figure 2.3.19: Typical voltage regulator.**

For instance, the Motorola A7805 takes in from 7 VDC to 35 VDC (through the In and Common pins), and produces a steady 5 VDC (through the Out and Common pins). These chips can be purchased to handle currents that range from 5 mA to 1.5 A, and they have internal current-limiting and thermal shutdown features that essentially make them immune to overload.

The need for constant voltages is ubiquitous in electronics, and the 5 volt voltage regulator is king because all digital circuits are based on 5 volt signals. A digital port, for instance, has 8 lines (generally labeled 0-7) and each line is either 5 VDC or off, 1 or 0 in binary. This is a logic called TTL (Transistor-Transistor Logic), which you'll learn more about next semester.

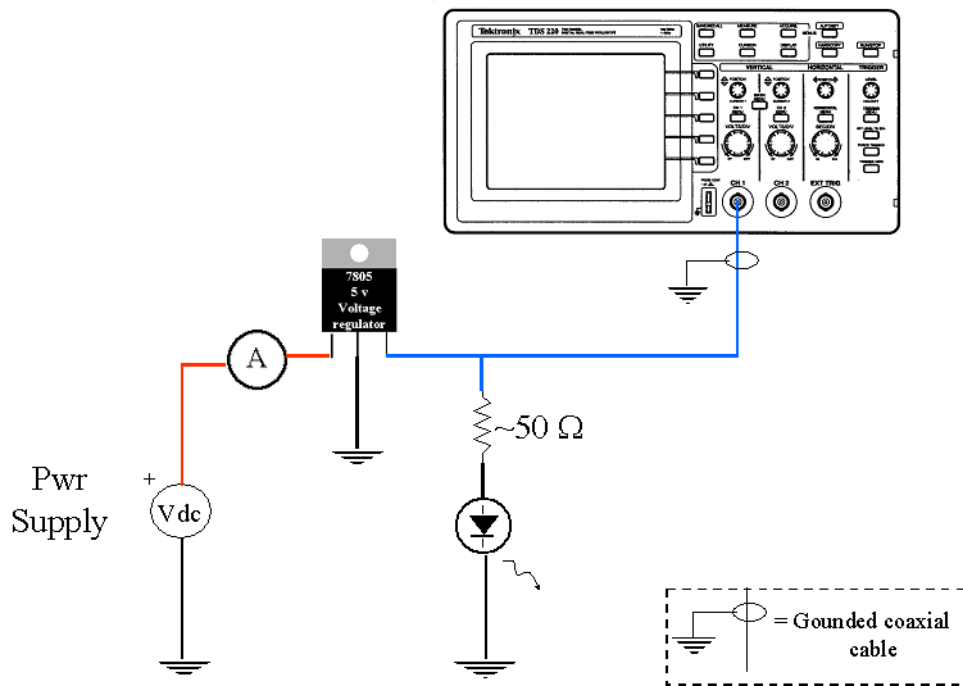


Figure 2.3.20: Voltage regulator circuit.

**Voltage Regulator exercise.** Build the circuit in Figure 2.3.20. This circuit protects the LED by limiting the voltage from the variable power source. Build this circuit and slowly increase the voltage up to the full capability of the power supply, while watching the oscilloscope. You should see that after the 7805 reaches biasing voltage ( $\sim 7$  VDC) the LED intensity stops increasing because the voltage to the LED is set at 5 VDC by the voltage regulator, and the current is limited by the resistor. **Demonstrate your working voltage regulator to an instructor before continuing.**

### 2.3.6 Voltage Dividers in Series: Input and Output Impedance.

Up to this point, we've been talking about resistance, but a broader concept of resistance is impedance, which we'll get into in later chapters. For now, however, we'll use the two terms interchangeably.

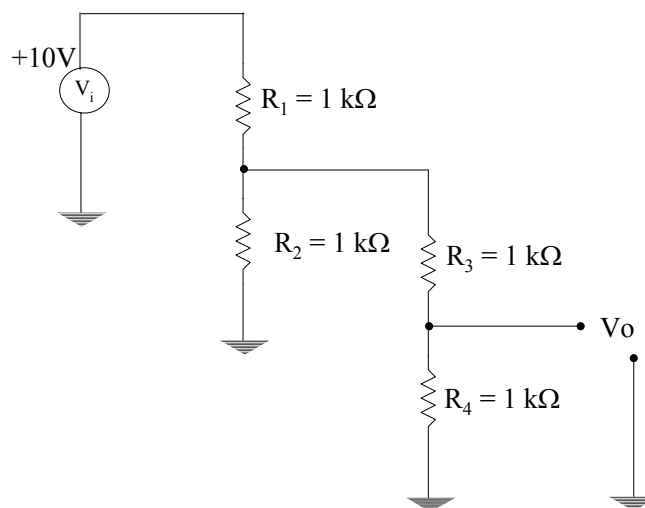


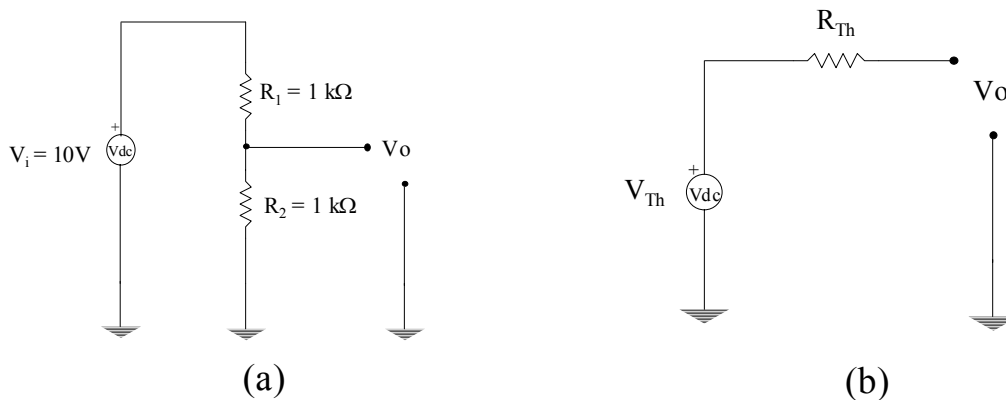
Figure 2.3.21: Two voltage dividers in series.

Analysis of voltage divider circuits is not always as straight forward as the series and parallel circuits we've shown. Figure 2.3.21 contains a schematic of two voltage dividers connected in series, and each one *divides by*

two, i.e.  $V_o/V_i = 0.5$ . Construct this circuit and measure  $V_o$ . You might expect the combined circuit to divide by 4, but it actually divides by about 5.

**Physical Analysis of the circuit.** Why does this circuit divide by 5? To begin to understand this circuit, we first note that our previous assumption of no load has been violated: we now have a load in parallel with  $R_2$ . Significant current is being drawn by the second voltage divider so we no longer have the same current passing through  $R_1$  and  $R_2$ . One of the voltage dividers is dividing by two and one is not. Use the Fluke multimeter to determine which is which.

