Department of Electrical Engineering

ELEG 2101L Lab Manual

Revision 1.0

Spring 2010

Table of Contents

Safety Instructions/Considerations

Please make sure that you read through this safety instruction carefully before you conduct any experiment!

Why do electrical accidents occur?

Fatal electric shock is one cause of death at research and development facilities. The most likely victim of an electrical accident is someone that even has a lot of experience with electrical equipment and is very familiar with the specific equipment being used. In addition, unlike many other industrial accidents, electrical accidents often happen to professional and supervisory staff. Many people have experienced some type of electric shock. Survival of a mild shock is dangerously misleading. Unfortunately, survival of an electric shock, even if minor, often results in unreasonable expectations for future survival.

Proper grounding and circuit breakers provide protection even if an internal fault exists. The greatest danger from equipment occurs when its protective enclosure is violated, when it is improperly installed, or when it is damaged during use.

When a person (YOU) come in contact with an energized circuit the damage depends on many things including but not limited to age, perspiration level, and physical condition. For these reasons a shock from 50 V may be as dangerous as a shock from 5,000V. For example if a person (YOU) has a lot of perspiration, the body may have as little as 1,000 ohms of resistance. Once contact to an energized wire is made, the resistance decreases. This is the result of the fluid in the body acting as the conductor. So it is important to stay alert even with lower voltages.

A person (YOU) must also be careful with jewelry. If a ring with a resistance of 0.5 ohms becomes lodged across 5 volts, the current through the ring will be 10A!!! The power dissipated (heat) in the ring will be 50 Watts. This will cause server burns.

What does electric shock do to the human body?

Voltage determines how much current will flow through a body resistance path. Current is the killing factor in electrical shock. Resistance of the current path will determine the current level that passes through the body. The pathway that the current takes can significantly affect the impact of the shock. Three basic types of injury result from electric current running through the body: shock to the nervous system, tissue burns, and mechanical injury.

The following statements apply to the person of average physical condition.

• 1 to 8 mA:

    1. Sensation of shock.
    2. Individual can let go at will.

• 8 to 15 mA:

    1. Painful shock.
    2. Individual muscular control is not lost (let go).

• 15 to 20 mA:

    1. Painful shock.
    2. Individual muscular control of adjacent muscles is lost.

• 20 to 100 mA:

    1. Painful shock.
    2. Severe muscular contractions (can’t let go).
    3. Breathing is difficult.

• 100 to 200 mA:

    1. Ventricular fibrillation.
    2. Loss of consciousness.
    3. Electrical current holds victim to circuit as long as current flows.
    4. Unless normal heartbeat is restored, with a large DC (or AC) pulse of current, death may result.
    5. Fibrillation threshold can range from 1amp depending on the duration of the shock and the persons physical condition.

• Greater than 200 mA:

    1. Severe burns.
    2. Muscular contractions severe enough to cause chest muscles to clamp the heart and stop it throughout the duration of shock.
    3. The heart may be too weak to restart when the current flow stops.

Circuit breakers and fuses are designed to protect circuits and equipment and, in general, are rated far in excess of the amount of current necessary to cause fatal electric shock. Whereas over-current devices are designed to protect equipment, they provide little or no protection for people.

Lab 1: Introduction to Circuits

Objectives

In this laboratory exercise, you will learn how to identify the common equipments that will be used throughout the semester. You also learn how to use the PSPICE software for simulating circuits. In addition, you will learn safety guidelines of the equipments by power capabilities and common mistakes.

Materials

Oscilloscope, DC Power Supply, Function Generator, Digital Multi-meter, Breadboard, RC box, Computer, and PSPICE simulation software

Reference

1. Circuit I Lab Manual: Safety instructions
2. Circuit I Lab Manual Appendix I: Identification of Equipment with Explanations
3. Circuit I Lab Manual Appendix II: Introduction to PSPICE
4. Electric Circuits, Nilsson & Riedel, 2008, Chapters 3.6 and 3.7

Pre-Laboratory Assignment

1. Read *Reference* 1 to understand lab safety issues before you come to the lab
2. Read *Reference* 2 to familiarize you with the equipments used in this semester
3. Read *Reference* 3 to familiarize you with the basic operation of PSPICE simulation software

Laboratory Assignment

Today, you have identified the major equipments including oscilloscope, DC Power supply, function generator, digital multi-meter, breadboard, and RC box that you will be used frequently. You have also learned about the safety concerns that all electrical engineers should know. In addition to the physical aspects of the lab, PSPICE has been presented as a simulation tool that electrical engineers use before stepping into the lab. In almost all cases, engineers do most of their work and have a good idea of the results expected before performing an experiment. This crucial concept is the difference between engineering and “trial and error”.

Now, you will need complete theexercises outlined below to demonstrate your new knowledge of PSPICE.

Problem 1 (30 points)

Simulate the circuit as shown in Figure 1 by the following steps:

(1) Complete a bias point analysis on the circuit;

(2) Sweep the input voltage from 10V to 100V at increments of 10V;

(3) Plot V1 versus V2 and V1 versus V3.

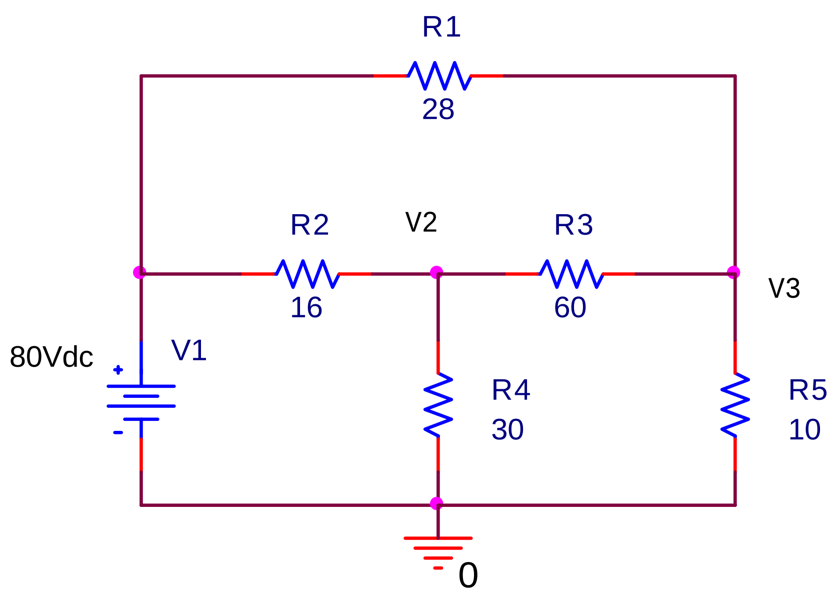


Figure 1: The first circuit to be simulated

Problem 2 (30 points)

Simulate the circuit as shown in Figure 2 by the following steps:

(1) Complete a bias point analysis on the circuit;

(2) Sweep the input voltage from 10V to 100V at increments of 10V;

(3) Plot V1 versus V2 and V1 versus V3;

(4) Compare the two curves obtained in Problem 2 with the two curves obtained in Problem 1, what conclusion can you draw? (Hint: Read *Reference* 4 to find the answer)

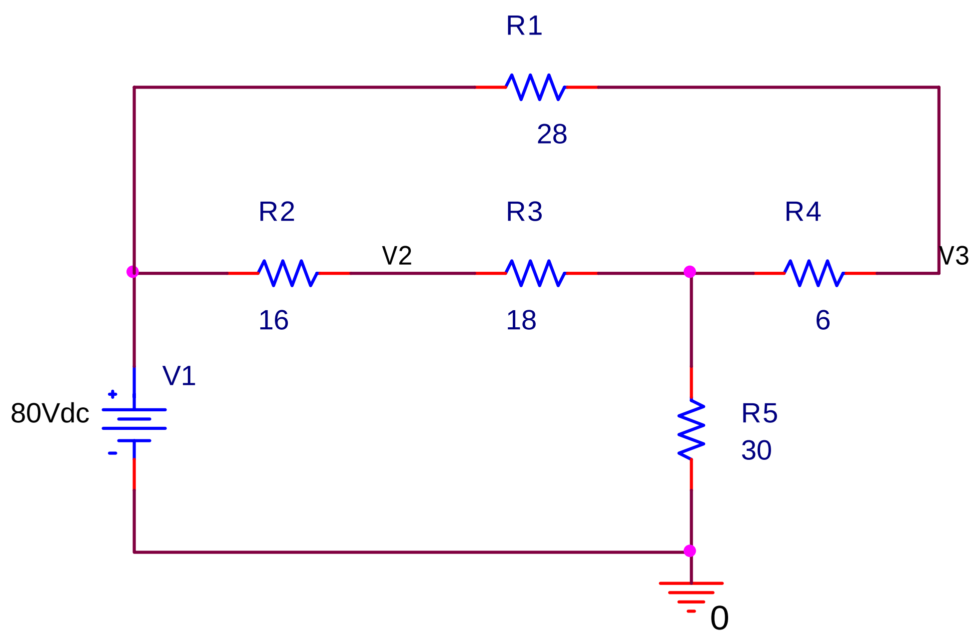


Figure 2: The second circuit to be simulated

Problem 3 (40 points)

Simulate the circuit as shown in Figure 3 by the following steps:

(1) Complete a bias point analysis on the circuit;

(2) Sweep the value of R4 to find when the current going through R5 is zero. (Hint: You can first choose a large range of the value of R4 to find the approximate value range and then gradually refine the sweep range to find the final value.

(3) Use the result you find to compare the ratios R1/R2 and R3/R4, what conclusion can you get into? (Hint: Read *Reference* 4 to find the answer)

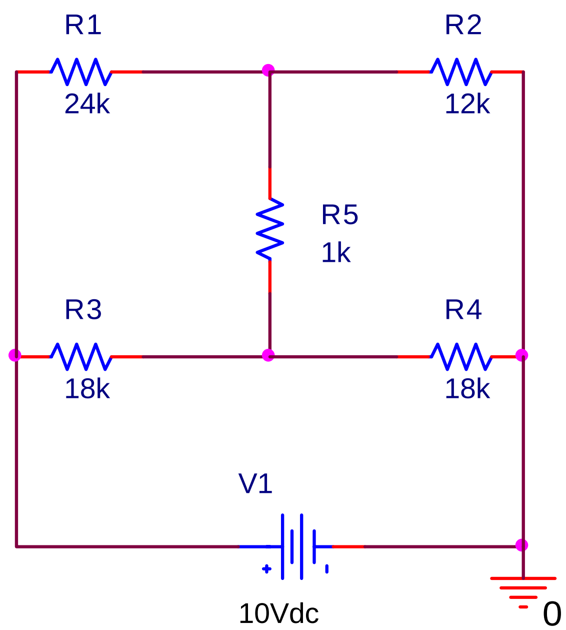


Figure 3: The third circuit to be simulated

Please remember that you need show your results to the TA before you leave. Please include the necessary steps (schematic drawing, simulation results, plots, etc) leading to the final answer in your final report.

Lab 2: DC Power Supplies, Resistors, and Digital Multi-Meters

Objectives

In this laboratory, you will construct an actual circuit based on a circuit schematic and test its simple behavior. You will gain the knowledge of using a DC power supply, resistors and a digital multi-meter (DMM) for testing the circuit. You will also carry on current-voltage (I-V) relationship measurement for this simple circuit.

Materials

DC Power Supply, Breadboard, Resistors (1K, 10K, 22K, 47K), Digital Multi-meter

Reference

1. Circuit I Lab Manual Appendix III: Using DC Power Supply, Resistors, and Digital Multi-meter
2. Electric Circuits, Nilsson & Riedel, 2008, Chapter 2.4
3. Electric Circuits, Nilsson & Riedel, 2008, Chapters 4.2 and 4.5

Pre-Laboratory Assignment

1. Read *Reference* 1 to farmilarize you with using DC power supply, resistors, and digital multi-meter (DMM)
2. Read *References* 2 and 3 to learn the basic knowledge of circuit analysis techniques

Problem1 (30 points)

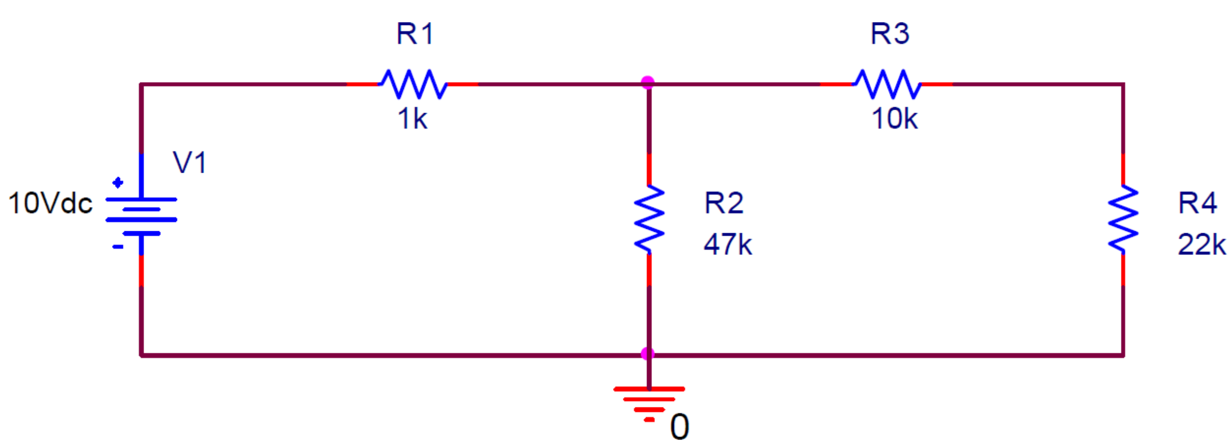


Figure 1: Assigned Circuit

Figure 1 gives a circuit schematic which will be used to construct an actual circuit in Lab 2. Please keep in mind that we have a standard process which needs to be done before we construct an actual circuit. We can use PSPICE to perform bias simulation. Based on such a simulation, an analytical treatment is required next. In areas of research, analysis and simulation go hand in hand. In almost all cases, analysis is done first as a means of laying sound groundwork and supplying an expectation of intended results. The art of simulation then provides an intermediary glance of 'real world' operation and it can also illuminate unseen pitfalls from analysis. In the end, both simulation and analysis are necessary and they can corroborate each other to provide stronger validity of the results. If analysis and simulation do not closely match each other, then an error of some sort is , is suspect whether by assumption or oversight.. In engineering, this process is done all the time before actually building and testing any kind of system or device.

For the analytical treatment of the circuit shown as in Figure 1, please use nodal or mesh circuit analysis technique to determine the following quantities:

(1) Check voltage drop across all the resistors

(2) Check currents I_1 and I_2 in the two loops

(3) Find the power delivered and the power adsorbed in the circuit

After having done these, please complete Table 1 below, noting that the formula for % Error is as follows

$$\%\text{Error} = \frac{|\text{Simulated Value} - \text{Calculated Value}|}{\text{Calculated Value}} * 100$$

Table 1

	Simulated Values	Calculated Values	%Error
V_{R1}			
V_{R2}			
V_{R3}			
V_{R4}			
I_1			
I_2			

Power Delivered			
Power Absorbed			

Please answer the following questions in your report: do the simulated values match up with your calculated values? If not, why? Explain the reasons in details. As alluded to earlier, this analysis and simulation need to be completed before you start building and testing the actual circuit.

Problem 2 (30 points)

Please first read through the entire lab and then do the task and answer questions below:

(1) Draw both the schematic diagram and a sketch of the power supply setup for the following:

a.) +/- 5V b.) +/- 6V c.) +12V d.) -8V

(2) What are the color codes for a 1k, 10k and 20k resistors?

a) 1k = ____

b) 10k = ____

c) 20k = ____

(3) What are the values of resistors with the following color codes (please give units)?

a) RED RED YELLOW

b) BROWN BLACK BROWN

c) GREEN BLUE VIOLET

(4) What maximum current can a 1kΩ, 0.25 W resistor handle?

(5) What maximum current can a 47kΩ, 0.25 W resistor handle?

(6) What maximum voltage can a 10kΩ, 0.25 W resistor handle?

(7) What maximum voltage can a 22kΩ, 0.25 W resistor handle?

Problem 3 (40 points)

In real world applications, we usually describe the characteristics of one electrical component (see Figure 2a and note the definition of passive sign convention) described by a table (or curve) of current versus voltage relationship. In Figure 2b, we present one circuit with two voltage sources and one resistor connected in series. Voltage source V2 and resistor R1 in the dash line box form one “component” and we will simulate and measure its current-voltage characteristics.

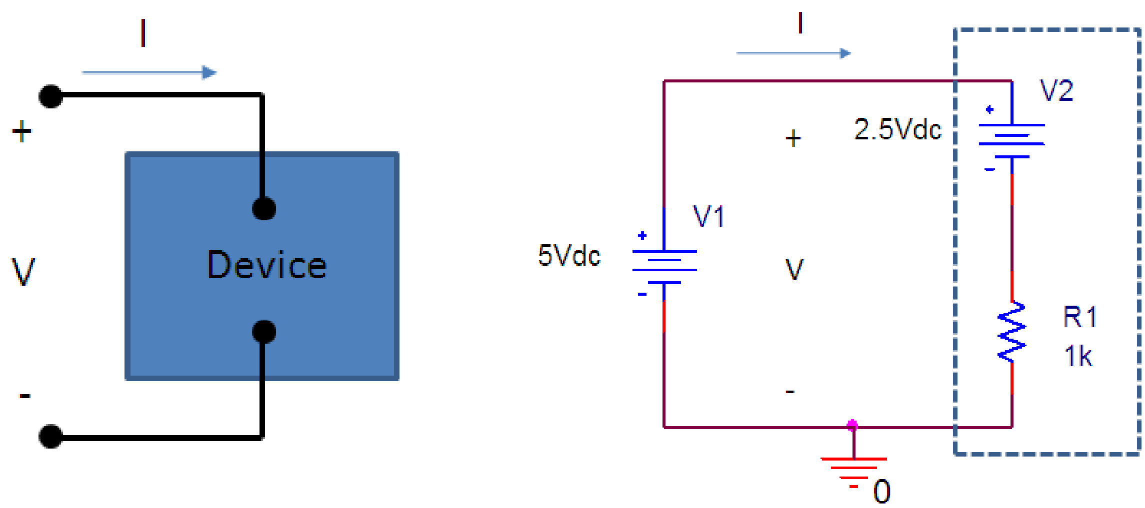


Figure 2 Assigned circuits (Left, a: A schematic of one electrical component; Right, b: The real circuit to be simulated)

Please use PSPICE to simulate the circuit shown in Figure 2b. Sweep the voltage source V1 and plot the curves of V1 versus I when voltage source V2 equals to 2.5V and 5V. Please answer the following questions. If the power rating for R1 is 0.25 W, what is the range of V1 to sweep when voltage source V2 equals to 2.5V and 5V? (Do you think that you can construct the circuit as shown in Figure 2 dash line box if the current-voltage curve is given?)

Laboratory Assignment

Problem 1 (50 points)

Now that you have an expectation of how the circuit in Figure 1 should behave (from previous simulation and analysis) as well as full knowledge of the physical equipment we will be using, it is time to produce some results. For each of the resistors chosen for building the circuit, begin by measuring the actual resistance using the DMM. Create a table to show the nominal resistance values, the actual measured values, and the percent error. After assembling the circuit in Figure 1 on a breadboard, apply power to the circuit and make all voltage and current measurements indicated in Table 1 (using measured V and I for the Power calculations). Create your own table with the addition of a measured values column and take the %Error to be the error between the calculated value and the *measured value*. Be sure to comment on your results. Are they as you expected them to be? What could be any sources of error? Next, please answer the following questions in your final report.

(1) What is the most *positive* voltage that the HM7042 can supply? Please sketch the setup.

(2) What is the most *negative* voltage that the HM7042 can supply? Please sketch the setup.

(3) What is the largest +/- voltage that the HM7042 can supply? Please sketch the setup.

(4) With the circuit still assembled and the **power off**, re-measure the resistance across R_2. Is this value the same as before? Please explain your finding.

Problem 2 (50 points)

Construct a circuit show in Figure 2b on the breadboard. Use DMM as a current meter and insert it to the proper position so that you can perform current-voltage measurement by sweeping V1 and reading the output of power supply and DMM simultaneously. Generate a table to record your measurement results for V2 is equal to 2.5 V and 0 V. Compare your measurement results with your simulation results and explain why they are different. List as many as possible of the reasons that might cause errors for the measurements and briefly describe how you correct them in the experiment. (For example, the DMM current reading is not ZERO even though there is no current going through the circuit.)

Lab 3: Voltage-Divider and Meter Loads of Digital Multi-Meter

Objectives

In this laboratory, you will study the impacts of external loads to the output of a voltage-divider circuit. You will also study to design a simple circuit to measure the internal resistance of a digital multi-meter under voltage measurement and current measurement modes.

Materials

DC Power Supply, Digital Multi-meter, Resistors (100, 1k, 10k, 100k, 1M), Breadboard, RC box

Reference

1. Electric Circuits, Nilsson & Riedel, 2008, Chapter 3.3

Pre-Laboratory Assignment

Problem 1 (50 points)

At times-especially in electronic circuits-developing, more than one voltage level from a single voltage supply is necessary. One way of doing this is by using a voltage-divider circuit. Considering a circuit shown in Figure 1, voltage source V_1, resistors R_1 and R_2 form one standard voltage divider circuit and voltage output is measured across the two terminals of R_2.

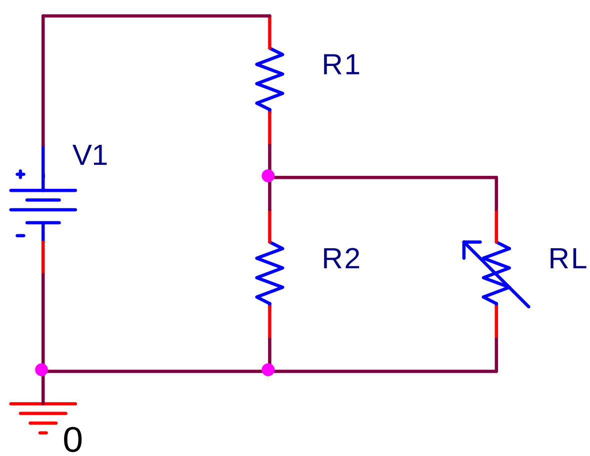


Figure 1: A circuit of voltage divider

The voltage output for the voltage divider without external load R_L is

$$V = \frac{R_2}{R_1 + R_2} V_1 \tag{1}$$

Since the ratio in equation (1) is always less than 1.0, the output voltage V is always less than the source voltage V_1. Consider that a resistor R_L is connected in parallel with R_2, as shown in Figure 1. The resistor R_L acts as a load on the voltage-divider circuit. A load of any circuit consists of one or more circuit elements that draw power from the circuit. With the load R_L connected, the expression for the output voltage becomes

$$V = \frac{R_{eq}}{R_1 + R_{eq}} V_1 \tag{2}$$

Where

$$R_{eq} = \frac{R_2 R_L}{R_2 + R_L} \tag{3}$$

Substituting Eq. (3) into Eq. (2) yields

$$V = \frac{R_2}{R_1[1 + (R_2/R_L)] + R_2} V_1 \tag{3}$$

Assuming $V_1 = 5.5V$, $R_2 = 10R_1$, please answer the following questions.

(1) What is the output voltage V if there is no external load connected?

(2) Please calculate the output voltage V for a list value of R_1 and R_L and finish the table below:

	$R_1 = 100\Omega$	$R_1 = 1k\Omega$	$R_1 = 10k\Omega$	$R_1 = 100k\Omega$	$R_1 = 1M\Omega$
$R_L = 100\Omega$					
$R_L = 1k\Omega$					
$R_L = 10k\Omega$					
$R_L = 100k\Omega$					
$R_L = 1M\Omega$					

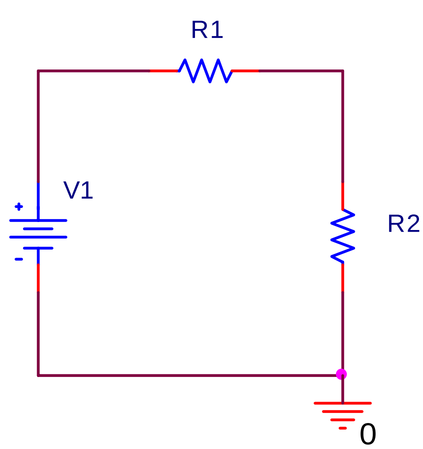
(3) For the case that $R_1 = 10k\Omega$ and no load is connected, if both R_1 and R_2 have a tolerance of $\pm 5\%$ and $\pm 10\%$, please find the maximum and minimum value of output voltage V and fill the following table.

	maximum	minimum
$\pm 5\%$		

$\pm 10\%$		

Problem 2 (50 points)

When DMM is used to measure the voltage and current, the user should be aware of that a DMM has internal resistance. This resistance might have significant impacts to the measurement results and introduce large errors. In principle, for voltage measurement, in order to change the circuit as little as possible, the internal resistance of the voltmeter is very high. This allows almost all of the current to flow through the resistor being measured just as it had been before the voltmeter was put in parallel. If the resistance being measured is greater than 10% of the voltmeter's internal resistance, the measurement will not reflect the actual voltage. The internal resistance of the voltmeter used in this lab is about 10 MΩ. When an ammeter is put in series with a resistor by breaking the circuit and re-closing it with the ammeter probes, it is like putting a resistance in series with the resistor being measured. Just as with the voltage measurement, this changes the equivalent resistance in the circuit and could significantly affect the current measurement. In order to change the circuit as little as possible, the internal resistance of the ammeter is relatively small. This keeps the voltage drop across the ammeter very low and allows the voltage across the resistor being measured to stay nearly the same as it was before the ammeter was put in series. Again, the 10% rule should be observed; if the ammeter resistance is greater than 10% of the value of the resistor being measured, the measurement will not be the actual current. The internal resistance of the ammeter used in this lab varies with the RANGE setting. The resistance drops as the range value increases. On a RANGE of .2, the internal resistance is 1KΩ.



In this experiment, you will need to design a simple circuit to measure the internal resistance of a Digital Multi-meter (DMM) under voltage meter mode as well as under current meter mode (On a RANGE of .2). A circuit shown in left can be used for the DMM internal resistance measurement, where V_1 is the voltage source, R_1 is the bias resistor and R_2 represents the internal resistor of DMM to be measured. $\mathrm{V_1}$ and $\mathrm{R_1}$ have to be chosen for two different measurements as $\mathrm{R_2}$ can change from MΩ to kΩ when a DMM works under voltage meter mode or current meter mode. When DMM works under the voltage meter mode, the voltage across $\mathrm{R_2}$ can be directly read from DMM. This voltage is not necessarily equal to $\mathrm{V_1}$ if the values of $\mathrm{R_1}$ and $\mathrm{R_2}$ are comparable. Using equation (1) and plugging in the values of $\mathrm{V_1}$ and $\mathrm{R_1}$, $\mathrm{R_2}$ can be calculated. When DMM works under the current meter mode, the current going through $\mathrm{R_2}$ can also be directly read from DMM. Using Ohms law, the total resistance of $\mathrm{R_1}$ and $\mathrm{R_2}$ can be calculated. Certainly, $\mathrm{R_1}$ can be directly measured using a DMM under resistance mode and then $\mathrm{R_2}$ can be calculated. Apparently, there are many other ways to measure the internal resistance of a DMM. You can design your own circuits to perform the measurement. In this prelab,

(1) Please provide the detailed design and explain how this circuit is used for the internal resistance measurement.
(2) Clearly explain how to choose the values of voltage source and resistors used in the circuits and sequential steps to perform the measurements. One factor that you want to keep in mind is that how you can obtain the maximum accuracy with the given components and instruments.

Please remember that you need show your design to the TA to get permission to carry on the actual measurement.

Laboratory Assignment

Problem 1 (50 points)

(1) Construct the circuit shown in Figure 1 and measure the voltage V across $\mathrm{R_L}$ using a DMM. Record all the values of $\mathrm{R_1}$, $\mathrm{R_2}$ and $\mathrm{R_L}$ during the measurements and generate a table to present your results.
(2) Provide a new drawing of the circuit to include the effect of DMM internal resistance. Compare the measurement results with your calculated results and discuss where the errors are from.For example, you can consider the non-ideal voltage source with internal resistor, the tolerance of the resistors used in the circuits or the impact of DMM internal resistance. .

Problem 2 (50 points)

(1) Perform the internal resistance measurement for a DMM as you designed in the prelab and record all values measured in the experiment. List the measured internal resistance for both voltage meter mode and current meter mode.

(2) Discuss the accuracy of your measurement results and where the error might be from.

Lab 4: Measuring Resistance Using the Wheatstone Bridge Circuit

Objectives

In this lab, you will gain some experiences to design and construct a circuit mimicking the function of a Wheatstone bridge to carry on resistance measurement. You will gain an understanding of the impacts of internal resistance of a current meter for the measurement accuracy. You will also gain some experiences using RC box as a high precision variable resistor.

