ECE240L

Electrical Engineering Fundamentals

Laboratory Manual

California State University, Northridge

Revised by:Soraya Roosta

CSUN | Engineering and Computer Science | Electrical and Computer Engineering

Acknowledgements

Special thanks to my T.A., Dominic Guerrero, for his exceptional editing skills and for his help in modifying the experiments.

I would also like to acknowledge the support and encouragement of Dr. Ashley Geng.

Introduction

This lab manual is designed to provide an in depth detailed practical experience for DC and AC circuits. Users will become familiar with common lab equipment such as Digital multimeters, function generators, power supplies, and oscilloscopes. Circuits used in the coming text include series, parallel, combination, Thevenin, Wheatstone bridge, operation amplifiers, RC, RL and RLC. Most of the experiments have been designed to utilize PSPICE to simulate DC and AC circuits.

TABLE OF CONTENTS

EXPERIMENT 1 : INTRODUCTION TO LAB EQUIPMENT

OBJECTIVE

Be able to measure resistance, voltage, and current.

EQUIPMENT NEEDED

- Power Supply
- Digital Multimeter
- Breadboard
- Resistors

BREADBOARD

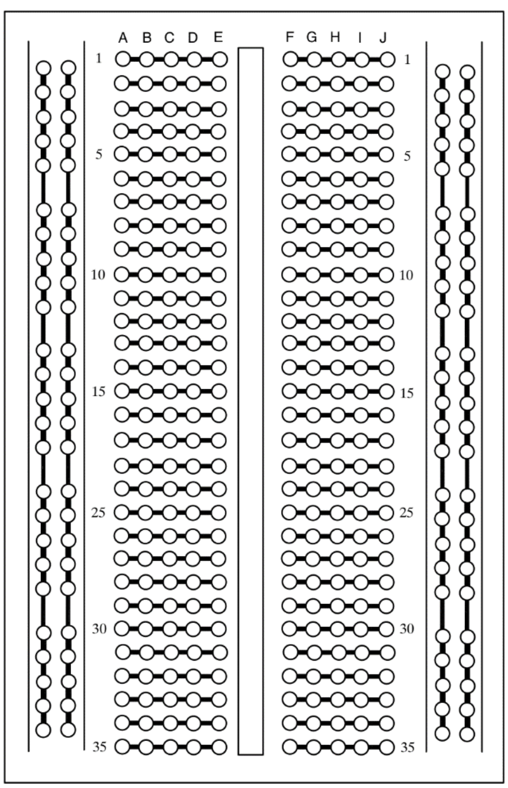


Figure 1.1

Figure 1.2

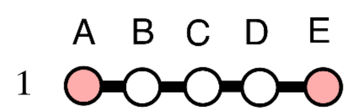


Figure 1.3

The breadboard has two basic internal connections, the vertical and horizontal. The verticals are connected from top to bottom, if 5 volts is placed on the very first pin, 5 volts will be present at the very last pin and any pin in between; this is demonstrated in Figure 1.2 from red pin to red pin. The horizontal strips are connected by every 5 pins. Looking at Figure 1.3 pin A1 to E1 are connected internally.

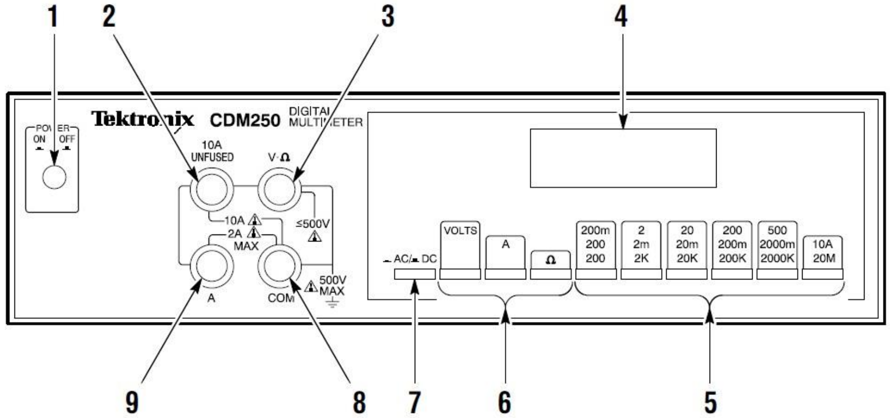


Figure 1.4

1	Power button
2	10A jack for measuring up to 10A. This jack will be unused throughout this course.
3	Voltage jack for measuring voltage. Commonly the red test lead is placed here for the positive.
4	Display
5	Range selection for the desired measurement.
6	Measurement type selection, voltage, amperage or resistance.
7	AC (depressed) or DC (extended) button.
8	Common jack; this port will always be utilized and is commonly occupied by the black test lead.
9	2A jack for measuring up to 2A. When measuring current in this course the red test lead will be connected here.

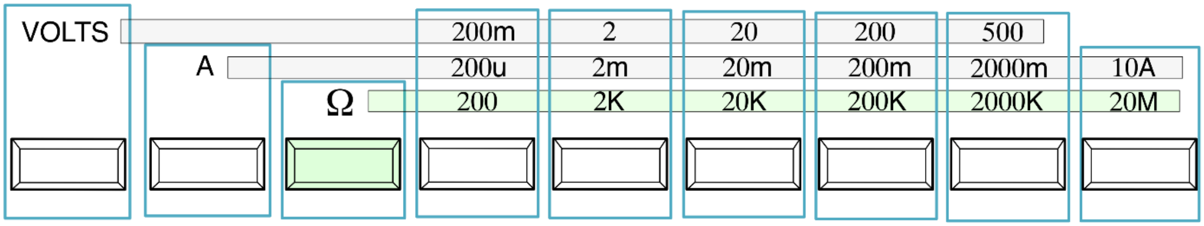


Figure 1.5

When measuring a 1.1k resistor the Ω button should be selected and the corresponding range values are indicated on the bottom row (green highlight in Figure 1.5). The range to be selected is 2K because 2K is the next value above 1.1k.

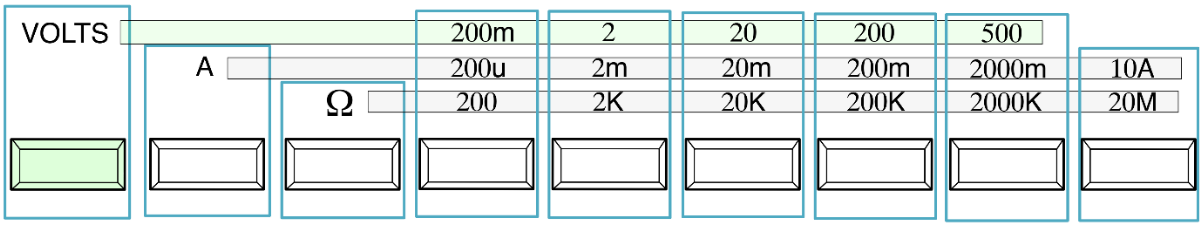


Figure 1.6

When measuring a node that is expected to be 6 volts, the volts button should be selected and the range values are based on the top row of values. The range to be selected is 20 because 20 is the next value above 6.

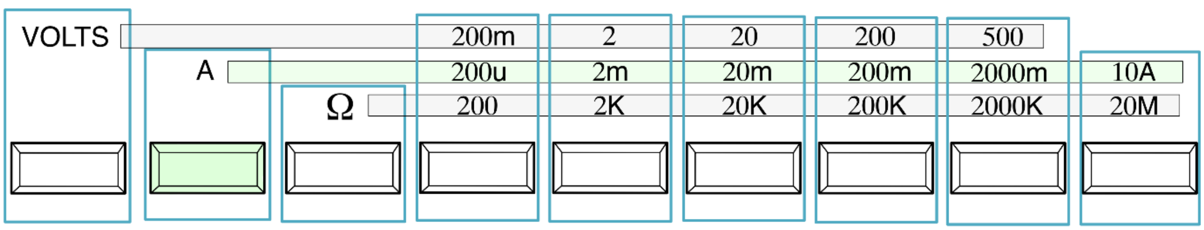
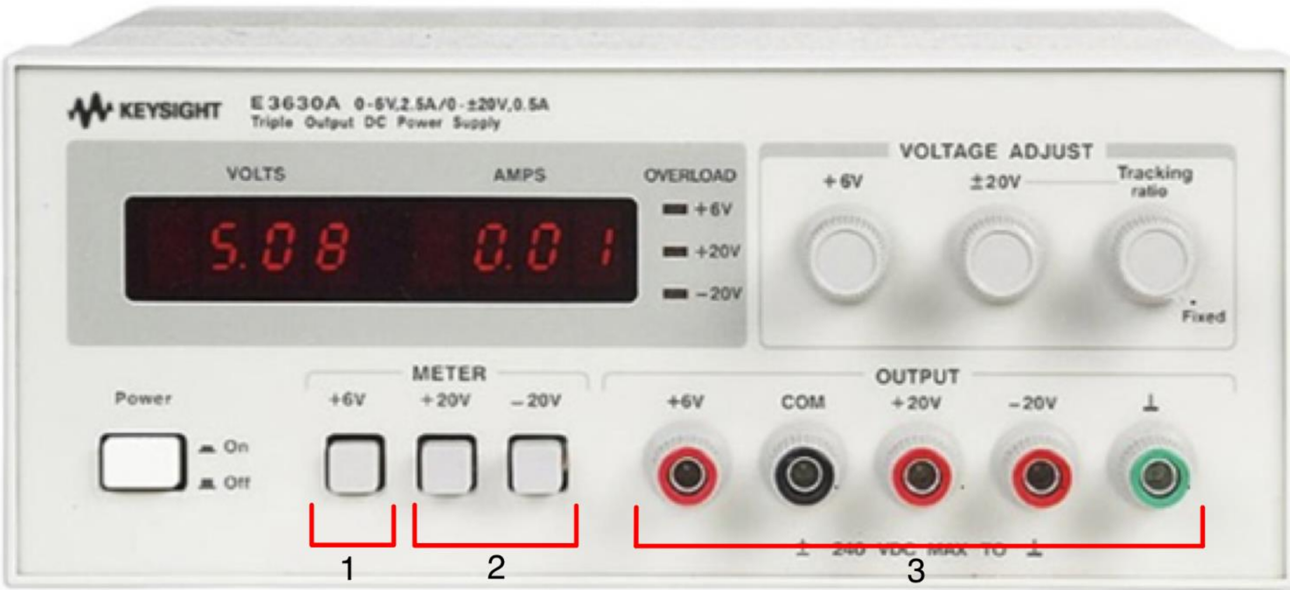


Figure 1.7

When measuring amperage at a node that is expected to be 1mA the amperage button should be selected and the corresponding range values are indicated on the middle row (green highlight in Figure 1.7). The range to be selected is 2m because 2m is the next value above 1m.



1	The "+6V" changes the voltage when the meter "+6V" is depressed.
2	The "$\pm 20V$" changes the voltage when the meter "+20V" or "-20V" respectively.
3	The output region is always active while the power supply is on, meaning each terminal is putting out the voltage that was selected regardless of the meter button that is selected.

THEORY

Ohms law is a relationship between voltage, current and resistance. In order to utilize this law, two of the three variables must be known. The following combinations are as follows:

$$V = IR \qquad I = \frac{V}{R} \qquad R = \frac{V}{I}$$

By memorizing the first equation it is possible to find the other two by isolating the variable in question. When summing resistors in series (in a row one after another), one just added the two values together to find the total resistance of the two combined.

PRELIMINARY CALCULATIONS

1. Calculate R_T, V_1,V_2 and I for Figure 1.8.

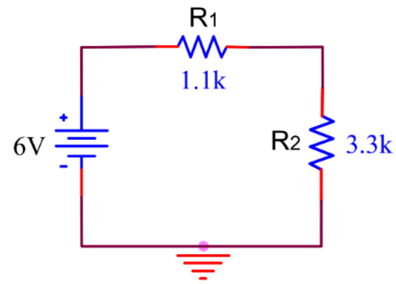


Figure 1.8

	R_T	V_1	V_2
Calculated			

2. Calculate R_T, V_1,V_2 and I for Figure 1.9.

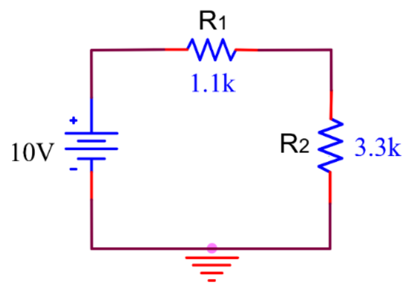


Figure 1.9

	R_T	V_1	V_2
Calculated			

PROCEDURE

1. Measure R_1 and R_2 record the values in Figure 1.10.

2. Place R_1 and R_2 as depicted in Figure 1.10. Measure and record R_T as shown in Figure 1.10.

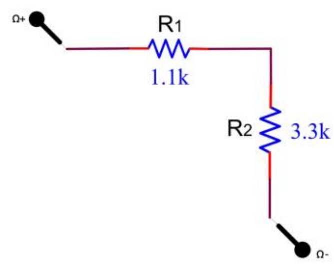


Figure 1.10

3. Turn on the power supply and adjust power supply to 6V by adjusting the "+6V" knob until the display reads approximately 6V; turn off the power supply. Connect the red cable to the "+6V" output and the black to "COM".

4. Connect the 6V power supply to R_1 and R_2 as depicted in Figure 1.11. Measure and record V_1 and V_2.

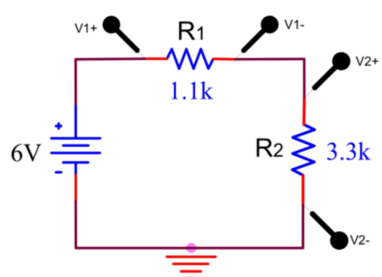


Figure 1.11

5. Turn off the power supply and break the connection between R_1 and R_2 to measure current (Figure 1.12). The positive lead of your DMM will be connected to R_1 and the negative lead will be connected to R_2. Once the probe placement is correct turn on the power supply and record the measurement.

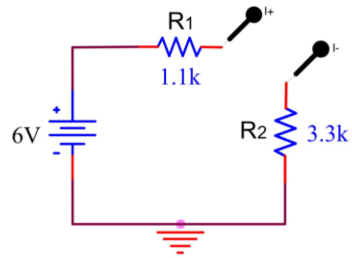


Figure 1.12

	R_1	R_2	R_T	V_1	V_2	I
Calculated						
Measured						
Percent Error						

6. Reconnect the circuit as depicted in Figure 1.13 to measure voltage. Measure V_1 and V_2 and record the obtained values.

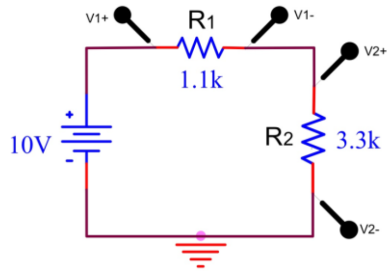


Figure 1.13

7. Turn off the power supply and move the positive cable from "+6V" terminal to the "+20V" terminal. Change the meter button from "+6V" to "+20V". Turn on the power supply and adjust the "$\pm 20V$" knob till the display reads approximately 10V.

8. Turn off the power supply and break the connection between R_1 and R_2 to measure current (Figure 1.14). The positive lead of your DMM will be connected to R_1 and the negative lead will be connected to R_2. Once the probe placement is correct turn on the power supply and record the measurement.

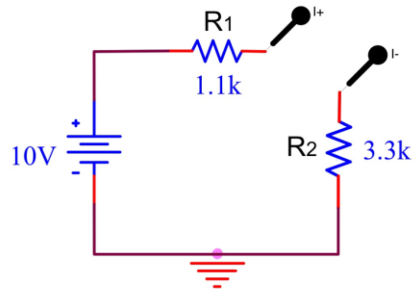


Figure 1.14

	R_1	R_2	R_T	V_1	V_2	I
Calculated						
Measured						
Percent Error						

EXPERIMENT 2 : SERIES AND PARALLEL CIRCUITS

OBJECTIVE

Comprehending series and parallel circuits.

EQUIPMENT NEEDED:

- DC Power Supply
- DMM
- Resistors

THEORY

Kirchhoff's Laws:

Kirchhoff's Voltage Law: The algebraic sum of the voltages around any closed path is zero.

$$\sum_{i=1}^{N} v_i = 0$$

Kirchhoff's Current Law: The algebraic sum of the currents at any node is zero.

$$\sum_{i=1}^{N} i_i = 0$$

Series Circuits:

A series circuit has the same current through each element but different voltages. The total series resistance R_S is given by:

$$R_T = R_1 + R_2 + \cdots + R_{N-1} + R_N$$

The Kirchhoff's voltage law indicates that:

$$V_S = V_1 + V_2 + \cdots + V_{N-1} + V_N$$

Parallel Circuits:

A parallel circuit has the same voltage across each branch but different currents. The total parallel resistance, *Rp* is given by:

$$\frac{1}{R_T} = \frac{1}{R_1} + \frac{1}{R_2} \ldots + \frac{1}{R_{N-1}} + \frac{1}{R_N}$$

Kirchhoff's current law states:

$$I_P = I_1 + I_2 + \cdots + I_{N-1} + I_N$$

PRELIMINARY CALCULATIONS:

1. Calculate R_T, V_1,V_2, and I for Figure 2.1.

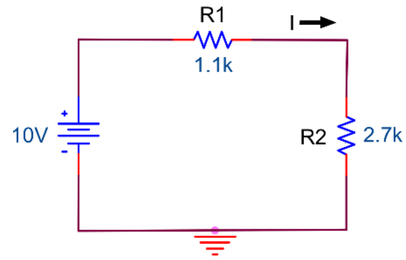


Figure 2.1

	R_T	V_1	V_2	I
Calculated				

Table 1

2. Calculate R_T, V_1,V_2, V_3 and I_1, I_2, I_3, I_T,for Figure 2.2.

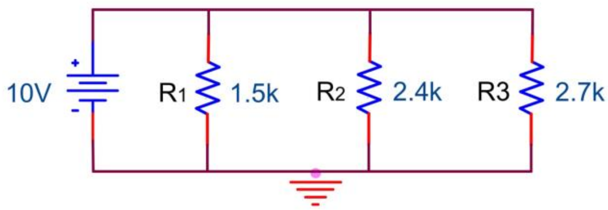


Figure 2.2

	R_T	V_1	V_2	V_3	I_1	I_2	I_3	I_T
Calculated								

PROCEDURE

1. Obtain and measure 1.1K resistor (R_1) and a 2.7K resistor (R_2). Record the values.
2. Construct the circuit in Figure 2.3 and measure R_T. Record the values.

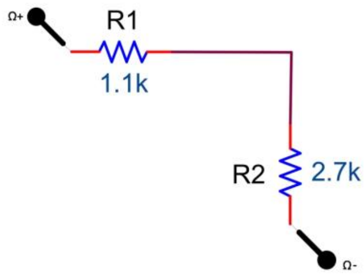


Figure 2.3

3. Construct Figure 2.4 and measure and record V_1 and V_2. Calculate I based on measured voltages and resistances.

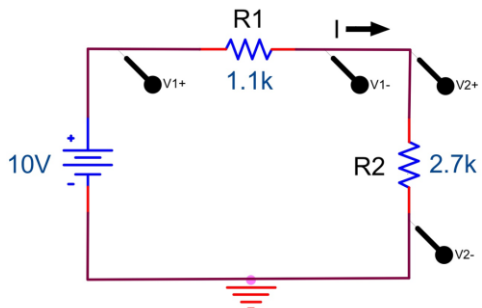


Figure 2.4

	R_1	R_2	R_T	V_1	V_2	I_T
Calculated						
Measured						
Percent Error						

$$\frac{Measured - Theoretical}{Theoretical} \times 100 = Percent\ Error$$

3. Measure R_1, R_2, and R_3 (independently and record the readings. Construct the circuit in Figure 2.5 and measure R_T. Record the value.