**Mathematical Analysis of the Circuit.** By probing the physical circuit, it is possible to determine *where* the problem lies, but it does not provide much guidance for designing a circuit that will behave in the desired manner (i.e. to divide by 4). To better understand the voltage divider, we will construct a **Thevenin equivalent circuit**. This is a transformation of our circuit into a simpler one consisting of a voltage source and a resistor in series. Let's consider only the power supply and left voltage divider as shown in Figure 2.3.22.a. Horowitz and Hill state Thevenin's theorem as "*any two terminal network of resistors and voltage sources is equivalent to a single resistor  $R_{Th}$  in series with a single voltage source  $V_{Th}$* ". We want to transform this circuit to the equivalent but simpler circuit shown in Figure 2.3.22b.



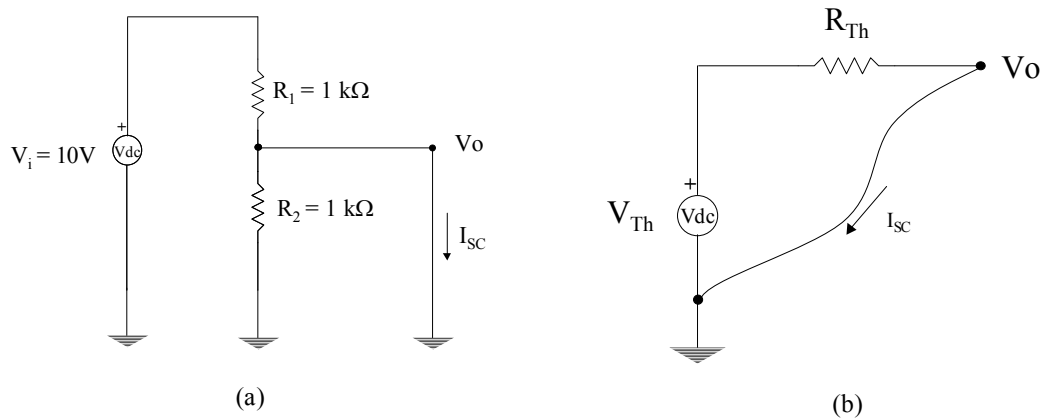
**Figure 2.3.22: Desired Thevenin equivalent of the double series circuit.**

We must therefore find  $V_{Th}$  and  $R_{Th}$ . To determine  $V_{Th}$  and  $R_{Th}$  we must consider that the above circuits are equivalent for **any** applied load. If no load is applied to the circuit (the circuit is *open*), we can see that in the Thevenin equivalent circuit (Figure 2.3.22b)  $V_o = V_{Th}$ . But the original circuit is just a voltage divider so from Figure 2.3.22a,  $V_{Th} = V_o = V_i(R_2/(R_1+R_2))$ , which is 5V.

When the two ports of the Thevenin circuit are connected together (a short circuit, as shown in Figure 2.3.23), the current passing through these nodes is  $I = V_{Th}/R_{Th}$ . So therefore  $R_{Th} = V_{Th}/I_{SC}$ , where  $I_{SC}$  is the *short circuit current*. By examination of Figure 2.3.23a,  $I_{SC} = V_i/R_1$ . Simple algebra then tells us that:

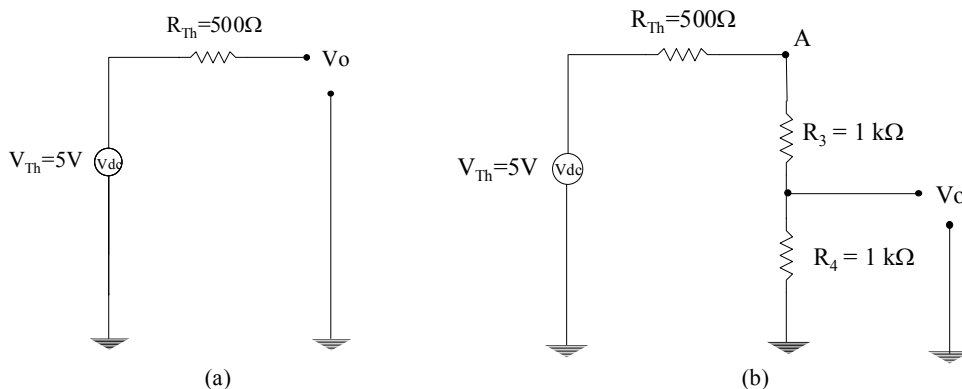
$$R_{Th} = \frac{R_1 R_2}{R_1 + R_2} \quad (2.5)$$

This is just the equivalent resistance of  $R_1$  and  $R_2$  in parallel, or in this case,  $500\Omega$ .



**Figure 2.3.23: Thevenin equivalent short circuit current  $I_{sc}$ .**

**Output Impedance.** The Thevenin equivalent circuit of the power supply with the left voltage divider is given in Figure 2.3.24 (a). Now we can see much more easily what effect the left voltage divider has if we represent it as a Thevenin equivalent circuit in series with the right voltage divider. This is shown in Figure 2.3.24 (b). Although our real power supply is providing 10V,  $V_{Th}$  is 5V (10/2). This would be the desired result if it were not for  $R_{Th}$  which creates an additional voltage drop proportional to the current through it. This is why  $V_o/V_i = 0.2$  rather than 0.25! This reduced model also helps us to redesign the circuit to achieve the desired behavior. We need to reduce  $R_{Th}$ . Equation (2.5) shows that we can do that by reducing  $R_1$  and  $R_2$ . However, the resistances must not be reduced too much because this raises the current. Change  $R_1$  and  $R_2$  to 510Ω resistors in Figure 2.3.22 and measure  $V_o$  again. Better, but we'd like to have it within 10% of our goal. In Figure 2.3.24,  $R_{Th}$  is the **output impedance** of the left voltage divider.



**Figure 2.3.24. Completed Thevenin equivalent circuits.**

**Input Impedance - Characterizing the Load.** We can also simplify the second voltage divider; however, this time the reduction is trivial. Since (this time) we can assume that there is no load on the output, we can just consider  $R_3$  and  $R_4$  to be in series so the equivalent resistance is  $R_{eq} = 2k$ . If a voltage  $V$  is applied to the input, a current  $I$  flows through the resistors. The **input impedance** is the impedance *seen* by a voltage source applied to the input, or  $V/I$ . In this case, the input impedance of the second voltage divider (or load) is  $R_{eq} = 2k$ .

Recalling that our means of achieving the desired performance is to minimize the voltage drop across  $R_{Th}$ , it is now evident that we must increase the input impedance of the right voltage divider. We do this by increasing  $R_3$  and  $R_4$ . Confirm this by replacing  $R_3$  and  $R_4$  with 10k resistors. We are now within 10% of our goal.

**Impedance Summary.** We've seen that a load placed on the output of a voltage divider changes the behavior of that voltage divider. This concept applies to **any** circuit (not just voltage dividers), and the effect of the load

depends on the output impedance of the supply circuit ( $Z_{os}$ ), and the input impedance of the load ( $Z_{il}$ ). For minimal effect, we require that  $Z_{il} \gg Z_{os}$ .

**Measurement Instruments revisited.** In our earlier experiments when we used the oscilloscope or multimeter to measure  $V_o$  on the single voltage divider, we assumed that the instrument did not draw any current, or that its input impedance is infinite. But this is not the case. Both the oscilloscope and multimeters have finite input impedances. The instrument will act like a resistor in a parallel circuit, and so for the instrument to have minimum effect on your circuit, the impedance is high – generally greater than  $15\text{ M}\Omega$ . This is important because if you are measuring the voltage across a high impedance component, your instrument can actually affect your results by loading the circuit.