Materials

DC Power Supply, Digital Multi-meter, Resistors (100, 1k, 10k, 100k, 1M), RC box, Breadboard

Reference

1. Electric Circuits, Nilsson & Riedel, 2008, Chapter 3.6
2. Appendix I: Identification of Equipment with Explanations (RC box part)

Introduction

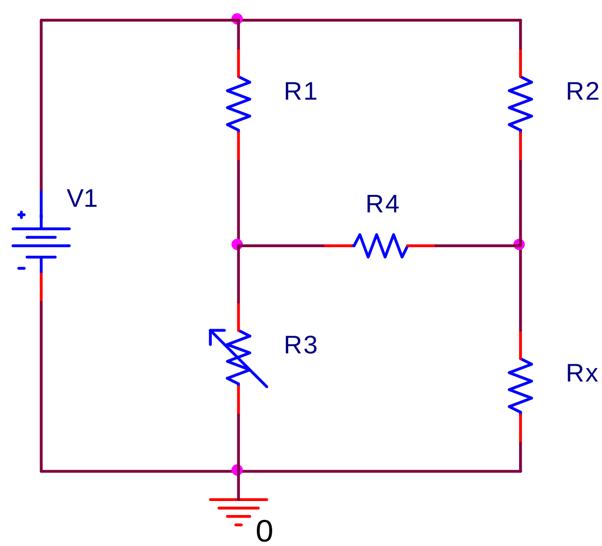


Figure 1. A schematic drawing of a wheatstone bridge (R_3 is a variable resistor and R_x is the resistor to be measured)

Many different circuit configurations are used to measure resistance. Using a Digital Multi-meter (DMM) might be the most straightforward one and has been used for many times in previous labs. This time we will focus on another one, the Wheatstone bridge. The

Wheatstone bridge circuit is used to precisely measure resistances of medium values, i.e., in the range of 1Ω to 1MΩ. In commerical models of the wheatstone bridge, accuracies on the order of ±0.1% are possible. A schematic drawing of a bridge circuit is given in Figure 1. The bridge circuit consists of four resistors (R_1, R_2, R_3, R_x), a DC voltage source V_1, and a detector (a highly sensitive current meter, represented as one equivalent resistor R_4). The resistance of R_3 can be varied. For accurate measurement, the DC voltage source has to be a highly stable battery. The detector is generally a d'Arsonval movement in the microamp range and is called a galvanometer.

To find the value of R_x, we adjust the variable resistor R_3 until there is no current in the galvanometer (R_4). We then calculate the unknown resistor from the simple expression

$$R_x = \frac{R_2}{R_1} R_3 \tag{1}$$

Equation (1) indicates that the voltage across the resistor R_4 is zero. For more detailed destrciptions of the Wheatstrone Bridge, you can read *Reference* 1.

Another modified measurement method is provided here, which does not require the resistance of R_1 and R_2 as long as they are stable. The measurement procedure is as following:

(1) Construct the circuit as shown in Figure 1 and adjust R_3 so that the current reading on R_4 is zero, and thenwrite down the resistance of R_3 as R_a

(2) Switch resistors R_3 and R_x and repeat the step (1) and write down the new resistance of R_3 as R_b

(3) Calculate the resistance of R_x by two measurement results using equation.

$$R_x = \sqrt{R_a R_b} \tag{2}$$

Pre-Laboratory Assignment

Problem 1

(1) Based on the abovementioned measurement procedure, derive equation (2) (10 points)

(2) The minimal current can be measured by the current meter (sensitivity) and the internal resistance of the current meter can play an important role for the measurement accuracy. Using the following values for the components given in Figure 1, perform PSPICE simulation (or directly calculate using the circuit analysis technique learned on

Circuit I class) to obtain the resistance of R_3 given current and resistantce of R_4 in Table 1and complete Table 1. (30 points)

V_1=10 V, R_1=R_2=R_x=1kΩ

Table 1

	Current going through R_4 (A positive current indicates that current flows from left to right)					
	10 μA	-10 μA	1 μA	-1 μA	0.1 μA	-0.1 μA
R_3 (R_4=1 kΩ)						
R_3 (R_4=10 Ω)						

(3) Assuming the sensitivity of a current meter is 10 μA, using the circuit shown in Figure 1 please design your own circuit to measure one resistor with resistance around 1 kΩ as accurate as possible. In you design, provide a list of selected values for every single component and explain why you want to chose those values. Please take the internal resistance of current meter (measured in previous lab) and the power rating of resistors into account in your design. (40 points)

(4) What is the maximum error for your design? (10 points)

Laboratory Assignment

Problem 1

(1) Show your prelab assignment to TA before you start the lab;

(2) Pick resistors that you need to construct the measurement circuit and measure them using DMM and record their resistance; (10 points)

(3) Construct the circuit and perform the measurement; (30 points)

(4) Calculate the resistance and compare this value with the measurement result using DMM in step (2) (20 points)

(5) Pick another 500 kΩ resistor and perform the same measurement, calculate and compare the result with the DMM measurement result (40 points)

Lab 5: Node Voltage and Mesh Current Methods

Objectives

In this lab, you will practice and compare two circuit analysis techniques(node voltage and mesh current methods) by performing PSPICE simulation and constructing the real circuits.

Materials

DC Power Supply, Digital Multi-meter, Resistors (1k, 2.2k, 4.7k), RC box, Breadboard

Reference

1. Electric Circuits, Nilsson & Riedel, 2008, Chapters 4.2 and 4.5

Pre-Laboratory Assignment

Problem 1

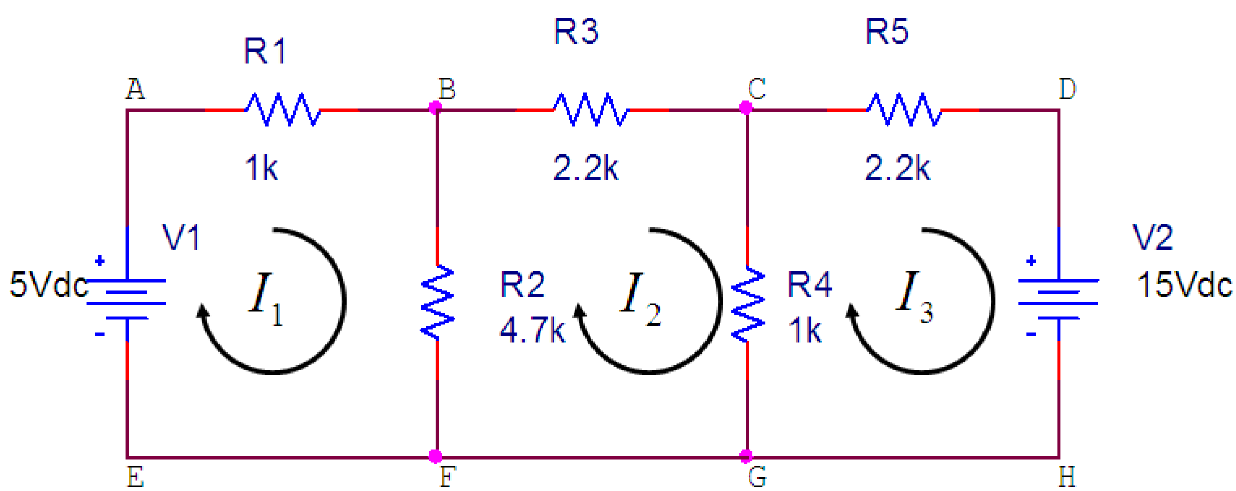


Figure 1. Circuit to be simulated

The circuit shown Figure 1 will be used in this lab to study the Node Voltage and Mesh Current analysis techniques. The definition of each mesh current is given in the figure. For future convenience, the node voltage will be represented as 'Vx', where 'x' represents one node from a list of nodes such as 'A, B, C, D, E, F, G, H'.

(1) Choose the ground node and write down two sets of symbolic equations leading to solve the circuit using Node voltage and Mesh current methods. Explain which method is preferred to use for calculations. (20 points)

(2) Use PSPICE to simulate the circuit show in Figure 1 using bias point mode. What are the values of the mesh currents I_1, I_2, and I_3. Use mesh current to calculate the branch currents of R_2 and R_4 and compare them with the directly calculated simulation results. In Schematic drawing program, turn on the "Bias voltage display" and "Bias current display" so that you can copy the circuit and the simulation results to your prelab report. (20 points)

(3) Replace the wire connecting nodes E and F with a resistor $R_6 = 1k\Omega$, choose the ground node and write down two sets of symbolic equations leading to solve the circuit using Node voltage and Mesh current methods, and explain which method is preferred in this case. (15 points)

(4) Use PSPICE to simulate the new circuit given in step (3). Use mesh current to calculate the branch current of R_2 and R_4 and compare it with the directly calculated simulation results. Copy the circuit and the simulation results to your prelab report. Compare the new node voltage with the node voltage obtained in step (2) and generate a table to summarize the results. (15 points)

(5) Based on the circuit described in step (3), further replace the wire connecting nodes G and H with a resistor $R_7 = 2.2k\Omega$, choose the ground node and write down two sets of symbolic equations leading to solve the circuit using Node voltage and Mesh current methods, and explain which method is preferred in this case. (15 points)

(6) Use PSPICE to simulate the new circuit given in step (5). Use mesh current to calculate the branch current of R_2 and R_4 and compare it with the directly calculated simulation results. Compare the new node voltage with the node voltage obtained in step (4) and generate a table to summarize the results. (15 points)

Laboratory Assignment

Problem 1

(1) Construct the circuit shown in Figure 1. Use DMM to measure the actual resistance and voltage output from the power supply and list them in your report. Measure the actual node voltages for all the nodes and generate a table to compare them with the simulation results. Based on the measured node voltages, calculate the mesh current

and the branch current of R_2 and R_4.g Generate a table to compare your results with the simulation results. (25 points)

(2) Construct the new circuit based on the description of step (3) in the pre-lab assignment. Include the measured resistance R_6 in your report. Measure the actual node voltages for all the nodes and generate a table to compare them with the simulation results. Based on the measured node voltage, calculate the mesh current and the branch current of R_2 and R_4. Generate a table to compare your results with the simulation results. (25 points)

(3) Construct the new circuit based on the description of step (5) in the pre-lab assignment. Include the measured resistance R_7 in your report. Measure the actual node voltages for all the nodes and generate a table to compare them with the simulation results. Based on the measured node voltage, calculate the mesh current and the branch current of R_2 and R_4. Generate a table to compare your results with the simulation results. (25 points)

(4) So far, all the mesh currents are measured indirectly. Actually, direct measurement of mesh current can be preformed as follows. First, change the DMM to “current measurement” mode. In the previous lab, you already did the measurement for the internal resistance of a DMM. Connect the DMM in series with the resistor of RC box. Replace R_5 by the connected DMM and RC box, adjust the resistance of RC box so that the sum of DMM resistance and RC box resistance equals to the resistance of R_5. Then, you can directly read the mesh current I_3 from DMM. Write down the reading of the current and compare it with the indirect measurement results. (25 points)

Lab 6: Superposition and Thevenin Equivalent

Objectives

In this lab, you will learn the superposition principle and Thevenin's theorem by physically constructing circuits and testing their behavior. You will gain an understanding of the power and simplicity from the superposition principle and Thevenin's theorem to electrical engineers.

Material:

DC Power Supply, Digital Multimeter, Breadboard, Resistors (10K, 22K, 4.7K, 1K, 2.2K, 800)

Reference

1. Electric Circuits, Nilsson & Riedel, 2008, Chapters 4.10-4.13
2. Appendix II: Introduction to PSPICE

Pre-Laboratory Assignment:

Problem 1 (30 points)

Build the circuit shown below in Figure 1 in PSPICE, and then answer the questions below.

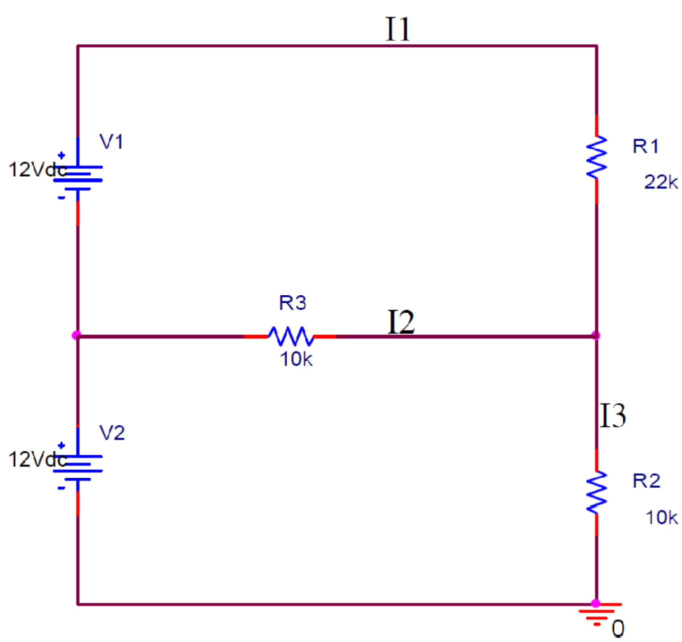


Figure 1. Circuit to be simulated

(1) Build the circuit shown in Figure 1 and simulate the circuit using 'Bias Point' Analysis. Show on the schematic the voltages and currents.

(2) Simulate the circuit from 0 to 10ms. Plot the current across R1, R2, and R3, and add the waveforms to your prelab report. (Refer *Reference* 2 for transient simulation technique using PSPICE)

(3) Calculate the currents I1 , I2 , and I3 shown in Figure 1, using either nodal or mesh analysis technique.

(4) Compare your calculated results of I1 , I2 , and I3 with the PSPICE results. Calculate the % error.

(5) Remove the lower voltage source and replace it with a short circuit and solve the currents in step (3)again and denote them as I1', I2', and I3'.

(6) Remove the upper voltage source and replace it with a short circuit and again solve the currents in step (3) and denote them as I1", I2", and I3".

(7) Combine (add) the solutions of steps (5) and (6) to find the currents I1 , I2 , and I3. Compare this result with the PSPICE results calculated in step (3). Calculate the % error.

Problem 2 (25 points)

Build the circuit shown below in Figure 2 in PSPICE, and then answer the questions below.

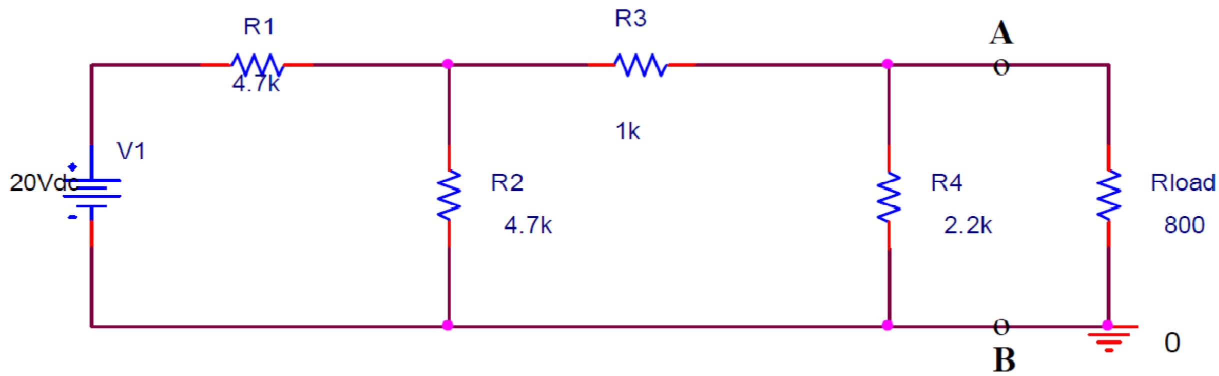


Figure 2: Thevenin Circuit Example

(1) Simulate the circuit shown in Figure 2 using PSPICE from 0 to 10ms, and plot the current and voltage across the 800 Ω load. Add the waveforms in your prelab report. (Hint – Remember the voltage, and current showed in different axis.)

(2) Calculate the voltage drop, and current across the 800 Ω load using either nodal or mesh analysis technique.

(3) Compare the results from steps (2) and (3). Find the % error.

Problem 3 (25 points)

Build the circuit shown in Figure 3 in PSPICE, and answer the questions below.

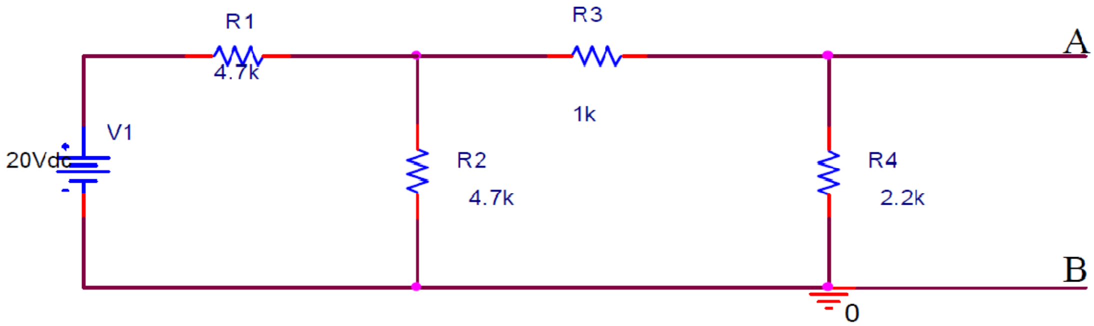


Figure 3: Circuit #3.

(1) Simulate the circuit shown in Figure 3 from 0 to 10ms. From the simulation, determine the Thevenin equivalent voltage and resistance with respect to terminals A and B. (Hint – Rerun the circuit with a modification to find the short circuit current.)

(2) Calculate the Thevinin's equivalent voltage and resistance with respect to terminals A and B

(3) Compare the results from steps (1) and (2). Find the % error.

Problem 4 (20 points)

Add the 800Ω load to your Thevenin equivalent circuit as shown below in Figure 4. Build the circuit in PSPICE, and answer the questions below.

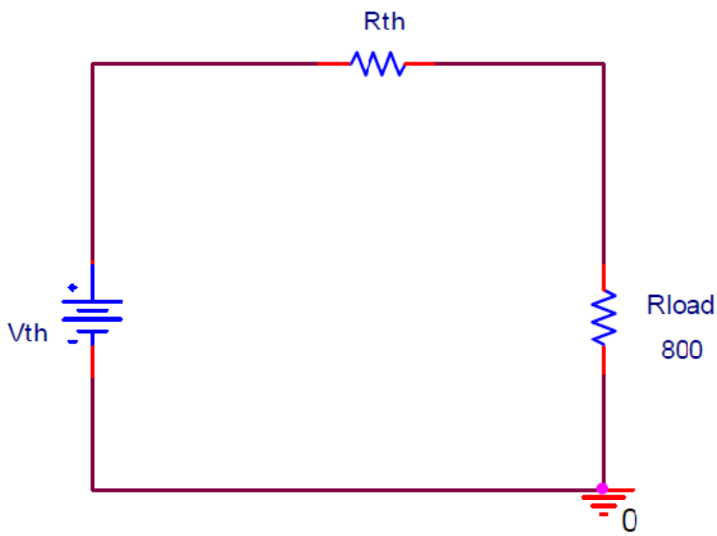


Figure 4: Circuit #4.

(1) Build the circuit shown in Figure 4 and simulate the circuit using 'Bias Point' Analysis. Show on the schematic the voltage and current across the 800Ω load.

(2) Calculate the voltage drop and current across the 800Ω load.

(3) Compare the results from (1) and (2). Find the % error.

Laboratory Assignment

Problem 1 (50 points)

For the illustration of superposition, simply go through the following steps at your lab station.

(1) Construct the circuit shown in Figure 1 of the Pre-Lab and measure the currents I1 , I2 , and I3 .

(2) Remove the upper voltage source. These can be done without destroying your current circuit setupBy simply turning your voltage supply off or setting it to 0 V. Again, re-measure the branch currents and label them as I1' ,I2' ,and I3'.

(3) Keep the upper voltage and remove the lower voltage source. Re-measure the branch currents and label them as I1", I2", and I3".

(4) Add the branch currents from steps (2) and (3). Does this equal the branch currents you measured in step (1) in this problem? Calculate the % error.

(5) Construct a table to easily compares all the branch currents, i.e., the PSPICE simulated values, your calculated values, and your measured values. Calculate a % error for your measured values and the calculated ones. Do your results validate the principle of superposition?

Problem 2 (50 points)

For the Thevenin Theory section of the lab, go through the following steps at your lab station.

(1) With the power supply turned off, construct the circuit from Figure 2. This circuit will be used to study Thevenin's theorem.

(2) With the output voltage of the power supply set to 20 volts, measure the open circuit voltage between terminals A and B with the DMM. Recall that the open circuit voltage is with the load removed. What is the significance of this voltage?

(3) The short circuit current (the load resistance goes to zero) between terminals A and B can be obtained by simply connecting an ammeter across terminals A-B. The positive

terminal of the ammeter should be connected to terminal A. Check the output voltage of the power supply and set it to 20V. Record the value of the short circuit current that exists between terminals A and B.

(4) On the basis of the data recorded in steps 2 and 3, derive the Thevenin equivalent circuit. Compare this experimentally derived circuit with the one derived by circuit analysis in the prelab.

Lab 7: Periodic Signals and Oscilloscope Measurement Techniques

Objectives

In this laboratory exercise, you will become familiar with the function of the Oscilloscope by practicing simple operational examples. You will gain an understanding of Oscilloscope fundamentals by some key menu operations highlighted as a solid starting point. You will also learn tooperate a Function Generator as a signal source, and learn to view it on the Oscilloscope. You will gain a deeper understanding of different types of periodic signals in terms of classification and properties and you will be able to produce a given periodic signal with a function generator and accurately measure the signal's properties with an Oscilloscope.

Materials

Oscilloscope, Function Generator, Breadboard, Resistors (1K, 47K, 10K, 22K), Capacitor (4.7μF)

Reference

1. Appendix II: Introduction to PSPICE
2. Appendix IV: Use Oscilliscope and Function generator
3. Appendix V: Periodical Signals and Oscilliscope Measurement Techniques

Pre-Laboratory Assignment

Problem 1

Please read *References* 2 and 3 carefully toobtain the knowledge using oscilliscope and function generator for periodical signal generation and measurement as preparation for implementing their operations step by step in the lab.

Problem 2

First, draw the circuit shown in Figure 1 in the schematic window. Note that a sinusoidal voltage source is used as the input. Please refer to *Reference* 1 regarding how to do a transient or AC analysis using PSPICE.

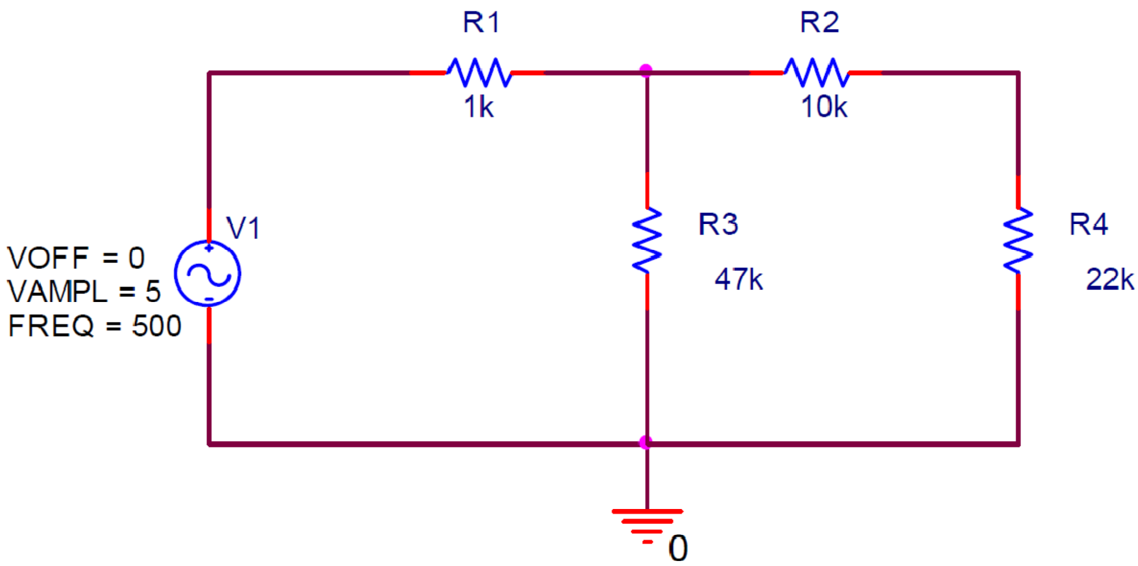


Figure 1: Assigned Circuit

After finishing the PSPICE simulation, complete the following tasks:

(1) Plot both the input voltage and the voltage across the 22kΩ resistor in one figure.

(2) Simulate the circuit in Figure 1 using a 500Hz square wave input signal. Plot both the input voltage and the voltage across the 22kΩ resistor in one figure. (Hint: Change the sinusoidal voltage source at the input to a square wave signal. The part name of the voltage source is 'Vpulse'. Set V1=0, V2=5, TD=0, TR=1ns, TF=1ns, PW=1ms, and PER=2ms.)

(3) Simulate the circuit in Figure 1 using a 500Hz triangular wave input signal. Plot both the input voltage and the voltage across the 22kΩ resistor in one figure. (Hint: Change the sinusoidal voltage source at the input to a triangular wave signal. The part name of the voltage source is 'Vpulse'. Set V1=0, V2=5, TD=0, TR=2ms, TF=1ns, PW=1ns, and PER=2ms.)

(4) Do the three figures of the input voltages and voltages across the 22kΩ resistor look as expected using different input signals? Explain Yes or No in details.

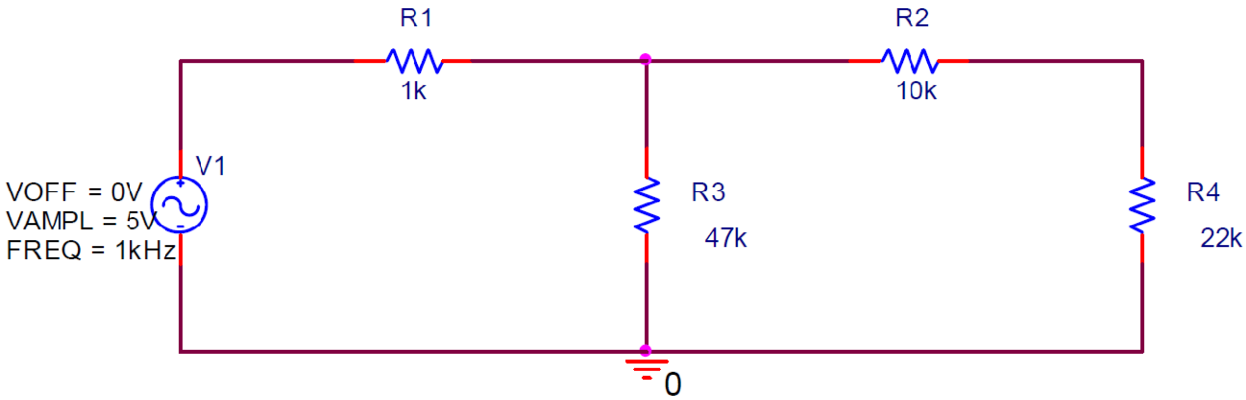


Figure 2: Assigned Circuit # 2

Problem 3

Change the frequency of V1 in Figure 1 to 1kHz as shown in Figure 2 and perform the following simulations:

(1) Simulate the circuit and plot only two periods of your waveform. Determine from the graph the peak-to-peak and 0-to-peak voltage across the 22kΩ resistance. Show the waveform marking the positive peak and period. (Hint – In the circuit schematic window, use voltage probe on top of the 22kΩ resistance. Then in PSPICE output waveform or PSPICE A/D window, select the 'trace'  option from the menus at the top. Then click on the button, , which moves the cursor to the positive peak of the waveform. Next, click on the button, , which marks the point of the peak. This shows both the x and y axis. Click on, , to move the cursor to the point that specifies one period, and then click on, , to mark the point of the peak)

(2) Measure the phase difference between the voltage waveform across the 22kΩ resistor with respect to the input voltage waveform. On the same graph, show both the input voltage waveform and the voltage waveform across the 22kΩ resistance (output).

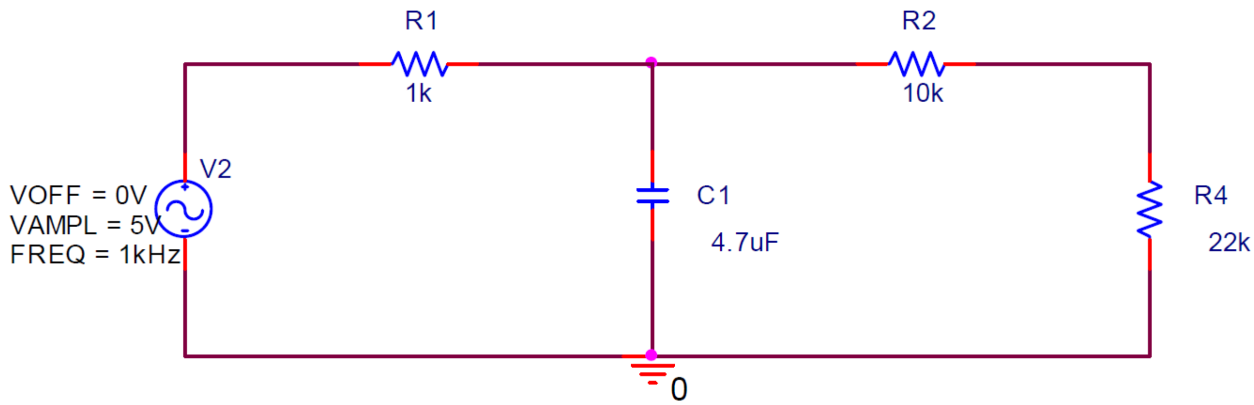


Figure 3: Assigned Circuit #3

Answer the next few questions based on the circuit shown in Figure 3. Notice that there is the addition of the capacitor in the circuit.