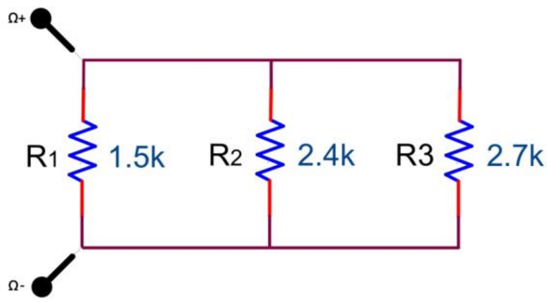


Figure 2.5

4. Construct Figure 2.6.Measure and record V_1, V_2, and V_3. Calculate the current using the taken measurements, annotate them.

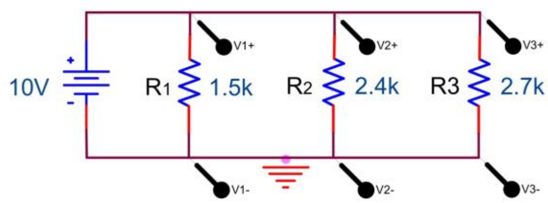


Figure 2.6

	R_1	R_2	R_3	R_T	V_1	V_2	V_3	I_1	I_2	I_3	I_T
Calculated											
Measured											
Percent Error											

EXPERIMENT 3 : SERIES-PARALLEL CIRCUIT

OBJECTIVE

Understanding mixed circuit logic.

EQUIPMENT NEEDED:

- DC Power Supply
- DMM
- Resistors

THEORY

Three basic laws can be used to solve a vast majority of circuits, Kirchoff's voltage law (KVL), Kirchoff's current law (KCL) and Ohm's law. KVL states that the sum of all the voltage drops in a circuit must equal zero. KCL states that the current entering a node must be equal to the current leaving the node. Ohms law shows the correlation between voltage, current, and resistance and is expressed as $V = IR$.

PRELIMINARY CALCULATIONS:

1. For the circuit of Figure 3.1 calculate the resistance between nodes:
   a. A and B
   b. A and C
   c. C and D

   Hint: Part c cannot immediately be reduced using series and parallel combinations.

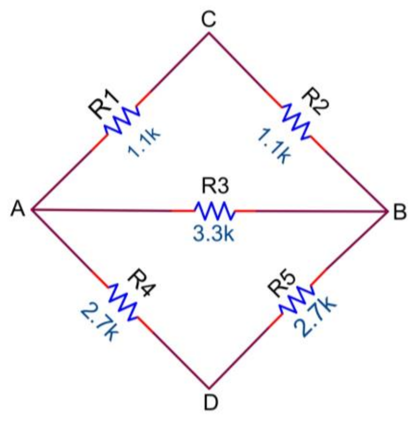


Figure 3.1

	R_{A-B}	R_{A-C}	R_{C-D}
Calculated			

2. Calculate V_3 and V_4 for the circuit in Figure 3.2.

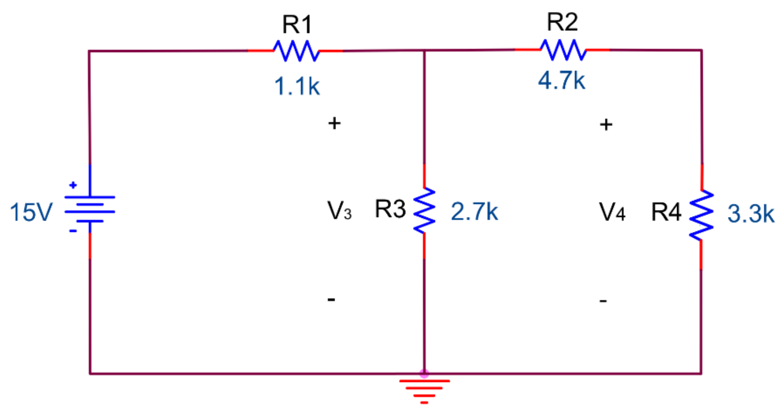


Figure 3.2

	V_3	V_4
Calculated		

3. Calculate all voltages and currents in Figure 3.3 and Figure 3.4.

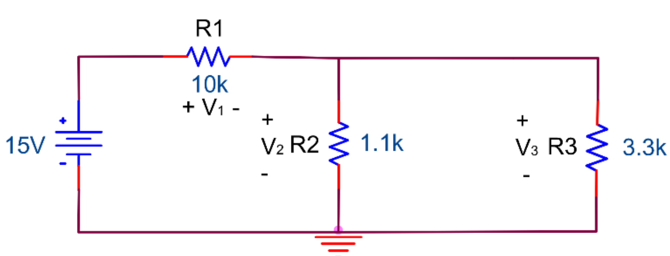


Figure 3.3

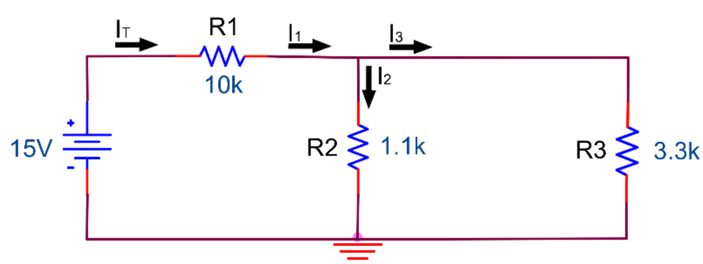


Figure 3.4

	V_1	V_2	V_3	I_1	I_2	I_3	I_T
Calculated							

4. Calculate all voltages and currents in Figure 3.5 and Figure 3.6.

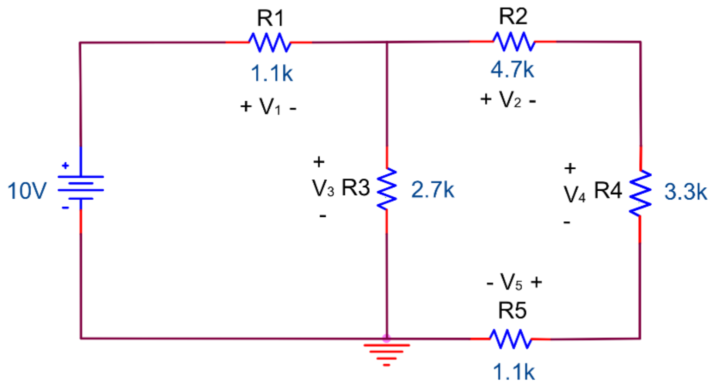


Figure 3.5

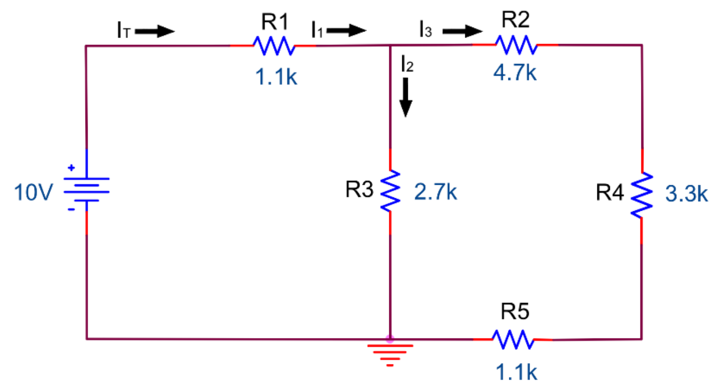


Figure 3.6

	V_1	V_2	V_3	V_4	V_5	I_1	I_2	I_3
Calculated								

PROCEDURE

1. Measure and record R_{ab} (Figure 3.7), R_{ac}(Figure 3.8), and R_{cd} (Figure 3.9).

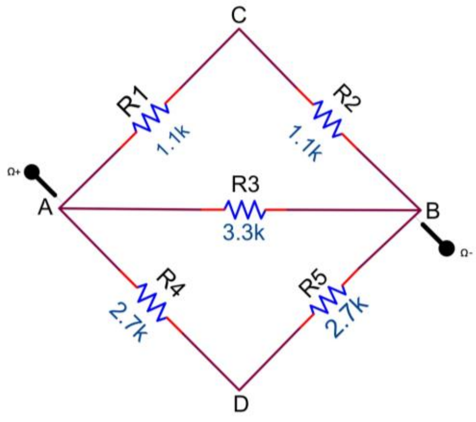


Figure 3.7

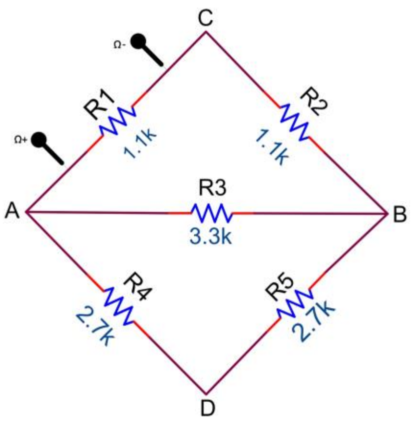


Figure 3.8

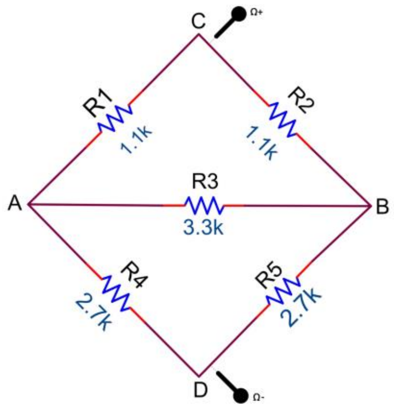


Figure 3.9

Resistance	Calculated	Measured	Percent Error
R_{ab}			
R_{ac}			
R_{cd}			

2. Construct the circuit in Figure 3.10. Measure V_3 and V_4 using the DMM. Calculate the percent error.

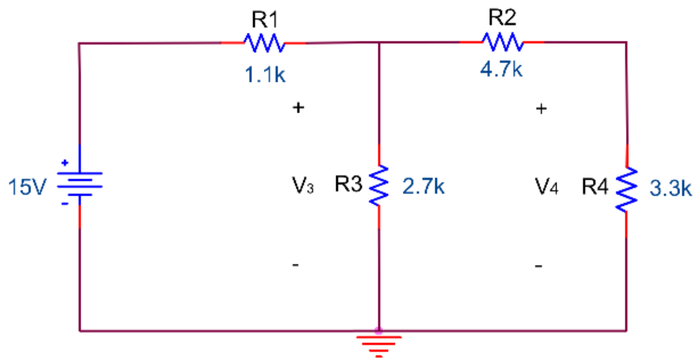


Figure 3.10

	V_3	V_4
Calculated		
Measured		
Percent Error		

3. Measure and record the voltages and currents depicted in Figure 3.11 and Figure 3.12.

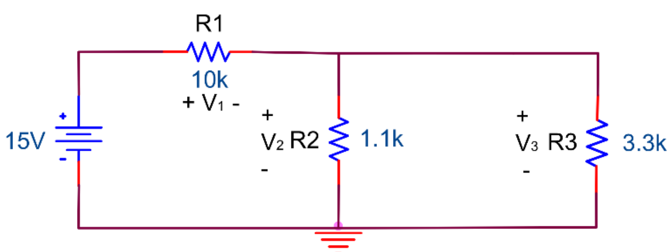


Figure 3.11

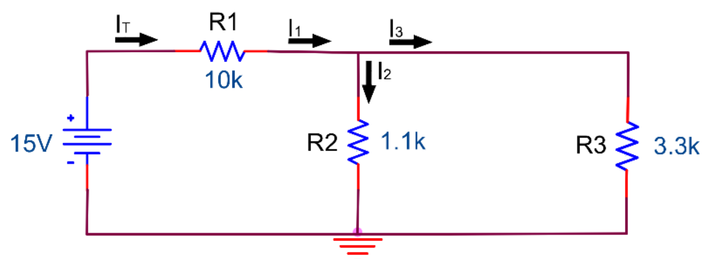


Figure 3.12

	R_1	R_2	R_3	V_1	V_2	V_3	I_1	I_2	I_3	I_T
Calculated										
Measured										
Percent Error										

4. Construct Figure 3.13. Using the DMM, measure each of the variables and calculate the percent error for each.

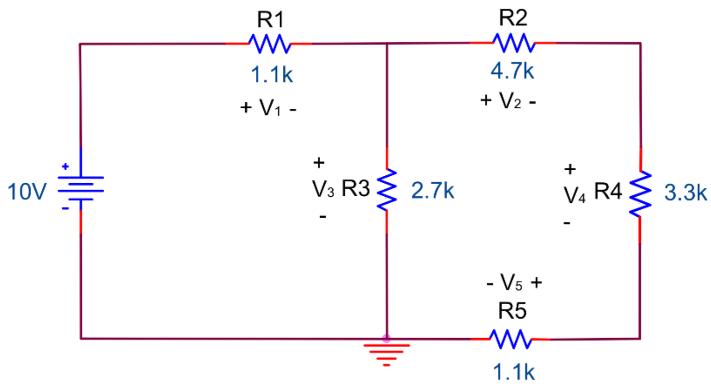


Figure 3.13

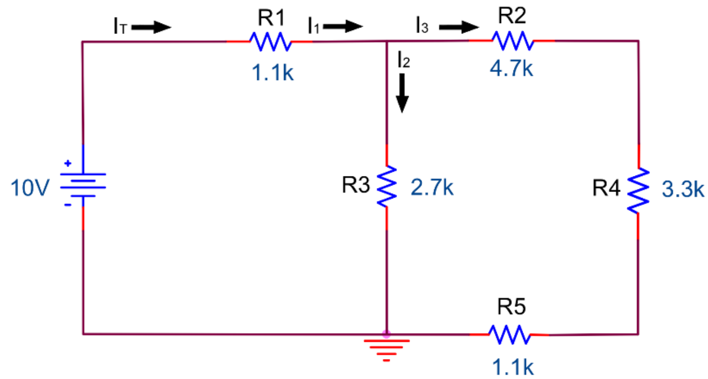


Figure 3.14

	V_1	V_2	V_3	V_4	V_5	I_1	I_2	I_3	I_4	I_5
Calculated										
Measured										
Percent Error										

EXPERIMENT 4 : COMPUTER SIMULATION

OBJECTIVE

Introducing basic operations of computer simulations for circuit design.

EQUIPMENT NEEDED

- Capture CIS 17.4

THEORY

Cadence is an overarching computer program that can simulate a vast number of tasks to include circuit design, PCB design and simulations. Capture CIS is the main program for building circuits and PSPICE is an embedded feature that simulates the built circuit. In this course Capture CIS and PSPICE will be utilized heavily to validate lab results.

a. To start a new project navigate to the top left of the page and click on the icon that looks like a page with an addition symbol.

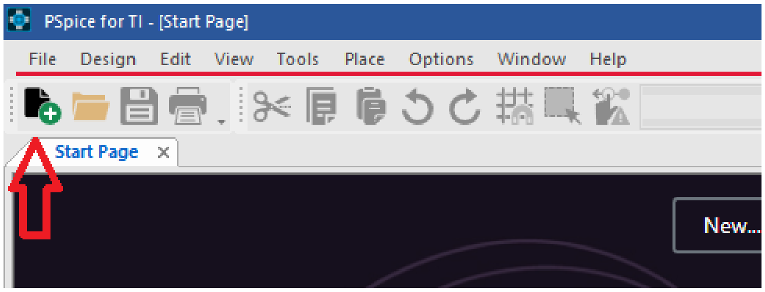


Figure 4.1

b. It is crucial that “Create a blank project” is selected.

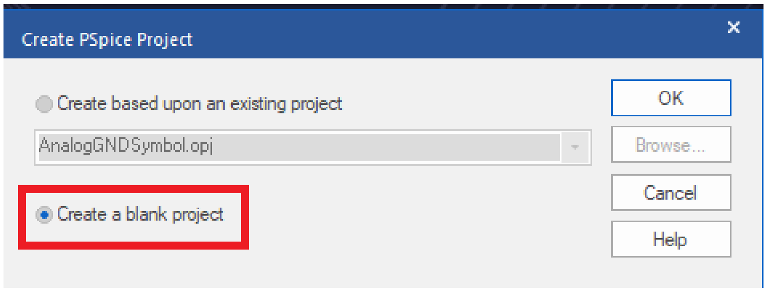


Figure 4.2

c. Name the document and save it to a location that you desire.

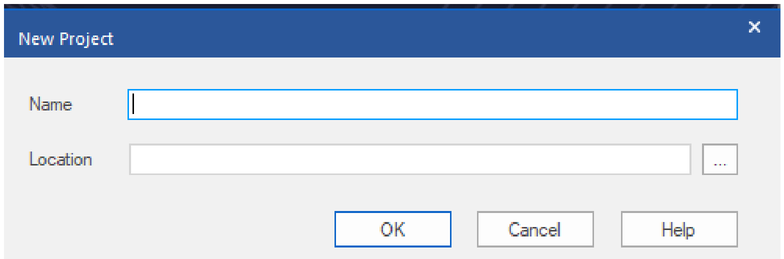


Figure 4.3

d. Parts can be placed by going up to the header and selecting “Place”. All the components for DC circuits can be found here. The voltage source can be found under “**Source**”→ “**Voltage Sources**”→ “**DC**”.

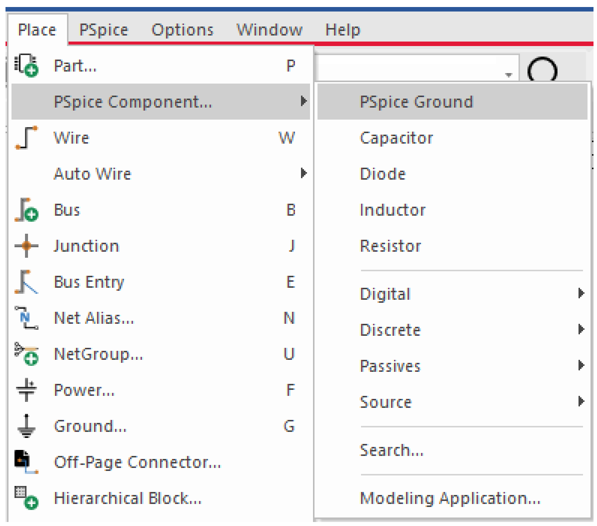


Figure 4.4

e. To connect parts together press “W” on your keyboard. Each component has two small red or white boxes, these boxes are where you click when trying to wire two components together. When a successful connection has been made the wire will look like Figure 4.5.

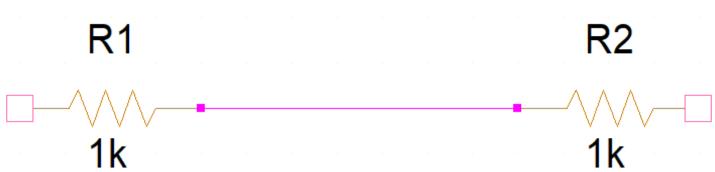


Figure 4.5

f. After all components have been connect correctly (looks like circuit from the lab manual) a simulation profile needs to be generated to begin testing. The simulation profile is under the tab "**PSpice**"→"**New Simulation Profile**". Name the file and continue.

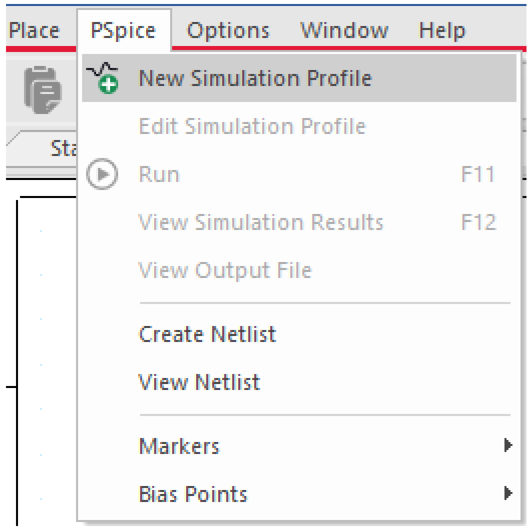


Figure 4.6

g. The following screen is where adjustments can be made to the specifications of the simulation. For DC purposes these parameters are not important, these parameters are important for AC analysis. For the time being, click "OK".