Ammeters have just the opposite problem. These instruments are connected in line with your component, and act as a resistor in series. To minimize the voltage drop through the ammeter, its impedance must be low – generally less than  $10\ \Omega$ . But if you are measuring a low impedance component such as a cold light bulb, your instrument can again affect your results. The difference in impedance requirements for voltage and current measurement is the reason that we must use different terminals on the Fluke meter. The Keithley automatically adjusts impedance when you set the operating mode.

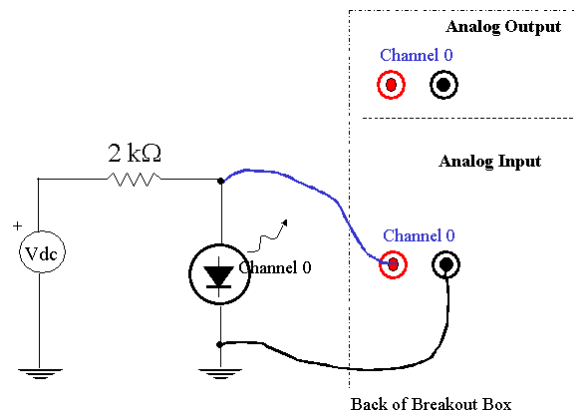
### 2.3.7 Introduction to LabVIEW.

LabVIEW is graphical programming language, and although it is capable of performing the same functions as C++, FORTRAN, BASIC, and other languages, its strength is data acquisition and manipulation. In this course, you will use and write programs written in LabVIEW to control processes, and acquire and present data.

In the following exercises, we will look at the LabVIEW interface and write some simple codes to read and write analog and digital signals.

#### Analog Input: Reading a signal with LabVIEW

Build the circuit in Figure 2.3.25. The *Breakout Box* will be sitting on your lab bench, and connects the data acquisition card in your computer to your circuits. Follow the instructions below to write a LabVIEW program which measures the voltage across the LED.



**Figure 2.3.25. Measuring voltage across an LED with a DAQ card.**

1. Go to the computer, and on the Desktop, double click the **LabVIEW** icon, and then when prompted, click **New VI** (VI stands for Virtual Instrument).
2. **Right** click on the gray screen, find **Numeric** on the pop-up **Controls** menu, and find and drag a **Digital Indicator** to the gray screen, as in Figure 2.3.26.



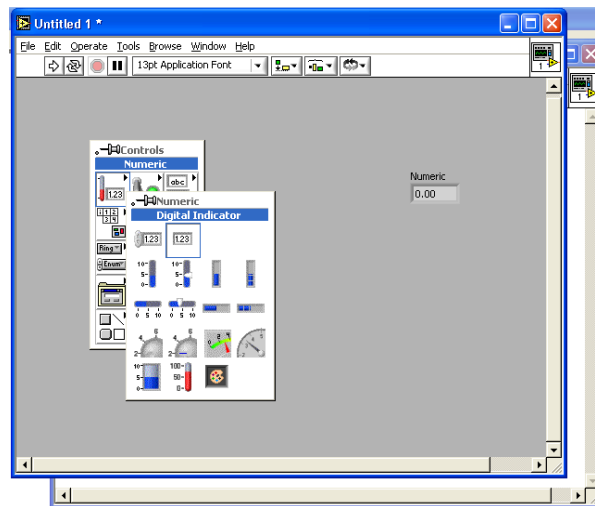


Figure 2.3.26. LabVIEW Front Panel.

3. On the gray **Front Panel**, select **Window**, then **Show Diagram** on the pull-down menu (or press Ctrl E to toggle between the Front Panel and Diagram windows). The Front Panel is the User Interface, whereas the diagram contains the source code. The **Diagram** panel should be white, and you should see the back of the **Numeric Indicator**. Right click on the white panel to pop up the **Functions** menu, and select **Data Acquisition** -> **Analog Input** -> **AI Sample Channel.vi**, as in Figure 2.3.27.

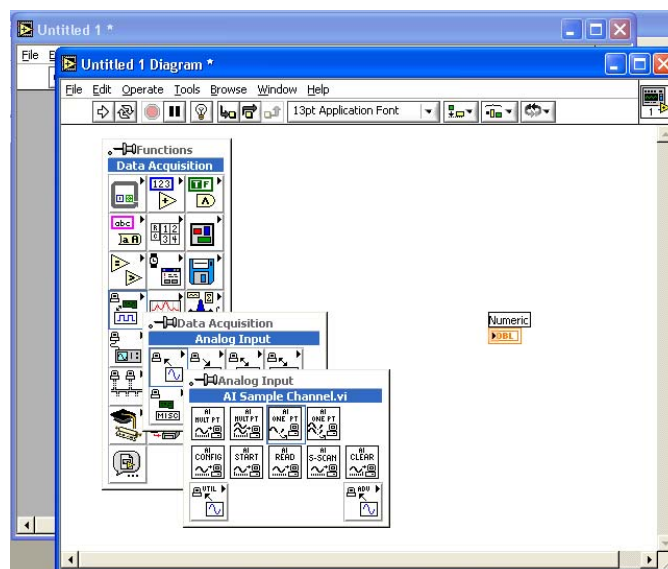


Figure 2.3.27. LabVIEW Diagram panel, and different menus.

4. Drag the **AI Sample Channel.vi** icon to the white **Diagram Panel**. This vi (virtual instrument) is a subroutine that takes in external voltages. Now with the cursor somewhere on the white Diagram panel, hit the Tab button until your cursor is a spool of wire (the “wiring tool”), and then move the wiring tool over the **AI Sample Channel.vi** until the **Sample** terminal flashes, then wire the **Sample** terminal to the **Numeric Indicator** (click on one, and then the other) as in Figure 2.3.28. If you make a mistake at any point, you can undo the action using Ctrl-Z.

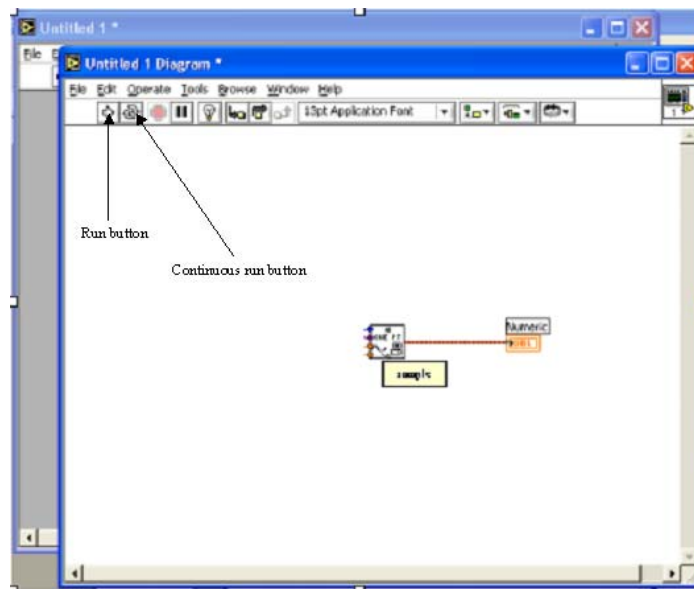


Figure 2.3.28. Wiring the Analog Input vi to the indicator.