(1) Simulate the circuit and plot only two periods in your waveform. Determine from the graph the peak-to-peak and magnitude of the voltage across the 22kΩ resistor. Show the waveform while marking the positive peak and period.

(2) Measure the phase difference between the voltage waveform across the 22kΩ resistor with respect to the input voltage waveform. On the same graph, show both the input voltage waveform and the voltage waveform across the 22kΩ resistance (output).

Laboratory Assignment

Problem 1

Please go through step-by-step instructions in *References*. 2 and 3 and make sure that you can answer all the questions listed in the instructions. Talk with TA first to get permission and then move forward to work on the following Problems 2 and 3.

Problem 2 (50 points)

(1) Build the circuit shown in Figure 1 on a breadboard in the lab.

(2) Apply a 500Hz sinusoidal signal with 5 Vp-p magnitude to the input of the circuit. Obtain the waveform of the input signal and the voltage across the 22kΩ resistor using the scope. Save the waveform and turn it in as a part of your lab report. Please compare this result with your PSPICE simulation obtained in the pre-lab.

(3) Apply a 500Hz square signal with 5 Vp-p magnitude to the input of the circuit. Obtain the waveform of the input signal and the voltage across the 22kΩ resistor using the scope. Save the waveform and turn it in as part of your lab report. Please compare this result with your PSPICE simulation obtained in the pre-lab.

(4) Apply a 500Hz triangular signal with 5 Vp-p magnitude to the input of the circuit. Obtain the waveform of the input signal and the voltage across the 22kΩ resistor using the scope. Save the waveform and turn it in as a part of your lab report. Please compare this result with your PSPICE simulation obtained in the pre-lab.

(5) Summarize whether your actual waveforms look very close to the simulated waveforms generated from PSPICE.

Problem 3 (50 points)

(1) Build the circuit shown in Figure 2 on a breadboard in the lab.

(2) Apply a 1000Hz sinusoidal signal with 5Vp-p magnitude to the input of the circuit. Obtain two periods of the input waveform and the voltage waveform across the 22kΩ resistor using the scope. Also using the scope, measure the Vp-p, V0-p, and frequency of both waveforms. Again using the scope, measure the phase difference. Save the waveforms and turn them in as a part of your lab report. Please compare this result with your PSPICE simulation obtained in the pre-lab. (A table would be strongly recommended to show the comparision result here.)

(3) Build the circuit shown in Figure 3 on a breadboard in the lab.

(4) Apply a 1000Hz sinusoidal signal with 5Vp-p magnitude to the input of the circuit. Obtain two periods of the input waveform and the voltage waveform across the 22kΩ resistor using the scope. Also using the scope, measure the Vp-p, V0-p, and frequency of both waveforms. Again using the scope, measure the phase difference. Save the waveforms and turn them in as a part of your lab report. Please compare this result with your PSPICE simulation obtained in the pre-lab as well as your theoretical calculation of phase. (A table would be strongly recommended to show the comparison result here.)

(5) What does the minus (-) sign mean in the phase?

(6) What do you think causing the phase difference in Circuit #3 shown in Figure 3?

Lab 8: Operational Amplifier

Objectives

In this lab, you will practice building real operational amplifier components as such inverting, summing, and non-inverting circuits. You will also practice using the knowledge learned in Circuit I to analyze the circuits and making comparison with the PSPICE simulation results.

Materials:

Oscilloscope, Function Generator, Breadboard, Resistors (1K, 4.7k, 10K, 22K, 47K), RC box, µA741 Op AMP

Reference

1. Electric Circuits, Nilsson & Riedel, 2008, Chapter 5.1-5.5

Pre-Laboratory Assignment

Problem 1 (20 points)

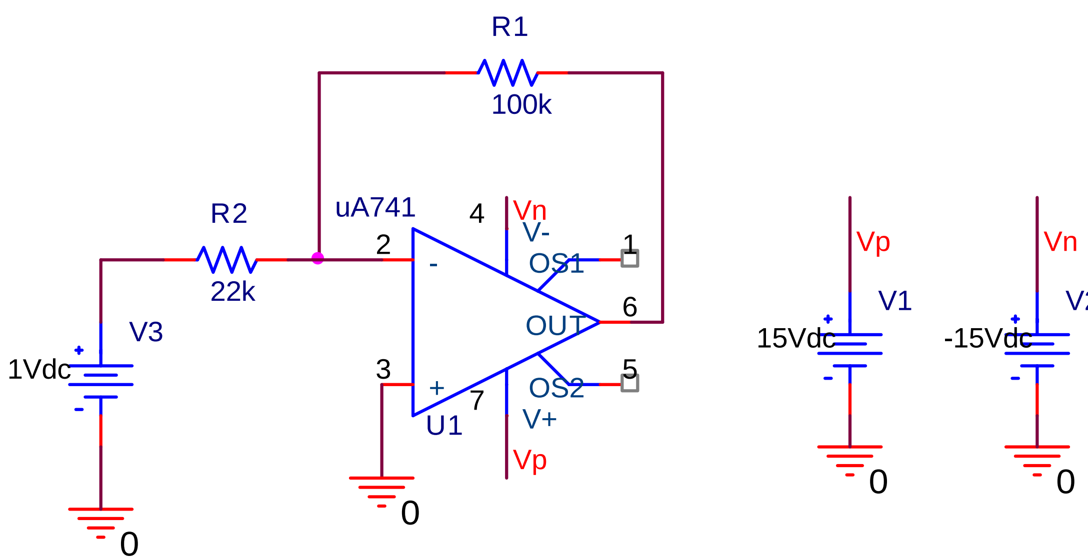


Figure 1. Circuit #1

First, construct the circuit shown in Figure 1 in PSPICE. The operational amplifier U1 in the PSPICE simulation is uA741, a very commonly used operational amplifier in the market. In Figure 1, the voltage sources (V1 and V2) are not physiclly shown connected to operational amplifier U1's positive/negative power supply terminals. However, since two "net" labels "Vp" and "Vn" are used, the "15Vdc" is indeed connected with U1's positive power supply terminal and the "-15Vdc" is indeed connected with U1's negative power supply terminal. This way of drawing makes the schematic easy to read. Then, do the tasks below:

(1) Calculate the output voltage for the six values of V3: 0.4, 2.0, 3.5, -0.6, -1.6, and -2.4V.

(2) Perform the bias simulation for the six values of V3 in step (1) and compare the results.

(3) What is the voltage of inverting input for the values of V3 in step (1) if the operational amplifier is ideal? What are the simulation results? Can you estimate the gain of the operational amplifier based on the simulation results?

(4) Specify the range of V3 required to aviod amplifier saturation. Perform the simulation and find the range and compare that range with your theoritical calculation.

Problem 2 (20 points)

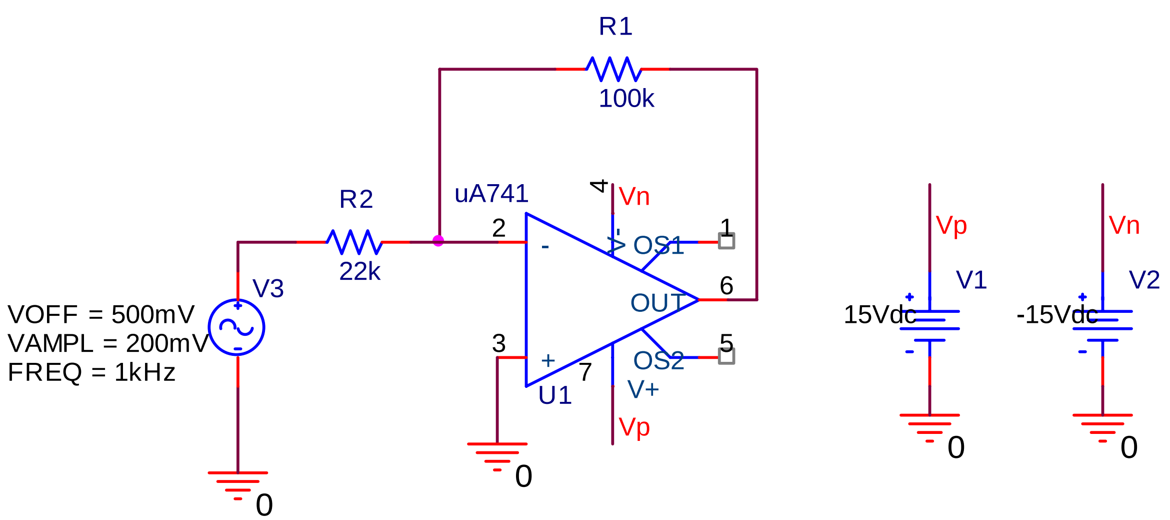


Figure 2. Circuit #2

First, change the DC source shown in Figure 1 to an AC source as shown in Figure 2 and perform the transient analysis for 10ms. Then, do the tasks below:

(1) Plot the output and the input in the same drawing and explain your simlulation results based on your understanding of OP.

(2) Change the VOFF to -500mV and redo the simultion. Plot the output and the input in the same drawing. Explain your simulation results based on your understanding of OP.

(3) Change VOFF=2V and VAMP=1V and redo the simulation. Plot the output and the input in the same drawing. Explain your simulation results .

Problem 3 (20 points)

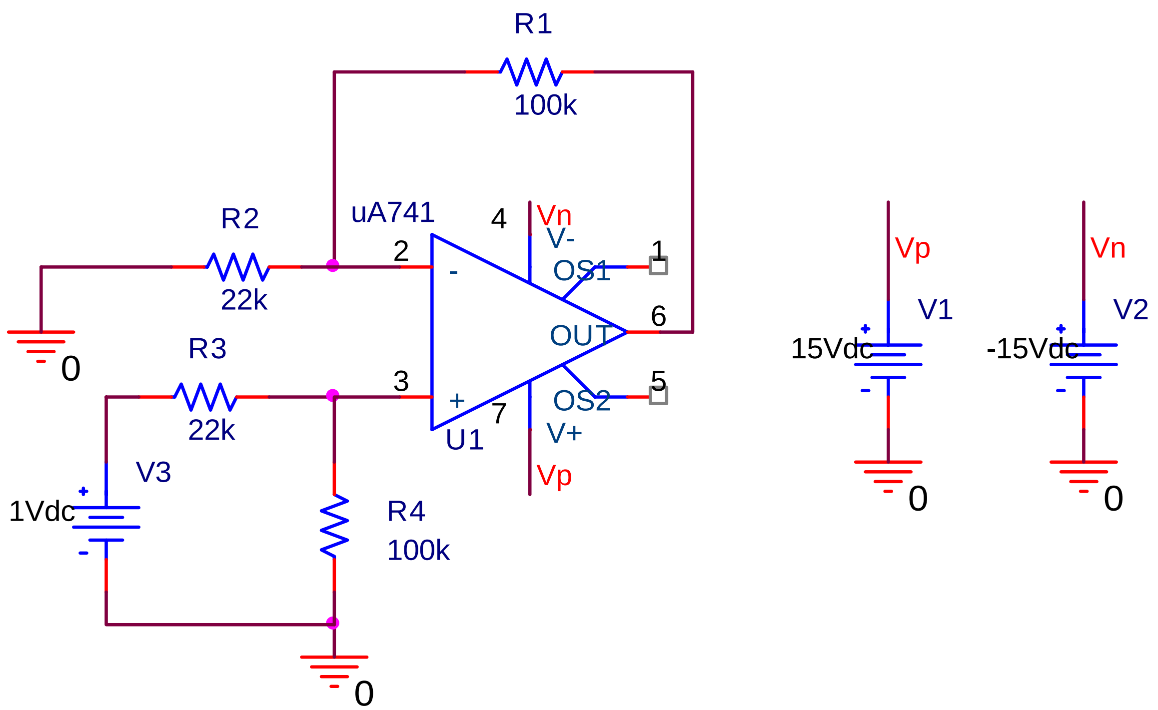


Figure 3． Circuit #2First, construct the circuit shown in Figure 3 in PSPICE. Then do the tasks below:

(1) Calculate the output voltage for the six values of V3: 0.4, 2.0, 3.5, -0.6, -1.6, and -2.4V.

(2) Perform the bias simulation for the six values of V3 in step (1) and compare the results.

(3) What is the voltage of inverting input for the values of V3 in step (1) if the operational amplifier is ideal? What are the simulation results?

(4) Specify the range of V3 required to aviod amplifier saturation. Perform the simulation and find the range and compare that range with your theoretical calculation.

Problem 4 (20 points)

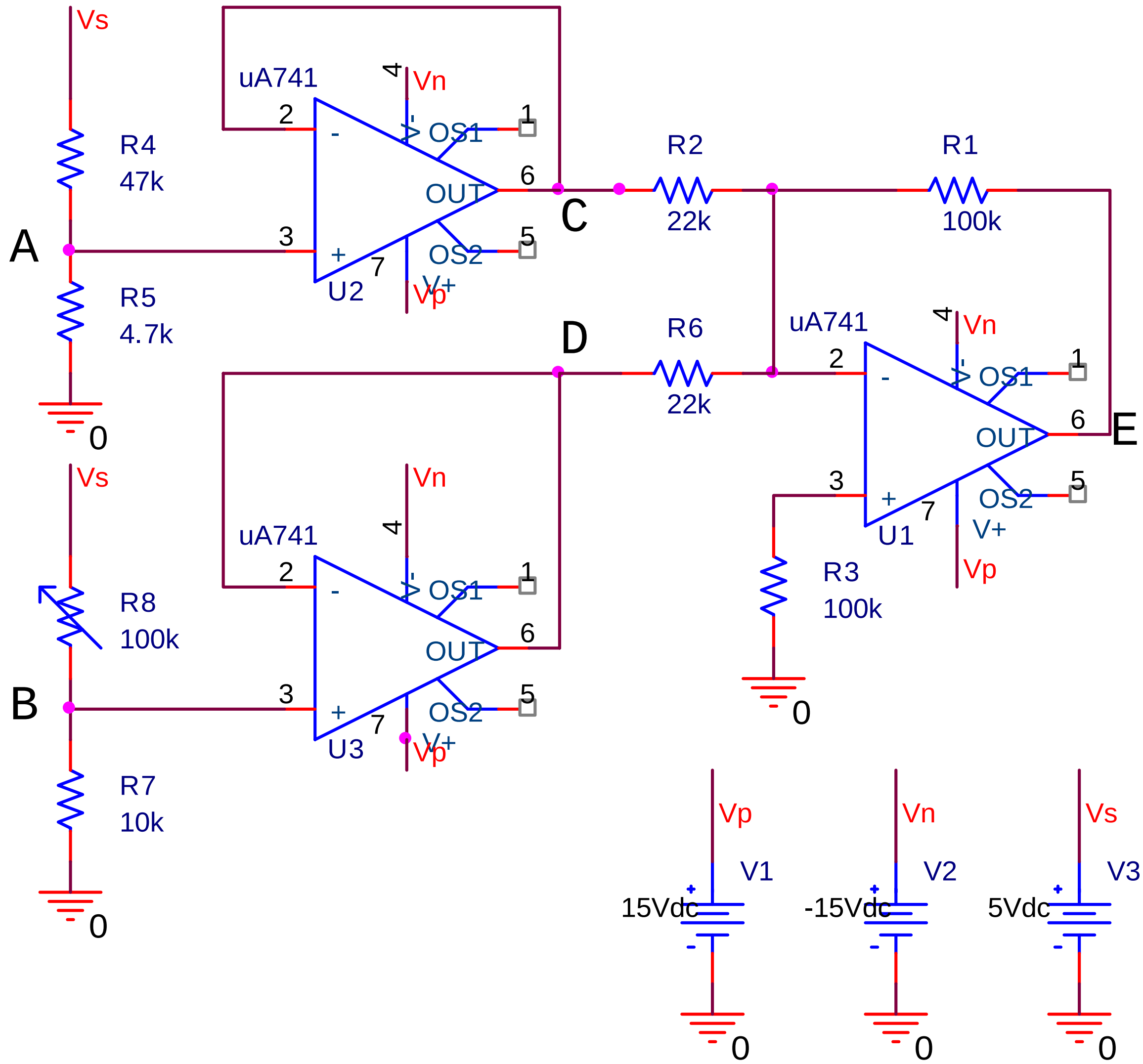


Figure 4: Circuit #4

First, construct the circuit shown in Figure 4 in PSPICE. Use a fixed value of resistor to replace the variable resistor R8 for your following simulation otherwise the resistance of R8 used in the simulation will be only 50% as shown in the drawing. Then do the following tasks:

(1) Calculate the voltage output at Node E for the five values of R8: 20k, 40k, 60k, 80k, 100k. What is the function of operational amplifers U2 and U3?

(2) Perform the simulation and list the output at Node E for the five values of R8: 20k, 40k, 60k, 80k, 100k. Are the voltages of Node D and B always same?

(3) What is the function of this circuit?

Problem 5 (20 points)

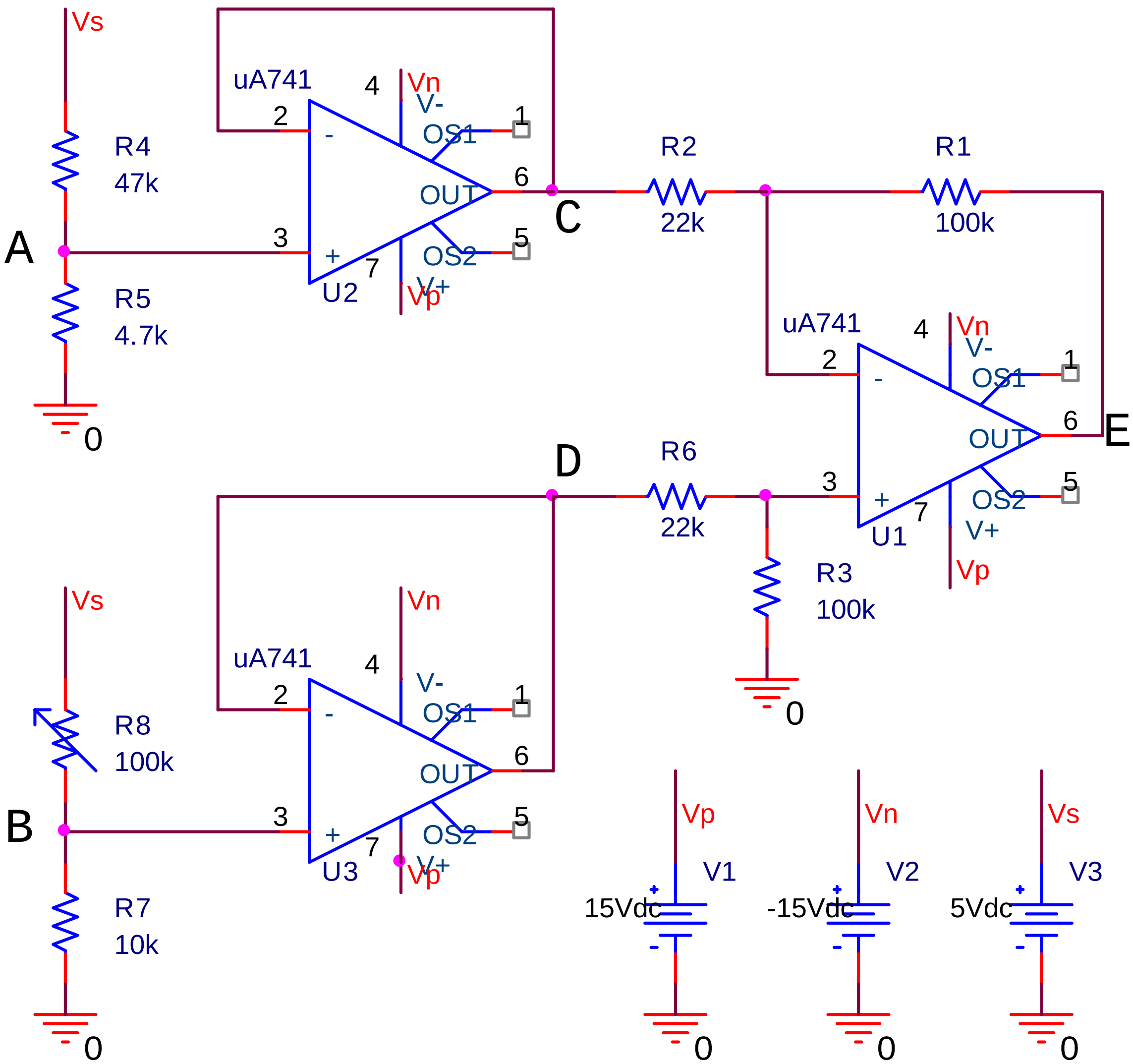


Figure 5: Circuit #5

First, construct the circuit shown in Figure 5 in PSPICE. Then, do the following tasks:

(1) Calculate the voltage output at Node E for the five values of R8: 20k, 40k, 60k, 80k, 100k. What is the function of operational amplifers U2 and U3?

(2) Perform the simulation and list the output at Node E for the five values of R8: 20k, 40k, 60k, 80k, 100k. Are the voltages of Node D and B always same?

(3) What is the function of this circuit?

Please make sure that you include all the schematic drawings of the circuits in your prelab report.

Laboratory Assignment

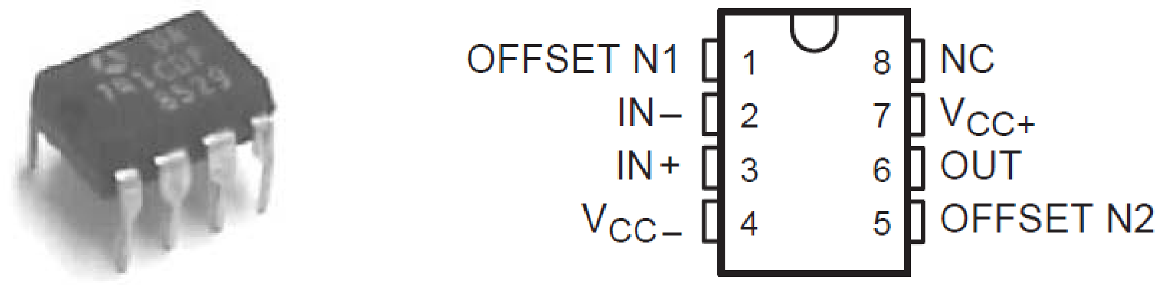


Figure 6: A picture of μA741 operational amplifier (Left) and schematic of top view for pin assignment (Right)

In this lab, we will be using **μA741 operational amplifier** with 8 pin dual inline packaging (DIP). The picture and the pin assignment are shown in Figure 6. We will only use pin 2, 3, 4, 6, and 7. Make sure that you connect the pins properly before you conduct any measurement.

Problem 1 (20 points)

Construct the circuit shown in Figure 1 in the lab. Then do the tasks below:

(1) Use DMM to measure the output voltage for the six values of V3: 0.4, 2.0, 3.5, -0.6, -1.6, and -2.4V and compare the measurements with your simulation results.

(2) Use DMM to measure the voltage of inverting input for the six values of V3 in step (1) and compare the measurements with your simulation results. What conclusion can you get?

(3) Change the voltage of V3 and simutaneously measure the output to determine the voltage range without amplifier saturation. Compare your results with your simulation results.

Problem 2 (20 points)

Change the DC source shown in Figure 1 to an AC source as shown in Figure 2 in the lab. The do the followling tasks:

(1) Use oscilliscope to measure both source and operational amplifier output by two channels. Adjust the setting to show two periods of waveform. Save the waveform and include it in your lab report. Explain your results based on your understanding of OP.

(2) Change the VOFF to -500mV and redo the measurement. Save the waveform and include it in your lab report. Explain your results based on your understanding of OP.

(3) Change VOFF=2V and VAMP=1V and redo the measurement. Save the waveform and include it in your lab report. Explain your measurement results?

Problem 3 (20 points)

Construct the circuit shown in Figure 3 in the lab. Then do the following tasks:

(1) Use DMM to measure the output voltage for the six values of V3: 0.4, 2.0, 3.5, -0.6, -1.6, and -2.4V and compare the measurments with your simulation results.

(2) Use DMM to measure the voltage of inverting input for the six values of V3 in step (1) and compare the measurements with your simulation results. What conclusion can you get?

(3) Change the voltage of V3 and simutaneously measure the output to determine the voltage range without amplifier saturation. Compare your results with your simulation results.

Problem 4 (20 points)

Construct the circuit shown in Figure 4 using RC box resistor as R8 in the lab. Then do the following tasks:

(1) Measur the voltage at nodes A, B, C, D, E for the five values of R8: 20k, 40k, 60k, 80k, 100k and list the results in a table.

(2) Compare the measured voltage of node E with the simulated voltage of node E and calculate the percent error.

Problem 5 (20 points)

Construct the circuit shown in Figure 5 using RC box resistor as R8. Then do the following tasks:

(1) Measur the voltage at nodes A, B, C, D, E for the five values of R8: 20k, 40k, 60k, 80k, 100k and list the results in a table.

(2) Compare the measured voltage of node E with the simulated voltage of node E and calculate the percent error.

Lab 9: Capacitors, Inductors and Response of First-Order RL and RC Circuits

Objectives

In this lab, you will gain an understanding of the operation and equations that govern the behavior of capacitors and inductors. You will also construct some simple circuits using capacitors and inductors to view their operations.

Materials

Oscilloscope, Function Generator, Breadboard, RC Box, Inductor (47mH), Resistor (470)

Reference

1. Electric Circuits, Nilsson & Riedel, 2008, Chapters 6.1-6.2
2. Electric Circuits, Nilsson & Riedel, 2008, Chapters 7.1-7.4

Pre-Laboratory Assignment

Problem 1 (20 points)

Build the circuit below shown in Figure 1 in PSPICE, and then complete the subsequent steps.

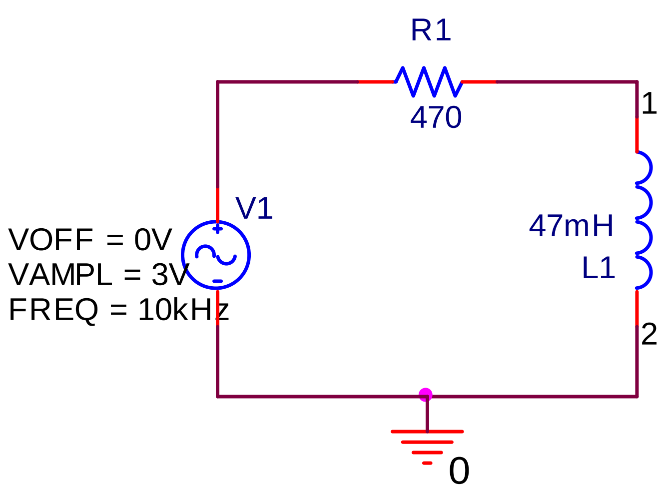


Figure 1: Circuit #1

(1) Simulate the above circuit in PSPICE from 0 to 2 ms. (It takes 0.5 ms to stablize signals). Determine from the graph the voltage drop across the output inductor (L_1). **Save the waveforms and add it to your prelab report**.

(2) Assuming the waveform of the voltage drop across L_1 is $v_L = V_{Lp}\cos\omega t$, determine the amplitude (V_{Lp}, in 'Volt') and the radian frequency (ω, in 'rad/sec') from the waveform.

(3) Plot the waveform v_R for the voltage across R_1 in PSPICE. **Save the waveform and add it to your prelab report**. (Hint – To measure the voltage drop across the R_1, you need to add a differential probe. The differential probe can be found near the top of the schematic window next to the voltage probe radio button. Its symbol is ''. You need select the differential probe radio button and differential probe across the resistance as shown in Figure 2 below:

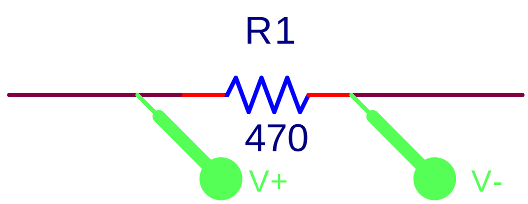


Figure 2: Differential Probe

(4) Assuming the waveform of the voltage drop across R_1 is $v_R = V_{Rp}\sin\omega t$, determine the amplitude (V_{Rp}, in 'Volt') and the radian frequency (ω, in 'rad/sec') from the waveform.

(5) Apply equation $v_L = L_1 \frac{d(V_{Rp}\sin\omega t)}{dt}$ to find the calculated V_{Lp}, and then compare it with PSPICE simulated V_{Lp} found in step (2) and compute the percent error.

Problem 2 (20 points)

Build the circuit below shown in Figure 3 in PSPICE, and then complete the subsequent steps.

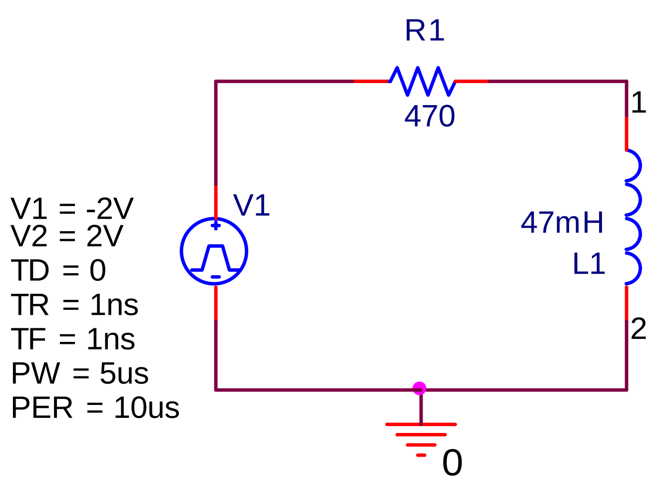


Figure 3: Circuit #2

(1) Simulate the above circuit in PSPICE from 0 to 3 ms. (It takes 0.5 ms to stablize signals.) The section of the waveform from 2-3 ms will be similar to the waveform seen from the scope. Zoom in this section to show one period of waveforms of v_R and v_L. Save the waveforms and add them to your prelab report.