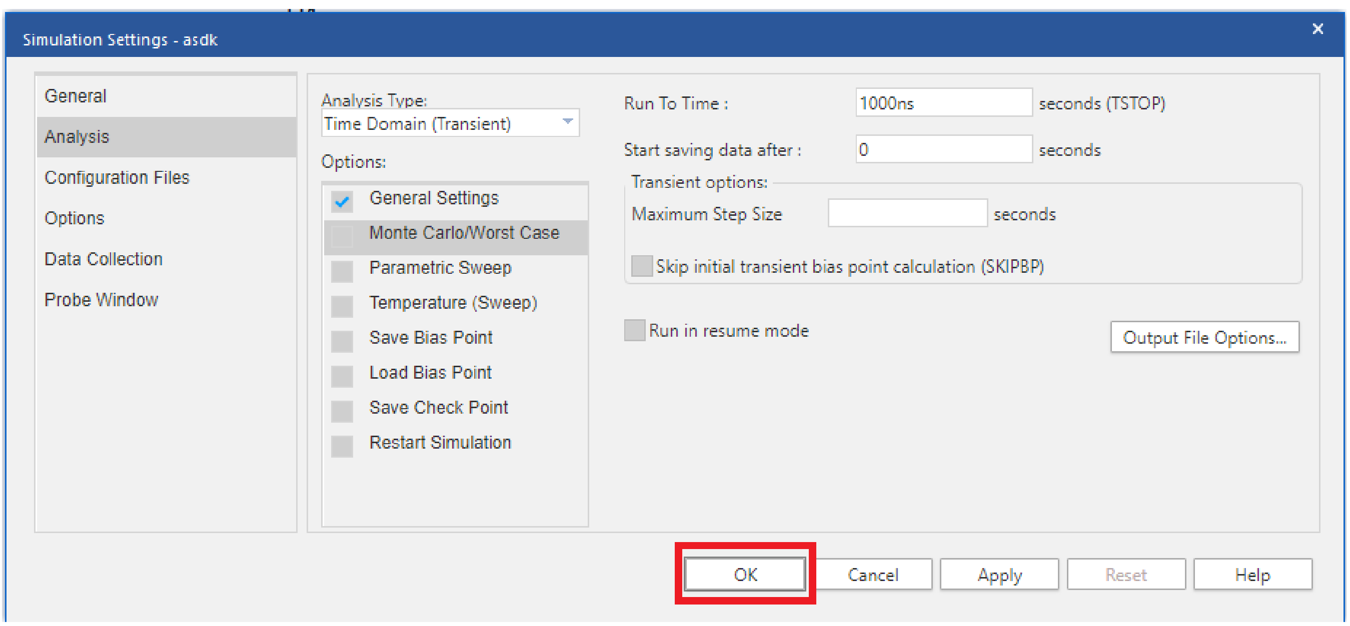


Figure 4.7

h. After the successful completion of the simulation profile it must be executed. At the top of the screen there is a button shaped like the **YouTube button**, click it to run the simulation.

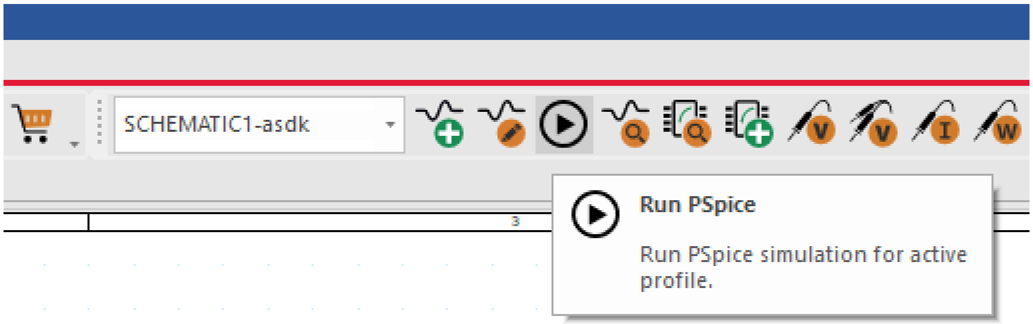


Figure 4.8

i. A secondary window will popup after the simulation profile is executed. This window may be closed, it is vital for AC analysis but not DC. Back at the main screen where the circuit was built there will be a "**VD**" and "**CD**" symbol at the top of the screen. These two symbols will turn on/off the voltage and current at all nodes. Turn both on.

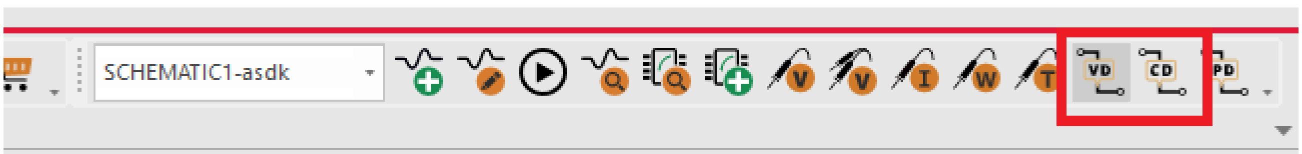


Figure 4.9

j. When the "**VD**" and "**CD**" buttons are activated, the circuit will then display the activated parameters at each node. Refer to figure Figure 4.10.

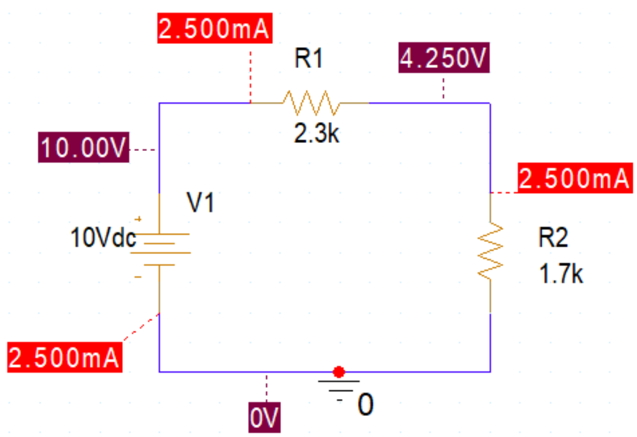


Figure 4.10

PRELIMINARY CALCULATIONS:

1. Calculate all voltages and currents in the Figure 4.11.
2. Simulate Figure 4.14. Ensure that the voltages and currents are displayed on the circuit; take a screen shot. Compare these values with your calculations.

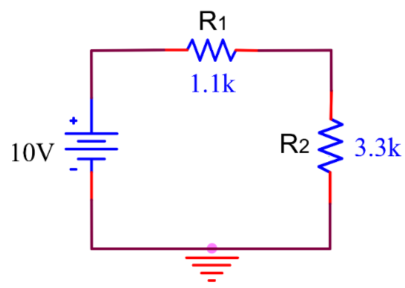


Figure 4.11

DC Circuit Simulation

1. Simulate Figure 4.12 in Capture CIS.
2. Using PSPICE, display all voltage and currents. Take a screenshot or copy and paste the circuit into Microsoft Word.

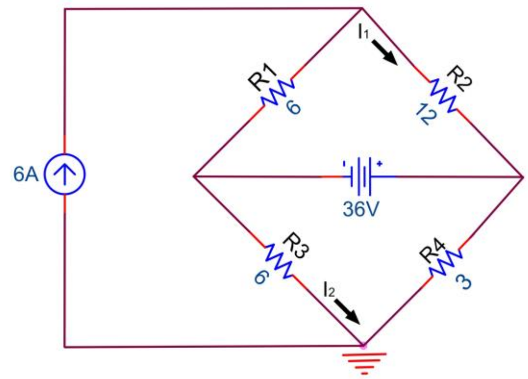


Figure 4.12

AC Circuit Simulation

3. Construct the circuit show in Figure 4.13. The source depicted is a "pulse" voltage source. Source parameters are 1KHz and 30 volts peak to peak. "Place"→"Pspice Component"

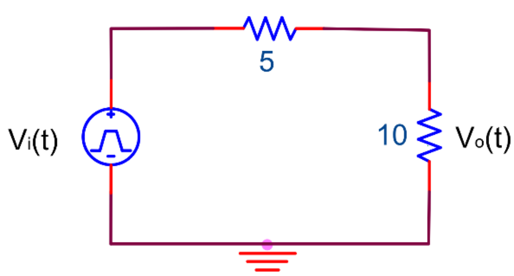


Figure 4.13

Pulse Voltage Source Parameters
V1 = voltage used in the lab V2 = voltage used in the lab TD = 0 TR = 1n TF = 1n PW = 0.2m PER =0.5m

4. Use transient analysis to display the first 2 cycles of $V_o(t)$. Save this output for the lab report.
5. Adjust the simulation parameters to plot the 10th and 11th cycles of $V_o(t)$. Save this output for the lab report.

6. Change the voltage source to a sine wave with a frequency of 1KHz and 30 volts peak to peak. Capture the first two cycles of $V_o(t)$. Save this output for the lab report.

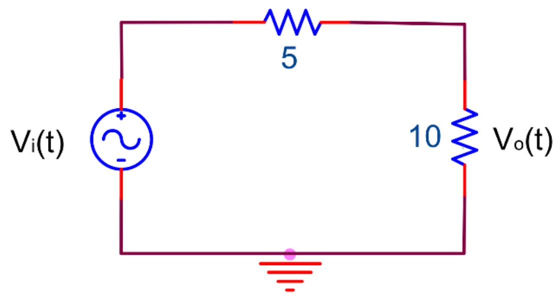


Figure 4.14

Sine Voltage Source Parameters
VOFF = 0 VAMPL = Voltage testing in the lab in peak, NOT peak to peak FREQ = Frequency used in the lab AC = 0

EXPERIMENT 5 : VOLTAGE/CURRENT DIVIDER CIRCUIT & DESIGN I

OBJECTIVE

Understanding how to use voltage/current division and be able to apply concepts thus far in a student designed circuit.

EQUIPMENT NEEDED

- Power Supply
- Digital Multimeter
- Resistors

THEORY

Thus far, Ohm's law, KVL and KCL have been used to determine unknown voltages in any given circuit. When given a circuit or section of a circuit that is in series, one can use voltage division to assertain a voltage drop. To use voltage division the follwing generic equation is used:

$$V_X = \frac{R_X}{R_T}(V_S)$$

$$desired\ unknown\ voltage = \frac{sum\ of\ resistors\ that\ make\ up\ unknown\ region}{sum\ of\ all\ resistors\ that\ make\ up\ the\ series\ connection} \times known\ voltage\ across\ the\ series$$

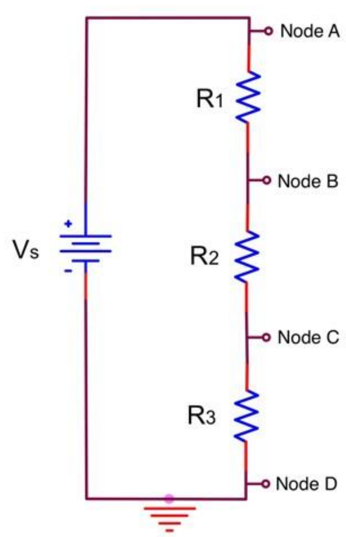


Figure 5.1

$$Voltage\ drop_{AB} = \frac{R_1}{R_1 + R_2 + R_3}(V_s)$$

$$Voltage\ drop_{AC} = \frac{R_1 + R_2}{R_1 + R_2 + R_3}(V_s)$$

$$Voltage\ drop_{BD} = \frac{R_2 + R_3}{R_1 + R_2 + R_3}(V_s)$$

$$Voltage\ drop_{BC} = \frac{R_2}{R_1 + R_2 + R_3}(V_s)$$

When there are multiple resistors in parallel it is possible to use current division to ease the analysis. There are two possible routes depending on the circuits current state. If there are two resistors in parallel, it will be easier to use equation one. When the number of parallel resistors exceeds two, Equation Two can provide a more efficient path.

Equation 1

$$I_x = \frac{R_T}{R_x + R_T} I_T$$

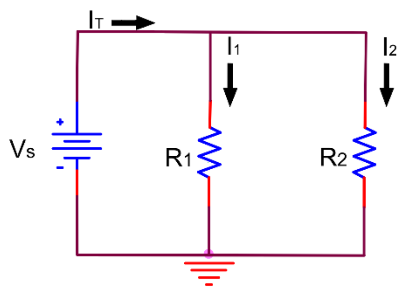


Figure 5.2

Equation 2

$$I_x = \left(\frac{\frac{1}{R_x}}{\frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} \cdots} \right) I_T$$

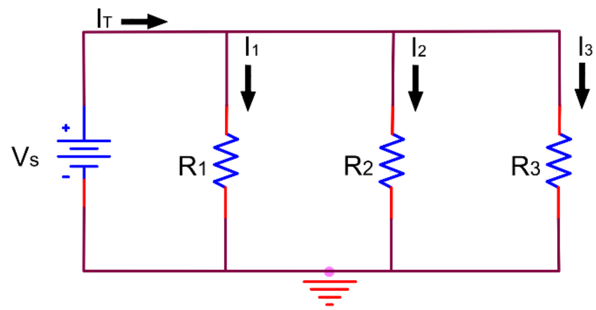


Figure 5.3

PRELIMINARY CALCULATIONS:

1. Calculate the voltage drop across at the nodes listed below for each diagram in Figure 5.4 using voltage division.

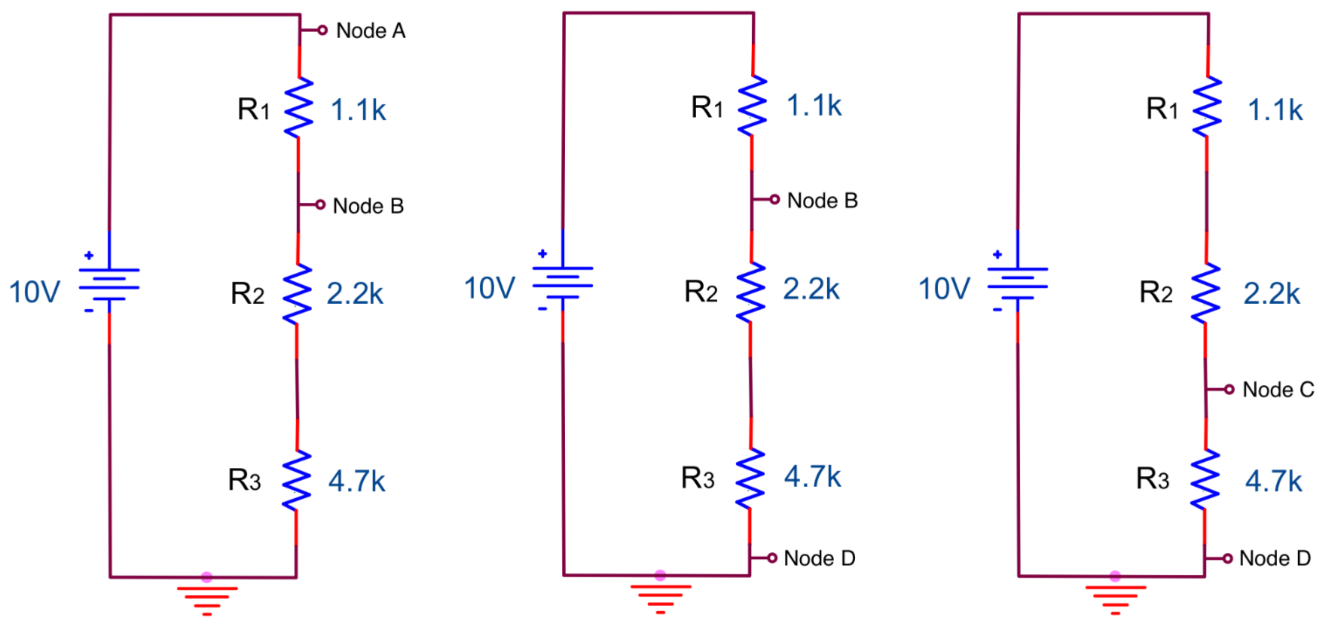


Figure 5.4

	V_{AB}	V_{BD}	V_{CD}
Calculated			

2. Calculate the voltage drop across each resistor using voltage division.

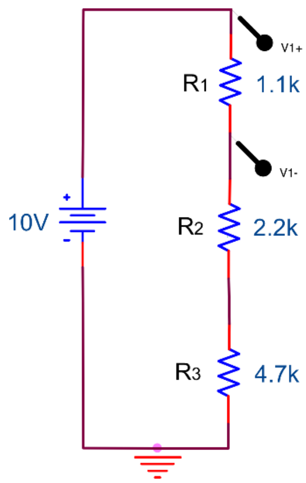


Figure 5.5

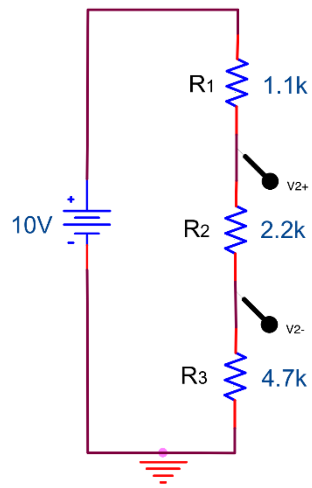


Figure 5.6.

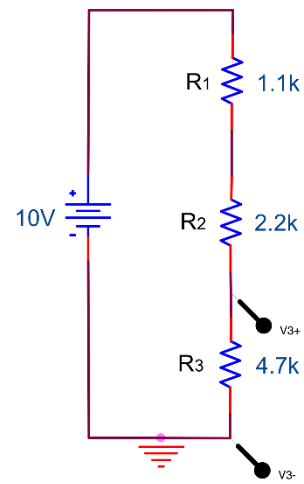


Figure 5.7

	V_1	V_2	V_3
Calculated			

3. Calculate all currents by using current divider rule.

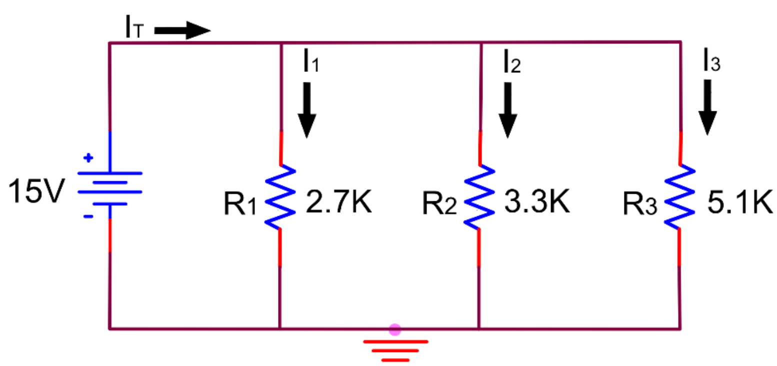


Figure 5.8

	R_T	I_T	I_1	I_2	I_3
Calculated					

4. Choose values for resistors R_1,R_2,R_3, and R_4, and R4 in Figure 5.9 to produce the voltages shown in the schematic.