5. Go back to the gray **Front Panel** by either clicking on the gray window, or selecting Show Panel on the Window menu (or pressing Ctrl E), and then press the **Continuous Run** button indicated in Figure 2.3.28. Turn on the power supply and turn the voltage knob up and down, and you should see the voltage on the **Numeric Indicator** change. If you have a broken **Run** button, or a dashed line where the wire should be, this probably means you didn't wire the items correctly.

Press the red **Stop** button to halt program execution.

#### Analog Output: Creating a signal with LabVIEW.

1. Turn off and disconnect the power supply from your circuit, and wire your circuit as shown in Figure 2.3.29. On your LabVIEW program **Front Panel** (the gray screen), go through the same steps you used to pull up the **Numeric Digital Indicator**, except this time pull up a **Numeric Digital Control**. Hit the tab key until the text editing cursor appears, and overwrite the words **Numeric** and **Numeric (2)** with **Voltage Across LED** and **Voltage Out**, respectively.
2. On the **Diagram Panel** go through the same steps as before, except this time pull up the **AO Update Channel.vi** by right clicking the background to pop up the **Functions** menu and selecting **Data Acquisition -> Analog Output -> AO Update Channel.vi**. This is a subroutine that sends a voltage out to the breakout box. Wire the **value** terminal of the vi to the **Voltage Out** control. The **Front** and **Diagram Panels** should look like Figure 2.3.30.

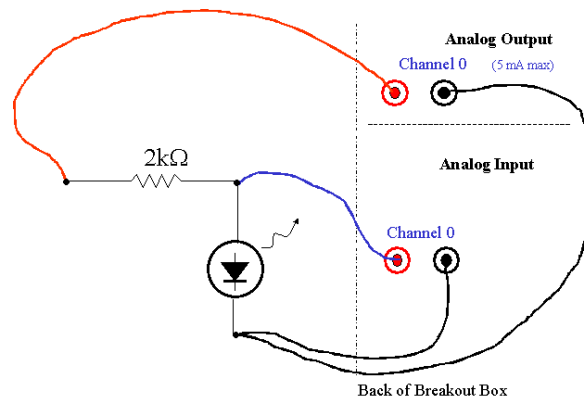


Figure 2.3.29. Circuit for Analog Input and Output.

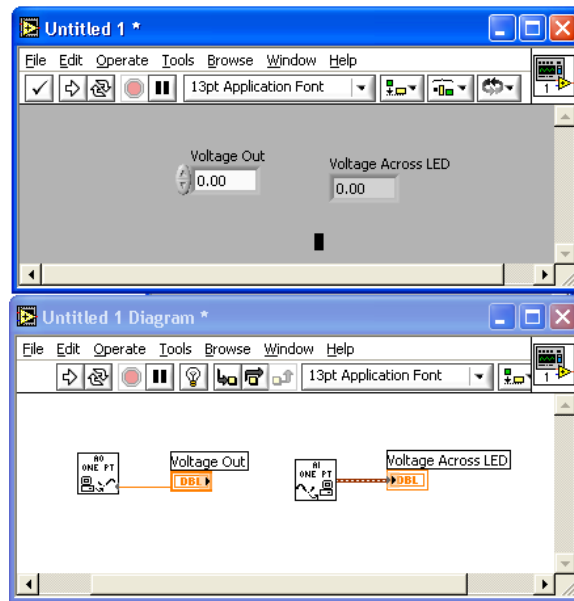
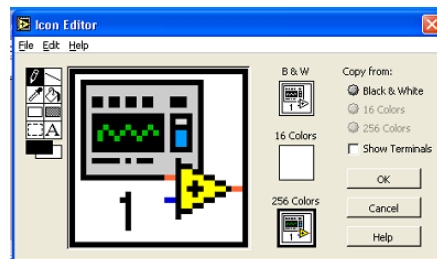


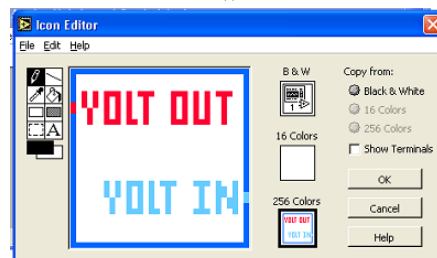
Figure 2.3.30. LabVIEW setup for voltage in and out

3. Return to the front panel and press the **Continuous Run** button, then increase the **Voltage Out** value using the up button on the **Voltage Out** control. If all is working well, you should see the **Voltage Across LED** value increase, and the LED change intensity. Before stopping the program, make sure the Voltage Out control is set to zero to depower the circuit. Now you basically know how to send out and take in voltages using LabVIEW. Save this program as **Analog Volt In and Out Lab 1**, using **File, Save As...**, and put it in your personal folder.

4. In future labs, we'll be using this program as a subroutine. To create this program as a subroutine, you'll have to "wire" it. On either the **Diagram Panel** or the **Front Panel**, double click the icon in the upper right corner. The icon editor will appear as in Figure 2.3.31a. Use the controls on the palette to change it to look like Figure 2.3.31b, and then hit **OK**.



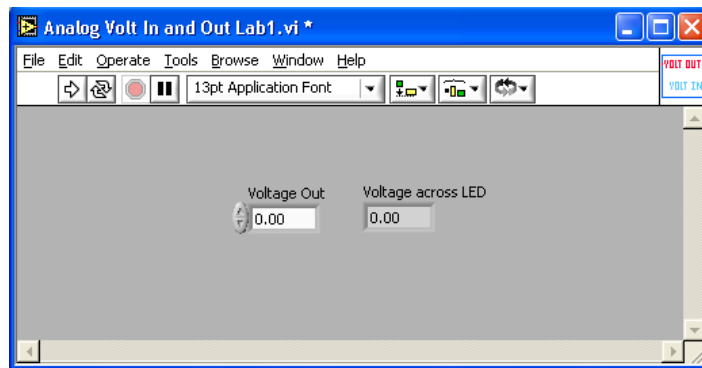
(a)



(b)

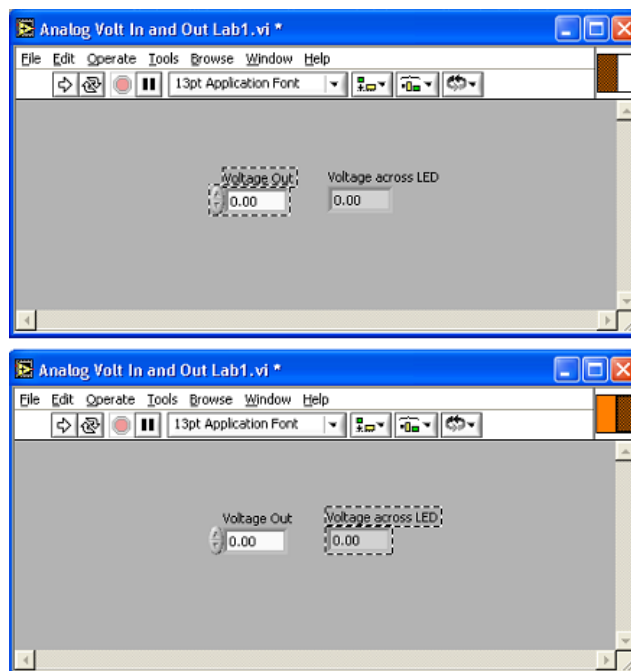
Figure 2.3.31. LabVIEW Icons.