(2) What are dv_R (the "rise") and dt (the "run") for the voltage across R_1 in one period of waveform?

(3) Calculate $\frac{L_1}{R_1} \cdot \frac{dv_R}{dt}$, compare it with the peak voltage V_{Lp} of the waveform for the voltage v_L across L_1, and then find the percent error.

Problem 3 (20 points)

Build the circuit below shown in Figure 4 in PSPICE, and then complete the subsequent steps.

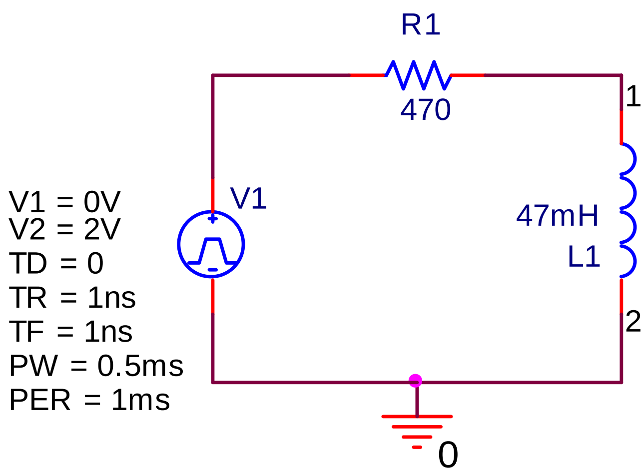


Figure 4: Circuit #3

(1) Simulate the above circuit in PSPICE from 0-3 ms and save the waveform of v_L and add it to your prelab report.

(2) The inductor and resistor in Figure 4 forms one first-order RL circuit.What is the time constant τ for this RL circuit?

(3) Consider one period of stimulus signal shown in Figure 4. When voltage rises from 0V to 2V and it holds more than 5 times of τ, the waveform of v_L can be treated as step response. Calculate the step response of the circuit shown in Figure 4 and compare it with the simulated waveform from 0-0.5 ms. What is the initial voltage and what is the

final voltage? Measure the time constant from the PSPICE simulation result and compare it with the calculated result in step (2)

(4) When the voltage drops from 2V to 0V, the waveform of v_L can be treated as the natural response with the initial condition determined by the end of waveform obtained in step (3). Calculate the natural response of the circuit and compare it with the simulated waveform from 0.5-1.0 ms. What is the initial voltage and what is the final voltage used in your calculation?

Problem 4 (20 points)

Build the circuit below shown in Figure 5 in PSPICE, and then complete the subsequent steps.

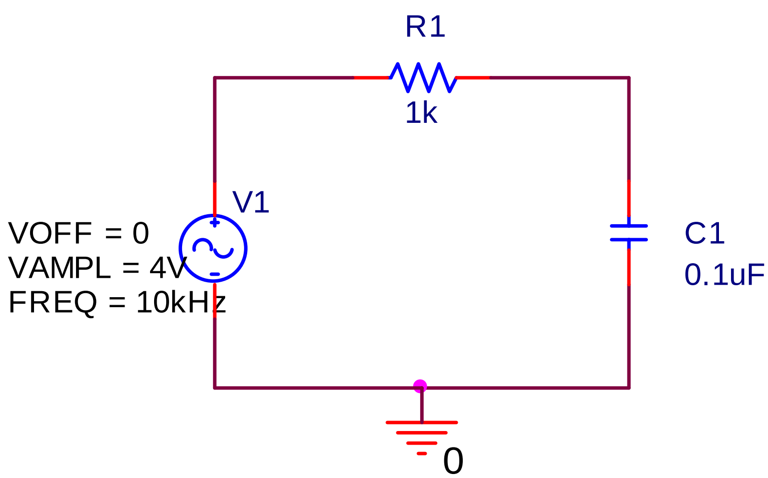


Figure 5: Circuit #4

(1) Simulate the above circuit in PSPICE from 0 to 2 ms. Determine from the graph the voltage drop across the output capacitor (C_1). **Save the waveforms and add it to your prelab report**.

(2) Assuming the waveform of the voltage drop across C_1 is $v_C = V_{Cp}\sin\omega t$, determine the amplitute (V_{Cp}, in 'Volt') and the radian frequency (ω, in 'rad/sec') from the waveform.

(3) Plot the waveform v_R for the voltage across R_1 in PSPICE. **Save the waveform and add it to your prelab report**.

(4) Assuming the waveform of the voltage drop across R_1 is $v_R = V_{Rp}\cos\omega t$, determine the amplitute (V_{Rp}, in 'Volt') and the radian frequency (ω, in 'rad/sec') from the waveform.

(5) Calculate $C\frac{dv_C}{dt}$, compare the calcuated amplitute with $\frac{V_{Rp}}{R_1}$, and give the percent error.

Problem 5 (20 points)

Build the circuit below shown in Figure 6 in PSPICE, and then complete the subsequent steps.

(1) Simulate the circuit in PSPICE from 0 to 3 ms. (It takes 0.5 ms to stablize signals.) The section of the waveform from 2-3 ms will be similar to the waveform seen from the scope. Zoom in this section to show one period of waveforms of v_R and v_C. Save the waveforms and add them to your prelab report.

(2) What are dv_C (the "rise") and dt (the "run") for the voltage across C_1 in one period of waveform?

(3) Calculate $C_1 \cdot \frac{dv_R}{dt}$ and compare it with the peak current calculated by $\frac{V_{Rp}}{R_1}$ of the waveform for the voltage v_R across R_1 and find the percent error.

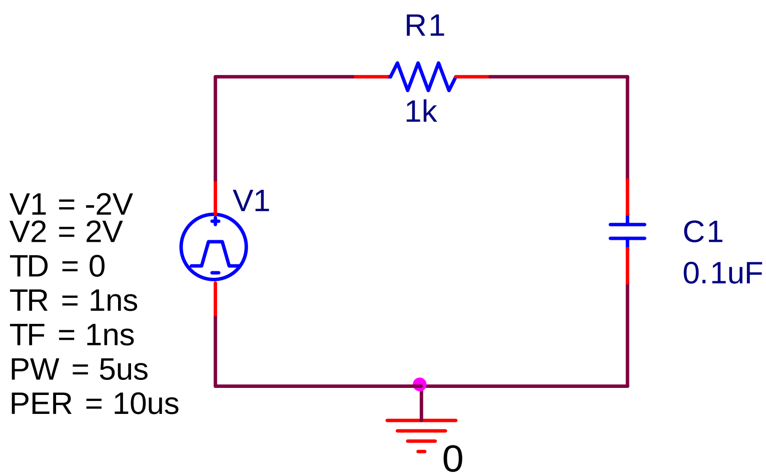


Figure 6: Circuit #5

Problem 6 (20 points)

Build the circuit below shown in Figure 7 in PSPICE, and then complete the subsequent steps.

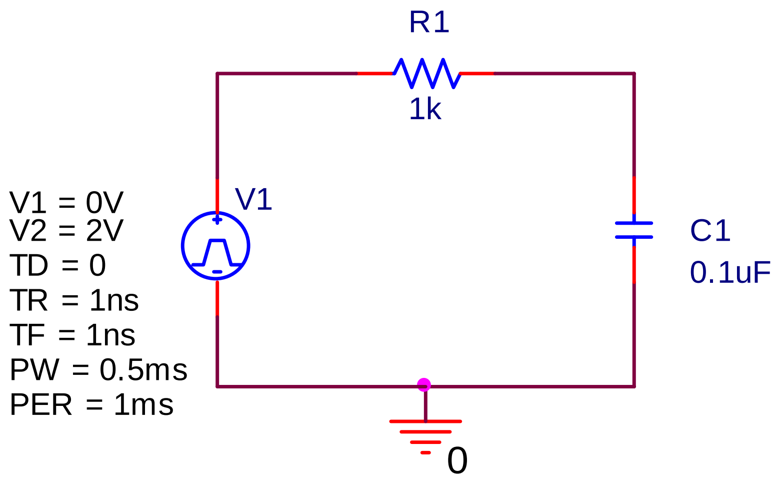


Figure 7: Circuit #6

(1) Simulate the above circuit in PSPICE from 0-3 ms, save the waveform of v_C, and add it to your prelab report.

(2) The capacitor and resistor in Figure 7 forms one first-order RC circuit. What is the time constant τ for this RL circuit?

(3) Consider one period of stimulus signal shown in Figure 7. When the voltage rises from 0V to 2V and it holds more than 5 times of τ, the waveform of v_C can be treated as step response. Calculate the step response of the circuit shown in Figure 7 and compare it with the simulated waveform from 0-0.5 ms. What is the initial voltage and what is the final voltage? Measure the time constant from the PSPICE simulation result and compare it with the calculated result in step (2)

(4) When the voltage V1 drops from 2V to 0V, the waveform of v_C can be treated as the natural response with the initial condition determined by the end of waveform obtained in 3). Calculate the natural response of the circuit and compare it with the simulated waveform from 0.5-1.0 ms. What is the initial voltage and what is the final voltage used in your calculation?

Laboratory Assignment

Problem 1 (20 points)

Construct the circuit, as seen from Figure 1 in the prelab and set the signal generator properly. Then do the following tasks:

(1) Measure $v_L = V_{Lp}\cos\omega t$ to determine the amplitude (V_{Lp}, in 'Volt') and the radian frequency (ω, in 'rad/sec') from the waveform. Print v_L and add it to your lab report.

(2) Measure $v_R = V_{Rp}\sin\omega t$, to determine the amplitude (V_{Rp}, in 'Volt'). Print v_R and add it to your lab report.

(3) Apply equation $v_L = L_1 \frac{d(V_{Rp}\sin\omega t)}{dt}$ to find the calculated V_{Lp}, and then compare it with measured V_{Lp} in step (2) and give the percent error

Problem 2 (20 points)

Use the circuit constructed in problem 1 and change the stimulation source to the one shown in Figure 3. Then do the following tasks:

(1) Adjust the scope so that only one period of v_R is shown and save it for your report.

(2) The waveform of v_R should look like a triangle wave. Adjust the vertical sensitivity of the oscilloscope so that the triangle wave is as large as possible but still completely on the screen. The definitations of dv_R (the "rise") and dt (the "run") for the voltage across R_1 are shown in Figure 8. Find dv_R and dt.

(3) Measure the waveform for the voltage v_L across L_1 and save it for your report.

(4) Based on the measurement results, calculate $\frac{L_1}{R_1} \cdot \frac{dv_R}{dt}$, compare it with the peak voltage V_{Lp} of the waveform for the voltage v_L across L_1 and find the percent error.

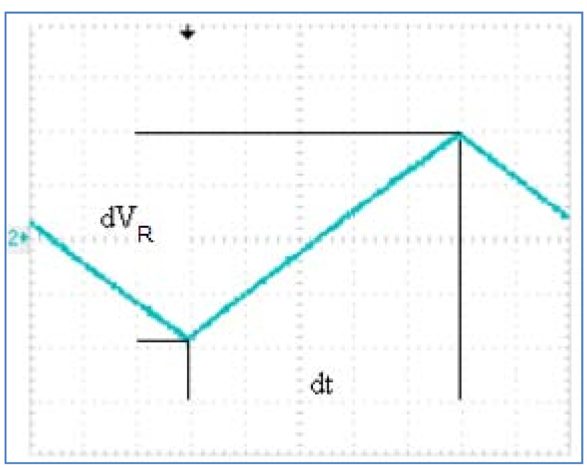


Figure 8: Schematic of waveform to find 'Rise/Run' of a resistor

Problem 3 (20 points)

Use the circuit constructed in problem 1 and change the stimulation source to the one shown in Figure 4. Then do the following tasks:

(1) Measure the waveform of v_L and add it for your report.

(2) Write down the expressions of step response and natural response based on your measured waveform. Mark down the parameters extracted from the waveform for the expressions and explain briefly how you get the results.

Problem 4 (20 points)

Construct the circuit, as seen from Figure 5 in the prelab and set the signal generator properly. The do the following tasks:

(1) Measure $v_C = V_{Cp}\sin\omega t$, and determine the amplitute (V_{Cp}, in 'Volt') and the radian frequency (ω, in 'rad/sec') from the waveform. Print v_C and add it to your lab report.

(2) Measure $v_R = V_{Rp}\cos\omega t$, and determine the amplitude (V_{Rp}, in 'Volt'). Print v_R and add it to your lab report.

(3) Calculate $C\frac{dv_C}{dt}$ and compare the calcuated amplitute with $\frac{V_{Rp}}{R_1}$ and give the percent error.

Problem 5 (20 points)

Use the circuit constructed in problem 4 and change the stimulation source to the one shown in Figure 6. Then do the following tasks:

(1) Adjust the scope so that only one period of v_C is shown and save it for your report.

(2) The waveform of v_R should look like a triangle wave. Adjust the vertical sensitivity of the oscilloscope so that the triangle wave is as large as possible but still complete on the screen. Find dv_C and dt.

(3) Measure the waveform for the voltage v_R across R_1 and save it for your report.

(4) Based on the measurement results, Calculate $C_1 \cdot \frac{dv_R}{dt}$, compare it with the peak current calculated by $\frac{V_{Rp}}{R_1}$ of the waveform for the voltage v_R across R_1, and find the percent error.

Problem 6 (20 points)

Use the circuit constructed in problem 4 and change the stimulation source to the one shown in Figure 7.

(1) Measure the waveform of v_C and save it for your report.

(2) Write down the expressions of step response and natural response based on your measured waveform. Mark down the parameters extracted from the waveform for the expressions and explain briefly how you get the results.

Lab 10: Natural and Step Response of RLC Circuits

Objectives

In this lab, you will gain an understanding of the natural response and step response of parallel RLC circuits and series RLC circuits. You will also practice designing and constructing circuits to observe the response for over damped, under damped, and critically damped conditions.

Materials

Oscilloscope, Function Generator, Breadboard, RC Box, Inductor (10mH, 47mH), resistors and capacitors

Reference

1. Electric Circuits, Nilsson & Riedel, 2008, Chapters 8.1-8.4

Pre-Laboratory Assignment:

Problem 1 (75 points)

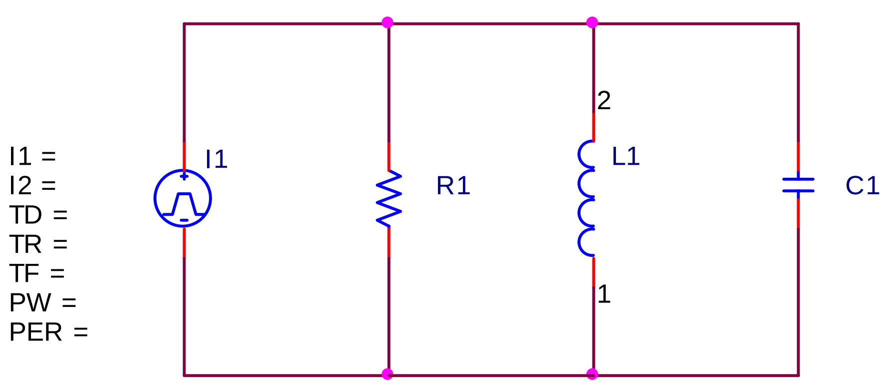


Figure 1

The circuit shown in Figure 1 will be used to study the natural and response of a parallel RLC circuit. In this lab, you will need to select the components with proper values so that response for over damped, under damped, and critically damped can all be observed.

(1) Assume that the available inductors are 10 mH and 47 mH, the available capacitors are 0.047 μF, 0.1 μF, 0.22 μF, 0.47 μF, 1 μF, and the resistance can be varied from 1 Ω to 1

MΩ by using the RC box, find the resistance to get circuit critically damped and finish Table 1. (**Hint**: Neper frequency $\alpha = 1/2RC$ and Resonant radian frequency $\omega_0 = 1/\sqrt{LC}$. For critically damped, $\alpha = \omega_0$. Therefore, the critically damped condition is $R_0 = \sqrt{L/4C}$. When R is larger than R_0, the circuit is under damped while when R is smaller than R_0, the circuit is over damped.)

Table 1

L=10 mH				
C (μF)	R_0(Ω)	α(rad/s)	$1/\alpha$ (s)	$\alpha/10$(Hz)
0.047				
0.1				
0.22				
0.47				
1				
L=47 mH				
C (μF)	R_0(Ω)	α(rad/s)	$1/\alpha$ (s)	$\alpha/10$(Hz)
0.047				
0.1				
0.22				
0.47				
1				

Please note that the role of $1/\alpha$ in a RLC circuit is similar to τ(time constant) in a RL or RC circuit. When a 50% duty cycle square wave is used as the stimulus, the pulse width has to be longer than five times of α to stabilize the signal. The item '$\alpha/10$' in Table 1 gives a rough estimation of selecting the frequency of stimulus signal.

(2) Based on the calculation results in Table 1, choose R, L, and C with appropriate values so that the circuit is over damped.

R=___________,   L=___________,   C=___________

Calculate $\alpha, \omega_0, s_1, s_2$ and finish Table 2.

Table 2

$\alpha = 1/2RC$	$\omega_0 = 1/\sqrt{LC}$	$s_1 = -\alpha + \sqrt{\alpha^2 - \omega_0^2}$	$s_2 = -\alpha - \sqrt{\alpha^2 - \omega_0^2}$

The frequency of stimulus signal should be less than $-s_1/10$.

Choose your stimulus signal frequency f=________________

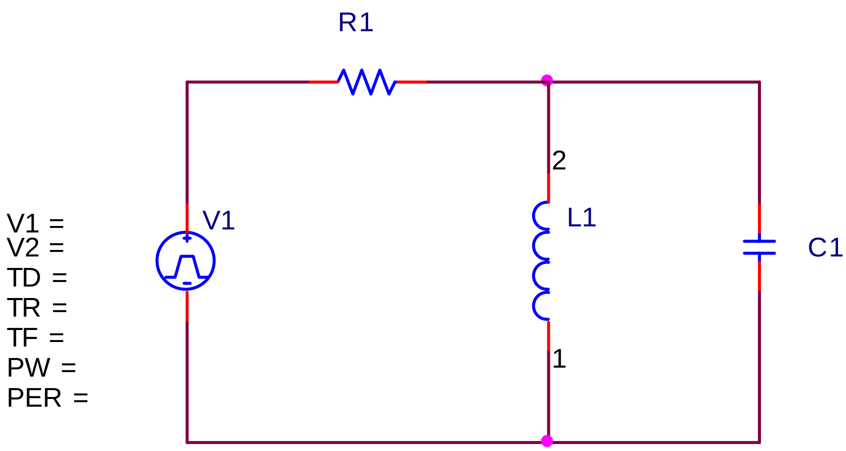


Figure 2

Since the signal generator is a voltage source, we will simulate the circuit shown in Figure 2 for the convenience so that we can compare the simulation results and experimental results. (The voltage source and the resistor in series are equivalent to a current source and a resistor in parallel by using source transformation). Use **V1=0V**, **V2=2V**, **TD=0**, **TR=1ns**, **TF=1ns**, choose your own PW and PER based on the predetermined f, use the R, L, and C values chosen for over damped to perform PSPICE simulation.

Measure the waveform across R1 and save it for your prelab report. How do you interpret the waveform using the concepts of step response and natural response?

(3) Based on the calculation results in Table 1, choose R, L, and C with appropriate values so that the circuit is under damped.

R=___________,   L=___________,   C=___________

Calculate $\alpha, \omega_0, s_1, s_2$ and finish Table 3.

Table 3

$\alpha = 1/2RC$	$\omega_0 = 1/\sqrt{LC}$	$s_1 = -\alpha + \sqrt{\alpha^2 - \omega_0^2}$	$s_2 = -\alpha - \sqrt{\alpha^2 - \omega_0^2}$

The frequency of stimulus signal should be less than $\alpha/10$.

Choose your stimulus signal frequency f=________________

Use **V1=0V**, **V2=2V**, **TD=0**, **TR=1ns**, **TF=1ns**, choose your own PW and PER based on the predetermined f to perform PSPICE simulation, use the R, L, and C values chosen for under damped to perform PSPICE simulation for the circuit shown in Figure 2.

Measure the waveform across R1 and save it for your prelab report. How do you interpret the waveform using the concepts of step response and natural response?

(4) Based on the calculation results in Table 1, choose R, L, and C with appropriate values so that the circuit is critically damped.

R=__________, L=__________, C=__________

Calculate α:_____________ The frequency of stimulus signal should be less than $\alpha/10$.

Choose your stimulus signal frequency f=________________

Use **V1=0V**, **V2=2V**, **TD=0**, **TR=1ns**, **TF=1ns**, choose your own PW and PER based on the predetermined f to perform PSPICE simulation, use the R, L, and C values chosen for critically damped to perform PSPICE simulation for the circuit shown in Figure 2.

Measure the waveform across R1 and save it for your prelab report. How do you interpret the waveform using the concepts of step response and natural response?

Problem 2 (75 points)

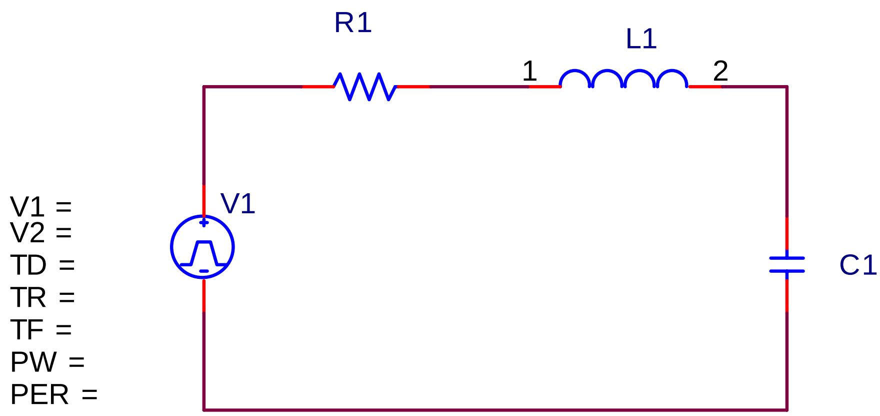


Figure 3

The circuit shown in Figure 3 will be used to study the natural and response of a series RLC circuit. In this lab, you will need to select the components with proper values so that response for over damped, under damped, and critically damped can all be observed.

(1) Assume that the available inductors are 10 mH and 47 mH, the available capacitors are 0.047 µF, 0.1 µF, 0.22 µF, 0.47 µF, 1 µF, and the resistance can be varied from 1 Ω to 1 MΩ by using the RC box, find the resistance to get circuit critically damped and finish Table 4. (**Hint**: Neper frequency $\alpha = R/2L$ and Resonant radian frequency $\omega_0 = 1/\sqrt{LC}$. For critically damped, $\alpha = \omega_0$. Therefore, the critically damped condition is $R_0 = \sqrt{4L/C}$. When R is larger than R_0, the circuit is over damped while when R is smaller than R_0, the circuit is under damped.)

Table 4

L=10 mH				
C (µF)	$R_0(\Omega)$	α(rad/s)	$1/\alpha$ (s)	$\alpha/10$(Hz)
0.047				
0.1				
0.22				
0.47				
1				

L=47 mH				
C (μF)	$R_0(\Omega)$	α(rad/s)	$1/\alpha$ (s)	$\alpha/10$(Hz)
0.047				
0.1				
0.22				
0.47				
1				

Please note that the role of $1/\alpha$ in a RLC circuit is similar to τ(time constant) in a RL or RC circuit. When a 50% duty cycle square wave is used as the stimulus, the pulse width has to be longer than five times of α to stabilize signals. The item '$\alpha/10$' in Table 4 gives a rough estimation of selecting the frequency of stimulus signal.

(2) Based on the calculation results in Table 4, choose R, L, and C with appropriate values so that the circuit is over damped.

R=___________, L=___________, C=___________

Calculate $\alpha, \omega_0, s_1, s_2$ and finish Table 5

Table 5

$\alpha = R/2L$	$\omega_0 = 1/\sqrt{LC}$	$s_1 = -\alpha + \sqrt{\alpha^2 - \omega_0^2}$	$s_2 = -\alpha - \sqrt{\alpha^2 - \omega_0^2}$

The frequency of stimulus signal should be less than $-s_1/10$.

Choose your stimulus signal frequency f=_________________

Use **V1=0V**, **V2=2V**, **TD=0**, **TR=1ns**, **TF=1ns**, choose your own PW and PER based on the predetermined f, use the R, L, and C values chosen for over damped to perform PSPICE simulation.

Measure the waveform across R1 and save it for your prelab report. How do you interpret the waveform using the concepts of step response and natural response?

(3) Based on the calculation results in table 4, choose R, L, and C with appropriate values so that the circuit is under damped.

R=___________, L=___________, C=___________

Calculate $\alpha, \omega_0, s_1, s_2$ and finish Table 6

Table 6

$\alpha = R/2L$	$\omega_0 = 1/\sqrt{LC}$	$s_1 = -\alpha + \sqrt{\alpha^2 - \omega_0^2}$	$s_2 = -\alpha - \sqrt{\alpha^2 - \omega_0^2}$

The frequency of stimulus signal should be less than $-s_1/10$.

Choose your stimulus signal frequency f=_________________

Use **V1=0V**, **V2=2V**, **TD=0**, **TR=1ns**, **TF=1ns**, choose your own PW and PER based on the predetermined f, use the R, L, and C values chosen for under damped to perform PSPICE simulation.

Measure the waveform across R1 and save it for your prelab report. How do you interpret the waveform using the concepts of step response and natural response?

(4) Based on the calculation results in Table 4, choose R, L, and C with appropriate values so that the circuit is critically damped.

R=___________, L=___________, C=___________

Calculate α:______________ The frequency of stimulus signal should be less than $\alpha/10$.

Choose your stimulus signal frequency f=_________________

Use **V1=0V**, **V2=2V**, **TD=0**, **TR=1ns**, **TF=1ns**, choose your own PW and PER based on the predetermined f to perform PSPICE simulation, use the R, L, and C values chosen for critically damped to perform PSPICE simulation for the circuit shown in Figure 3.

Measure the waveform across R1 and save it for your prelab report. How do you interpret the waveform using the concepts of step response and natural response?

Laboratory Assignment

Problem 1 (50 points)

(1) Construct the circuit shown in Figure 2 based on your design in Prelab problem 1 to obtain the waveform for the conditions of over damped, under damped, and critically damped. Design the signal generator to have 50 Ω internal resistance, and make sure that you take it into account when you construct the circuit.

(2) Save all the waveforms for your lab report, and compare them with your simulation results. (What are the reasons to cause the difference?) List the actual resistance used to obtain those waveforms.

(3) Explain how you obtrain the critically damped waveform during the experiment? Save the waveforms of a few critical steps leading you to obtain the critically damped waveform and provide a brief explanation.

Problem 2 (50 points)

(1) Construct the circuit shown in Figure 3 based on your design in Prelab problem 2 to obtain the waveform for the conditions of over damped, under damped, and critically damped. Design the signal generator to have 50 Ω internal resistance, and make sure that you take it into account when you construct the circuits.

(2) Save all the waveforms for your lab report, and compare them with your simulation results. (What are the reasons to cause the difference?) List the actual resistance used to obtain those waveforms.

(3) Explain how you obtain the critically damped waveform during the experiment? Save the waveforms of a few critical steps leading you to obtain the critically damped waveform and provide a brief explanation.

Appendix I: Identification of Equipment with Explanations

1. Digital Oscilloscope

Figure 1: View of an Oscilloscope

Just as the brush is an extension of a painter's hand, or a microscope is an extension of a microbiologist's eye, the oscilloscope (O-scope) serves as the eyes of an electrical engineer. All the signals of the circuit, no matter at what point therein, can be viewed and measured by this extraordinary piece of equipment by first attaching the O-scope's probe/s. However, simply physically connecting the probe does not guarantee an accurate or reasonable view of what's going on in the circuit. Indeed, there is an art to capturing a signal and/or keeping it stable on the O-scope's display. So much so, that a future lab will be dedicated to its function in details. For now, an example of what an O-scope looks like is shown in Figure 1. We will be using a digital O-scope in this lab. However, no matter whether the O-scope is analog or digital, all scopes have an output screen, a certain number of connecting probes (or channels), dials that control the vertical and horizontal spacing (voltage and time, respectively), and triggering controls for "syncing" to (or capturing) a signal. Once these principles are known, it is just a matter of adapting to specific functionality or the 'bells and whistles' of different O-scopes.

2. DC Power supply

In circuit schematics, many times there is a symbolic source of some kind, i.e., an element that supplies energy to the circuit. We can describe our source as being a power or signal source. There are a number of classifications of signals, i.e., deterministic/random, causal/non-causal, periodic/aperiodic, AC/DC, analog/discrete, to name a few. To go into details on these classifications would not be appropriate here, but don't worry, you'll have numerous courses later in the curriculum that will specifically, sufficiently (and painfully) deal with this topic. For now, what pertains to us is the equipment that we have and that we will use throughout the semester to physically supply energy, whether as a constant source or periodic signal of some kind.