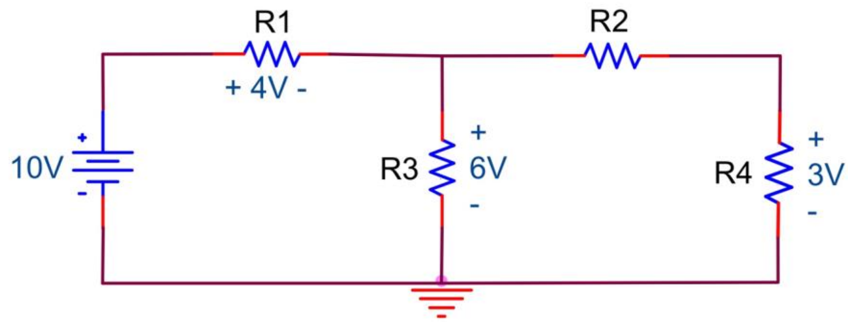


Figure 5.9

	R_1	R_2	R_3	R_4
Calculated				

PROCEDURE

1. Construct Figure 5.10. Measure and record R_1, R_2, and R_3.

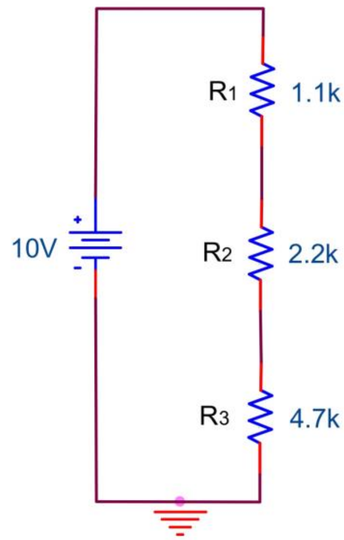


Figure 5.10

	R_1	R_2	R_3	R_T
Calculated	1.1k	2.2k	4.7k	8k
Measured				
Percent Error				

2. Measure and record V_{AB}, V_{BD} and V_{CD} as depicted in Figure 5.11.

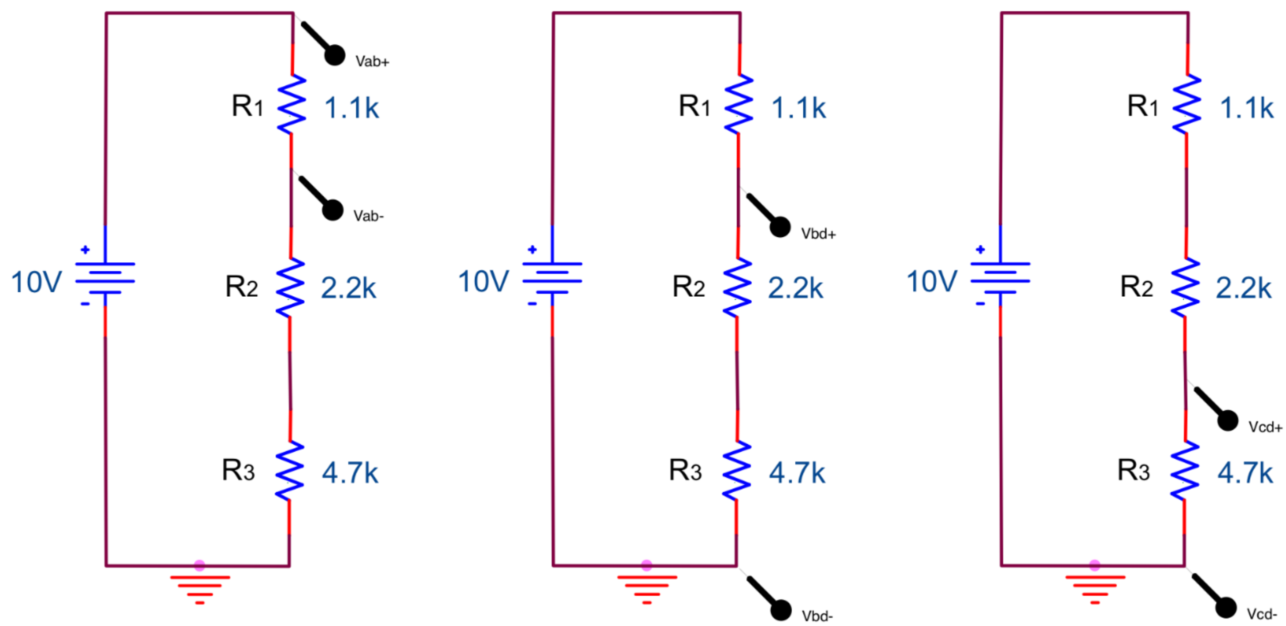


Figure 5.11

	V_{AB}	V_{BD}	V_{CD}
Calculated			
Measured			
Percent Error			

3. Measure and record the voltage drop across each resistor.

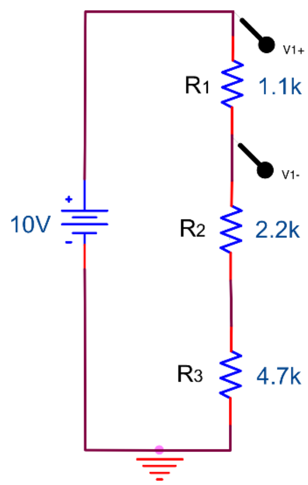


Figure 5.12

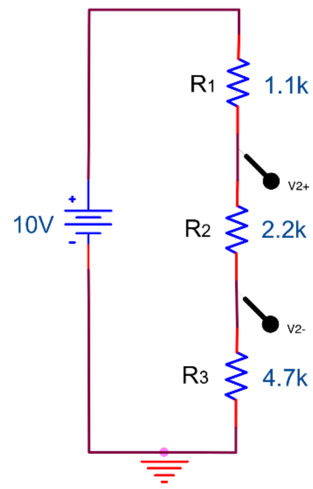


Figure 5.13.

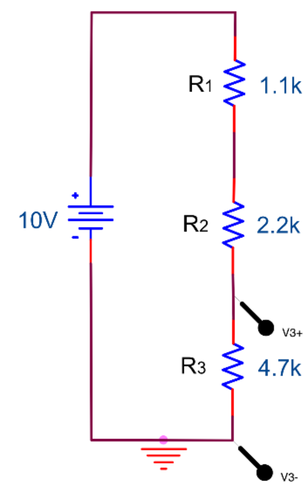


Figure 5.14

	R_1	R_2	R_3	V_1	V_2	V_3
Calculated						
Measured						
Percent Error						

4. Construct Figure 5.15. Measure the voltage for each resistor and calculate each current.

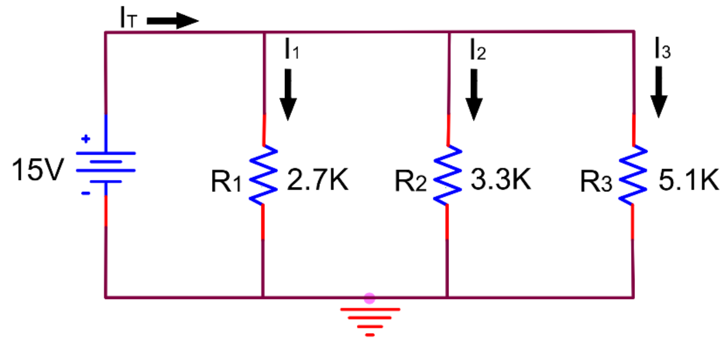


Figure 5.15

	V	R_T	R_1	R_2	R_3	I_T	I_1	I_2	I_3
Calculated									
Measured									
Percent Error									

5. Contruct Figure 5.16 with the values chosen in the prelab. Ensure that the resistors are measured individually prior to placing on the breadboard.

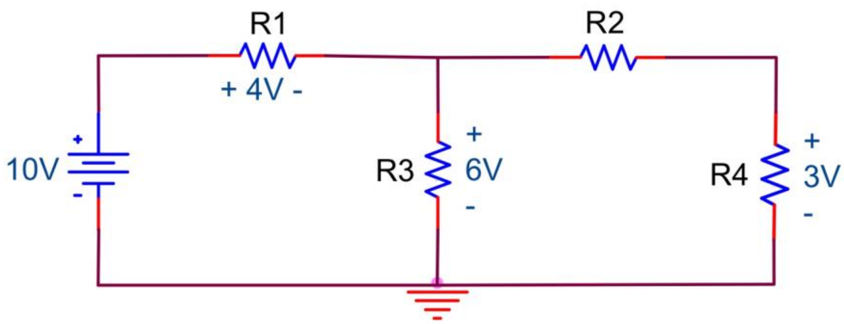


Figure 5.16

	R_1	R_2	R_3	R_4
Calculated				
Measured				

6. Measure the voltage drop across each resistor using the DMM.

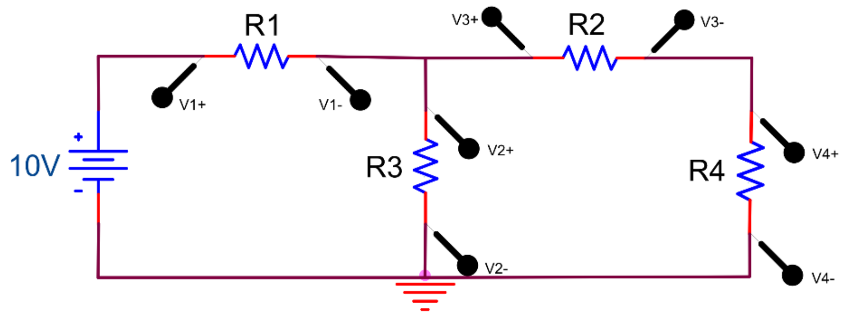


Figure 5.17

	V_1	V_2	V_3	V_4
Calculated				
Measured				
Percent Error				

1. Use PSPICE to display all voltages and currents, refer to Figure 5.18. Compare these values to the ones obtained in the laboratory.

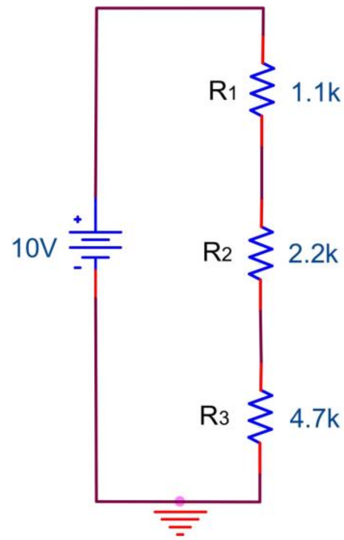


Figure 5.18

2. Use PSPICE to display all voltages and currents, refer to Figure 5.19. Compare these values to the ones obtained in the laboratory.

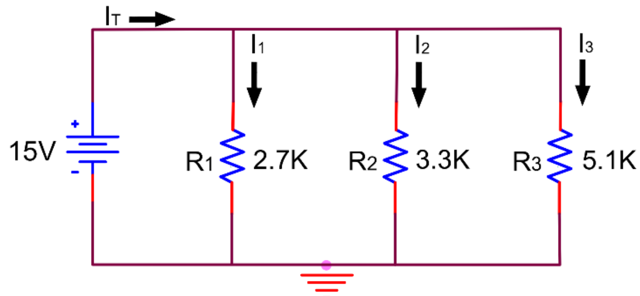


Figure 5.19

3. Build and simulate Figure 5.20 with the values used in the laboratory. Ensure that all voltages and currents are displayed. Compare these values to the ones obtained in the laboratory.

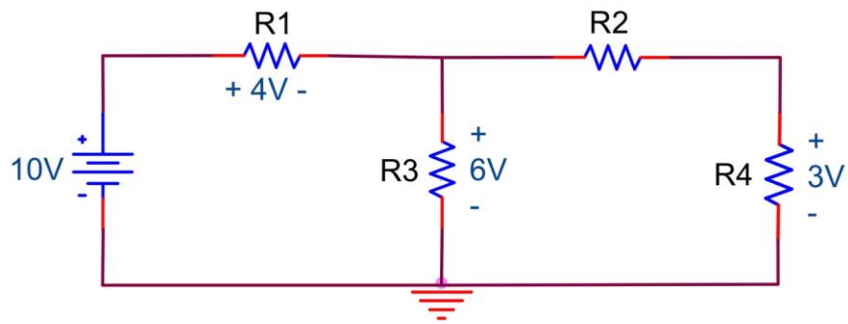


Figure 5.20

EXPERIMENT 6 : THEVENIN & SUPERPOSITION CIRCUITS

OBJECTIVE

Be able to apply Thevenin principles to simplify circuits to analyze complex logic.

EQUIPMENT NEEDED

- DC power supply
- DMM
- Resistors

THEORY

<u>Thevenin's Theorem:</u>

Any two-terminal linear resistive circuit can be replaced by an equivalent circuit with a voltage source and a series resistor.

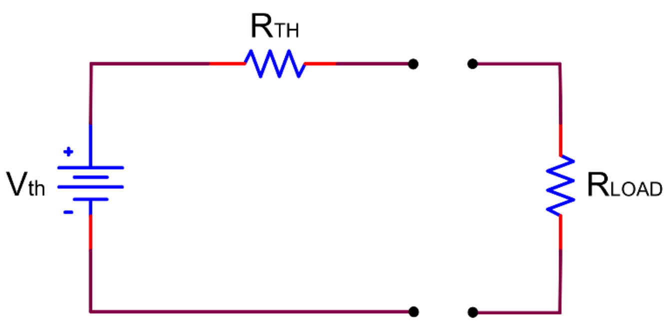


Figure 6.1

The voltage source is denoted by V_{oc} (open circuit voltage), and the resistor by R_{TH} (Thevenin resistor). The objective is to evaluate V_{oc} and R_{TH} . The procedure of obtaining Voc and RTh is stated below for the circuits containing independent sources only.

a. Remove the portion of the circuit external to which the Thevenin’s equivalent circuit is to be found.

b. Compute the voltage across the open-loop terminals. This voltage is V_{oc}.

c. Eliminate all the sources and compute the resistance across the open-loop terminal. This resistance is R_{TH}. A voltage source is eliminated by replacing it with a short circuit and a current source is eliminated by replacing it with an open circuit.

d. Draw the Thevenin equivalent circuit by placing the load resistor across the open-loop terminal (Figure 6.1).

Norton's Theorem:

Any two-terminal linear resistive circuit can be replaced by an equivalent circuit with a current source and a parallel resistor.

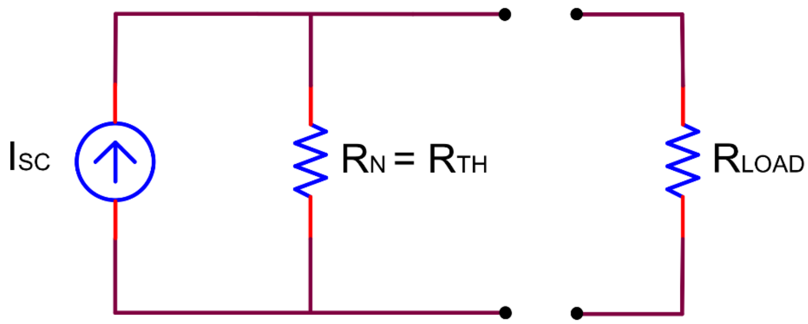


Figure 6.2

The current source is denoted by I_{sc} (short circuit current), and the resistor by R_N (Norton resistor). The value of R_N is the same as R_{TH}. The objective is to evaluate I_{sc} and R_N. The simplest way is to convert the voltage source of Figure 6.1 to the current source of Figure 6.2. This indicates:

$$R_N = R_{TH} \qquad\qquad I_{sc} = \frac{V_{oc}}{R_{TH}}$$

The general procedure of obtaining I_{sc} and R_N is stated below for circuits containing independent sources only.

a. Remove the portion of the circuit external to which the Norton equivalent circuit is to be found.

b) Place a short across the open-loop terminals and evaluate the current through this portion of the circuit. This current is I_{sc}.

c. Eliminate all sources and compute the resistance across the open-loop terminals. This resistance is R_N, which is identical to R_{TH}.

d. Draw the Norton equivalent circuit by placing the load across the open-loop terminals (Figure 6.2).

The procedure of obtaining R_{TH} (R_N) for the circuits containing dependent sources is different than outlined above. In such cases R_{TH} is obtained by

$$R_{TH} = \frac{V_{oc}}{I_{sc}} \qquad\qquad R_{TH} = R_N$$

Maximum Power Theorem

Figure 6.3 shows a Thevenin circuit with a load resistor R_L. The maximum power theorem states that for the load resistor R_L to dissipate the maximum power, its value must be equal to the Thevenin resistance, that is:

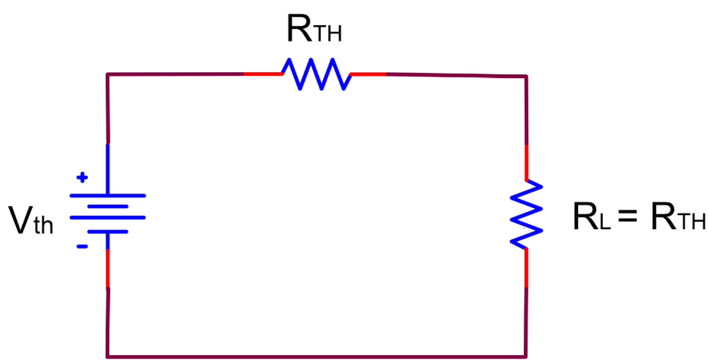


Figure 6.3

In such a case, the power dissipated by R_L is maximum and is given by the following equation:

$$P_{\max\,(load)} = \frac{V^2{}_{OC}}{4R_{TH}}$$

Superposition Theorem

In any linear resistive circuit containing two or more independent sources, the current through or voltage across any element is equal to the sum of the currents or voltages produced independently by each source.

The procedure of using the superposition theorem is stated below:

a. Eliminate all independent sources except one.

b. Obtain the currents and/or voltages desired. Record the proper directions of currents and polarities of voltages.

c. Repeat steps a and b for all the other independent sources in the circuit.

d. Combine the results algebraically.