The completed **Front Panel** should look like Figure 2.3.32. Save your program now.



**Figure 2.3.32. Completed LabVIEW Front Panel.**

Next you'll have to wire the icon, which becomes the actual subroutine. While in the **Front Panel**, right click on your new icon, and select **Show Connector** on the pop-up menu.



**Figure 2.3.32. Back of subroutine icon showing terminals.**

Using any cursor, wire the icon by clicking the left terminal and then the **Voltage Out** control. Click on the right terminal, and then the **Voltage across LED** indicator. Display the icon by right clicking the icon again and selecting **Show Icon**. Save your work again to update the changes.

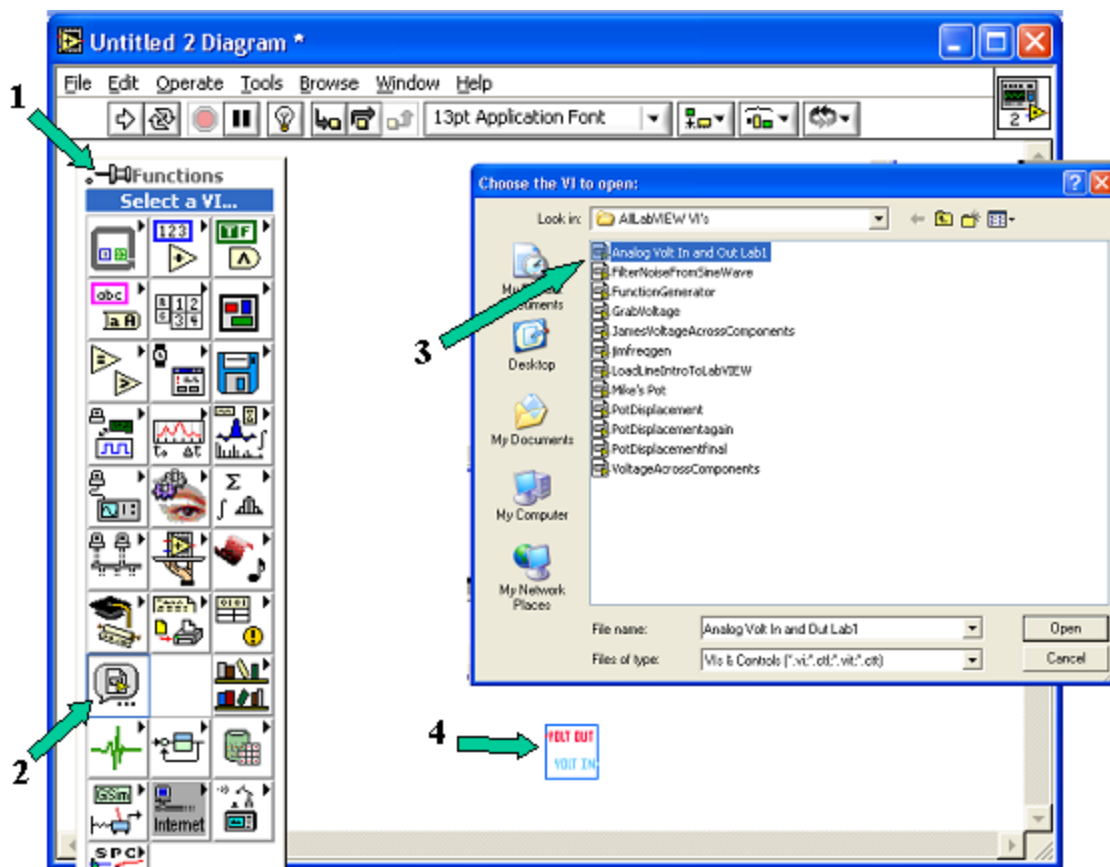


Figure 2.3.33. Steps to open a subroutine.

To use your this as a subroutine, open a new vi (File, New VI) and then on the **Diagram Panel**, under the **Functions Menu**, click **Select a VI...**, and then double click your subroutine, dragging it to the **Diagram Panel**. This is shown in the four steps of Figure 2.3.33. Now, as shown in Figure 2.3.34, wire a control and an indicator to the terminals of the subroutine. Press the **Run Continuously** button and change the **Voltage Out** control values. The **Voltage Across LED** indicator should change as it did before. Again, make sure you set Voltage Out to zero before stopping the program.

A note on wiring subroutines: There are two ways to create controls and indicators for use wit any subroutines, or any LabVIEW functions. One is by bringing up controls or indicators on the **Front Panel**, as you've done previously. A faster method is while on the **Diagram Panel**, use the tab key until the wire spool is the cursor, and then *right* click one of the terminal on the subroutine or the function. Now scroll down to **Create**, and then click **Control** or **Indicator**.

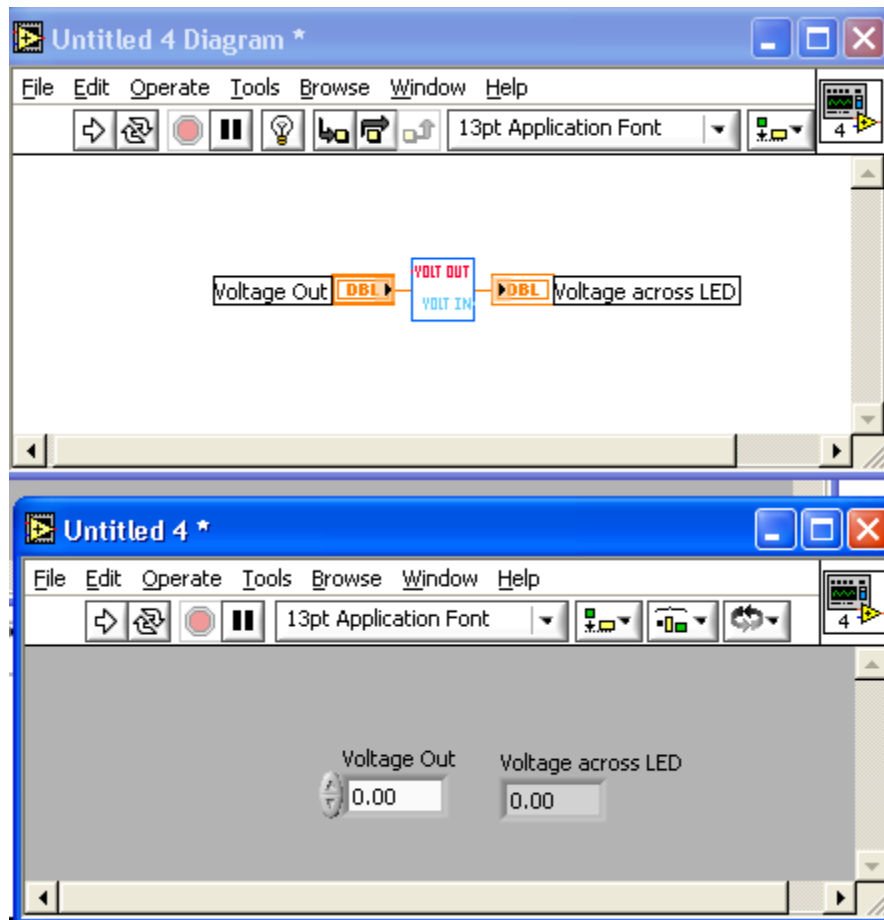


Figure 2.3.34. Wiring a subroutine.