If the source represents a constant voltage (or current) supply, then we have what is commonly known as an independent DC voltage/current supply. These sources associated with supplying power to a circuit are called power supplies. What differentiates a good power supply from a not so good power supply, is the ability for that supply to hold a constant value regardless of what is going on in the circuit, as well as the amount of power it is able to supply. The DC power supply that we will be using in lab is shown in Figure 2.

Figure 2: DC Power Supply (courtesy of HAMEG)

Although our supply is not known as 'high power' supply, it can deliver enough power to create burns and severe shock. So pay close attention in the safety section for this device. Notice this equipment provides three separate sources, one fixed, and two that are variable. They can be combined in series as well as produce +/- supply voltages. This is only a glimpse of the excitement we have to look forward to in subsequent labs.

3. Function generator

Figure 3: Function Generator (Courtesy of HAMEG)

Now, if the source represents a type of periodic signal (repeating waveform), then we can physically create this with what's called a function generator. We will also devote a future lab to its detailed operation. For now, let's observe that we can create, or generate, a number of different periodic signals, or functions. A pulse wave, a triangle wave, a square wave, a sawtooth wave, and of course everybody's favorite, a sinusoidal wave. Figure 3 shows what a typical function generator looks like and the one we will be using in class. Some function generators can also be programmed for a user-defined waveform and some can also create modulated waves, i.e., AM, like on your radio.

Figure 4: Digital Multi-meter (Courtesy of Fluke)

4. Digital Multimeter

Besides the O-scope, there is another measurement device called a multimeter. Figure 4 shows what a multimeter looks like. A multi-meter can measure quantity values such as resistance (Ohms), capacitance (Farads), inductance (Henrys), DC voltage (Volts), DC current (Amps). AC voltages and currents can also be measured. However for AC, there are a number of different ways to express a quantity. The physical units are still in Volts or Amps, but since the signal is AC, the measurement can be in terms of peak-to-peak, 0-to-peak, or rms quantities.

Further explanations of these terms, as well as the different methods to measure voltages or currents, will be taught as a future lab. Multi-meters are more convenient and quick to use than O-scopes, but are of course more limited in their capability. For this reason, lots of multi-meters are portable and no electrical engineer should be without one, whether testing a battery, continuity, or troubleshooting a circuit or power grid. They are relatively inexpensive compared to the O-scope.

5. Breadboard

For simple circuits that we will be constructing for this semester and beyond, we will use what's called a breadboard. A breadboard is a solderless, reusable prototyping board that is used to build and test circuits. Observe what a typical breadboard looks like in Figure 5.

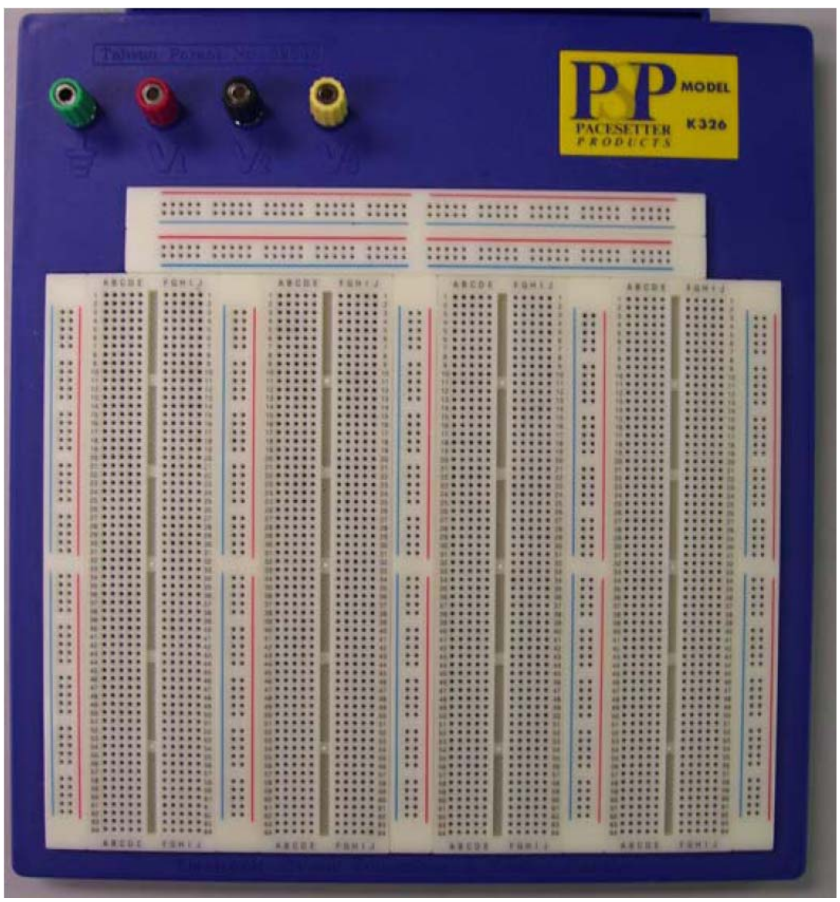
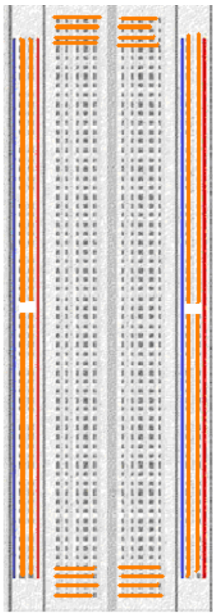


Figure 5: Breadboard

We can see many tiny holes on the surface, but we can't see is what is beneath the surface. Below are tiny metal rails (usually copper) that connect some of the holes together, either vertically or horizontally. If you look closely at Figure 5 you can see board actually made of several sections. To demonstrate the usage and the internal connection of the board we will focus only one section.



The figure to the left shows the layout of the interconnected holes. This layout is designed to assist the designer in connecting component ends together to form single node, i.e., instead of 'tying' 3 resistors together, we can push them in 3 different holes that are connected on the breadboard. Remember care must be taken because components can easily be shorted causing device damage or poorly functioning circuits. What is a 'short' you ask! A 'short' is a connection that exists between components. This can be a good thing or a bad thing. On example where it is bad is if two sides of a resistor are plugged into two holes that are on the same node. In this situation, no current will flow through the resistor since there exists a path of zero resistance, or what is called a "short" for the current to flow through. This situation has, for all practical purposes, eliminated the resistor from the circuit, which could cause other devices to be damaged.

Rows or columns that are connected together are commonly known as 'buses'. A power bus is one that makes power conveniently available for the whole board. One final thought is to note that these types of breadboards, because of the connecting metal rails underneath, do have power limitations and also add a tiny capacitance to the circuit making them unsuitable for high-power or high-frequency applications.

6. Circuit elements

Up until now, we've identified equipment that is used to measure or supply circuits. We have not even talked about any elements that are actually in a circuit. That is until now. Figure 6 below shows some typical circuit elements.

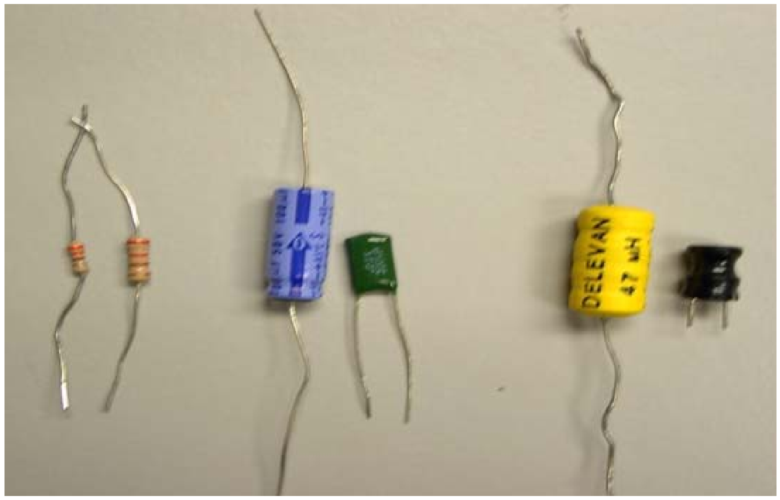


Figure 6: Resistors, Capacitors, and Inductors (grouped right to left)

A typical circuit element can either be passive or active. We will be talking about passive elements, i.e., elements that do not require a power supply to properly function. Resistors are passive elements that relate voltage and current by Ohm's law. The way they relate voltage and current (the slope) determines the resistance value (in Ohms). Resistors, as opposed to capacitors and inductors, purely dissipate power usually in the form of heat (capacitors and inductors are energy storing elements). A question should arise, "how much heat?" Well, that is dependent on the resistor's power capabilities. Most of our resistors are rated at a 1/4 Watt, and exceeding that power is not a good idea as will be seen in the safety section. Capacitors and inductors have similar specifications, but are laden with additional considerations that determine how they are specifically made, i.e., material. Polarity and frequency may play a role in these elements' operations.

7. RC box

For circuit element vendors, it is not economical to manufacture and stock elements with all different kind of values. Normally, they are made for a series of values and the any needed value can be obtained by combine several elements in series or parallel. For our circuit lab, often time we have a need to use either a resistor or a capacitor which is not a standard product. A RC box is the equipment to be used to substitute a resistor or a capacitor in a circuit with arbitrary value. A picture of a RC box is given in Figure 7.

Figure 7: Resistance-capacitance substituter (Courtesy of IET LABS)

Appendix II: Introduction to PSPICE

The objective of this part of the lab experiment deals with learning the fundamentals of the computer aided circuit simulation using Pspice. Pspice is a computer aided circuit analysis tool that is used for various analog and digital circuits. In this lab, we will concentrate on DC analysis. Later experiments in the semester deals with time and frequency domain analysis.

<u>Getting Started with Pspice:</u>

First, log on the computer workstations in front of you by using your uark e-mail login ID and password. Then, find the program link by first clicking on the 

menu > Programs > Orcad 16.0 > Capture CIS. Select **Allegro PCB Librarian XL** and click **OK** in the pop up dialogue box to start the program. Next, you should see a window pop open that looks something like Figure 1.

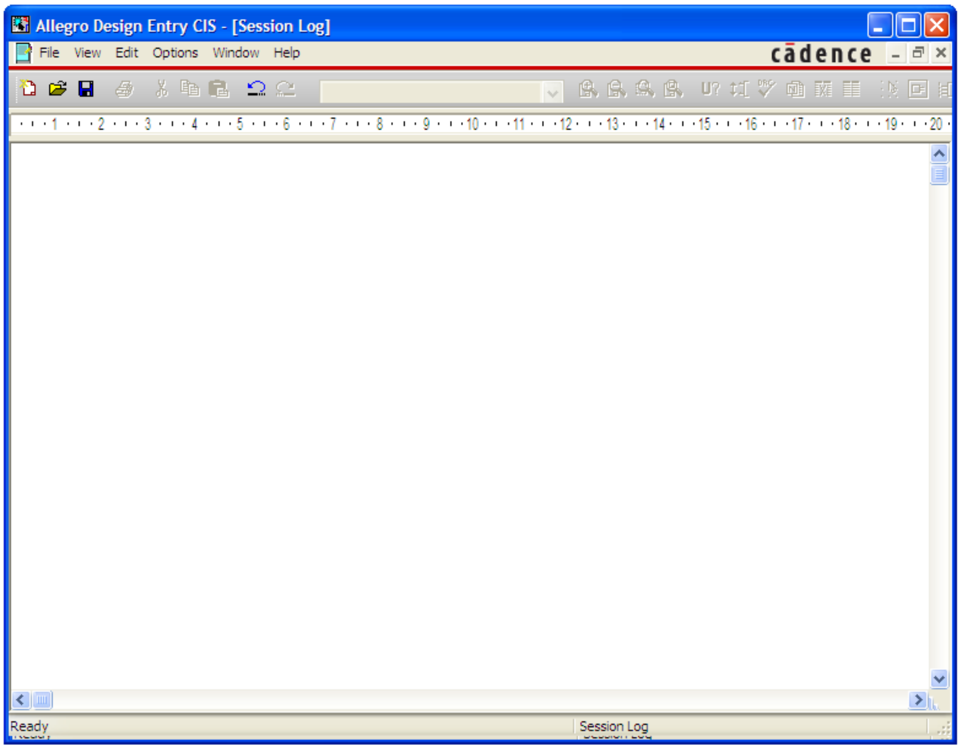


Figure 1: ORCAD Capture Screenshot

Next, click on this icon  or click File > New > Project. This will create a new project file. You will notice a new window pop up like the one in Figure 2 shown below.

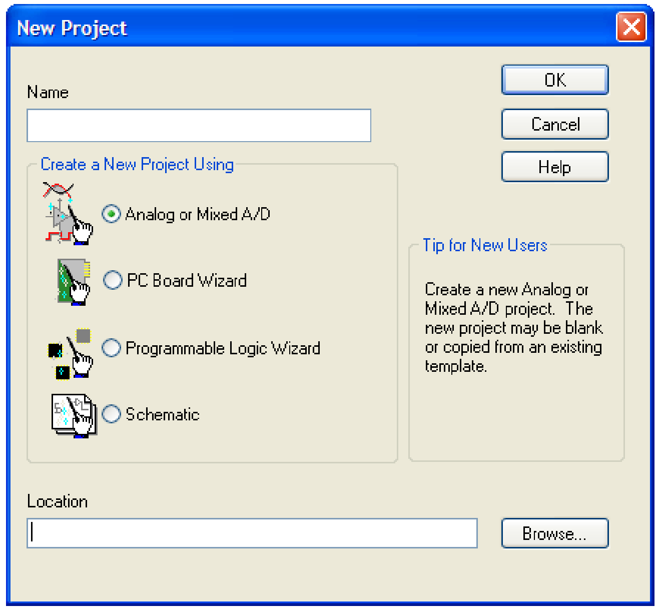
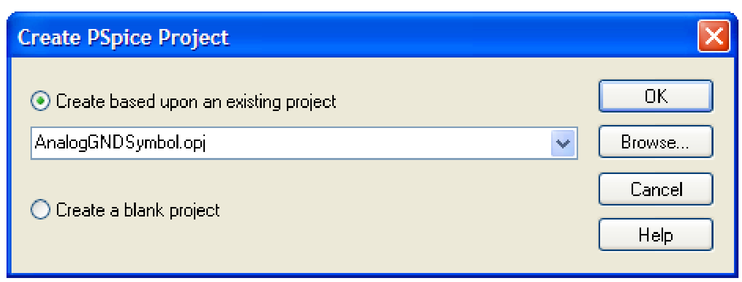


Figure 2: New Project Window

Then, type a name for the project (For example, Lab1_Prelab) and select the directory to save the project. You must select the **"Analog or Mixed A/D"** and then click **OK**. If you did everything correctly, the window shown below will pop up:



Now, select **"Create a blank project"** and then click **OK**. Upon following that step, a new window will pop up that is called the schematic window, shown below in Figure 3. This is the window where you can draw your circuit schematic.

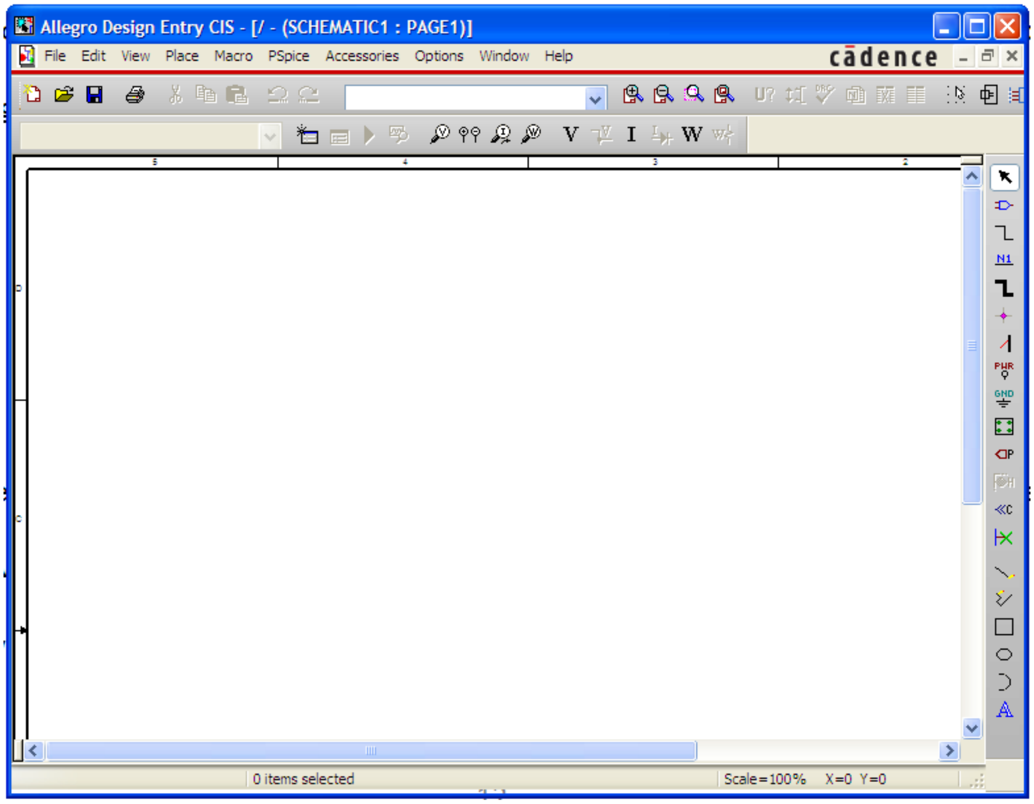
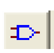
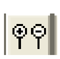


Figure 3: Schematic Window

As you can see from Figure 3, there are different types of radio buttons on the right hand side of the window, which specify different sets of commands. For example, the second radio button  from the top is the *place part* command, the third radio button  is the *place wire* command, and the ninth radio button  is the *place ground* command. Also, on the top of the window you will notice the radio buttons for voltage , current , watts , and differential voltage  probes.

Drawing a Circuit Schematic in the Schematic Window:

We will be learning how to draw a simple circuit given by the schematic in Figure 4.

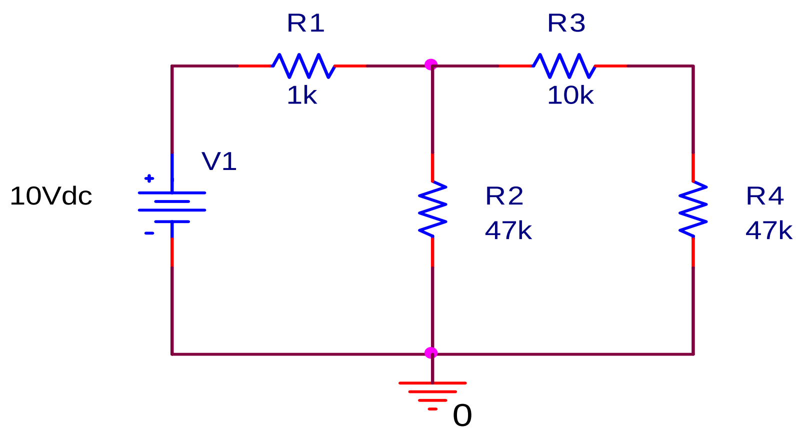
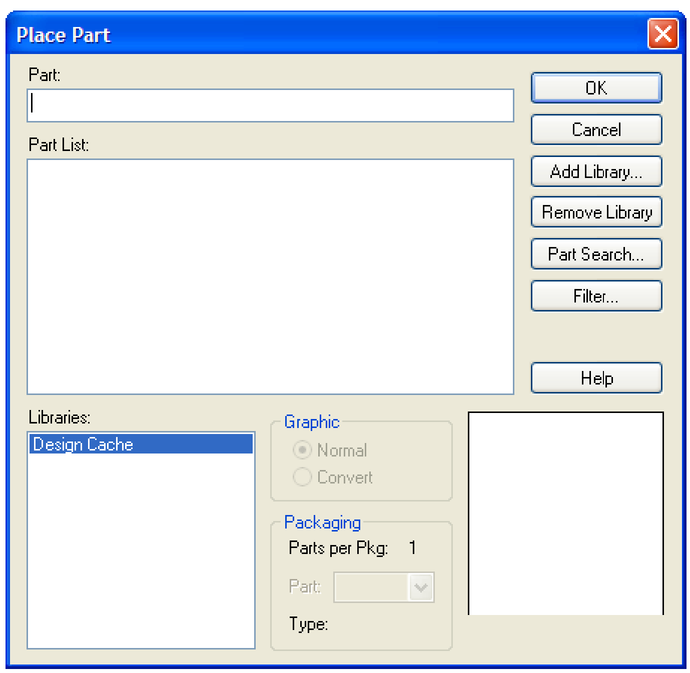


Figure 4: Simple Circuit Example

Adding a Library:

In order to place the appropriate PSPICE part, we need click on the  radio button (place part) on the right hand side of the window (we can also place part by doing the following Place > Part). If this step was done correctly, the window shown below will pop up:



Next click on **"Add Library"** in order to find the library that has the appropriate part we need for the circuit schematic. To do that, select the first library file, and then press **"Ctrl + A"** which will highlight all the library files. Then, press *open*, and then press *cancel*. Now you are ready to build the circuit on the schematic window.

Building the Circuit:

The circuit shown in Figure 4??? requires six components, which are:

1) Voltage Source

2) Four Resistors

3) Ground

To place the voltage sources

1) Click on the radio button (or from the Place menu, choose Part) to display the Place Part dialog box.

2) Add the library for the parts you need to place: Click the Add Library button, Select SOURCE.OLB (from the PSpice library) and click Open.

Note: There are two sets of library files supplied with Capture and PSpice. The standard schematic part libraries are found in the directory \TOOLS\CAPTURE\LIBRARY. The part libraries that are designed for simulation with PSpice are found in the sub-directory \TOOLS\CAPTURE\LIBRARY\PSPICE. In the Part text box, type VDC.

3) Click OK.

4) Move the pointer to the correct position on the schematic page and click to place the part.

5) Right-click and choose End Mode to stop placing parts.

To place the resistors

1) Click on the radio button to display the Place Part dialog box.

2) Add the library for the resistors you need to place: Click the Add Library button, Select ANALOG.OLB (from the PSpice library) and click Open.

3) Follow similar steps as described for the source to place the resistors (R).

Note: To rotate the part so the arrows are pointing in the correct direction, place the part, select it, then press *R* one or more times to rotate the part to the desired orientation.

To place the zero ground part

1) To place the ground parts (0), click the GND button display the Place Ground dialog box.

2) Add the library for the parts you need to place: Click the Add Library button, Select SOURCE.OLB (from the PSpice library) and click Open.

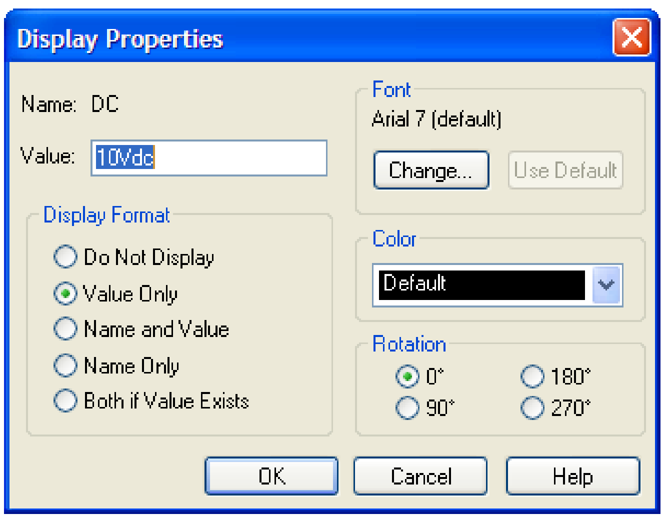
3) Place the 0 ground part from SOURCE.OLB. You must use the 0 (zero) ground part from the SOURCE.OLB part library. You can use any other ground part only if you change its name to 0 (zero).

To place the wires

Click on the  radio button to add wires and connect the voltage source, resistors, and ground together.

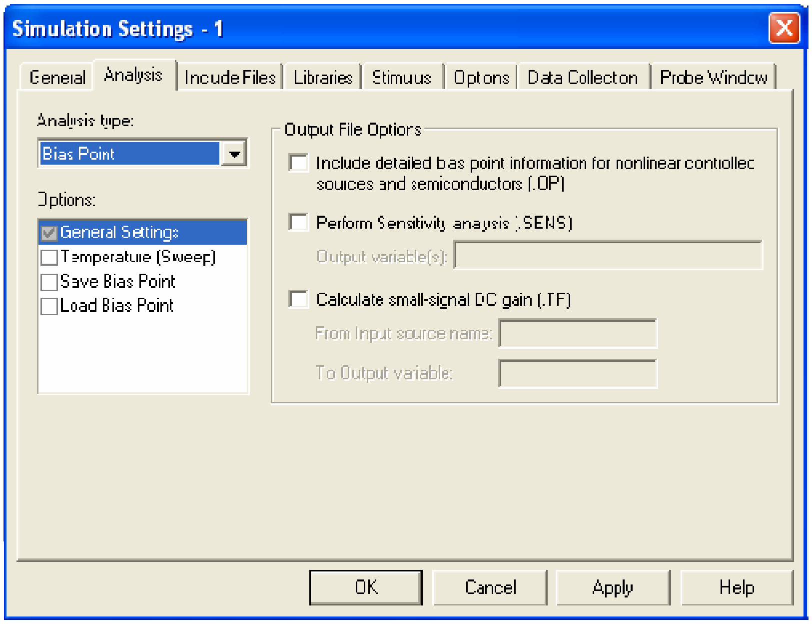
To change the values

Next, we need to change the value of the **"V1"** to 10V. To do this, double click on the "0Vdc" and a new window will pop up as shown below. Type in **"10Vdc"** in the value box and click "**OK**." You can also change the value of the voltage source by doing the following steps: Select the voltage source > Right click > Edit properties > Change the value of the "DC" column from "0Vdc" to "10Vdc" > Apply > Close the property Editor window. Follow the same steps for changing the value of the resistors. A dialogue box to edit the display properties is given in below.



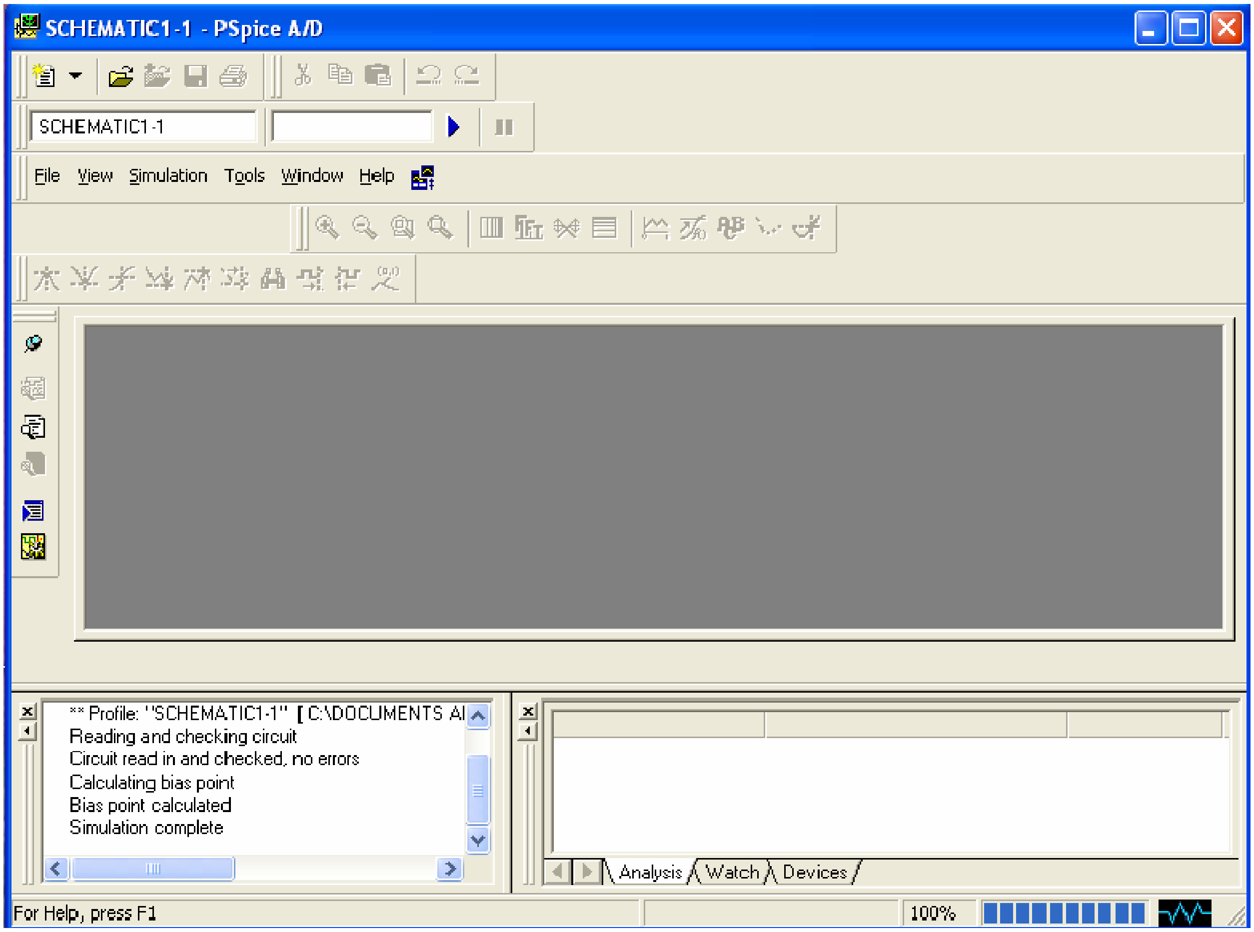
Simulating the Circuit:

Once the circuit is built in the schematic window, it's time to simulate the circuit. In the first lab, we will concentrate on DC analysis. To simulate the circuit we need to click on Pspice (Menu on top) > New Simulation Profile > Type in a name for your simulation > Click on *Create*. Then the Simulation Setting window will pop up.