PRELIMINARY CALCULATIONS

1. Calculate the Thevenin and Norton Equivalents for the circuit to the left of RL in Figure 6.4.

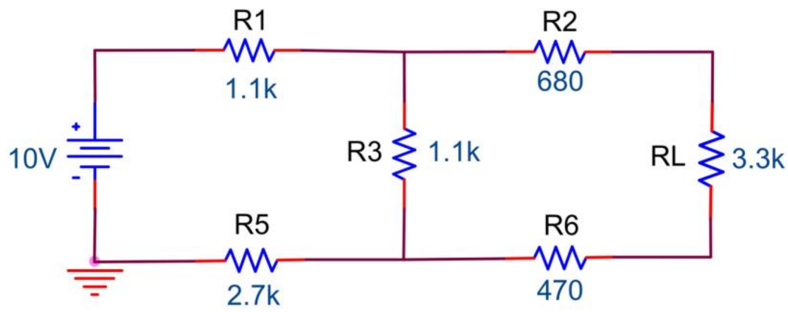


Figure 6.4

	V_{TH}	R_{TH}
Calculated		

2. For the circuit in Figure 6.5, calculate V_1 and V_2.

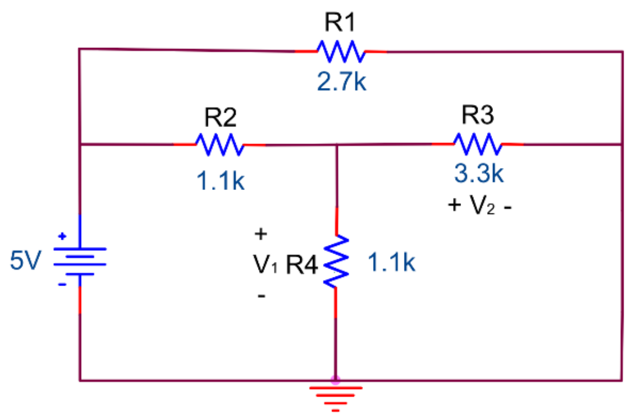


Figure 6.5

5V Circuit	V_1	V_2
Calculated		

3. For Figure 6.6 calculate V_1 and V_2.

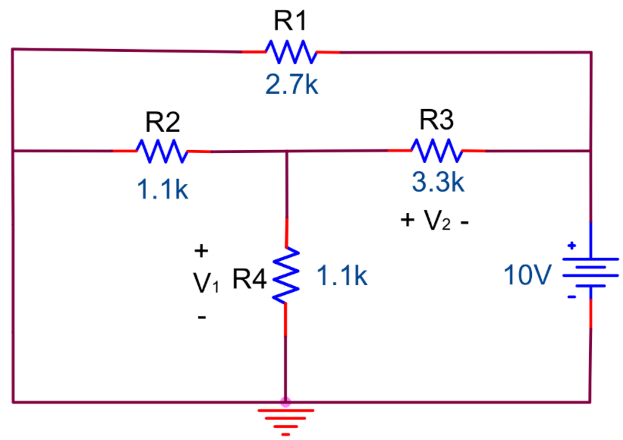


Figure 6.6

10V Circuit	V_1	V_2
Calculated		

4. For Figure 6.7 calculate V_1 and V_2 utilizing the superposition principle.

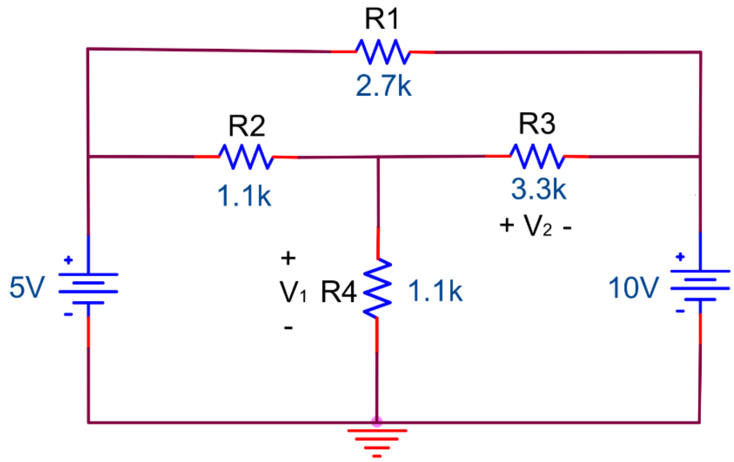


Figure 6.7

	V_1	V_2
10V Circuit		
5V Circuit		
Total		

PROCEDURE

1. Construct the circuit of Figure 6.8. Measure the voltage, V_{ab}.

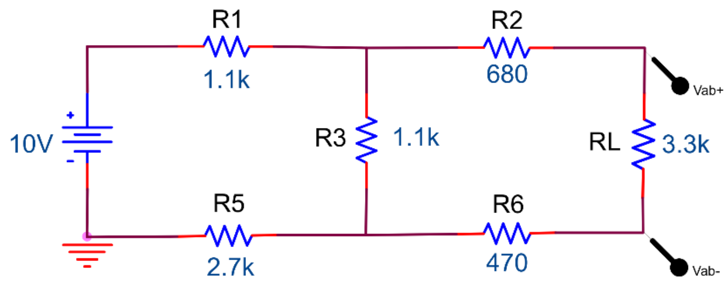


Figure 6.8

	V_{ab}
Measured	

2. Remove the 3.3kΩ resistor and measure and record V_{TH}.

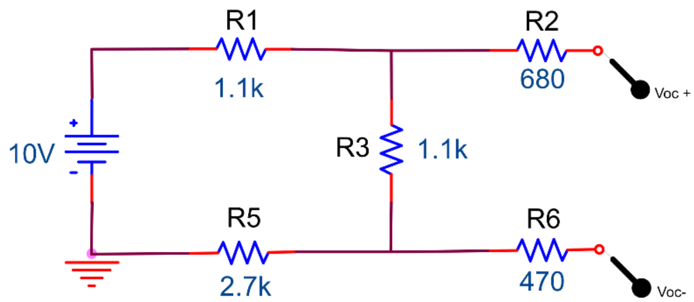


Figure 6.9

	V_{oc}
Calculated	
Measured	

3. Place a short circuit between the terminals a-b and measure the current, I_{sc}, flowing through this short circuit.

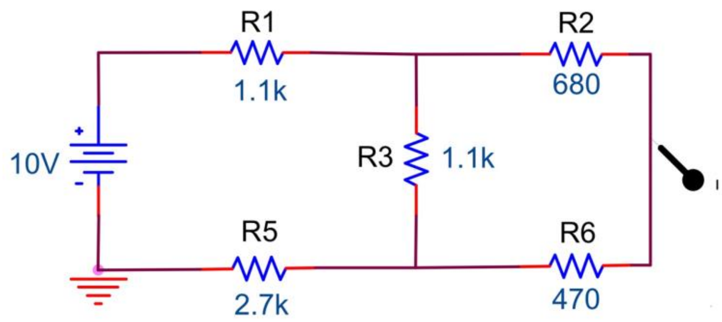


Figure 6.10

	I_{sc}
Measured	

4. Remove the 10 volt source and replace it with a short circuit. Use an ohmmeter to measure R_{TH}.

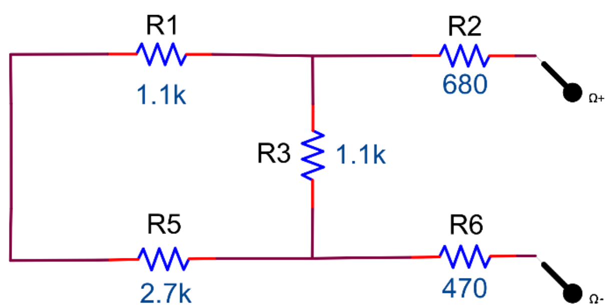


Figure 6.11

	R_{TH}
Measured	

5. Using the measured values of V_{oc} and R_{TH} (using closest available standard values) construct the Thevenin equivalent of the circuit. Add the 3.3kΩ resistor across a-b and measure the voltage across it. Compare to the value measured in step 1.

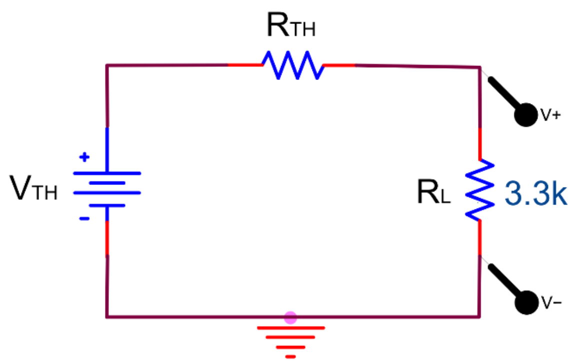


Figure 6.12

	V_{RL}
Measured	

6. Construct the circuit of Figure 6.13 and measure V_1 and V_2.

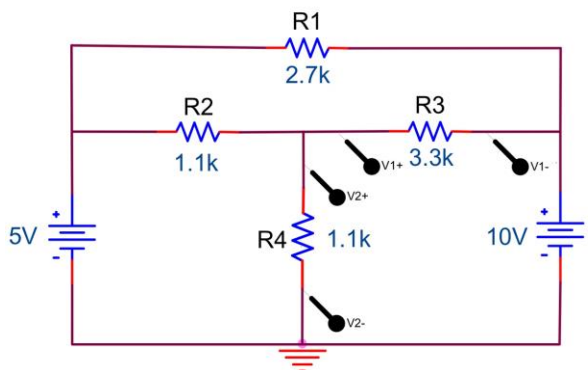


Figure 6.13

	V_1	V_2
Measured		

7. Remove the 5 volt source and replace it with a short circuit. Measure V_1 and V_2.

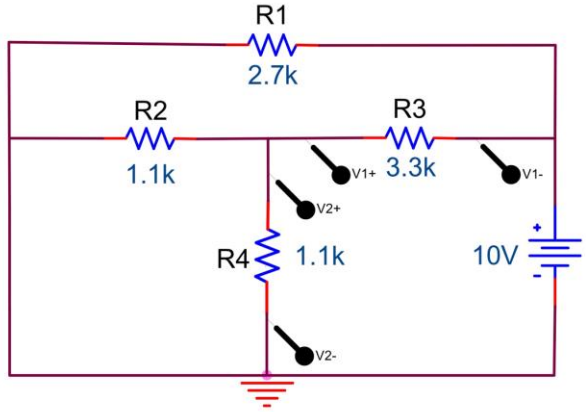


Figure 6.14

	V_1	V_2
Measured		

8. Replace the 5 volt source and remove the 10 volt source and replace it with a short circuit. Measure V_1 and V_2 again.

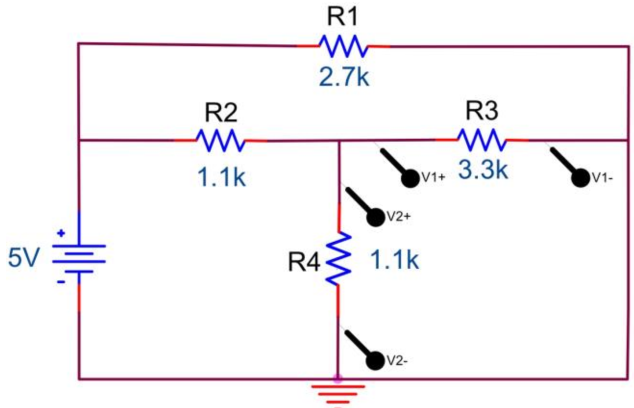


Figure 6.15

	V_1	V_2
Measured		

9. Use your results to verify the principle of superposition. Also, compare these values to those obtained in the pre-lab.

	V_1	V_2
10V Circuit		
5V Circuit		
Total		

	V_1	V_2
Calculated		
Measured		
Percent Error		

1. Use PSPICE to measure $V_{oc} = V_{TH}$. If the circuit is built like the lab procedure PSPICE will not be able to run due to open connections. It is imperative that the circuit be built as shown in Figure 6.16.

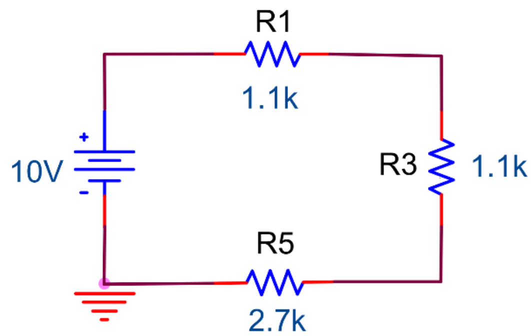


Figure 6.16

2. Using PSPICE measure R_{TH}. Once the circuit has been built ensure that the node voltages and currents are displayed. Use ohms law at the source to find the R_{TH}.

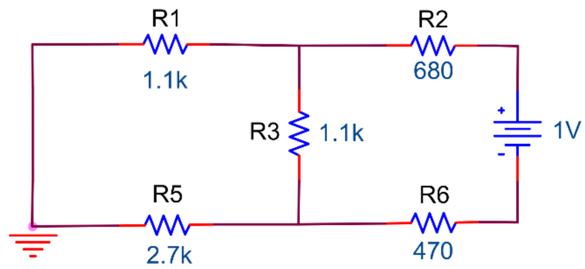


Figure 6.17

3. Use PSPICE to analyze the circuit down below. Ensure that all voltages are displayed. Validate these voltages with the values obtained from the lab.

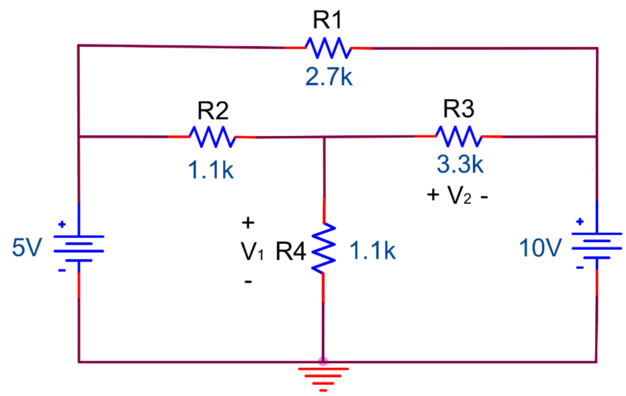


Figure 6.18

EXPERIMENT 7 : WHEATSTONE BRIDGE & CIRCUIT DESIGN II

OBJECTIVE

Leverage previous experiments to deliver maximum power to a specific load.

EQUIPMENT NEEDED

- DC Power Supply
- DMM
- Resistors

THEORY

In the previous Thevenin experiments it was important to isolate and remove the load resistor to properly calculate the Thevenin equivalent. It is possible to find a resistive value that delivers the most power to this load based on the rest of the circuit, this is known as the Maximum Power Transfer Theorem. When a load resistor is equal to the rest of the circuit's resistance, $R_{TH} = R_L$; anything previous or after this value will yield a lower amount of power.

PRELIMINARY CALCULATIONS

1. Calculate R_{TH} in Figure 7.1
2. Calculate V_{TH}; $V_{TH} = V_A - V_B$
3. Using the maximum power transfer theorem, Calculate the value of R which will result in the maximum power being delivered to R (see Figure 7.1).

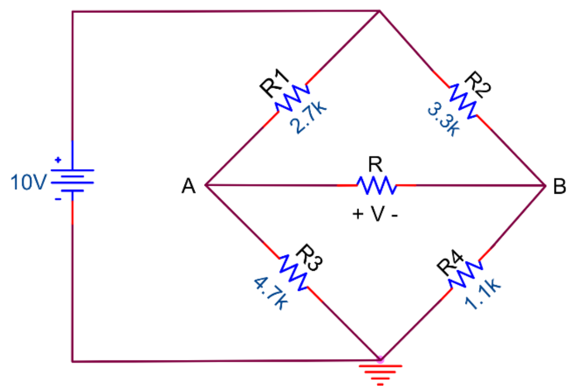


Figure 7.1

	R_{TH}	V_{TH}
Calculated		

4. For the circuit shown in Figure 7.2, choose resistor values that will maximize the power transferred to the 2.7kΩ resistor.

   Hint: As a designer you can choose how much current flows through each section of the circuit.

5. Use any method to calculate the voltage and current at R_4.

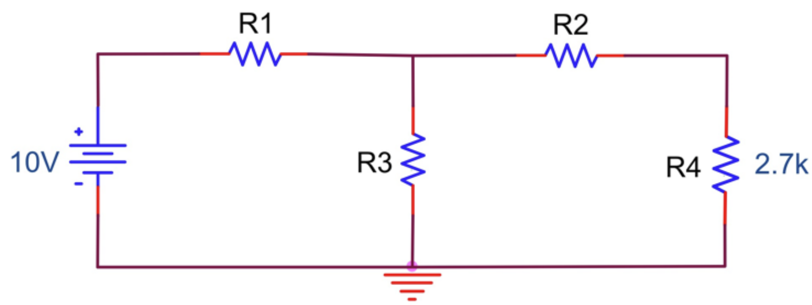


Figure 7.2

	V_4	I_4
Calculated		

PROCEDURE

1. Construct and measure R_{TH}, V_{TH}.

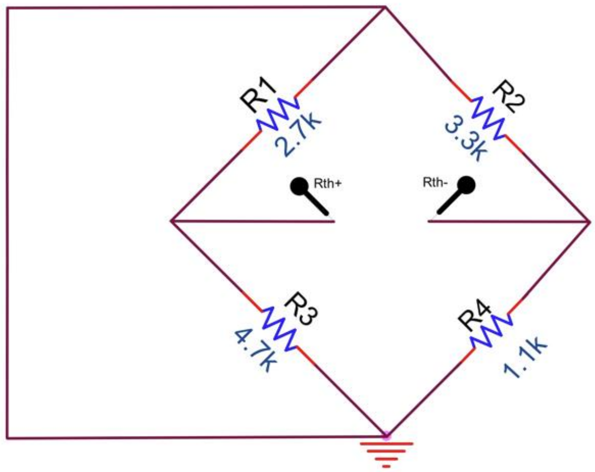


Figure 7.3

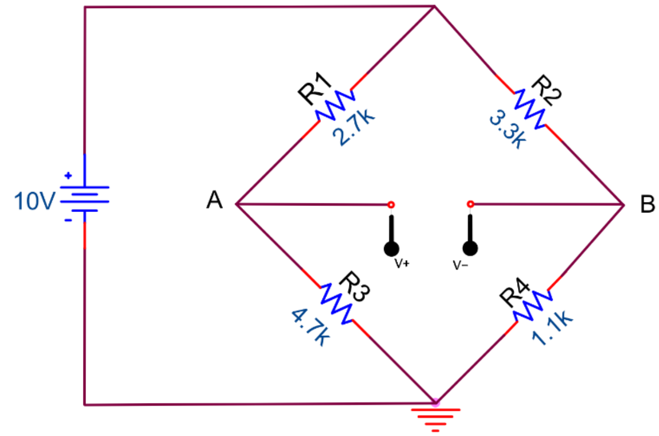


Figure 7.4

	R_{TH}	V_A	V_B	V_{TH}
Calculated				
Measured				
Percent Error				

2. Construct Figure 7.5and cycle resistors listed in the table below for "R". Measure V_R, R, and calculate power. Plot Power vs Resistance using Microsoft Excel.

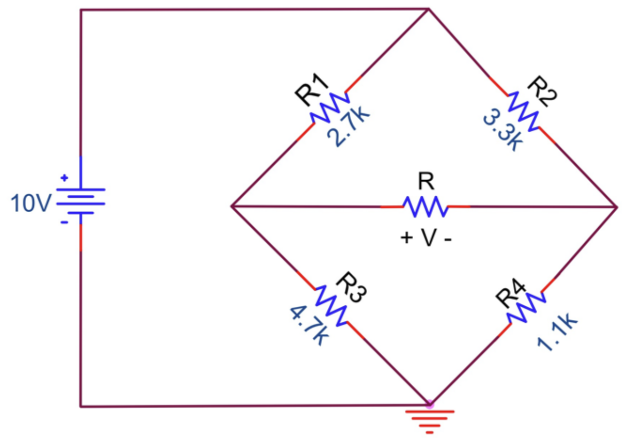


Figure 7.5

R		V_R	$P = \frac{V_R{}^2}{R}$
100Ω			
270 Ω			
470 Ω			
680 Ω			
1.1k Ω			
2.2k Ω			
2.4k Ω			
2.7k Ω			
4.7k Ω			
6.8k Ω			
10k Ω			

3. Construct Figure 7.6with the values chosen in the preliminary calculations.

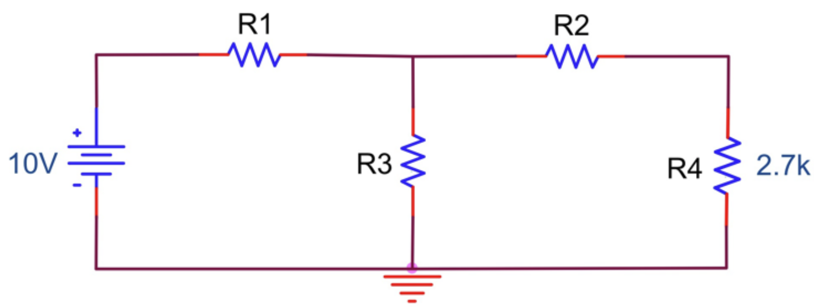


Figure 7.6

4. Measure and record the voltage at R_4. Calculate the current going through the R_4 in Figure 7.7.

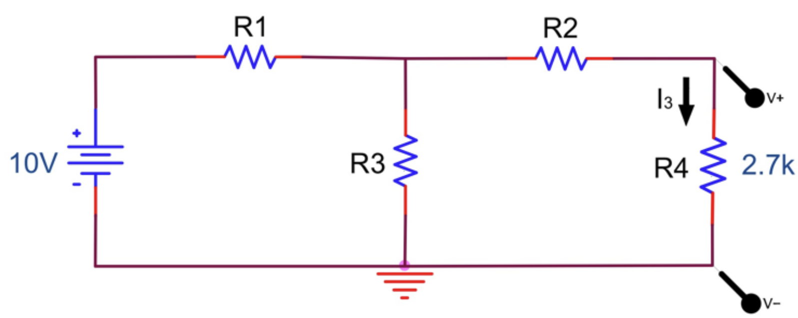


Figure 7.7

	R_4	V_4	I_3
Calculated			
Measured			

5. Verify that the circuit is operating correctly (Figure 7.8) by changing R_4 for the resistor values located in the table below. Plot the graph using Excel when completed (Power vs Resistance), verify that the plot looks correct.