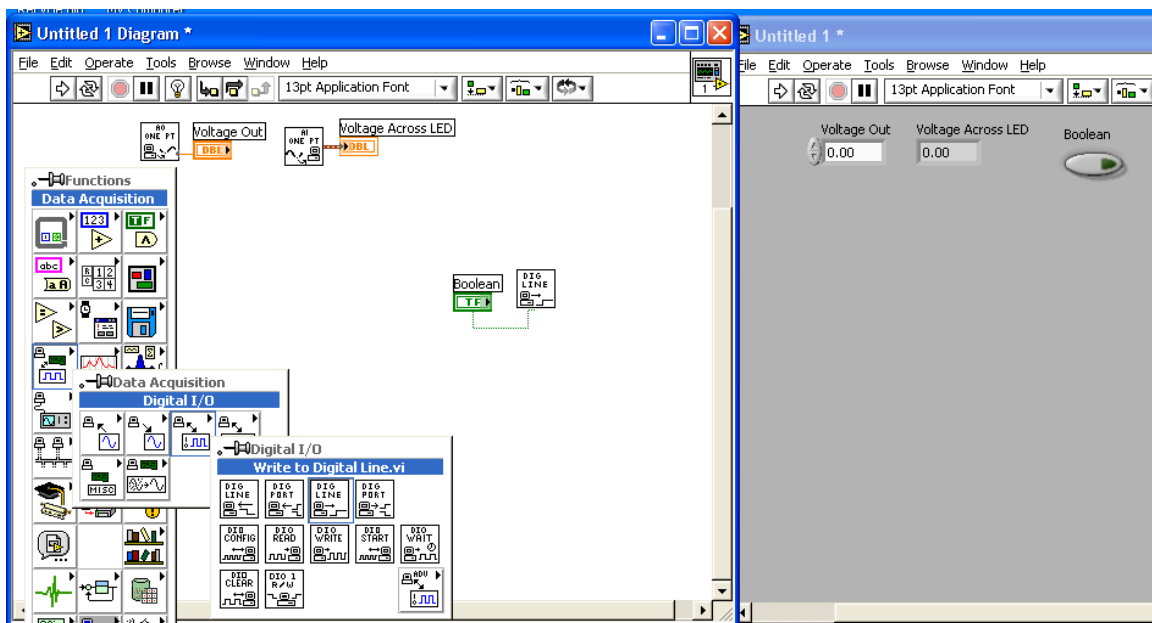


Figure 2.3.35. The Digital Output vi.

## Digital Output:

On the **Diagram Panel** of your LabVIEW program, pull up the **Write to Digital Line.vi** as in Figure 2.3.35, then on the **Front Panel**, bring up a **Push Button** (under the **Boolean** menu), and wire it to the **line state** terminal of the **Write to Digital Line** vi. Using wires with dual banana plugs, connect a Fluke multimeter to the Digital Output Channel 0 as in Figure 2.3.36. Select the DC voltage setting and press the small yellow button on the Fluke momentarily so that MAN is indicated on the display. This means that the meter is in *manual range* mode. You can specify the location of the decimal place by momentarily pushing the yellow button repeatedly until the display is configured as you desire. Press LabVIEW's **Run Continuously** button. Click the Boolean Push Button, and you should see the Fluke voltage go from 0V to 5V, to 0V. This is a digital signal. To return to *auto range* mode, push the button for one second – the meter will then display the maximum number of decimal places possible for the voltage level you are reading.



Figure 2.3.36. Digital connection.

**Demonstrate your LabVIEW program to an instructor before continuing.**

## Review of Data Acquisition basics.

Whether it's sending a signal to your printer or reading an image from your digital camera, computers can only send or receive small voltages - never current. For standard computer interfaces (such as printer ports) the output can be either 5 volts or zero volts. This is a *digital* signal. Through a data acquisition board, however, the computer can send or receive not just 5V, but voltages that vary from zero to as high as +/-10 volts or more. You saw this with the LED experiment earlier in this section. This is called an *analog* signal. Within the computer the input or output signal is still digital (either 0 or 5 volts), but the data acquisition board uses an *analog to digital converter* (ADC) and a *digital to analog converter* (DAC) as translators between the digital computer and the analog world. The Data Acquisition board (sometimes referred to as a DAQ – pronounced 'dack') mounted in this computer is built by National Instruments, the same company that makes the LabVIEW program, and along with some digital lines and some timer lines, it has 16 analog input and 2 analog output lines.

The ADC and DAC in the National Instruments card are 16 bit converters, meaning that the card can distinguish  $2^{16}=65536$  different voltage levels within its analog voltage range. That is, if the voltage range is -10V to +10V (i.e. 20V range), then the smallest detectable step is  $300\mu\text{V}$  (20V divided by 65536). So if you're sending out a voltage, ramping up from 0V to say 1V, then the first voltage will be zero, and then the next will be  $300\mu\text{V}$ , the next will be  $600\mu\text{V}$ , etc. You would not be able to achieve an analog output voltage of  $310\mu\text{V}$ , and conversely, an analog input voltage of  $310\mu\text{V}$  would probably be interpreted by the computer as  $300\mu\text{V}$ . This is a source of *uncertainty* in any signals generated or read by the computer. We will discuss uncertainties in general later in this course.

In this course, most of the signals we will encounter are analog signals generated by sensors (i.e. devices that measure temperature, displacement, pressure, etc.). Over the next few weeks we will study electronic circuits and data acquisition tools for turning these signals into usable measurements.

### 2.3.8 Introduction to Potentiometers.

A potentiometer, or *pot* for short is a three-terminal resistive device. Even if you have never built an electronic circuit, you have almost certainly used a potentiometer before (ie. the volume knob on a stereo is a potentiometer). The schematic symbol for a potentiometer is shown in Figure 2.3.37. It consists of a strip or layer of resistive material, connected at each end to an *end terminal*, and over which a *wiper* traverses. As the wiper is moved, the path lengths between the wiper and each of the end terminals changes, and therefore the resistance between the wiper and each terminal also changes. Resistance between end terminals is constant, and is part of the specification of the potentiometer. Typical values are 1k, 10k, and 100k. Potentiometers come in two mechanical configurations: linear and radial. Linear pots have a shaft which can be translated to move the wiper and on a radial pot, the wiper is moved by rotating a knob; it may have a range of one or more turns. Another important characteristic of a potentiometer is the taper: linear or exponential (unfortunately *linear* is also used in this context). Potentiometers with a linear taper have a constant resistance per unit length, while the resistance in an exponential potentiometer varies exponentially throughout its range. All pots used in this lab will have a linear taper. Other characteristics are the tolerance (just like with resistors), power rating and maximum voltage. To emphasize the construction and operation of a potentiometer, you will use a large hand-made wire-wound potentiometer in the following exercises.