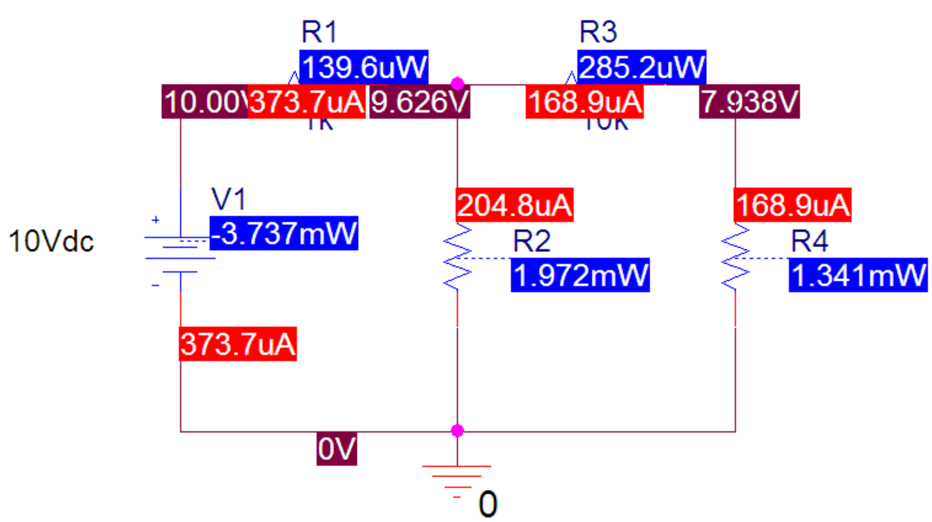


Bias Point:

Select from the **"Analysis type:"** drop down menu **"Bias Point".** Then click '**OK.'** Next, press the run radio button  on the top of the schematic window. A new window pops up that is called the Pspice simulator, or Pspice A/D shown below.

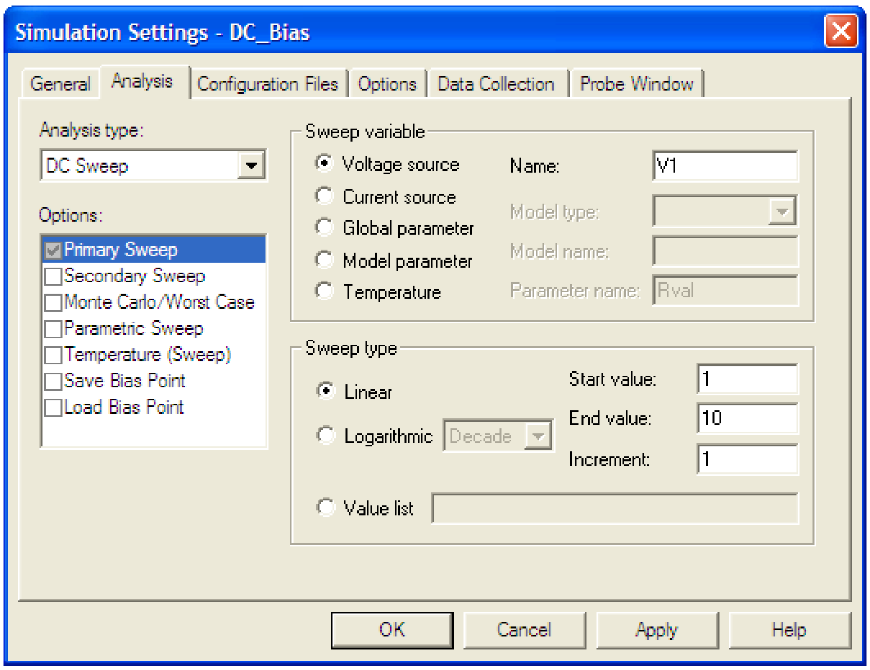


Now, go back to the schematic window and click V I and W radio buttons on the top of the window, which will show the voltages, currents, and watts at each node of your circuit. So at this point, your schematic will now look like the figure in the following.



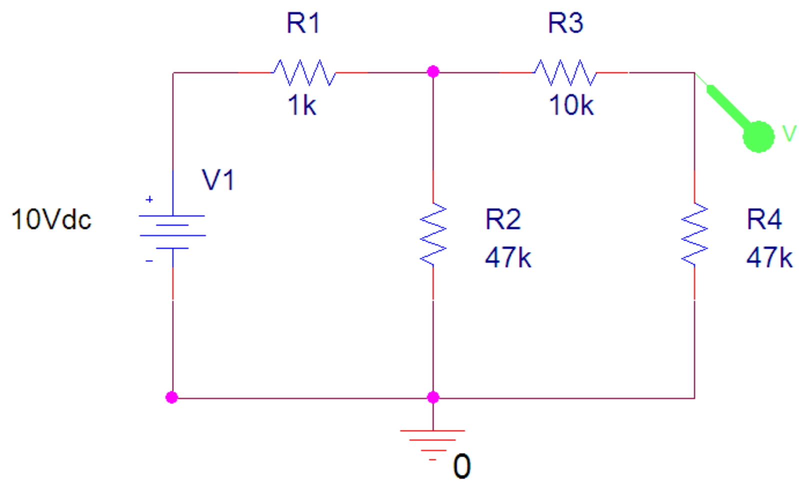
DC Sweep:

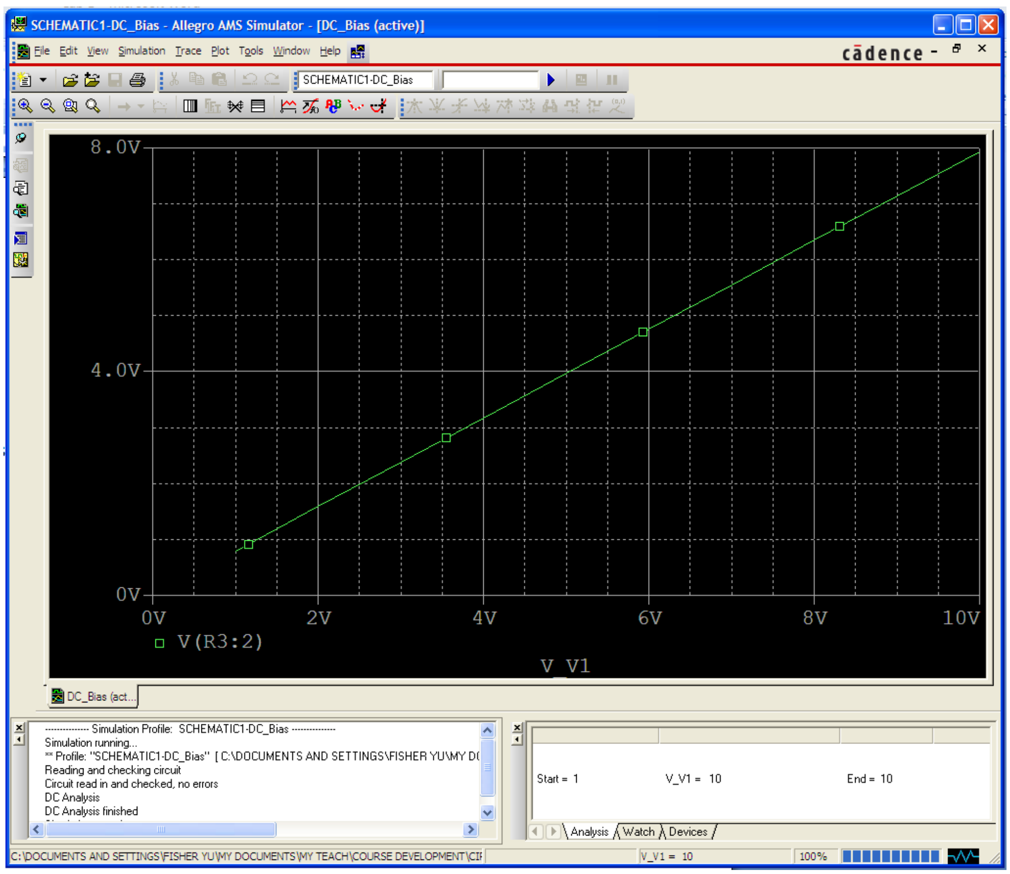
Next, we will do the "**DC sweep**" analysis on our circuit. So, in order to do that you need to change the "**Analysis type:**" from "**Bias Point**" to "**DC Sweep**". To do that you need to click on the radio button next to the run radio button or you can do the following Pspice > Edit Simulation Profile. Then, select the "**Analysis type:**" to "**DC Sweep**". In this lab, we are going to sweep the voltage source "V1" and see how the sweep affects the voltage across the resistor "R4" (which is the output voltage of the circuit). So, setup the Simulation Settings window as shown below, and then click **OK**.



Then, press the *run* radio button on the top of the schematic window. A new window pops up, which is the Pspice simulator or Pspice A/D.

Next, go back to the schematic window, unselect the V I and W radio buttons, and place the voltage probe at nonground side of R4, and your schematic should now look like the figure shown to the below. Next, go back to the Pspice simulator or Pspice A/D and look at the graph. Your graph should look like the graph shown below if done correctly.



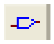


Parametric Sweep:

Resistance cannot be swept in the similar way as we have done for voltage sweep. In order to do so, we have to use parametric sweep.

1) Double-click the value (47k) of part R4 to display the Display Properties dialog box, In the Value text box, replace 47k with {Rval} and then click OK.

PSpice interprets text in curly braces as an expression that evaluates to a numerical value. This example uses the simplest form of an expression--a constant. The value of R4 will take on the value of the Rval parameter, whatever it may be.

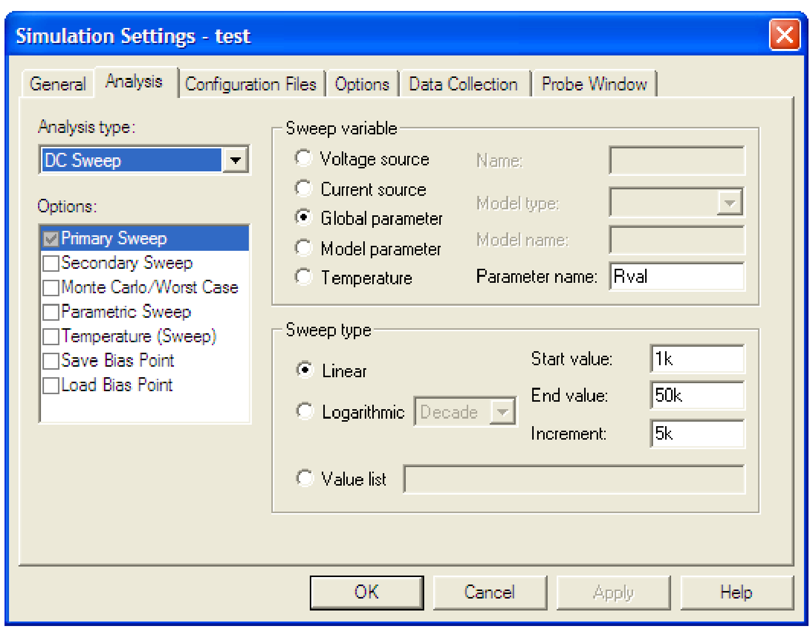
2) Click on the  radio button to display the Place Part dialog box, in the Part text box, type PARAM (from the PSpice library SPECIAL.OLB), then click OK, place one PARAM part in any open area on the schematic page.

3) Double-click the PARAM part to display the Parts spreadsheet, then click *New Column*. In the *Name* text box, enter Rval (no curly braces), then click OK. This creates a new property for the PARAM part, as shown by the new column labeled Rval in the spreadsheet.

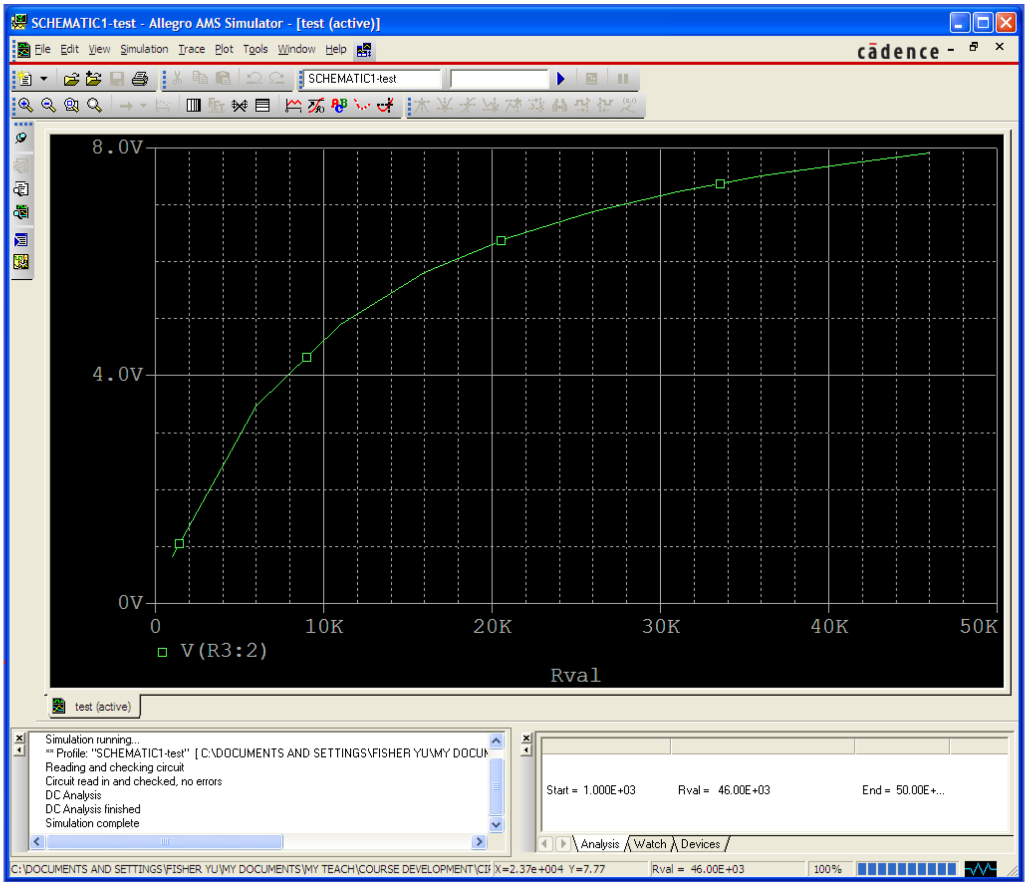
4) Click in the cell below the Rval column and enter 10 k as the initial value of the parametric sweep; While this cell is still selected, click *Display*. In the Display Format frame, select *Name and Value*, then click *OK*. Click *Apply* to update all the changes to the PARAM part. Close the Parts spreadsheet.

5) From the File menu, choose Save to save the design.

6) From the Pspice menu, select Edit Simulation Profile and change sweep variable to “Global parameter” and other parameters as shown in below.



Then, press the *run* radio button  on the top of the schematic window. The simulation result is shown in below.



Transient analysis

Take a look at the circuit shown in below, which should look familiar to you by now.

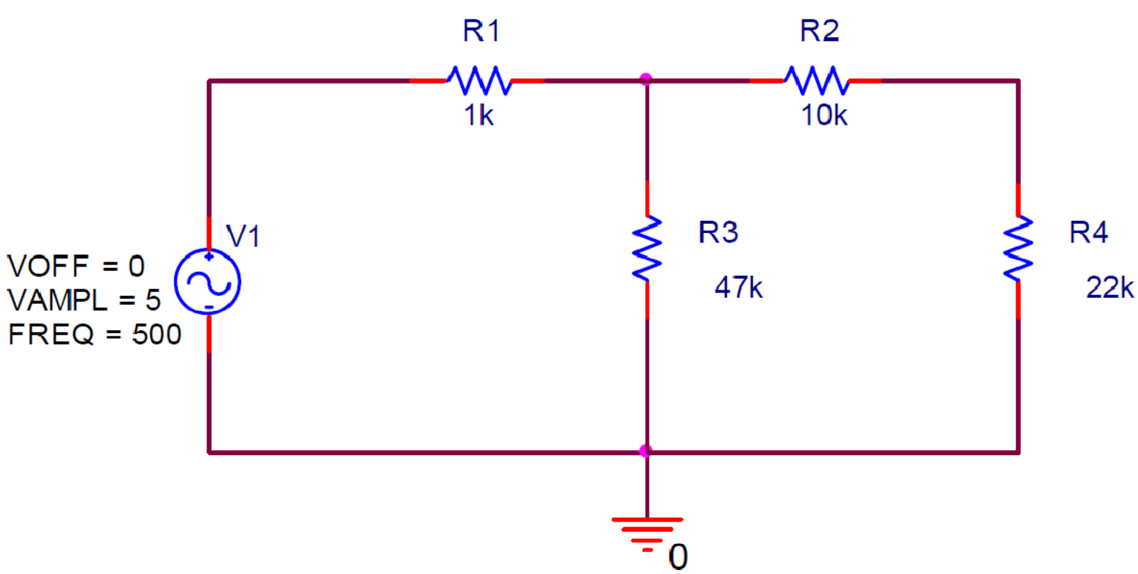


Figure:???

Now, draw the circuit shown in Figure ??? in the schematic window. Note that instead of inserting a DC source, we are inserting a sinusoidal voltage source at the input. The name of the sinusoidal voltage source part is 'Vsin'.

TRANSIENT ANALYSIS

Once you are done drawing the circuit in PSPICE, click on PSPICE > New Simulation Profile > Type in a name. Then setup the simulation settings window as shown in Figure 2:

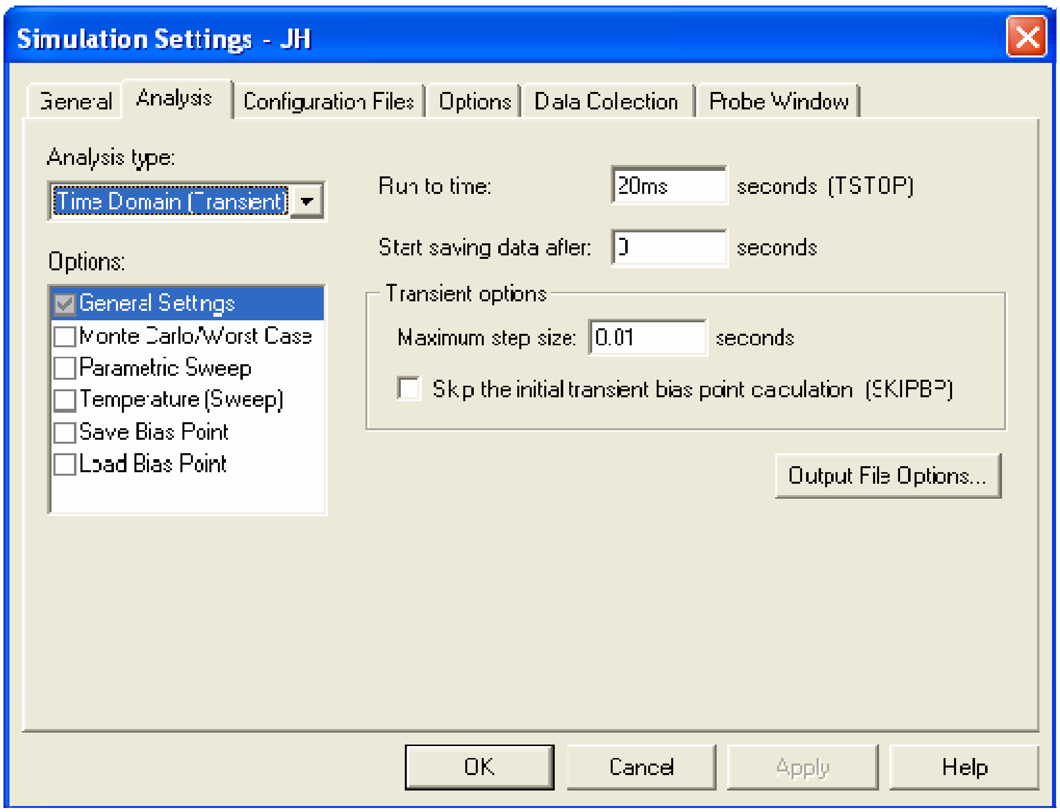


Figure 2: Simulation Settings Window

Click 'OK' and run the simulation. If your circuit is working properly, it will show the following message on the PSPICE output window or PSPICE A/D:

"Reading and checking circuit

Circuit read in and checked, no errors

Calculating bias point for Transient Analysis

Bias point calculated

Transient Analysis

Transient Analysis finished

Simulation complete”

Appendix III: Using DC Power Supply, Resistors, and Digital Multi-Meter

Part I: DC Power Supply

In general, a DC power supply simply sources a constant voltage to a circuit. Specifically, we will look at the operation of the Hameg 7042 Triple power supply. The Hameg 7042 consists of three *independent* power supplies. A rough sketch is shown in Figure 2. You should identify the plugs marked 1, 2, 3, etc. on your power supply.

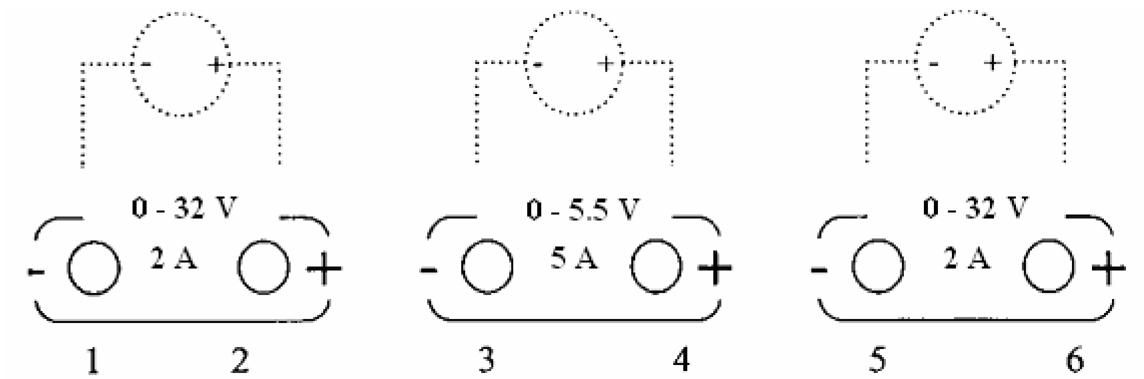


Figure 2: Front panel of the Hameg 7042 Triple Power Supply ??? Figure 1

The two outside power supplies are variable supplies; that is, they can be set by the operator at any voltage level from 0 – 32V and can output a maximum of 2A. The middle power supply is also variable from 0 – 5.5V; however it is capable of producing 5A. These supplies are the "real-world" equivalent of the voltage source circuit symbol commonly used in schematics (drawn directly above them).

Operation of the power supply is relatively simple. Some of the major features to note are the following:

Displays: The three variable sources have LED displays which indicate what voltage level is being sourced along with current in Amps.

Course/Fine knobs: Turning these knobs adjusts the voltage level of the respective supply. Course changes the values drastically, fine changes the level steadily.

Output On: This button is essentially a switch, which turns the power supplies on or off. When you have everything hooked up, be sure to turn on the supplies!

Electronic Fuse: This is a safety feature that will turn off the supply when there is a surge in current exceeding the power capability.

Overcurrent Indicator: When too much current is being drawn from the supply, the red light will turn on. If this happens, **TURN THE OUTPUT OFF!**

The three independent supplies can be arranged to establish many different types of DC sources. We will now go over a number of examples that are commonly seen and show how to hook up a source that matches that of a circuit schematic.

Example 1: Figure 3 shown below illustrates a simple 0-32V variable voltage source and its power supply counterpart that you will connect. The diagram on the left is the circuit symbol whereas the one on the right is the actual power supply hookup.

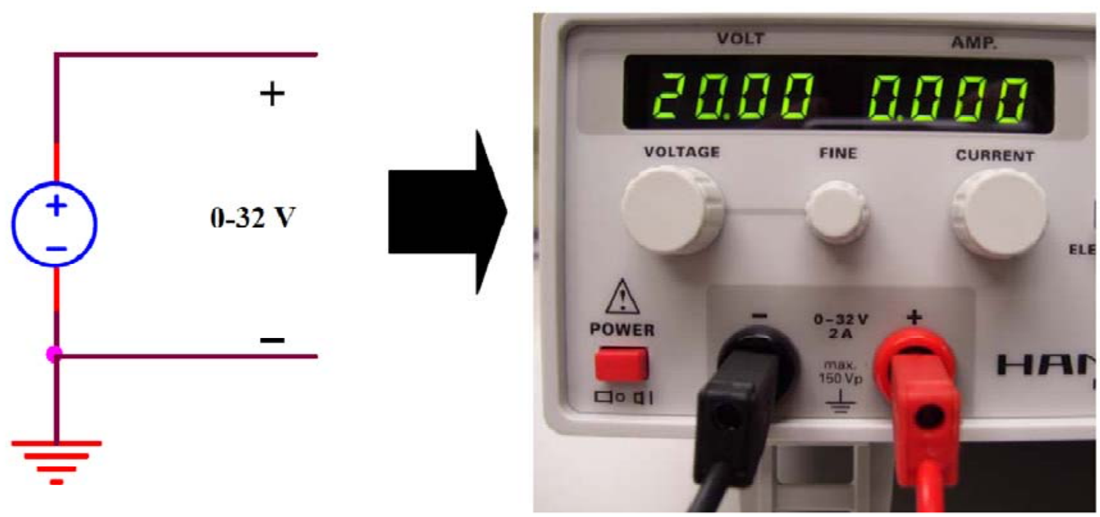


Figure 3: Variable 0-32V Voltage Source Figure 2???

By convention, the *red* lead is connected to the positive and the *black* lead to the negative. This merely serves as an identifier. By turning the coarse or fine knobs, this configuration can generate 0-32V. Note: Be sure that the **OUTPUT ON** button is truly on, and that you don't have the red and black leads touching or even close to each other touching.

Example 2:

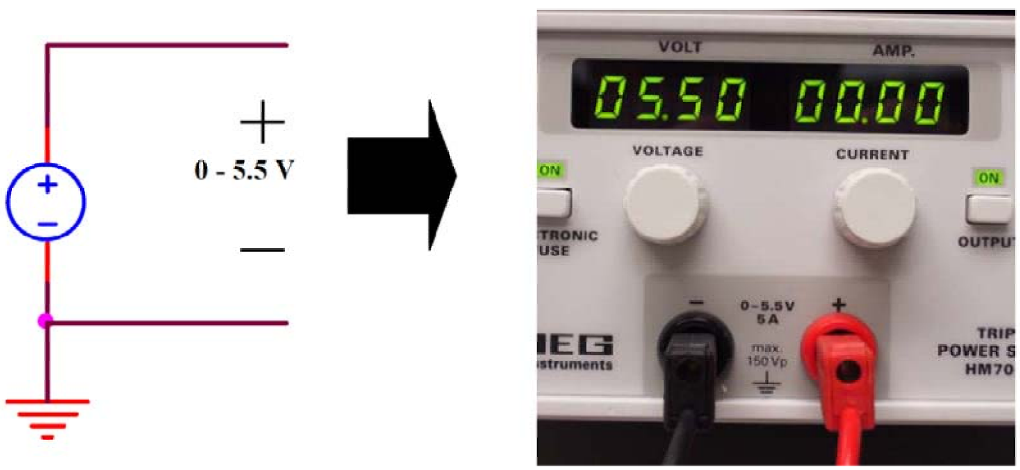


Figure 4: Variable 5.5V Voltage Source Figure 3???

The next example consists of a 5.5V variable supply. Observe the connections in Figure 4. Note that this supply is the middle supply shown in Figure 2 and can be used to source more current then the other sources (up to 5A). This source only has a coarse variable knob available.

Example 3:

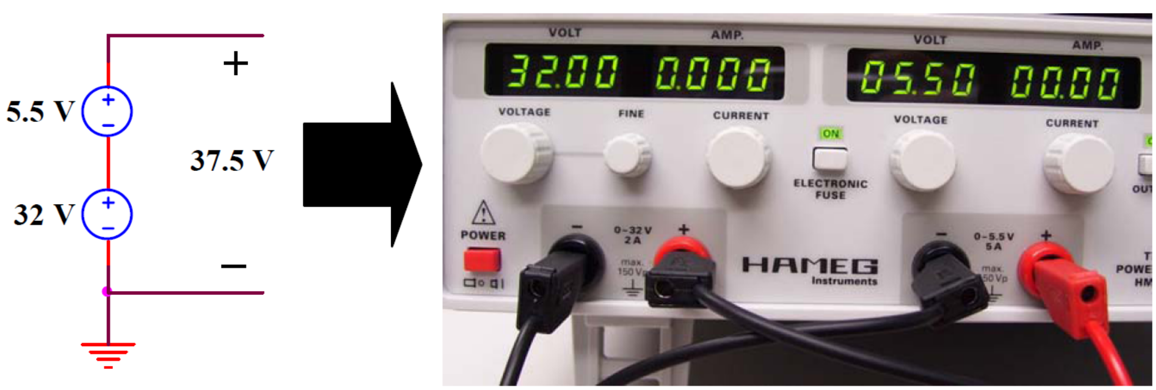


Figure 5: 37. 5 V Voltage Source Figure 4???