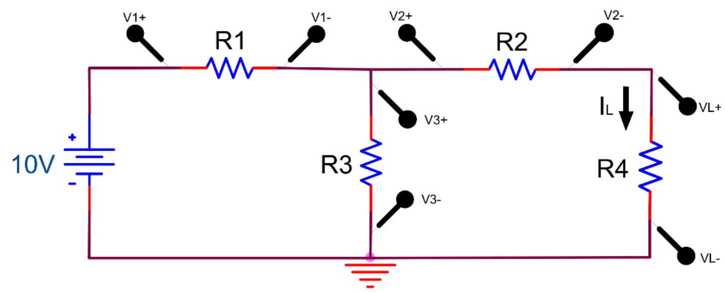


Figure 7.8

R_L	V_1	V_2	V_3	V_L	I_L	$P(w)$
180 Ω						
560 Ω						
1.8k Ω						
2k Ω						
2.4k Ω						
2.7k Ω						
3.3k Ω						
4.7k Ω						
5.1k Ω						
10k Ω						

1. For the following circuit find R_{th} and V_{th}.

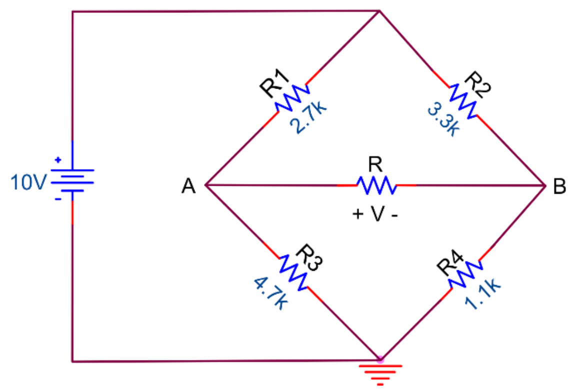


Figure 7.9

EXPERIMENT 8 : INTRODUCTION TO AC CIRCUITS

OBJECTIVE

Understand how to analyze AC circuits and how to take measurements on the oscilloscope.

EQUIPMENT NEEDED:

- Digital Oscilloscope
- Function Generator
- Resistors

THEORY

Until now the DC experiments were performed using a power supply and an DMM to measure the voltages in a circuit. In an AC circuit the power supply is replaced with a function generator and the measurements are taken with an oscilloscope.

An oscilloscope is a valuable tool that allows the user to measure a voltage and plot it against time. With the plot the user can then calculate the voltage of the signal and the frequency by looking at the display and performing basic arithmetic. The laboratory oscilloscope is a two-channel scope capable of monitoring two signals at once and displaying them on the screen; channel one is yellow or blue and channel two is green or pink.

Four values can be obtained from a signal on an oscilloscope such as voltage in peak and peak to peak, period, and frequency. A period is when a signal has achieved one complete cycle, meaning the signal is going to repeat again. The frequency of a signal can be found after the period has been obtained from the oscilloscope. A signal can be characterized in volts peak (V_p) aka amplitude or volts peak to peak (V_{pp}). In the laboratory V_{pp} is used but PSPICE components commonly use amplitude.

$$f = \frac{1}{T}$$

$$V_p = \sqrt{2}V_{RMS}$$

$$voltage = \#\ of\ divisions \left(\frac{amplitude}{division}\right)$$

$$period = \#\ of\ divisions \left(\frac{seconds}{division}\right)$$

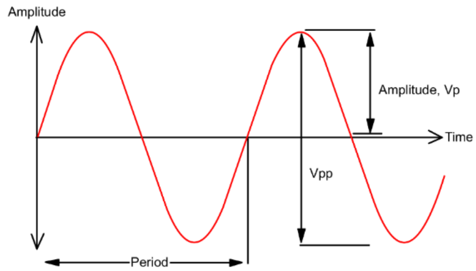


Figure 8.1

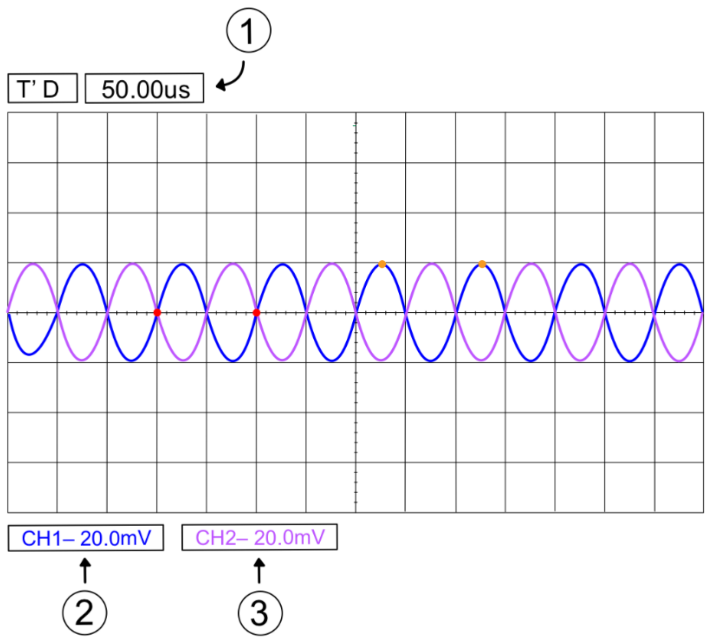


Figure 8.2

Figure 8.2 is like what the display of an oscilloscope will look like. Number one is the time base which indicates the time per division. Numbers two and three indicate the volts per division for each channel respectively. To calculate the period of the wave form above one must choose one cycle and count how many squares occupy that distance. For example, the two red dots or the two orange dots are one cycle; it is easier to choose the red dots given they are on the x-axis making it easier to count seconds per division. Channel one's signal has two squares going from the first red dot to the second meaning there are two divisions. The oscilloscope is set up for $50.00\mu s$ per division, using the period then frequency formula yields:

$$Period\ (T) = 2div\left(\frac{50.00\mu s}{1div}\right) = 100\mu s$$

$$f = \frac{1}{T} = \frac{1}{100\mu s} = 10KHz$$

To calculate the amplitude, look at the x-axis and look up till one of the peaks are found. The distance between the x-axis and the peak is the number of divisions. Figure 8.2 has one division for channel one. The oscilloscope is configured for $20mV$ on channel one (indicated by number 2). Using the voltage formula yields:

$$v_p = 1div\left(\frac{20.0mV}{1div}\right) = 20.0mV$$

$$v_{pp} = 2div\left(\frac{20.0mV}{1div}\right) = 40.0mV$$

PRELIMINARY CALCULATION

Calculate the following for Figure 8.3,Figure 8.4,and Figure 8.5.

1. Peak Voltage V_p.
2. Peak to peak voltage V_{pp}.
3. Period.
4. Frequency.

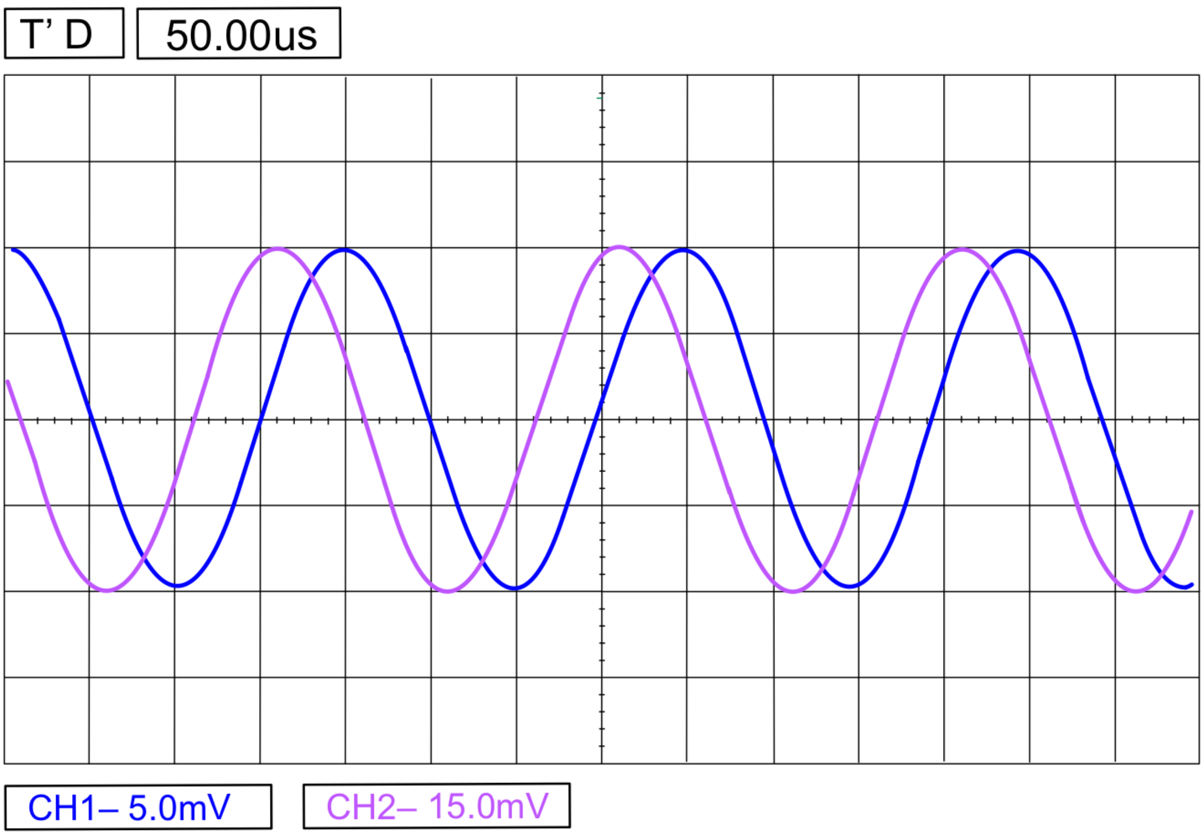


Figure 8.3

CH1	V_p	V_{pp}	Period (T)	Frequency
Calculated				

CH2	V_p	V_{pp}	Period (T)	Frequency
Calculated				

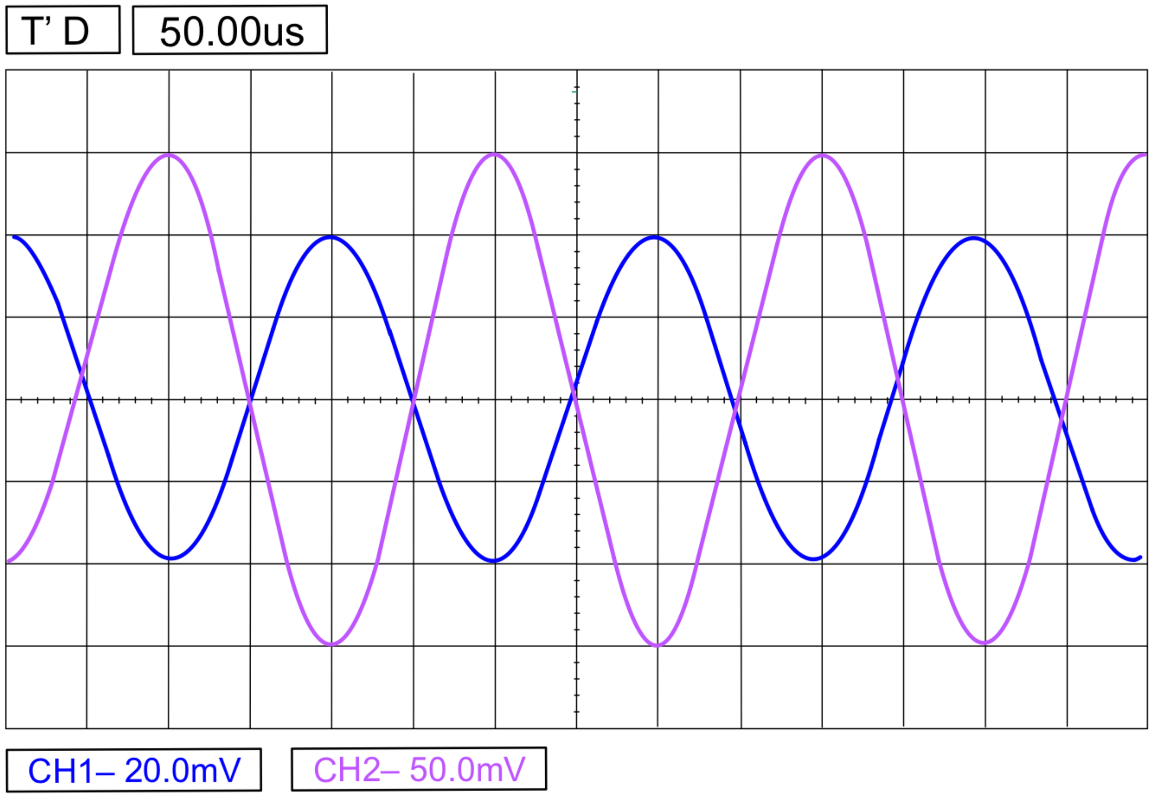


Figure 8.4

CH1	V_p	V_{pp}	Period (T)	Frequency
Calculated				

CH2	V_p	V_{pp}	Period (T)	Frequency
Calculated				

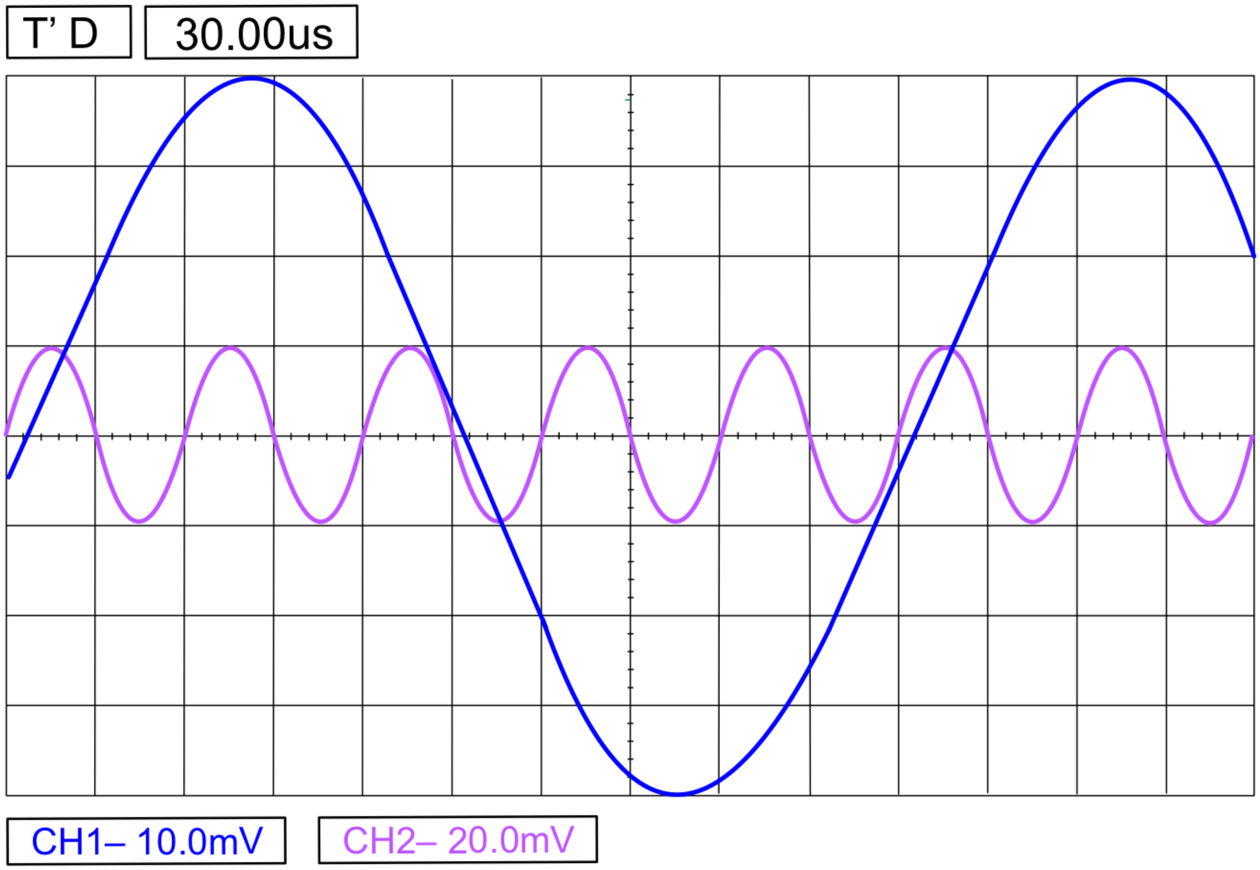


Figure 8.5

CH1	V_p	V_{pp}	Period (T)	Frequency
Calculated				

CH2	V_p	V_{pp}	Period (T)	Frequency
Calculated				

PROCEDURE

1. On the function generator, set amplitude to $10V_{pp}$ and frequency to 1KHz. Also set the output to High Z.
2. Attach the channel one of the oscilloscope to V1 and channel two to V2. Use the display to calculate the V_p, V_{pp}, period, and frequency. Take a picture of the display.

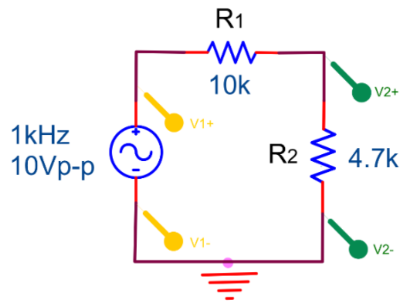


Figure 8.6

	$V_{in(pp)}$	$V_{out(pp)}$	Period (T)	Frequency
Calculated				
Measured				
Percent Error				

3. Construct and use the DMM to measure the output of the function generator. Measure and calculate V_p.

   Note: The measurement taken on the DMM is V_{RMS}. Ensure that the DMM is in AC mode.

$$V_p = \frac{V_{pp}}{2} \qquad V_{rms} = \frac{V_p}{\sqrt{2}}$$

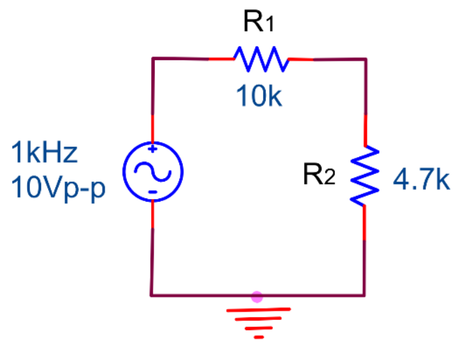


Figure 8.7

DMM	$V_{in}(rms)$	$V_1(rms)$	$V_2(rms)$
Calculated			
Measured			
Percent Error			

1. Construct the following figure and find V_p, V_{pp}, period, and frequency for both V1 and V2. Compare these values to the ones obtained in the lab. Label the output graph in V_{pp}.

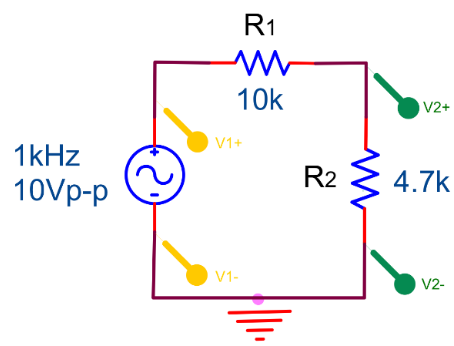


Figure 8.8

EXPERIMENT 9 : OPERATIONAL AMPLIFIERS

OBJECTIVE

Understand how to utilize pin out diagrams and effectively use active components.