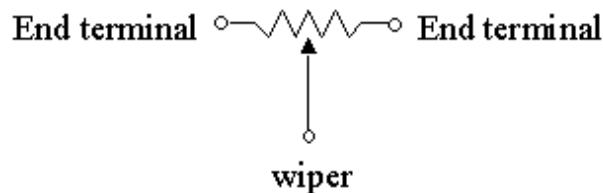


Figure 2.3.37: Potentiometer Schematic Symbol

**The Wire wound potentiometer.** All wire has some resistance, and the resistance of the copper wire we'll be using in these labs has about  $0.15\ \Omega$  per meter. We built this potentiometer by wrapping about 200 meters of thin copper lacquer coated wire around a  $2\frac{1}{2}$  cm radius x 70 cm long PVC tube, and then sanded off the lacquer where the metal ball will touch the copper wire. The resistance from left end of the wound wire to the right is about  $30\ \Omega$ . The ball completes the potentiometer as it rolls along the wound wire tube and the brass rod. It is the wiper.

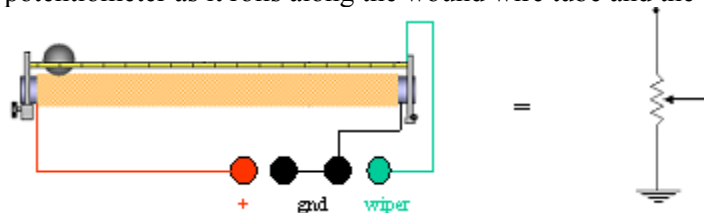


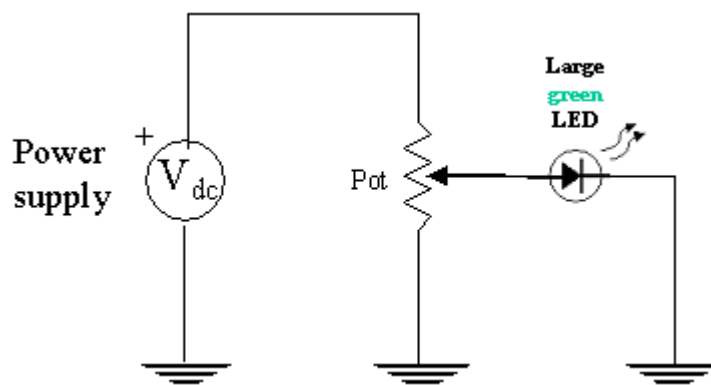
Figure 2.3.38: Wire wound potentiometer.

**The Potentiometer as a Variable Resistor.** A variable resistor can be created by using only two terminals of the potentiometer: the wiper and one of the ends. The other end is left unconnected. As the wiper is moved back and forth, the length of the electrical path changes, and hence so does the resistance. We will now construct a variable resistor using the wire-wound potentiometer, study its behavior, then use it to adjust the intensity of an LED connected in series.



Connect the left (**red**) and the right (**black**) leads of the pot to the Keithley multimeter, and measure the total resistance of the pot. It should read roughly  $30\ \Omega$ . Now move the left (**red**) lead to the Wiper lead (**green**) and place the ball on the pot. Move the ball all the way up and down the pot, and watch the resistance change. Move it to the center and notice this resistance (it should read about  $15\ \Omega$  – about half the pot’s full value).

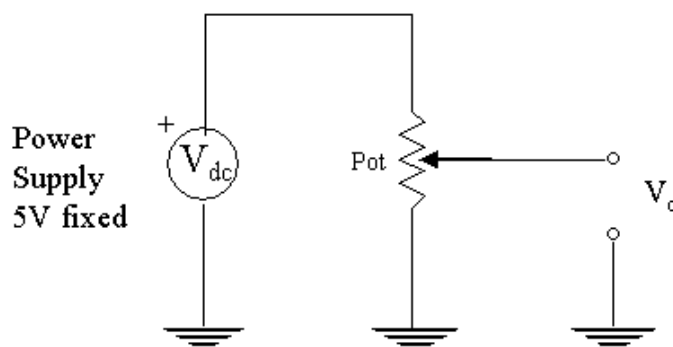
Connect the potentiometer to a large **green** LED and the power supply, as in Figure 2.3.39. As always, before connecting a power supply, ensure that the power supply’s voltage knob is zero and the power supply is off.



**Figure 2.3.39. Potentiometer circuit powering an LED.**

Place the ball in the middle of the pot, and turn the power on, setting the output voltage to 2.0V. Move the ball up and down the rod, and as the pot resistance changes, you should see the LED get brighter and dimmer. Other colors of LED will also work but we found that variations in intensity were easier to see on a green LED.

**The Potentiometer as a Voltage Divider.** We’ve already seen discrete voltage dividers with series circuits, but we can use the potentiometer’s variable resistance to construct a variable voltage divider.



**Figure 2.3.40: The Potentiometer as a voltage divider.**

To construct a voltage divider, apply 5V across the end terminals of the potentiometer using the 5V fixed output on the power supply, as shown in Figure 2.3.40. Read the voltage level of the wiper (referenced to ground) using the oscilloscope while rolling the ball back and forth. Note that for any given position of the ball, the length of wire between the positive supply terminal and the ball serves as  $R_1$  in the voltage divider equation, and the length of wire between the ball and the grounded terminal serves as  $R_2$ . Therefore, by moving the ball back and forth, we change the values of  $R_1$  and  $R_2$ , and hence the output voltage.

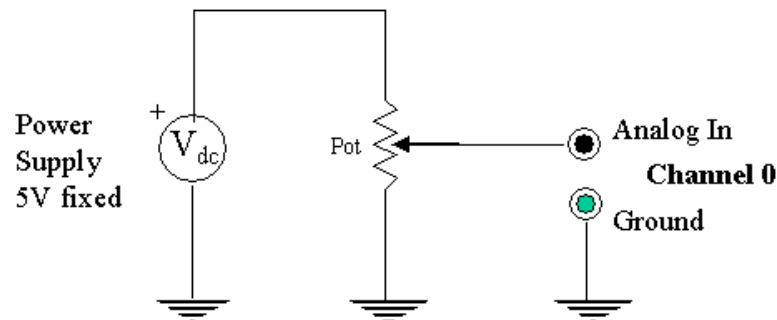
The wire-wound potentiometer on your bench is simply a teaching tool. You will almost certainly never use such a device again. If you have a need for a potentiometer in an electronic circuit, or to measure displacement in an experiment, you would buy a potentiometer. Obtain a linear and a radial potentiometer from a lab instructor. Make sure you can identify the terminals. What are their mechanical ranges (ie. linear displacement or rotation)? What are the resistances? Record this information in your notebook. Also look at the data sheets for these

devices and note any other information that looks important. Your homework assignment for this lab session is to produce a technical reference sheet for potentiometers so make sure you get all of the information you think you might need.

Now replace the wire-wound potentiometer with one of the commercially manufactured potentiometers that you have. Repeat the voltage divider experiment.

In the next activity we will use the variable voltage divider function of the potentiometer to measure the position of the wiper (ball).