The next example consists of a 37.5V voltage source. To obtain 25V from our supply, we must hook up two sources in series. The circuit schematic and its equivalent are shown in Figure 5. Make sure to always connect the positive terminal of one supply to the negative terminal of the other. **Never connect positive and negative terminals of the same supply together. This will cause a short circuit and create a safety hazard!**

Example 4:

Sometimes, applications call for a power supply that has both positive and negative voltage levels. This is accomplished by moving the ground, or the 0 *reference point*.

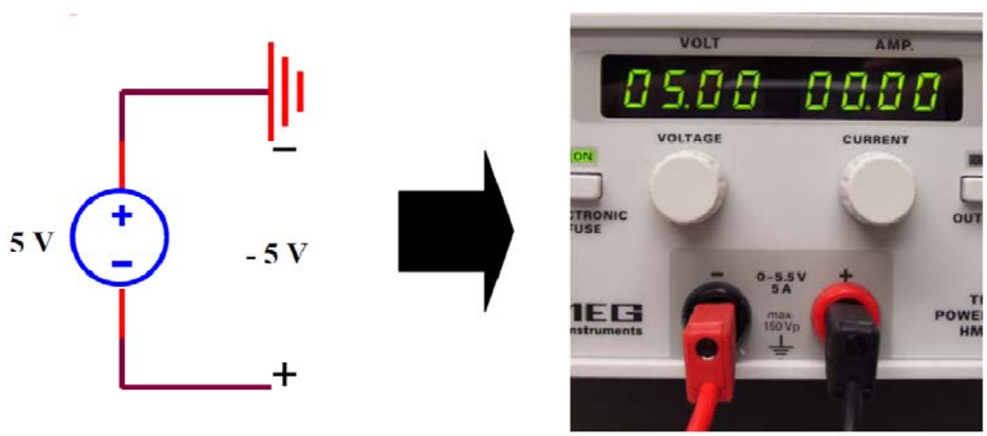


Figure 6: -5V Voltage Source ??? Figure 5

Let's start a simple -5V voltage source. The -5V voltage source is connected almost the same as the +5V source except the outputs are taken from *the opposite* terminals. This doesn't change the actual voltage present; it only moves the ground or 0V reference point. In this case, ground has been moved from 0V (the negative terminal) to 5V (the positive terminal). The "positive" output is now the "negative" terminal, which is 5V less than the "positive" terminal, giving us a

-5V source. Refer to Figure 6 for an accurate visual representation. A voltage source with voltage levels on *both* sides of ground can be constructed using two power supplies.

Example 5:

Sometimes there is a configuration where ground is *between* two voltage sources. Refer to Figure 7. Notice that we have access to 20V on both sides of ground, providing us with +/- 20V. This type of configuration is very common for supplying power to IC chips, especially amplifiers.

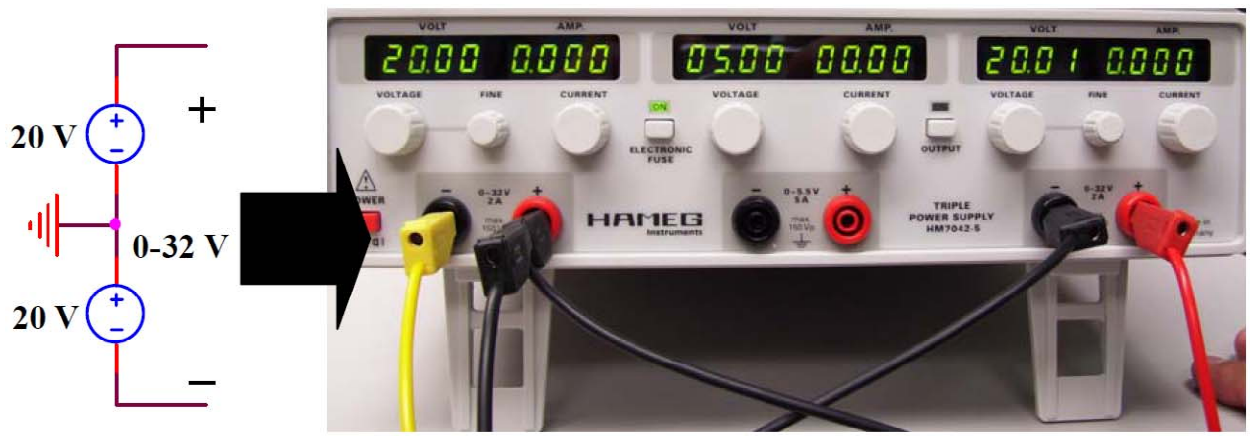


Figure 7: ±20V Voltage Source Figure 6???

Part II: Resistors

Resistors are devices through which current flows, creating a voltage drop between two points. Resistance is expressed in units called 'ohms,' or symbolically 'Ω.' Resistors rely on Ohm's Law, which states that the voltage drop across two points of a conductor is held in proportion to the current flowing through it. Or mathematically speaking,

$$V = IR \tag{1}$$

where V represents the voltage, I represents the current in amperes, and R represents the resistance. Notice this ideal equation exhibits linear behavior, i.e., the resistance is a constant slope or ratio of voltage to current.

Figure 8: Example of a Common Resistor Figure 7???

The resistors that we will be using in the lab are made of carbon composite material. An example of this type is shown in Figure 8. The resistance value can be determined by evaluating the color code on that particular resistor. Resistors that have 5% tolerance or greater have four color bands. The band at the end of the resistor signifies the most significant digit (A) followed by the digit (B), the next to last is the exponent (C), and the last band signifies the tolerance (D). So, the value of the resistance (R) is,

$$\mathrm{R} = \mathrm{A} * \mathrm{B} * 10^{\mathrm{C}}\ \Omega$$

In a similar manner, resistors that have 1% tolerance or less have five color bands instead of four color bands. Therefore, the value of this resistance is,

$$\mathrm{R} = \mathrm{A} * \mathrm{B} * \mathrm{C} * 10^{\mathrm{D}}\ \Omega$$

Now that we know what each band represents, we need to know which color corresponds to what digit. The color code is as follows:

BLACK	0	**GREEN**	5
BROWN	1	**BLUE**	6
RED	2	**VIOLET**	7
ORANGE	3	**GREY**	8
YELLOW	4	**WHITE**	9

The color bands for the tolerance band are:

NO BAND ±20%
SILVER ±10%
GOLD ±05%

Figure 9 pictorial description of the process to evaluate the resistance value from its color codes.

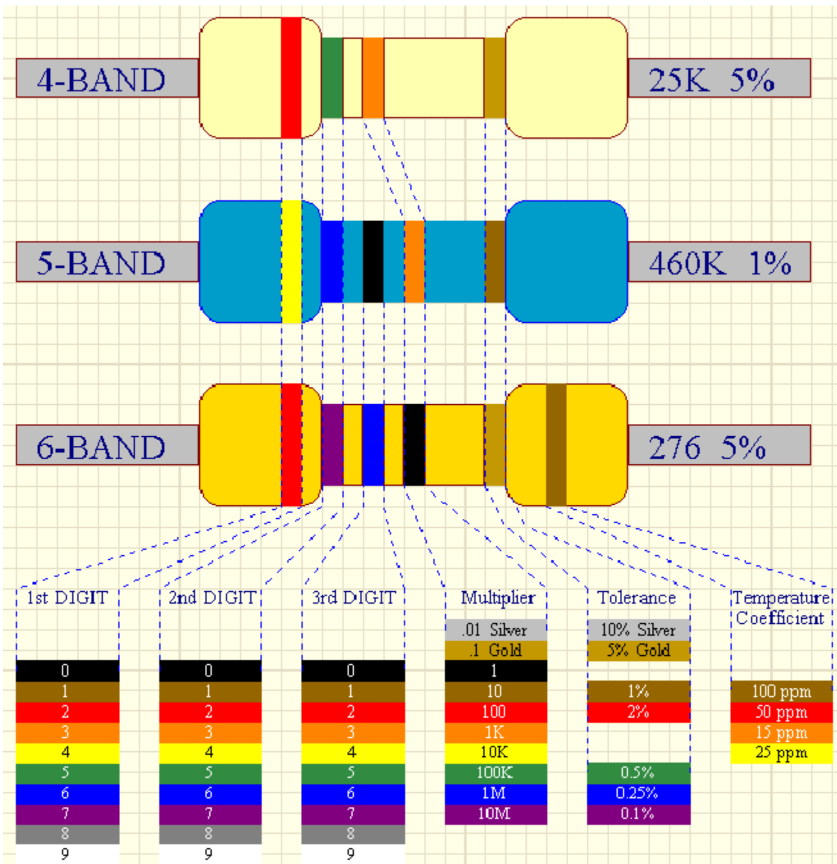


Figure 9: Pictorial Resistance Value Guide (courtesy of www.armory.com) Figre 8???

Power Rating :

One very important piece of information about resistors in general is the power rating. Just because a resistor has a value that fits nicely into Ohm's Law, doesn't mean it will operate under Ohm's Law for all values of voltage or current. A resistor will follow Ohm's law based on the power dissipation capability of the resistor. The resistors in the lab are rated at ¼ Watts. If this rating is exceeded, the resistors will start burning as indicated by smoke. As an example, consider the following:

A 1kΩ resistor rated at .25 Watts in the lab can only handle 0.0158 A, or 15.8 mA, by knowing that

$$P = V * I = I^2 * R \tag{2}$$

Therefore,

$$I = \sqrt{\frac{P}{R}} = \sqrt{\frac{0.25}{1000}} = 0.0158A$$

Part III: Digital Multimeter

At this point, we have learned how to set up various power supply configurations and know the power capabilities of the resistors we will be using along with how to read their values (without relying on the value marked on the bin we get them from). We are almost ready to construct our circuit. However, we need to know how to *measure* the actual values when our circuit is in operation. To do this, we need to be familiar with a piece of equipment known as the digital multimeter.

A digital multimeter (DMM) can be a very useful and accurate instrument when it is used correctly. The purpose of this section of the lab is to teach proper measurement techniques. This section will also show some common errors made and explain how the DMM actually changes the circuit under test. At the end of this lab you should be able to correctly measure resistance, DC voltage, and DC current.

Meter Loading:

Before we start using the DMM, the operator should be aware of the basic principles of how the DMM works for both voltage and current readings.

Voltmeters:

When voltmeter probes are put across a resistor in a circuit, it is like putting a large resistance in parallel with the resistor being measured. This changes the equivalent resistance in the circuit and could significantly affect the voltage measurement. In order to change the circuit as little as possible, the internal resistance of the voltmeter is very high. This allows almost all of the current to flow through the resistor being measured just as it had been before the voltmeter was put in parallel. If the resistance being measured is greater than 10% of the voltmeter's internal resistance, the measurement will not reflect the actual voltage. The internal resistance of the voltmeter used in this lab is about 10 MΩ.

Ammeters:

When an ammeter is put in series with a resistor by breaking the circuit and re-closing it with the ammeter probes, it is like putting a resistance in series with the resistor being measured. Just as with the voltage measurement, this changes the equivalent resistance in the circuit and could significantly affect the current measurement. In order to change the circuit as little as possible, the internal resistance of the ammeter is relatively small. This keeps the voltage drop across the ammeter low and allows the voltage across the resistor being measured to stay nearly the same as it was before the ammeter was put in series. Again, the 10% rule should be observed; if the ammeter resistance is greater than 10% of the value of the resistor being measured, the measurement will not be the actual current. The internal resistance of the ammeter used in this lab varies with the RANGE setting. The resistance drops as the range value increases. On a RANGE of .2, the internal resistance is 1KΩ. On a range of 20A, the resistance is 0.1Ω.

Important Things to Remember:

- Internal meter resistances should have minimal impact on the circuit being measured.
- Voltmeter internal resistance is high because the meter is put in parallel with the circuit.
- Ammeter internal resistance varies with the range setting and goes in series with the circuit.

Common Errors:

- Measuring resistances while they are connected to other resistances in the breadboard.
- Measuring resistances with your body touching the probes.

• Trying to measure current the same way you measure resistance and voltage.

Finally, we are now adequately prepared to make the following measurements with our DMM.

Resistance Measurement:

To setup the DMM for a resistance measurement, do the following steps:

1. Insert the test leads into the proper sockets of the multimeter: Black to **Com**, Red to **V/Ω**.

2. Set the DMM to Ω by turning the dial to the **Ω** setting. Note: Make sure you do not have the leads on a live circuit.

Next, to make a resistance measurement, follow these steps:

1. Touch the ends of the probes to the ends of a 10K resistor. Make sure nothing but the probes touch the resistor ends.

2. The resistance value should be displayed on the DMM. You can measure resistances from .1 Ω to 50 MΩ.

Voltage Measurement:

To setup the DMM for a DC voltage measurement, do the following steps:

1. Insert the test leads into the proper sockets of the multimeter: Black to **Common**, Red to **V/KΩ**.

2. Set the DMM to V or mV by turning the rotary switch to the**V** or **mV** position, respectively.

Next, to make a voltage measurement, follow these steps:

1. To measure the voltage across a resistor simply put the probes on each end of the resistor and read the value from the display. It is best practice to connect the ground or common lead first, then the positive voltage probe.

2. The voltage should be displayed on the DMM. The **mV** position will measure values from .1mV to 600mV and the **V** position will measure 1mV to 1000V.

Current Measurement:

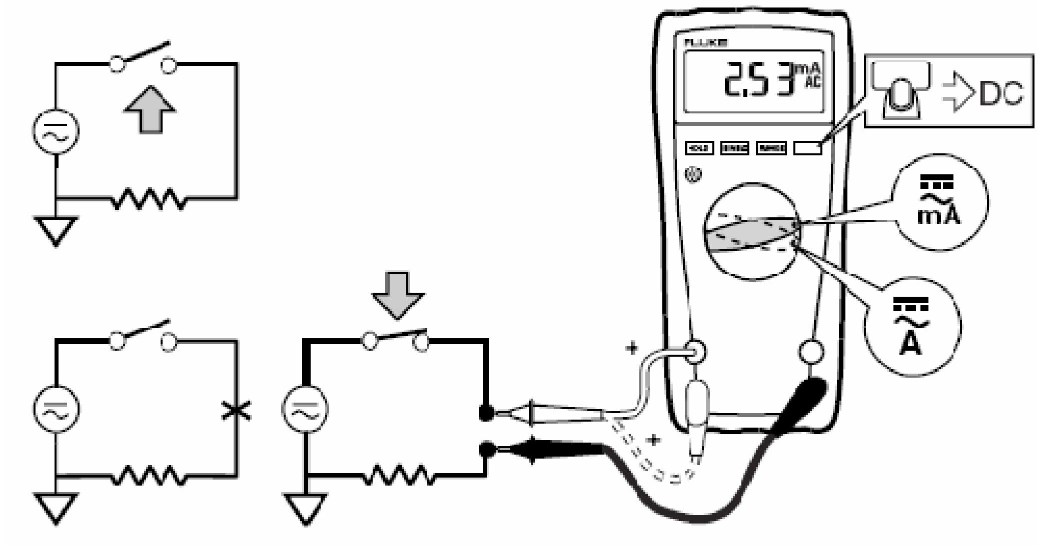


Figure 10: Ammeter Setup (Courtesy of Fluke) Figure 9???

The current measurement is a little different and a bit trickier. Notice that for a voltage measurement, your probes were in parallel with the resistor. For a current measurement, the probes must be in series with the resistor. To setup the DMM for a current measurement, do the following steps, referring to Figure 10:

1. Insert the test leads into the proper sockets of the DMM: Black to **Common**, Red to **10A** or **400mA** socket.

2. Set the DMM to mA of A by turning the rotary switch to the **mA** or **A** position. For DC measurements, press the **YELLOW** button. For appropriate settings, note that the **mA** position has the range of .01mA to 400mA and the **A** position f from .001A to 10A. The Red lead must be in the appropriate corresponding socket as well. This implies that you have some reasonable estimated knowledge of how much current you are expecting to measure.

Next, to make a current measurement on the breadboard, do the following, referring to Figure 11 as a physical example:

1. Make sure power to the circuit is turned off. Unplug one leg of the resistor through which you want to measure the current. This is known as “breaking” the circuit.

2. Insert the "unplugged" leg into an unused row on the breadboard, leaving the other leg attached to the circuit.

3. Put the red lead of the DMM where the "unplugged" leg used to be.

4. Put the black lead into the row where the "unplugged" leg is now so that the DMM "bridges the gap." The probes are now in series with the resistor, or can be seen as “completing” the circuit.

5. Turn the circuit power back on. Note the passive sign convention. If current flows into the Red lead, a positive sign is given. If current flows into the Black lead, or common, a negative sign is given.

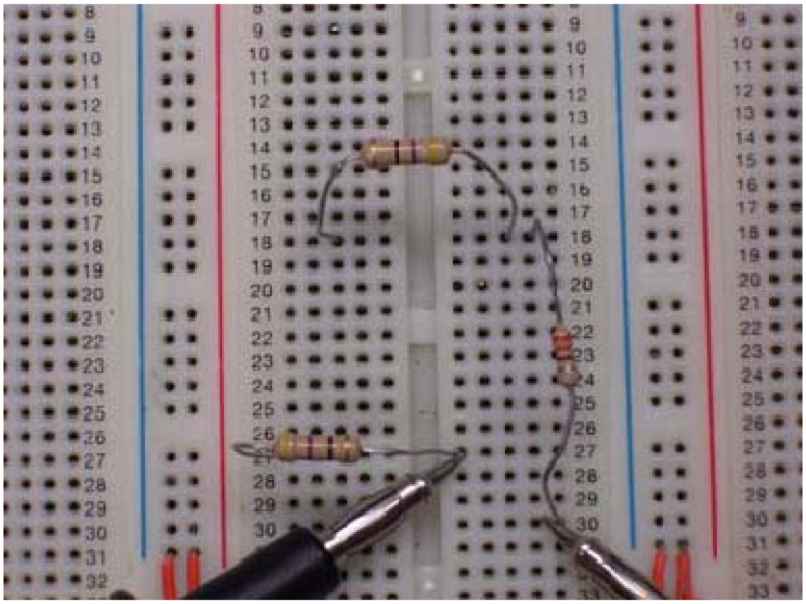


Figure 11: Example Current Measurement Figure 10???

Appendix IV: Using Oscilliscope and Function generator

1. Oscilloscope

Figure 1: The Digital Tektronix TDS 2012B Oscilloscope

Introduction

An Oscilloscope is a measurement device that displays a graph of an electrical signal. A graph of a waveform displayed on an oscilloscope conveys essentially the same information as a graph of a waveform on paper. The oscilloscope however has the advantage of displaying the waveform in real time. In addition, the oscilloscope can be adjusted to display the waveform in a number of ways, i.e., scaling x and y axes, locking onto certain voltage levels, and showing multiple waveforms simultaneously. Although we will be using a digital O-scope, the basic physical operation of an analog O-scope is worth noting. The analog O-scope works by firing an electron gun onto the back of a CRT panel. The panel is illuminated where the electron beam hits. The gun sweeps horizontally from the width of the screen from left to right in a certain amount of time. As the voltage of the signal being measured changes with time, the vertical position of the beam also correspondingly changes. While it is advisable to learn the theory behind an instrument such as an oscilloscope, it is not necessary to be an expert on its operation in order to be able to make some basic measurements.

All oscilloscopes have the same basic functional controls: a screen output, Vertical Input, and Horizontal Input. Most manufacturers conform to a similar control layout that has evolved over years of lab use; however, each particular oscilloscope model is still a little different from the others. This appendix will familiarize you with the functions in general and the controls of the digital TEKTRONIX TDS 2012B in particular, as shown in Figure 1.

Display Screen and Controls

The display screen is used to display the waveforms. When turned on, the screen is marked with grid squares to allow both vertical and horizontal measurements to be made. The screen can easily and quickly be adjusted for better viewing of the display.

Figure 2: Control Buttons for Menu Operation

As you read through the following functional descriptions, find these controls on your oscilloscope. As you find these controls, mark them with the corresponding number as depicted in Figure 2.

- ON/OFF Power Switch - Main power of the oscilloscope. No reference number on figure. Located on top of O-scope.

- Display Screen - Displays the waveforms. Grid squares are 1cm x 1cm. No reference number on diagram.
- ____ **Cursor** Control - Displays measurement cursors, horizontal or vertical, for accurately measuring time and voltage displacement respectively.
- ____ General Purpose Knob - Depending on the last menu item selected, the general-purpose knob selects various operations such as scrolling through options.
- ____ **AutoRange** - Similar to Autoset. In straightforward cases, this automatically detects and selects an appropriate voltage and time scale and positions the signal.
- ____ **Save/Recall** - Displays the menu for saving and recalling waveforms and front-panel setups.
- ____ **Print** - Can be used to print or quickly save waveforms to a file.
- ____ **Measure** - Displays the automated-measurement menu. User can select a variety and multiple measurements for multiple signals
- ____ **Help** - This help function will display a detailed help file for the given menu function that is last active. Very useful for learning any control on the O-scope.
- ____ **Autoset** - Automatically adjusts instrument controls to obtain a usable display.
- ____ **Ref Menu** - This menu allows you to recall and display saved waveforms called reference waveforms.
- ____ **Utility** - Displays options such as: system status, I/O, system calibration, and diagnostic menus.
- ____ **Display** - This control defines how the O-scope displays waveforms, and can change the overall appearance of the waveform, i.e., contrast.
- ____ **Default Setup** - Quickly restores most controls to factory setup as shipped
- ____ **Acquire** - This controls how the O-scope acquires waveform data, i.e., sample, peak detect, or averaging (which reduces noise)
- ____ **Run/Stop** – Starts and stops acquisition
- ____ **Single Seq** - This button acquires a single waveform then stops.
- ____ USB FLASH PORT - Used to store and print Waveforms. Insert a Flash Drive here.

After you have identified the above controls on your O-scope and marked them with their corresponding numbers shown in Figure 2, turn on your O-scope using the ON/OFF button located on the top of the O-scope. After a few moments of warming up, answer the following questions based on what you see from the screen.

Q1.

a) How many horizontal grid squares are there?

b) How many vertical squares?

c) What quantity does the horizontal direction represent?

d) What quantity does the vertical direction represent?

e) Each hatch mark represents what portion of a full division?

Vertical Input

The vertical input, also called the signal or voltage input, controls the vertical position of the signal displayed. You can identify the vertical input section by finding the knobs that control Volts/Div and also the coaxial inputs.

Figure 3: Vertical Controls

As you read through the following functional descriptions, find these controls on your oscilloscope. Once again as you find these controls, mark them with the corresponding number as seen in Figure 3.

- ____ Coaxial Connectors - Provides a path into the oscilloscope for the signal you want to measure. Oscilloscopes typically have 1 to 4 input channels. If there is more than one input, the channels will be labeled as Channel l, Channel 2, etc. Each channel provides an independent signal path into the oscilloscope. If there is more than one channel the oscilloscope can usually display all the channels at one time so the user can look at more than one signal.
- ____ **CH 1 Menu and CH 2 Menu** - Assigns instrument control to the selected channel indicated in the display. From here you also have the choice to select signal parameters.
- ____ **MATH MENU** - Displays the menu choices for the math operations and activates the math waveform.
- ____ **POSITION** Knobs - Moves the respective waveform vertically.
- ____ **Volts/Div** Knobs - Adjusts the voltage per division scaling for each channel accordingly.

After you have identified the above controls on your O-scope and marked them with their corresponding numbers shown in Figure 3, take a moment and familiarize yourself with these controls. Note that CH 1 and CH 2 have the same features that work independent of each other; as such, be sure to identify which 'CH' is highlighted or active when using the O-scope. For either channel CH1 or CH2 (not Math signal), use the Volts/Div knob and answer the following questions.

Q2.

a) What is the largest voltage per division that your scope can display?

b) What is the smallest voltage per division that your scope can display?

c) On the maximum setting, how much voltage does the entire screen represent?

Horizontal Input

Figure 4: Horizontal Controls

This section can also be called the ‘Time Base’ section. The controls in this section determine the horizontal screen scale (time). The triggering determines the voltage level of the incoming signal at which the oscilloscope will begin to display the signal. The subtleties of triggering will be discussed in more depth in later lab procedure. As you read through the following functional descriptions, find these controls on your oscilloscope. As you find these controls mark them with the corresponding number as seen in Figure 4.

- ____ **Position** - moves the waveform horizontally.
- ____ **Horizontal Menu** - Displays the menu choices for the horizontal system.
- ____ **Set To Zero** - Automatically displays the signal such that the '0' time is in the middle of the screen. Notice the white arrow on top of the display and how it moves with the horizontal position knob.
- ____ **Second/Div** - Adjusts the horizontal scale, or time scale for all waveforms.
- ____ External Trigger Connector - used for triggering purposes, not important for the time being.

After you have identified the above controls on your O-scope and marked them with their corresponding numbers shown in Figure 4, take a moment and familiarize yourself with these controls. Note that CH 1 and CH 2 are not independently controlled by the horizontal controls, i.e., changing the time scale affects both channels equally. Use the Sec/Div knob and answer the following questions:

Q3.

a) What is the minimum time per division that the oscilloscope can display?

b) What is the maximum time per division that the oscilloscope can display?

c) On the maximum setting, how much time does the entire screen represent?

Probes

Oscilloscope probes conduct the signal from its source to the oscilloscope. All probes have a probe tip, ground clip, and coaxial cable with a connector. We will learn more about the electrical impedance of the probes in a later lab. As the name implies, a probe is the actual tool that 'probes' inside the circuit at any point. For now, there are a few basic principles to know about oscilloscope probes. First, the probe tip is the part that attaches to the circuit where the signal is to be monitored. Secondly, the ground clip connects to ground, or reference point, and should be connected as close as possible to the source of the signal.

Signals

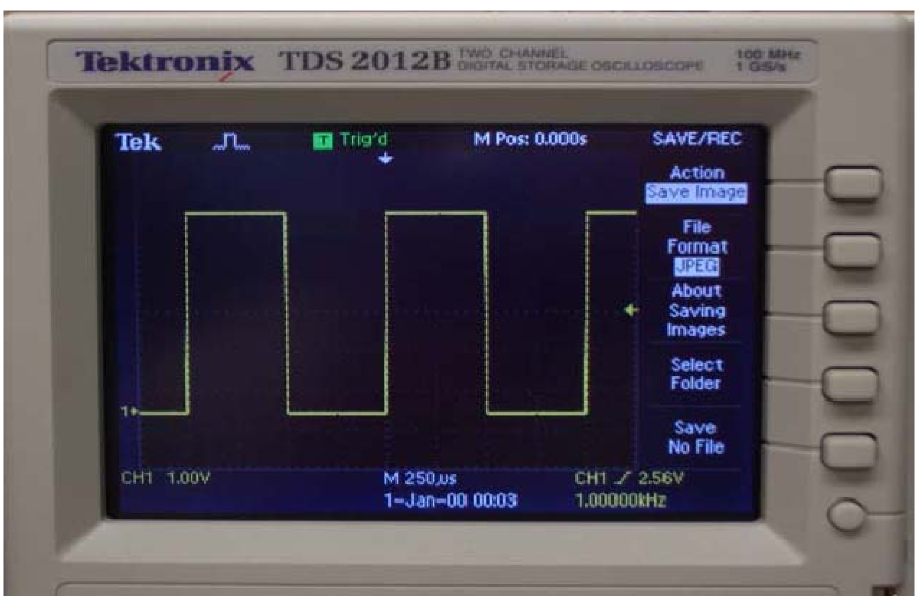


Figure 5: Example Square Wave on O-Scope

Even though you have not been introduced at this point to different waveforms, the only meaningful way of looking at a signal on the O-scope is by viewing an time-varying signal. So without discussing signals themselves, the focus here is to simply view a signal on the O-scope. The following procedure will walk you through the oscilloscope setup and let you view a signal on the screen.

1. Install the probe on CH 1. Connect the probe tip to PROBE COMP and the probe ground to the ground symbol on the front panel. This constantly outputs a 5V, 1 KHz square wave.
2. Set the VOLTS/DIV to 1 V/Div. Use the vertical POSITION knob to center the waveform vertically on screen.
3. Set the SEC/DIV to 250µs.
4. Check that a square wave signal of about five divisions in amplitude is on screen.
5. Check that one period of the square wave probe-compensation signal is about four horizontal divisions on screen.
6. Check that the horizontal POSITION knob positions the signal left and right on screen when rotated.
7. Press CH 1 Menu to remove CH 1 from the display.
8. Press CH 2 Menu and move the probe to the CH2 input.
9. Repeat steps 2 through 6 for Channel 2.

It is important to note that any changes made to the display settings do not in any way change the actual signal. After following the above procedure, you should see something on your screen like Figure 5.

Keep the signal displayed on the screen for the next section. Next, answer the following questions:

Q4. Adjust the following controls and note what effects they have on the signal display:

a) SEC/DIV

b) VOLTS/DIV

c) Vertical-Position knob

d) Horizontal-Position knob

Q5. Change the coupling from DC to AC. By using CH 1 Menu and selecting Coupling in Main menu, change coupling from DC to AC by using Side menu. What happened to the signal display? Why do you think this happened?