EQUPMENT NEEDED

- Oscilloscope
- DC power supply
- Function Generator
- DMM
- Resistors

THEORY

Operational Amplifier

The symbol for an operational amplifier is shown in Figure 9.1. The operational amplifier (op amp) has many terminals. These terminals include inverting input, noninverting input, output, dc power supply with positive, negative, and ground, frequency compensation, and offset null terminals. For the sake of simplicity Figure 9.1 shows the two input terminals and the output terminal.

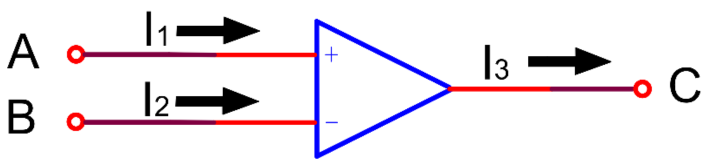


Figure 9.1

Operational amplifiers are usually available in the form of integrated circuits. The most important characteristics of op amps are stated below:

a. Currents into both input terminals are zero

$$I_1 = 0 \qquad\qquad I_2 = 0$$

b. Voltage between the input terminals is zero

$$V_{AB} = 0$$

c. The output current $I_3 \neq 0$ hence KCL does not apply here, that is,

$$I_1 + I_2 \neq I_3$$

For more information on the operation amplifiers, refer to your textbook.

PRELIMINARY CALCULATIONS

1. For the circuit in Figure 9.2 and Figure 9.3 Calculate V_o as a function of $V_i$and R_F.

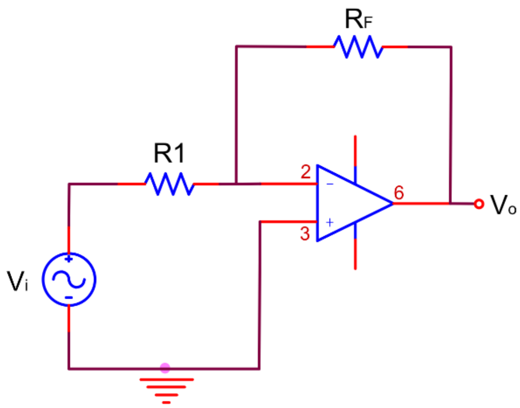


Figure 9.2

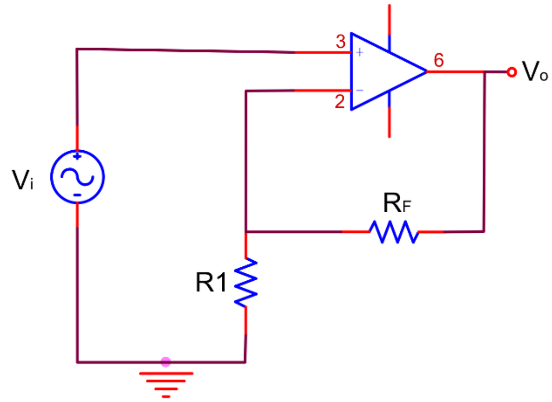


Figure 9.3

2. Using the formulas from step one, calculate A_v and V_o with an input voltage of $.6v_{pp}$ and $R_1 = 10k\Omega$.

Figure 9.2	A_v	V_o
$R_F = 10k\Omega$		
$R_F = 27k\Omega$		
$R_F = 100k\Omega$		

Figure 9.3	A_v	V_o
$R_F = 10k\Omega$		
$R_F = 27k\Omega$		
$R_F = 100k\Omega$		

3. Calculate the voltage V_{ab} across the 6.2k ohm resistor in Figure 9.4.

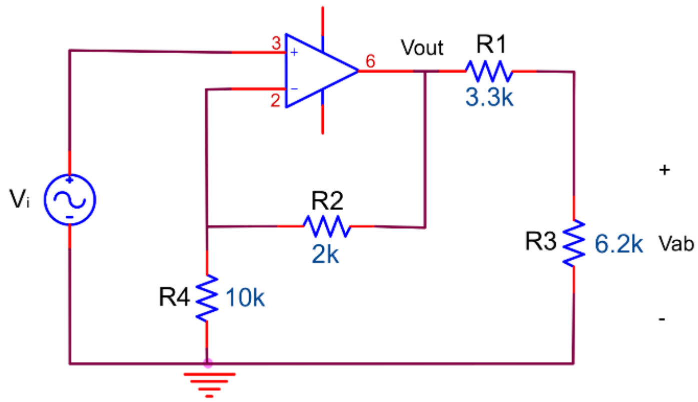


Figure 9.4

	A_v	V_{out}	V_{ab}
Calculated			

PIN NO.	NAME	DESCRIPTION
1	NU	NOT USED
2	V-	INVERTING INPUT
3	V+	NON-INVERTING INPUT
4	VNEG	NEG. DC PS VOLTAGE
5	NU	NOT USED
6	VO	OUTPUT VOLTAGE
7	VPOS	POS. DC PS VOLTAGE
8	NU	NOT USED

1. Construct the circuit in Figure 9.5. Record v_i and v_o; compare it to the value calculated in the pre-lab. What is the phase difference between V_i andV_o?

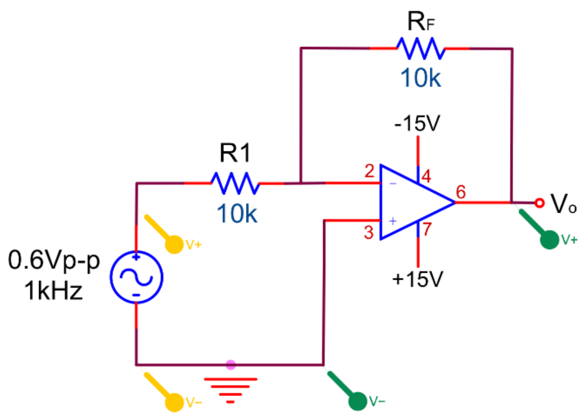


Figure 9.5

	v_{in}	A_v	v_o	θ
Calculated				
Measured				

2. Construct the circuit in Figure 9.6. Record v_i and v_o; compare it to the value calculated in the pre-lab. What is the phase difference between V_i andV_o?

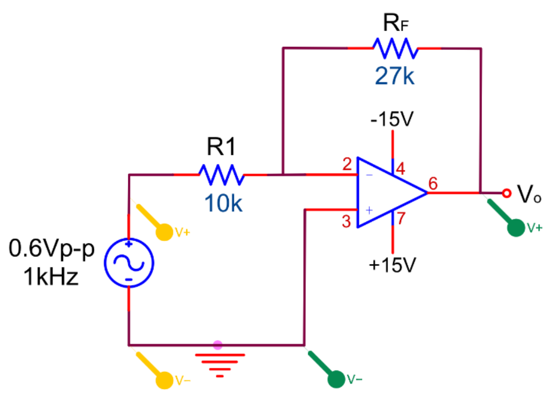


Figure 9.6

	v_i	A_v	v_o	θ
Calculated				
Measured				

3. Construct the circuit in Figure 9.7. Record v_i and v_o; compare it to the value calculated in the pre-lab. What is the phase difference between V_i andV_o?

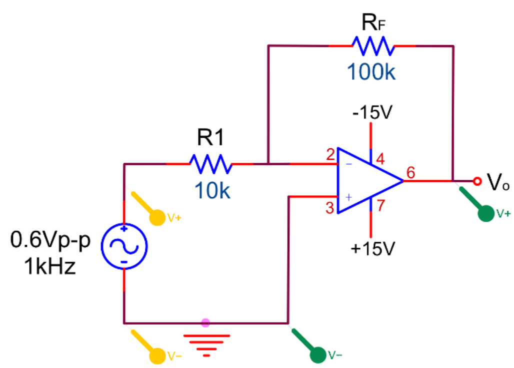


Figure 9.7

	v_i	A_v	v_o	θ
Calculated				
Measured				

4. Construct the circuit in Figure 9.8. Record V_i and V_o; compare it to the value calculated in the pre-lab. What is the phase difference between V_i andV_o?

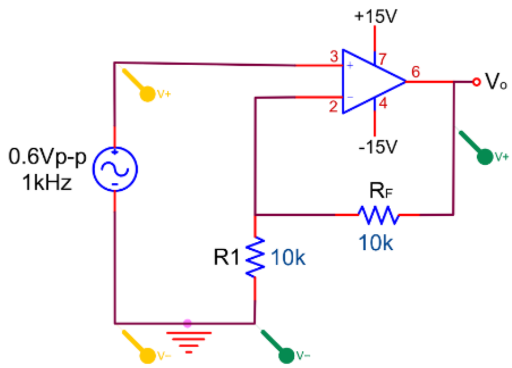


Figure 9.8

	v_i	A_v	v_o	θ
Calculated				
Measured				

5. Construct the circuit in Figure 9.9. Record v_i and v_o; compare it to the value calculated in the pre-lab. What is the phase difference between v_i andv_o?

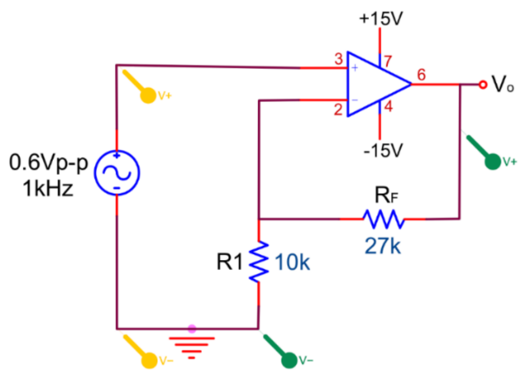


Figure 9.9

	v_i	A_v	v_o	θ
Calculated				
Measured				

6. Construct the circuit in Figure 9.10. Record v_i and v_o; compare it to the value calculated in the pre-lab. What is the phase difference between v_i and v_o?

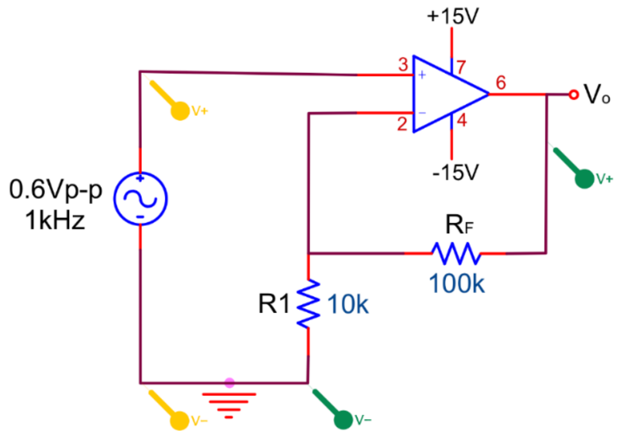


Figure 9.10

	v_i	A_v	v_o	θ
Calculated				
Measured				

7. Construct Figure 9.11, what is the peak value of v_o?

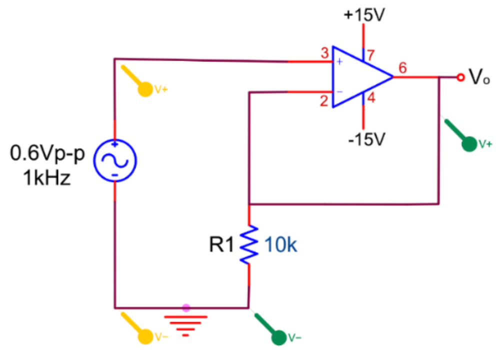


Figure 9.11

	v_i	A_v	v_o	θ
Calculated				
Measured				
Percent Error				

8. Construct the circuit of Figure 9.12. Measure v_o and the voltage across R_3 with an oscilloscope. Compare your result to that obtained in the pre-lab.

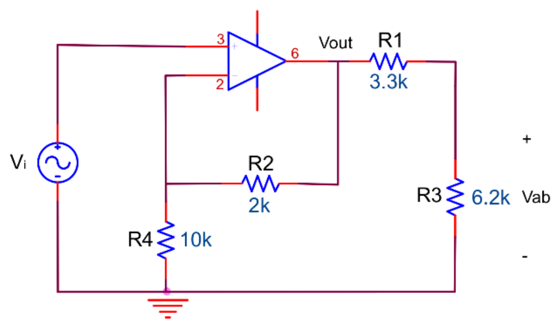


Figure 9.12

	v_i	A_v	v_o	v_{ab}
Calculated				
Measured				

Finding an OpAmp:

For the purposes of this lab the following generic OpAmp (Figure 9.13) will be sufficient to simulate the desired results. In the future if a specific OpAmp is required such as LM741, it must be searched for under “Place”→”Part”, alternatively “P” on the keyboard.

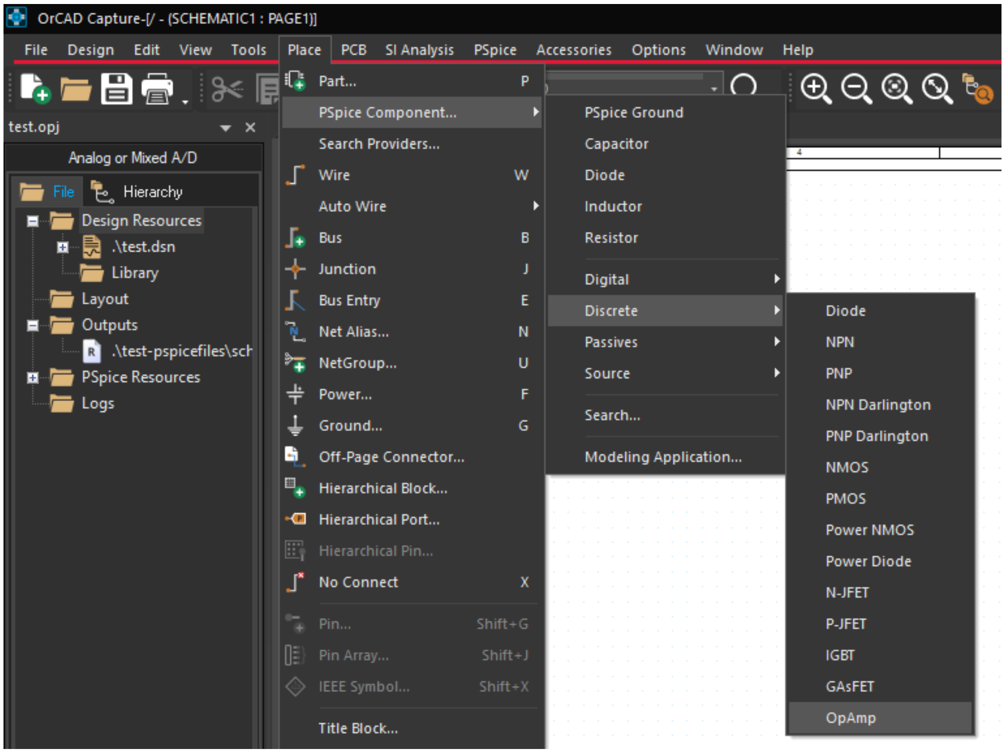


Figure 9.13

Some of the builds change the pin of the input, in order to construct the schematic neatly it is necessary to reorient the component. To do so, right click the component and “Mirror Vertically”; pins two and three will now be inverted.

Configuring Simulation Settings:

1. The “Run To Time” needs to be set such that you see three periods of the signal. In order to achieve this you must convert your frequency to time using $T = \frac{1}{f}$. The “T” is for one period so it is necessary to multiply by three to achieve three periods. The “Maximum Step Size” will always be 1 micro for the purposes of these labs.

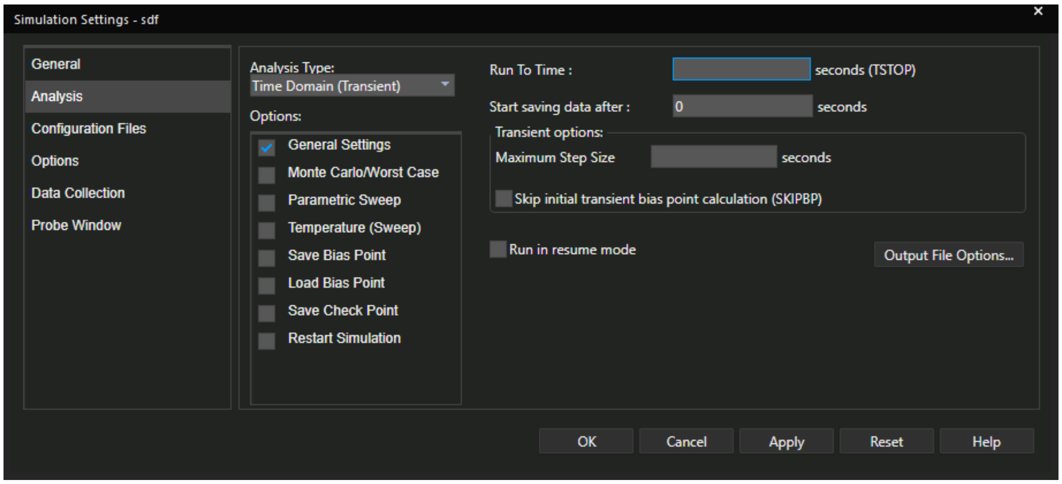


Figure 9.14

Measuring Signals:

1. After successfully probing the circuit you will have an output with different color traces. Down in the bottom left there will be two “Names” Each name is associated with one of the traces on the graph and can be distinguished by the color. In the example down below the green trace is called “V(V1:+)” and the red trace is “V(R1:2)”.