**A potentiometer application: dynamic displacement measurement.** Because the properties of our voltage divider vary with position of the wiper, it can be used as a displacement transducer (a device that converts position to voltage). To do this, we must **calibrate** the potentiometer so that we know the relationship between output voltage and position. Then, using LabVIEW, we can convert a time-changing voltage into position as a function of time, and therefore track the position of the ball as it rolls along the potentiometer. The following steps outline the process:



**Figure 2.3.41: The Potentiometer Configured for Displacement Measurement with LabVIEW.**

1. Open the Labview VI PotDisplacement.vi. Maximize the window so that all of the controls and indicators are displayed. The left side of the panel is for calibration, and the right side is for data acquisition. On the calibration side, there are two columns of values. The left column contains indicators that will display displacements corresponding to the notches in the wiper bar of the wire-wound potentiometer. The right column contains controls where you will enter voltage levels resulting from the displacements given in the left column. The 'Potentiometer Calibration' graph displays the relationship between voltage and displacement. During calibration, the two analog gauges in the bottom left corner will display the instantaneous voltage being read from the potentiometer, and the displacement calculated from the calibration data. On the data acquisition side is a button to initiate data collection, and an x-y graph displaying the displacement of the ball as a function of time.
2. In our displacement measurements, we will be using the wire-wound potentiometer as a voltage divider, so begin by connecting the end terminals of the potentiometer to the 5V fixed terminals on the power supply. Connect the wiper to the +ve side of Analog Input Channel 0, and the wiper reference terminal to the ground terminal on the LabVIEW breakout box. Note that this is the same configuration as you previously had except that the output will now be read by the DAQ card on the computer.
3. Start PotDisplacement.vi by clicking the start arrow (do not run continuously). Roll the ball back and forth on the potentiometer and verify that the 'Potentiometer Output (V)' gauge is changing in response to ball motion.
4. Now the potentiometer must be calibrated. Rotate the brass bar so the grooves are facing slightly toward the wire-wound pipe. This will allow the ball to rest in the grooves for calibration. Begin at zero displacement, and record the displayed voltage levels in the 'Voltage' column as you increment the position of the ball. As you enter points, they will be plotted on the graph. We will begin with the assumption that the potentiometer is linear – that is, that the resistance per unit length is the same in any portion of the potentiometer's range. Therefore, the program will try to fit a straight line to the data points. As the number of data points increases,

the line better fits the data (The algorithm uses **all** of the points to calculate the line. When some are still preset to zero, the representation is poor.).

5. When you have completed the calibration, all of your data points should be nearly on top of the line. Also, as you move the ball back and forth, the 'Displacement' gauge should reflect the true displacement of the ball. Note the *linearity* of the potentiometer. Try to describe or quantify the deviation of the data points from the best-fit line. Are the points randomly distributed about the line, or do they follow some pattern? What do you think is the cause of these deviations? This should be briefly discussed in the reference sheet that you will prepare after the lab.
6. Now you are ready to acquire data. Even though the potentiometer is not exactly linear, we will assume that it is and use our calibration (the straight line) to convert voltages into displacements. If the deviations are small, this will be a good approximation. In a more sophisticated experiment, we would quantify these deviations so that we could estimate the uncertainty in our measurements. This will be discussed in detail later, and will become very important in the thermodynamics experiments later in this course.
7. Rotate the brass bar so that the grooves face away from the pipe and the ball can roll freely. By loosening the thumbscrews at the end of the potentiometer, raise the end to produce a slight incline. Your angle should be such that it takes the ball about 3 or 4 seconds to traverse the entire length when started from rest. Release the ball from the top of the ramp immediately after clicking on the 'Take Data' button. Data acquisition will stop and the program will end when the ball reaches approximately the 60 cm mark. You can save copies of both of the graphs by clicking on one to highlight it, then copying (ctrl-C) and pasting (ctrl-V) in another software program such as Word, Powerpoint or Paint. You can then save that file. The dynamic displacement measurement can be repeated, if necessary, by restarting the program. The calibration will not be reset until the program is closed.

**Demonstrate your calibration and dynamic displacement data to an instructor before continuing.**

### 2.3.9 Clean Up

You have now completed all of the laboratory activities for this session. You may dismantle all of the circuits that you constructed, putting the components back in their proper places. **Any components that are burnt out or otherwise not functioning should be thrown away to avoid frustrating you and your colleagues.** Remove your pin board from the trainer and store it in your allocated spot in the storage cabinet. Make sure that you turn off the power on all instruments and that you leave your bench as you found it so that it is ready for the next group.

### 2.3.10 Constructing the Reference Sheets.

You should now be quite comfortable with using potentiometers and voltage dividers. Your assignment is to construct a reference sheet for potentiometers. If you have time you should try to complete as much of this as possible during the lab session so that you can make certain that you have collected sufficient data and ask questions. In particular, you may need to discuss with the lab instructors what items merit inclusion in the reference sheet. These will be due at the beginning of the next lab session.

Remember that the reference sheet should be useful to you in the future. Make sure that you include the following items:

- the schematic symbol for the potentiometer
- different ways of connecting a potentiometer (including schematic diagrams) and related applications
- any applicable equations (ie. voltage division)
- specifications and other information about commercially manufactured potentiometers
- any warnings or caveats that you think might be useful based on your experience with potentiometers

To construct schematic diagrams, you will have to use a software package called Orcad. Orcad is the foremost schematic generation software used in industry, and has capabilities far beyond that for which we will use it. The following notes will help you to get started. For further information, consult the Orcad reference on the MAE 221 web page.

- The Orcad program can be found on computers in room J-209 under **Start -> All Programs -> Orcad Family Release 9.2 -> Capture**, or by clicking the **Capture** icon on the desktop.
- To begin a new drawing, select **File -> New -> Project**. In the pop-up window, choose **Schematic** and give your project a name. You should also use the **Browse** function to specify the directory so that you can find it later. When you click OK, you will get a new window in which to draw your diagram.
- You don't have to draw components and symbols, Orcad has an extensive database of these that can be selected and placed in your diagram. To search the database, click the **Place Part** button (second button from the top) on the toolbar along the right edge of the screen. In the **Part** field, enter a number or word that describes the component that you require, or is part of the description. For example, to find a potentiometer, type `pot` and the **Parts List** field will display items that begin with those letters. If you don't know how the name begins but know a sequence of letters or numbers in the name, you can use wildcards, ie `*7805*`.
- You can then select the appropriate item from the **Parts List** field, click OK, and place it on your diagram by left clicking. When you have placed as many as you need, hit Esc. You will find the feel of Orcad to be much like LabVIEW. The components that you place have terminals and labels just like LabVIEW does. You can rotate the part by pressing ctrl-R repeatedly until the part has the desired orientation. Labels can be customized by double clicking them and supplying the information in the resulting pop-up window. Labels can be removed by clicking them once and hitting Delete on the keyboard.
- Once you have placed the components, you must connect them. Do this using the **Place Wire** button (third from the top). This works just like LabVIEW.
- Grounds and nodes are found on the side toolbar, not in the parts database.
- To put your schematic diagram into a text document, use the **Select** tool (the arrow) to make a box around your diagram, selecting it. Then copy and paste it into another application.

**In addition to completing the technical reference sheet, make sure you review Chapter 3 and answer the Pre-lab Questions in Chapter 3 prior to your next lab session. You will need to provide your answers to the lab instructors at the beginning of the period.**