Saving a Waveform to a File on your Computer

You can save images or data files of the results displayed on the oscilloscope to the computer connected to the equipment on your bench. You can later transfer the file to your own private media such as a USB flash drive, or email it to yourself, to use in your lab report.

There are two options when saving your data. You can either save it as an image or as data points. When saving an image, a screen shot of the oscilloscope display is saved as a chosen the image type. Saving as a JPEG file is recommended for good image quality and small file size. When saving data, the program saves the data in a .csv (coma separated values) file that can be accessed using any spread sheet program such as Microsoft Excel.

To save a waveform as an image file, use the same square wave signal on the display and work through the following steps. You may refer to Figure 6 as a visual aid for what the final output should look like.

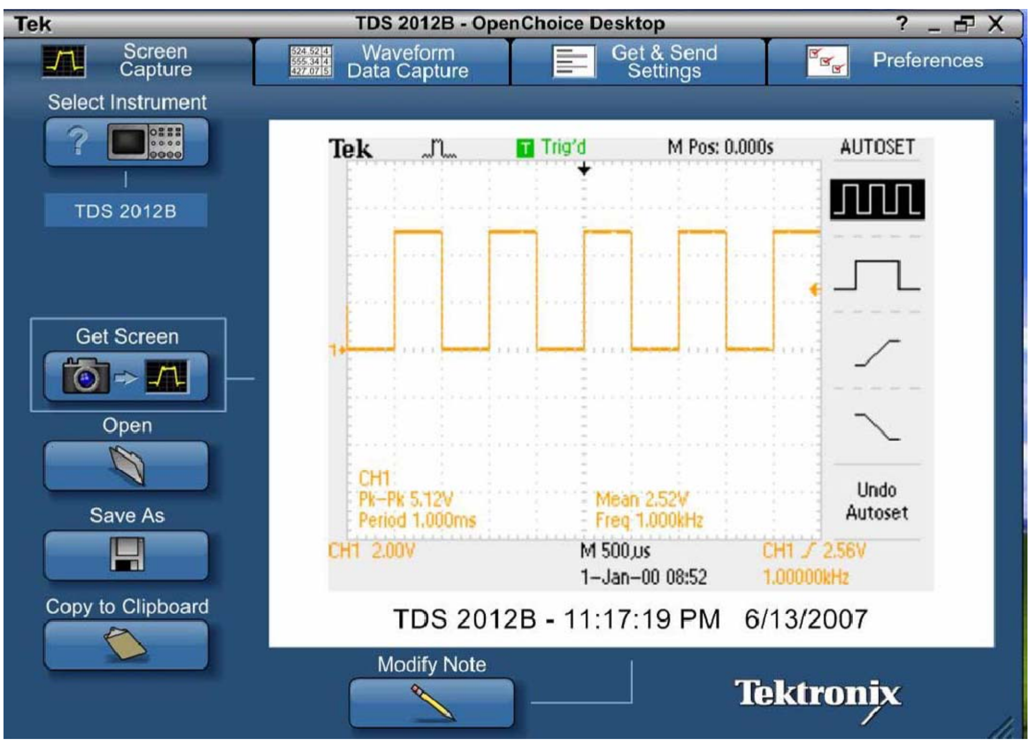


Figure 6: Saving an Oscilloscope Waveform as an Image File

1. Open “OpenChoice Desktop” from the desktop.

2. Once the program opens, you will be prompted to "select the instrument," choose the USB device.
3. To store a screen capture click on the "screen capture" tab.
4. Now click on "Get Screen." This should display the scope screen shot on your window.
5. Click the "Save As" button, specify a path, filename, and image type (JPEG). Click OK when done.

Alternatively, to save the waveform as a collection of data points, do the following steps while referring to Figure 9 on the next page as a visual aid for the final results:

1. Open "OpenChoice Desktop" from the desktop.
2. Once the program opens, you will be promoted to "select the instrument," choose the USB device.
3. To store a waveform capture, click on the "Waveform Data Capture" tab.
4. Select the targeted channel by clicking on the "Select Channels" button.
5. Now click on the "Get Data" button. This should display the scope screen on your window.
6. Click the "Save As" button, specify a path and filename, and then click OK.

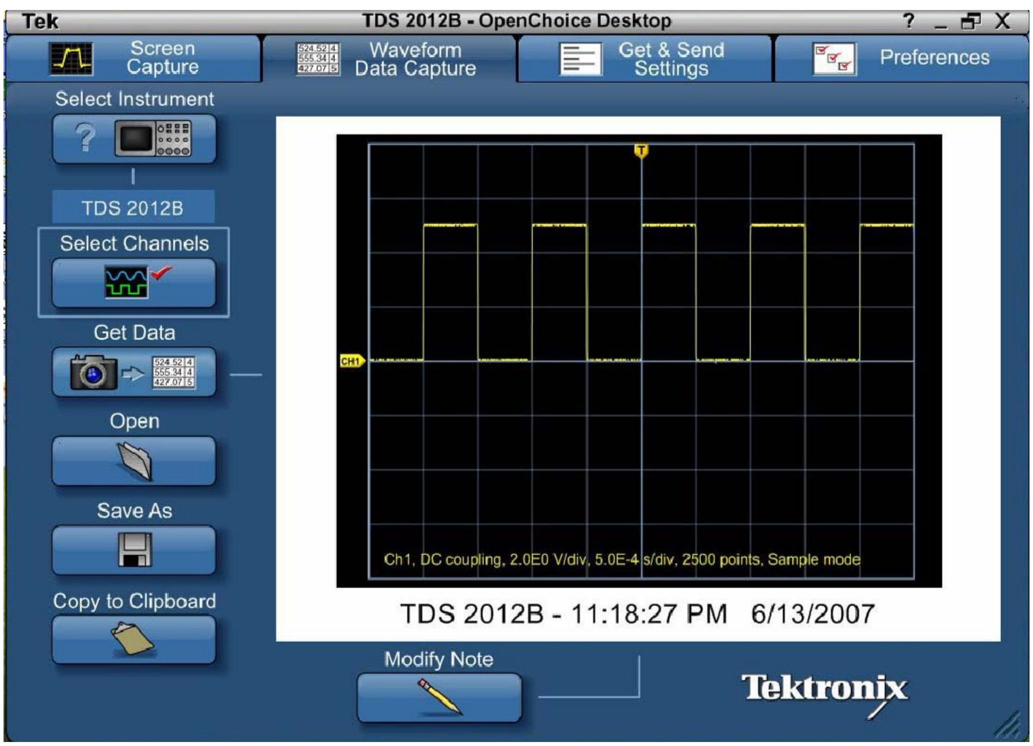


Figure 7: Saving an Oscilloscope Waveform as a Data File

Saving the waveform as a collection of data points allows access to the actual values of the samples of the waveform. For display purposes, you will have to create your own graph by means of another software program such as Excel. However, the real power of having the actual samples of the waveform is the ability to run data analysis from a myriad of software programs, i.e., Matlab.

Saving a Copy directly to a USB Flash Drive (optional)

The TEKTRONIX TDS 2012B is equipped with a USB flash drive port for saving digitized waveforms. The USB flash drive provides mass storage for an unlimited number of oscilloscope setups. Using the same square wave signal on the display, work through the following procedure for saving a waveform while being careful not to format your flash drive when connecting to the scope:

1. Insert Flash Drive into USB port on front panel of O-scope
2. Press the Save/Recall button
3. On the right-hand side of the display, press the button corresponding with the menu option Action until Save Image is highlighted.
4. On the right-hand side of the display, press the button corresponding with the menu option File Format until JPEG is highlighted.
5. On the right-hand side of the display, press the button corresponding with the menu option Save TEKxxxx.jpg, where xxxx are the numerical increments to which waveform you are saving to the file.

As a quick alternative, once you have your desired file format selected, in this case JPEG, go back to step 3 and go to the Save All selection under the Action menu. Then, under the PRINT Button option, press the corresponding button until Saves Image To File is highlighted. From now on, whenever you have a waveform up with whatever menu or measurements displayed on the screen, pressing the Print button will automatically save the screenshot to a file on your flash drive (does not overwrite, automatically creates new file name).

2. Function Generator

Introduction

Function generators are devices that generate various types of periodic signals such as square waves, sine waves, triangle waves and sawtooth waves. The frequency, amplitude, and DC offset of these signals can be adjusted within a wide range of values. Our function generator, shown below in Figure 8, can also be used to generate arbitrary waveforms, different types of noise (white or pink), and also amplitude modulated (AM) signals. A function generator is the "real life" equivalent of a theoretical signal source in a circuit schematic.

Figure 8: Programmable Function Generator HM8131 (courtesy of HAMEG)

As seen from Figure 8, when you power up the function generator, there is a default setting that produces a free-running, 10 Vp-p, 1 kHz, 0 offset sinusoidal signal. The output is automatically turned off though. For basic operation, the user should be able to manipulate these parameters with ease (Voltage level, Frequency, Offset, and type of signal). To do this, let's take a closer look at the front panel of our function generator shown in Figure 9.

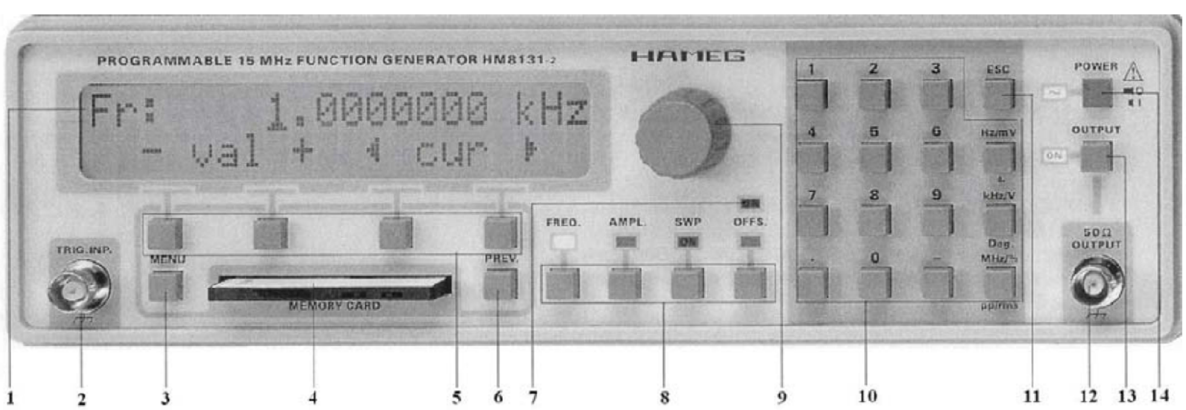


Figure 9: HM8131 front panel controls (courtesy of HAMEG)

Notice the numbers that correspond to the following operations or functions

1. LCD display with 2 lines of 20 characters each

2. Trigger input

3. Menu selection pushbutton

4. Memory card slot

5. Pushbuttons for menu control

6. Menu backspace pushbutton

7. Offset indicator

8. Pushbuttons for menu control

9. Incremental rotational control

10. Keyboard

11. Escape pushbutton, erases an input

12. 50 Ω output {BNC)

13. Pushbutton for switching the output on/off

14. Main switch

The operation of the HM 8131 will be predominantly via the displayed menus (1). Choosing menu options are selected by the menu control buttons (5). The parameters are either set using the keyboard (10) or the incremental control knob (9). Once turned on (14), connect a BNC plug to the output (12). The BNC connector is used to feed the signal to the circuit or to the scope. Even though connected, the output is not necessarily on. To switch the output off and on, use the button indicated by output (13). To facilitate your learning of the function generator, connect the function generator directly to the O-scope and 'find' the default signal. From here, go through the menu options (3) to familiarize yourself with the choices. Don't select options that you are not sure of what they do. Remember you can always move back by (6) or (11). Once you have scrolled through the numerous menus, perform the following functions.

Q6. Show on the O-scope a sine wave at 1.25 kHz and 5 Vp-p.

Q7. Show on the O-scope a square wave at 10.5 kHz, 2.5 Vp, and 1 V offset. Make sure your O-scope is set to DC coupling so the offset is shown. What does the offset do?

Appendix V: Periodic Signals

1. Introduction

Periodic signals are waveform patterns that repeat over and over on a regular time interval. Three common types of signals, sinusoids, square waves, and triangle waves are shown in Figures 1, 2, and 3 respectively.

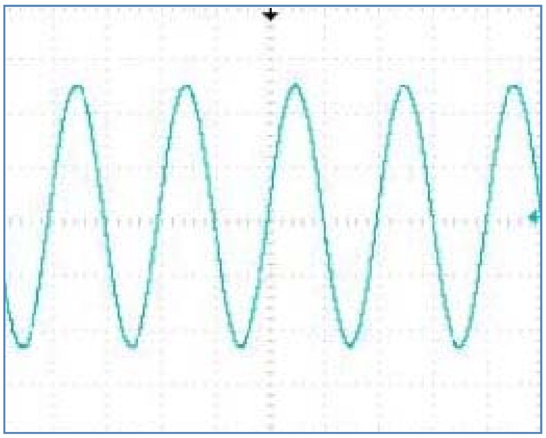


Figure 1: A Sinusoidal Periodic Signal

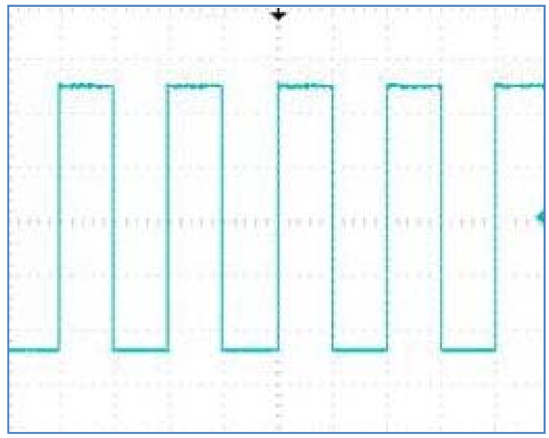


Figure 2: A Square Periodic Signal

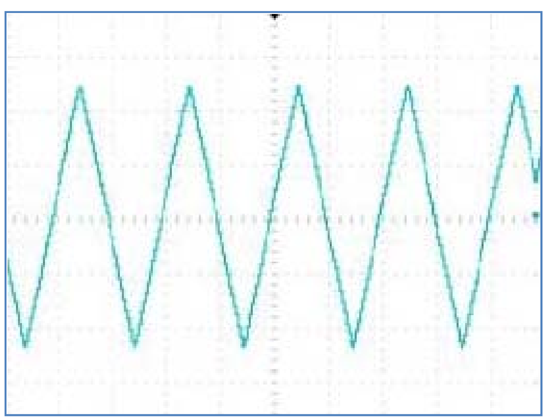


Figure 3: A Triangular Periodic Signal

Properties of Signals

The basic properties of a signal are amplitude and period, or frequency.

- Amplitude (V): Amplitude is a vertical (voltage) measurement. There are a few different ways to express the amplitude of a signal. For instance, the maximum value of a periodic waveform measured from 0 to its peak is often referred to as V_{0-p}. A second way to reference amplitude is from peak to peak ($V_{p.p}$). For the square wave in the Figure 2, $V_{0-p} = 2.5\text{ V}$, and $V_{p.p} = 5\text{ V}$.

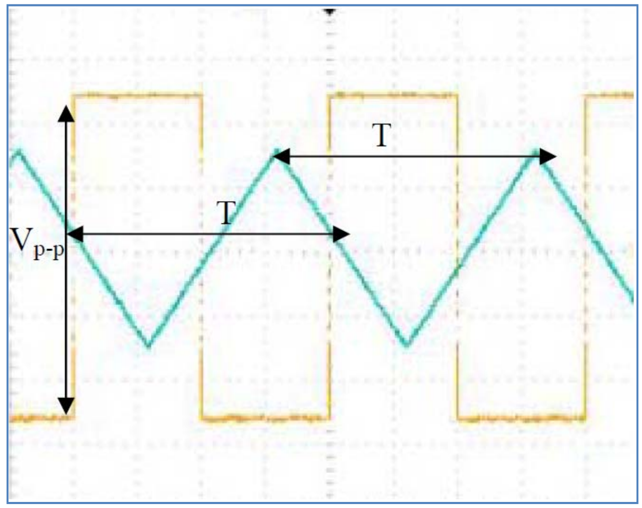


Figure 4: Illustration of Signal Properties

- Period (T): Period is a horizontal (time) measurement. The time interval between any repeated points on a waveform is called its period. A capital T is used to symbolize period. The period may be measured from peak to peak, zero-crossing to zero-crossing, or any other convenient reference point.
- Frequency (*f*): Frequency is also a time based measurement and is defined by the number of times a waveform pattern repeats (or cycles) per a second. Frequency is commonly measured in Hertz (Hz) and is simply calculated by the relationship $f = 1/T$ (Hz). Frequency can also be expressed by the units rad/sec, and is related to Hz by $\omega = 2\pi f$ (rad/sec).

Once again, looking at the two signals in Figure 4, you see a triangular and a square wave. The two signals have different amplitudes but the exact same period. The square wave has its period measured from one *zero crossing* to the next. The triangular wave has its period measured from one *maximum value* to the next. Notice that either way is valid and equivalent.

- DC offset: Another property of a periodic signal is the DC offset. This is the vertical distance (in volts) between the average value of the signal and 0 volts.

A useful measurement that is similar to the average value is the *root mean square*, or *rms* value. A signal's *rms* value is also another way of expressing amplitude. In a sinusoidal signal, the *rms* value is calculated by $\mathrm{V_{rms}} = \mathrm{V_{0-p}}/\sqrt{2}$

- Phase: The final property of periodic signals we will study is phase shift. Phase shift is the distance between reference points on two separate waveforms. This is usually measured in degrees. In a sine wave, one period (T) is 360°.

The phase difference between two waves having the same period is calculated by:

$$\Delta\theta = \frac{\text{Displacement Between Waves}}{\text{Period}} \times 360°$$

For the signals in Figure 6, the displacement between the two signals is approximately 2 squares and the period is 7 squares, so using the above equation we find the phase shift to be approximately 102°.

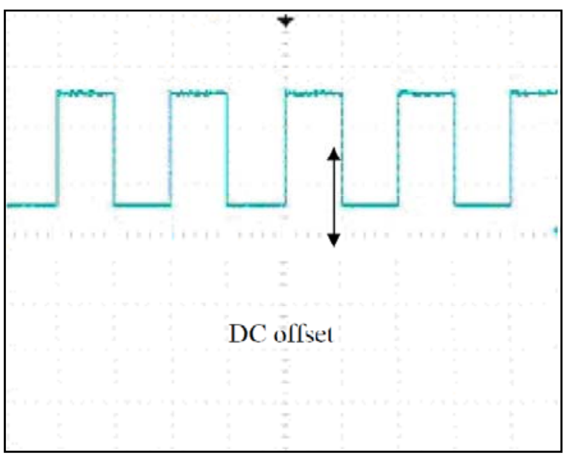


Figure 5: Illustration of DC offset

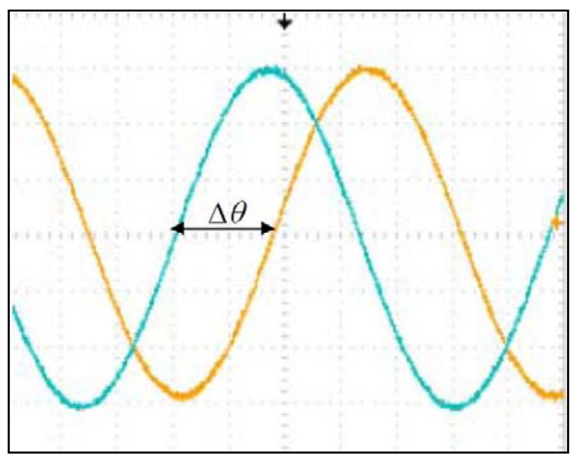


Figure 6: Illustration of Phase Shift

Part II: Oscilloscope Measurement Techniques

Introduction

Before we begin with basic oscilloscope measurement techniques, let's identify what we have learned that serves as a prerequisite to this task. We can categorize three key elements: characteristics of periodic signals, function generator operation, and familiarization with oscilloscope controls. This lab procedure brings these three things together. You will learn how to use the oscilloscope to make magnitude, frequency, and phase measurements. In addition, you will be introduced to the triggering section of the oscilloscope in more detail.

Magnitude Measurements

There are two different ways to make magnitude measurements, namely, manually or automatically. The manual method is most important in that it demonstrates true knowledge of the O-scope as well as enables the user to make 'accurate judgment calls' when the signal exhibits low signal-to-noise ratios (SNR), i.e., the signal is noticeably corrupted by spurious noise. The automatic method deals with scopes that have a built-in capability of making signal measurements in real-time. Not all scopes have this function, as it is not a necessity, and blind reliance on this function can lead to inaccurate measurements. The automatic method is convenient when a signal is 'locked in' and is very 'clean', i.e., no noticeable noise present.

For manually determining the magnitude of the signal, we go back our observation in our previous lab of squares on the display. Notice the "big squares" are considered one division, while the hatch marks represent 0.2 divisions. The word 'division' is exactly what is meant by the Volts/Div knob on your O-scope. Each "big square," or division, is given a voltage value set by the Volts/Div knob. For example, if the O-scope is set at 5V/div, then the height of each square would be 5V and the distance between each hatch would be 1V.

When measuring the amplitude of a signal, it is important to adjust the display so the signal takes up as much vertical space as possible. The oscilloscope is calibrated for "full scale vertical deflection." That is, the oscilloscope gives a better measurement when the signal takes up most of the vertical space. This technique also gives maximum resolution for counting hatch marks.

Let's work through the following example to set up and measure a 0.8 $V_{p.p}$sinusoidal signal.

1. Turn on the O-scope and press *CH 1 MENU*

2. Set the *Coupling* menu option to *Ground* and use the vertical position knob to center the trace on the screen. This is your zero reference point.

3. Change the *Coupling* menu option to *AC*. This option removes any offset which may be present in the signal. Offset will be explained later in the lab.

4. Connect the 50 Ω output of the function generator to CH 1 of the O-scope. (This does not use the probes.)

5. Set the function generator to output a sinusoid at 1 KHz. You should see some kind of signal on the screen unless your *Volts/Div* setting is way off from the actual amplitude of the signal.

6. Set your *Volts/Div* scale at 200 mV/div and your *Sec/Div* time scale to a reasonable value such as 250 µs/div.

7. Adjust the amplitude on the function generator such that the peak-to-peak voltage of the sinusoidal signal takes up exactly 4 divisions on the display. This is our desired 0.8 $V_{p.p}$ (from 4 div * 200 mV/div = 0.8 $V_{p.p}$). You should have something that looks like Figure 10.

To display an automatic measurement, simply push the *Measure* button followed by one of the measurement options in the side menu. This should give you the side menu as in Figure 10. Make sure the Source is set to *CH1* and the *Type* is set to *Pk-Pk*. Push the *Measure* button once more and you will then see the real-time value of your signal to be 800 mV. Changing the amplitude on the function generator will automatically change the measurement on the scope. Note that you can choose any number of measurement types by scrolling through the *Type* option. Now you can answer the following question.

Q1 Setup a 1 KHz sinusoidal signal with a magnitude of 6 Vp-p and save the waveform to a file.

a) What should the Volts/Div setting be to display the largest signal without going off the edge of the screen?

b) At this setting, how many vertical divisions should the signal take up?

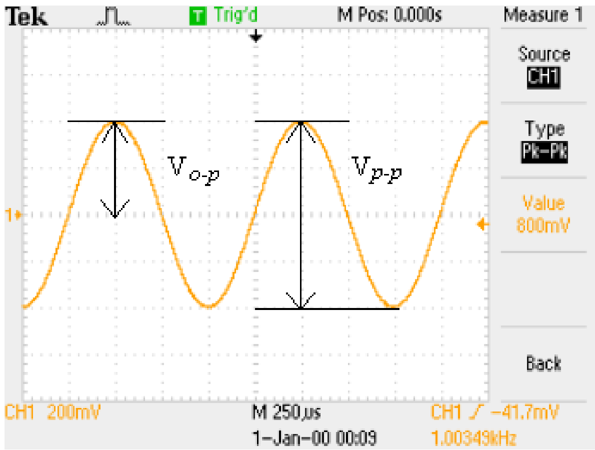


Figure 10: Magnitude Measurements

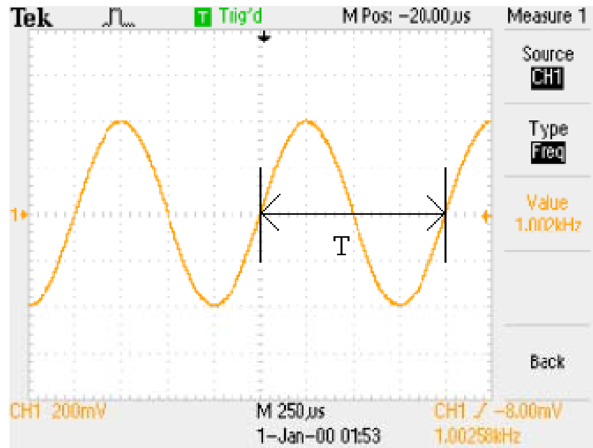


Figure 11: Timing Measurements

Timing Measurements

Following the same procedure as in the magnitude measurements section, re-obtain the original 0.8 $V_{p.p}$, 1 KHz sinusoidal signal back on the O-scope. When taking timing measurements, it is always important to adjust the display so that signal takes up as much horizontal space as possible. That is, a period measurement taken from a display of slightly more than 1 period is better than one taken from a display of 2 or more periods.

Work through the following example to measure the period of a sinusoidal signal:

1. After setting the O-scope and function generator to view a 0.8 $V_{p.p}$, 1 KHz sinusoidal signal, you should see 2 periods of the signal on the screen. (250µs/div time scale)
2. The number of divisions from "zero-crossing" to "zero-crossing" should be 4.
3. The period is then T = 4 Divisions * 250µs/div = 1 msec.
4. The frequency F = 1/T = 1/(1 msec) = 1 KHz, as expected. Check to see if you have something like Figure 11.

The automatic measurement method is nearly identical to the previous section. Push the *Measure* button followed by one of the measurement options in the side menu. Once again, this should give you the side menu in Figure 11. Make sure the *Source* is set to *CH1* and the *Type* is set to *Freq*. Push the *Measure* button once more and you will then see the real-time value of your signal to be approximately 1 KHz. Changing the frequency on the function generator will automatically change the measurement on the scope. You may notice that the automatic frequency measurement tends to vacillate a bit. This is common. Frequency stability and frequency drift are real, non-trivial issues. Now you can answer the following question.

Q2 Set up a 1.5 KHz sinusoidal signal with a magnitude of 6 $V_{p.p}$. Adjust the Oscilloscope to the best display for taking a measurement of the period.

a) What is the period of the signal?

b) From the measured period, calculate the frequency.

c) How does this frequency compare to the displayed frequency on the function generator? (a % Error calculation would be in order here)

d) Save your waveform to a file. Label the period on the waveform.

Phase Measurements

Two signals may have the same amplitude and frequency but still be different from each other because of differing phases. Phase describes when the period of one signal starts in time relative to another signal, or another point in time. The phase difference, or phase shift, between two signals is usually expressed in degrees, with one entire period being set to 360 degrees. This makes perfect sense for a periodic signal, that is, 360 degrees later you are back to where you started (1 period).

Triggering

Just as a trigger on a gun sets into motion a seemingly instant *reaction*, a trigger on an oscilloscopes sets into motion a seemingly instant capture. Triggering determines what voltage level on the signal the oscilloscope will “lock on to." It is used to synchronize the signal so it doesn't "move around" on the display. The actual voltage level that is being used to trigger can be seen as an arrow marker on the right hand side of the signal frame. You may have already noticed this in Figs. 10, 11. That arrow, which indicates the triggering voltage level, can easily be adjusted up or down by the *Trigger Level* control knob.

Let’s take a closer look at some basic options for triggering and what they mean. As a starting point, set up the signal described in **Q2**. Next, push the *Trig Menu* button to show the side menu on the display. Change the option *Mode* to *Normal*. Now adjust the trigger level such that it is just above or below the max. or min. signal level respectively. What do you observe happens? You should observe that the last good capture is frozen on the screen, indicated by it being shaded. The O-scope is no longer capturing a signal, because the signal does not reach the trigger level to set off a capture. Therefore, the Normal Mode displays waveforms only when the O-scope detects the trigger condition and displays older waveforms until the oscilloscope replaces them with new ones. Now press the *Force Trig* button and observe what happens. You should notice that the waveform 'jumps' to a different location on the screen. This is a result of forcing the trigger to take a single snapshot of the signal at that specific moment in time. The position of the signal is likely to change from one forced trigger to the next. To quickly get back to a reasonable trigger level, press the *Set to 50%* button and observe where the trigger level lands. You should note that the *Set to 50%* intuitively sets the trigger level to be approximately halfway between the minimum and maximum voltage levels.

Using the same signal, let's change the *Mode to Auto*. If the trigger level is at 50%, nothing seems to happen. Once again, let's move the trigger level such that it is just above or below the max. or min. signal level respectively. What do you observe this time? Although the waveform is still active and being captured, you should see the waveform racing across the screen and unsynchronized. Therefore, the Auto Mode forces the O-scope to trigger when it does not detect a trigger within a certain amount of time.