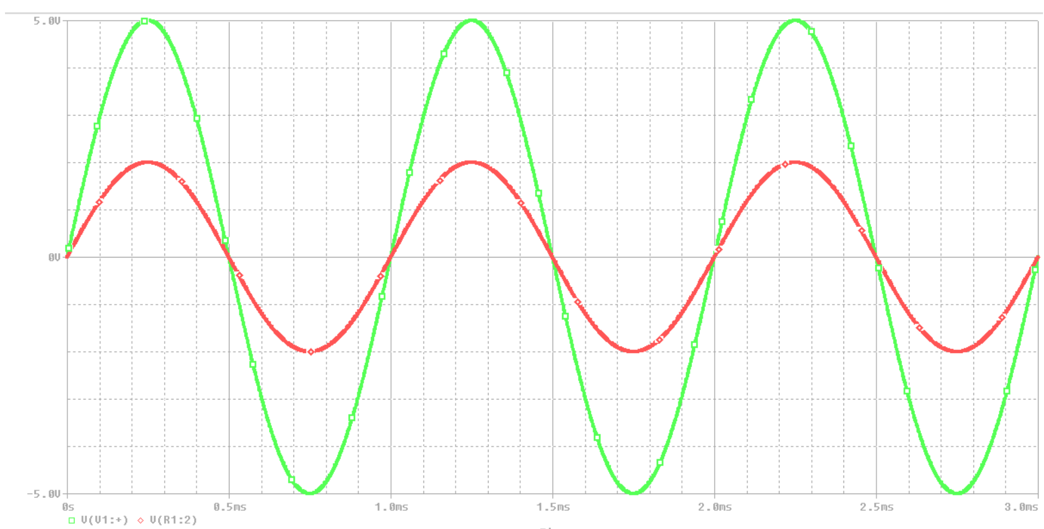


Figure 9.15

2. To take measurements go under “Trace” →”Evaluate Measurement..”

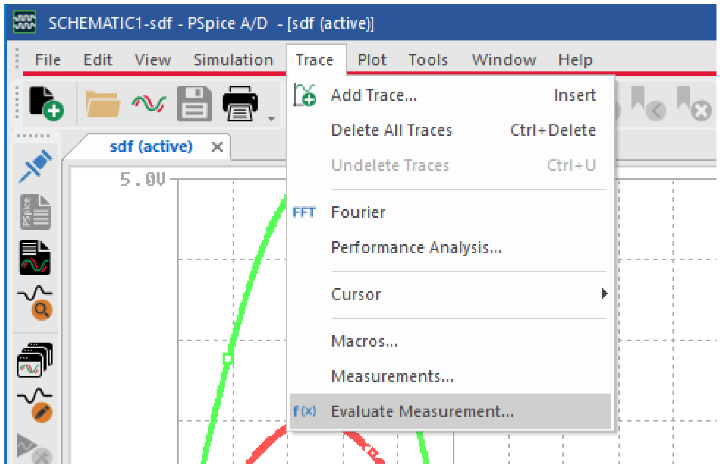


Figure 9.16

3. This window will let you enter functions to perform on the different traces on your plot. To find V_{pp} type a function that finds the max value subtracted from the minimum value. In the example the function will look like this, your function will have different names:

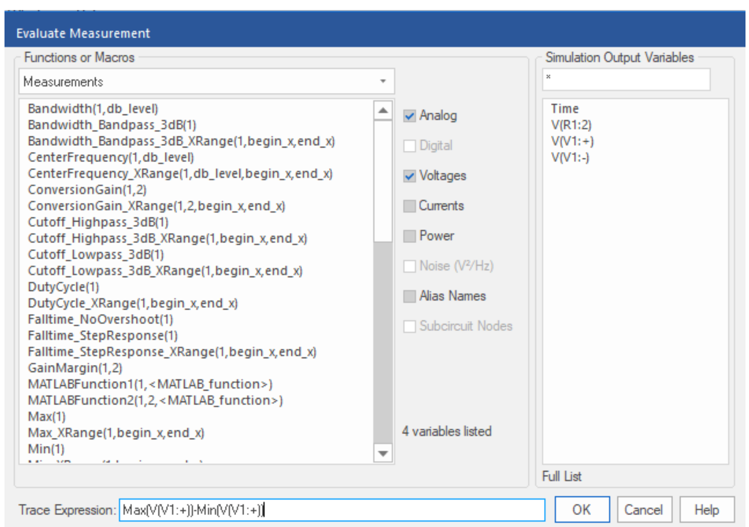


Figure 9.17

4. If the function was valid the main graph will have an “Measurement Results” table below it. In the following example the green trace has a $V_{pp} = 9.99$

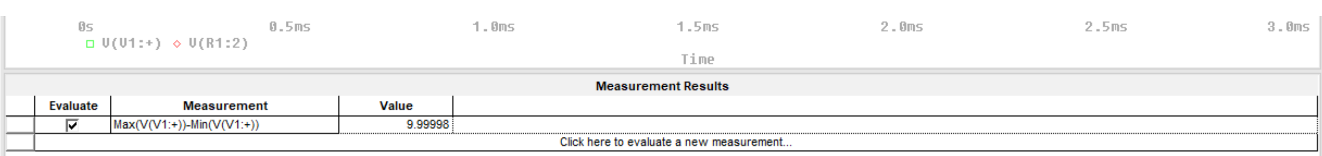


Figure 9.18

Labeling the Graph:

1. Add a text label by selecting “Text Label”.

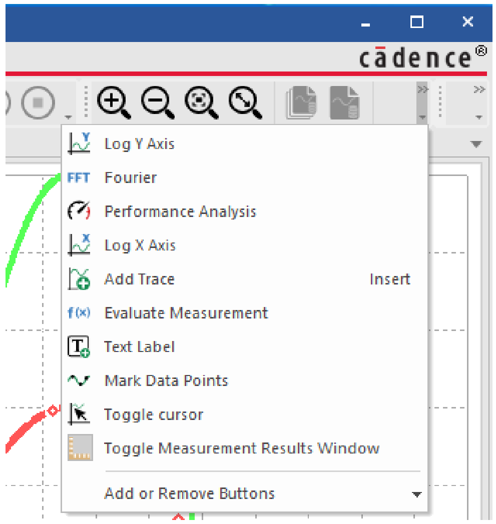


Figure 9.19

2. Add the desired text. For the running example the green trace is V_{in}.

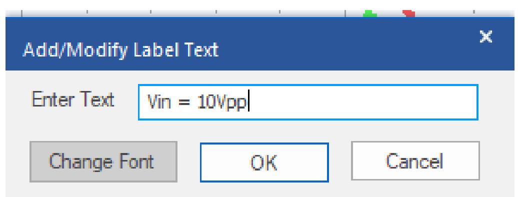


Figure 9.20

3. Change font so the text color matches the trace color and the text size is big enough to see when printed.

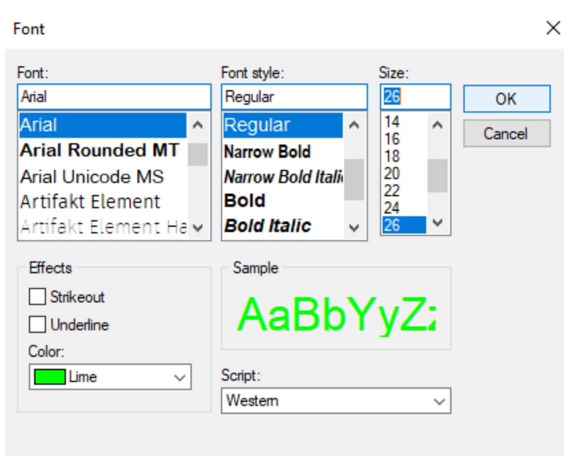


Figure 9.21

4. Place the text where it is not obstructing the plot.

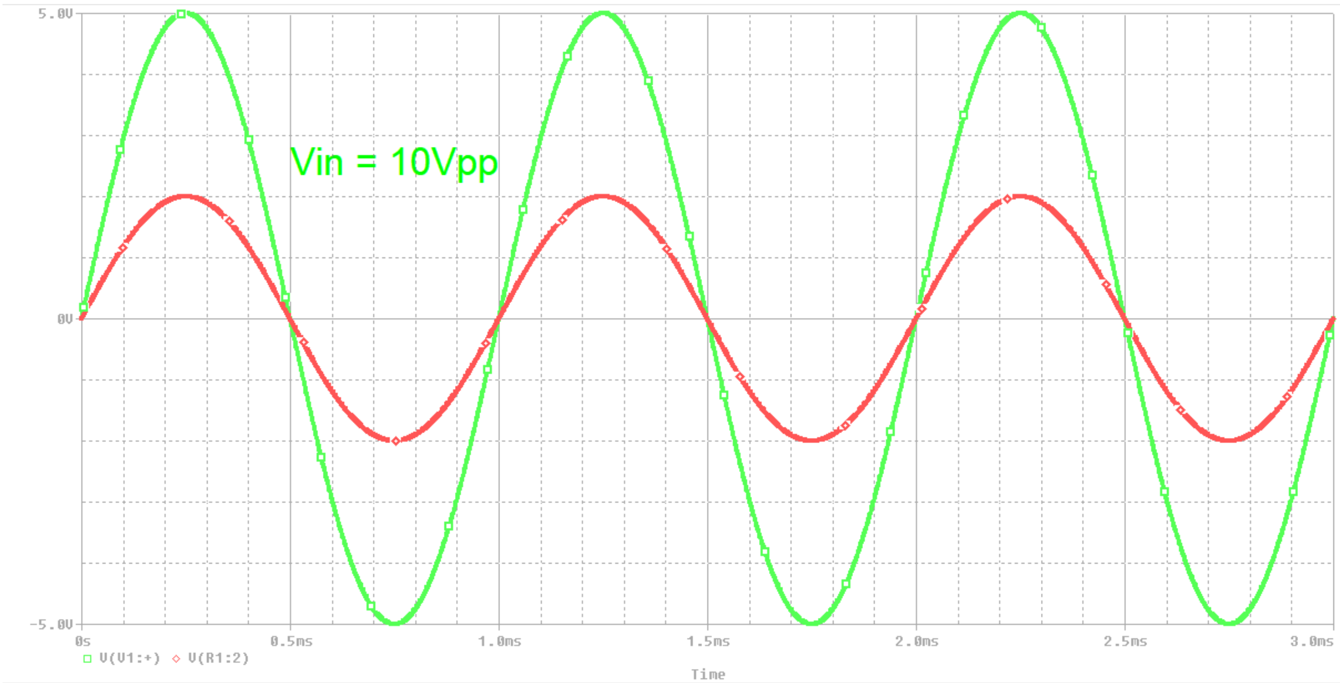


Figure 9.22

Note: For every simulation indicate the V_{pp} of each trace and the phase differExence between traces.

1. Simulate Figure 9.7
2. Simulate Figure 9.10
3. Simulate Figure 9.12

EXPERIMENT 10 : FIRST ORDER CIRCUITS

OBJECTIVE

To understand how to calculate and observe the time constant in an inductor or capacitor.

EQUIPMENT NEEDED

- Oscilloscope
- Function Generator
- Resistors, Capacitors, Inductors

THEORY

Voltage-Current Relationship for a Capacitor:

The voltage across a capacitor and the current through the capacitor are related as follows:

$$V_C(t) = \frac{1}{C}\int_{-\infty}^{t} i_C(t')dt' + V_c(-\infty)$$

$$i_C(t) = C\frac{dv_C(t)}{dt}$$

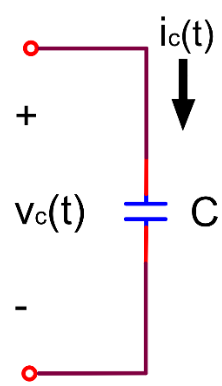


Figure 10.1

Equation (9.1) can be written as $(if\ V_C(-\infty) = 0)$

$$V_C = \frac{1}{C}\int_{-\infty}^{0} i_c(t')dt' + \frac{1}{C}\int_{0}^{t} i_c(t')dt'$$

With $V_0 = \frac{1}{C}\int_{-\infty}^{0} i_c(t')dt' \rightarrow V_C(t) = V_o + \frac{1}{C}\int_{0}^{t} i_c(t')dt'$

If the capacitor has no initial voltage ($V_o = 0$), then the equation reduces to:

$$V_C(t) = \frac{1}{C}\int_0^t i_c(t')dt'$$

It is also clear that, when the voltage v(t) across the capacitor is constant, the current through the capacitor is zero. Under such a condition the capacitor can be replaced by an open circuit. This occurs for the case of DC input, steady state condition.

Voltage-Current Relationship for an Inductor

The voltage across an inductor and the current through the inductor are related as follows:

$$V_L(t) = L\frac{di_L(t)}{dt}$$

$$i_L(t) = \frac{1}{L}\int_{-\infty}^t v_L(t')dt'$$

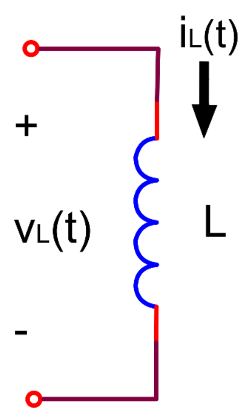


Figure 10.2

Equation (9.7) can be written as

$$i_L = \frac{1}{L}\int_{-\infty}^{0} v_L(t')dt' + \frac{1}{L}\int_{0}^{t} v_L(t')dt'$$

With

$$I_0 = \frac{1}{L}\int_{-\infty}^{0} v_L(t')dt' \rightarrow i_L(t) = I_0 + \frac{1}{L}\int_{0}^{t} v_L(t')dt'$$

If the inductor has no initial current ($I_0 = 0$), then reduces to:

$$i_L(t) = \frac{1}{L}\int_{0}^{t} v_L(t')dt'$$

It is also clear that, when the current iL(t) through the inductor is constant, the voltage across the inductor is zero. Under such a condition the inductor can be replaced by a short circuit. This occurs for the case of DC input, steady state condition.

Simple RC and RL Circuits:

The differential equation of a simple RC and RL circuits (first order differential equation) can be obtained by application of KVL or KCL.

For the case of a simple RC circuit the time constant τ in seconds is defined as;

$$\tau = RC$$

For the case of a simple RL circuit the time constant τ in seconds is defined as

$$\tau = \frac{L}{R}$$

The terms τ = RC and $\tau = \frac{L}{R}$ appear in the solutions of simple RC and RL circuits respectively. It is important to remember that for the case of DC input, it takes approximately 5τ (5RC) for the capacitor to become fully charged, and $5\tau\left(\frac{5L}{R}\right)$ for the inductor to become fully fluxed or energized, assuming initially there is no voltage across the capacitor and no current flowing through the inductor.

For the DC input, steady state solution, the capacitor is replaced by an open in the RC circuit and inductor is replaced by a short in the RL circuit.

For further information, review the sections on RC and RL circuits in your textbook.

1. A voltage source can be modeled as an ideal voltage source in series with a resistance, R_{int} as shown in Figure 10.3.

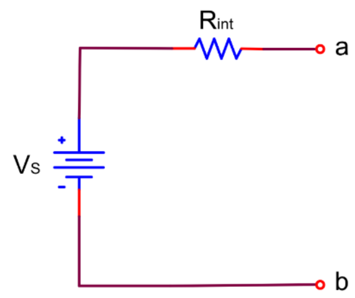


Figure 10.3

The internal resistance of the voltage source (R_{int}) can be determined by placing a known resistance of R across the open terminals of the circuit in Figure 10.3. In doing so, the following procedure allows the student to measure this internal resistance and use it in circuits that are energized by voltage sources.

If:

Voc = Vab, when a-b is open-circuited, and

VR = Vab, when a resistance, R, is connected between a and b

Show that

$$R_{int} = \frac{R(V_{OC} - V_R)}{V_R}$$

2. At t = 0, the switch in Figure 10.4 is moved from position 1 to position 2.

Calculate and sketch, vC(t), t ≥ 0.

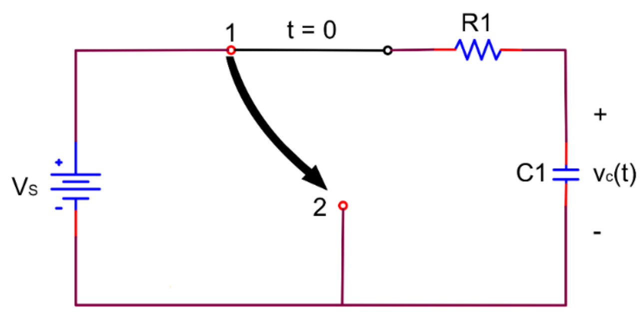


Figure 10.4

3. An "actual" inductor can be modeled as a resistor in series with an ideal inductor. If the circuit in Figure 10.5 is constructed with a one volt d.c. source, show that:

$$R_{dc} = \left(\frac{R}{1 - V_L}\right) V_L$$

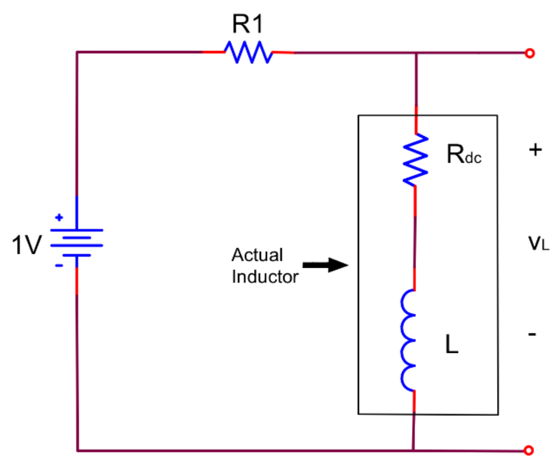


Figure 10.5

4. Assume the actual inductor in Figure 10.6 has a dc resistance, R_{dc} and an inductance, L. Calculate and sketch, VR(t), the voltage across the resistor.

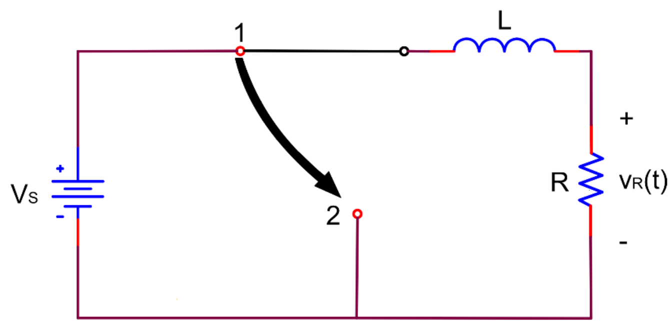


Figure 10.6

1. Use the technique described in part 1 of the pre-lab to determine R_{int} for your square wave generator, when the generator is set to 2kHz. Use values of R that give VR readings in the range of $\frac{1}{3}V_{oc} - \frac{2}{3}V_{oc}$. Be sure to include this as a part of your total resistance in the remainder of this and other labs. You should recheck the value if you vary the frequency later in the experiment.

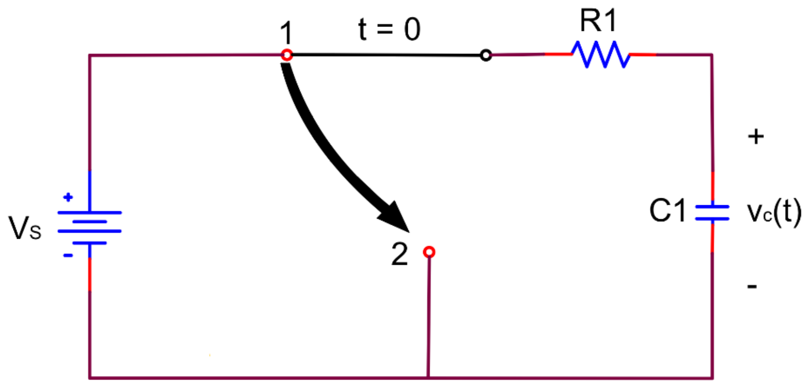


Figure 10.7

The natural response of RC, RL and RLC circuits can be demonstrated with the use of a square wave function generator. Consider Figure 10.7. The switch is left in position 1 for a "long time" so that the capacitor charges to V volts. When the switch is moved to position 2 (at t = 0) the capacitor discharges and we have the natural response of the RC circuit. The circuit in Figure 10.8 provides the same result repeatedly for the ease of viewing on the oscilloscope if $\frac{T}{2}$ is long compared to the circuit time constant, RC.

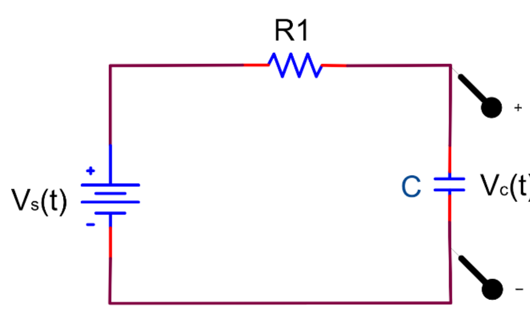


Figure 10.8

2. Construct the circuit in Figure 10.9.. Use the oscilloscope to observe, $v_c(t)$. Compare this to the theoretical time constant (be sure to include the internal resistance of the source).

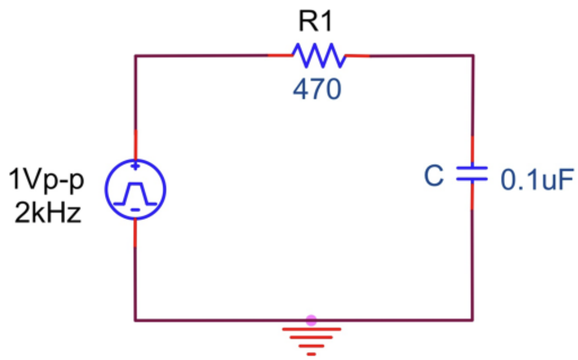


Figure 10.9

	τ	5τ	v_c
Calculated			
Measured			
Percent Error			

3. Construct Figure 10.10 and adjust the frequency of the function generator to obtain an adequate display (5τ).

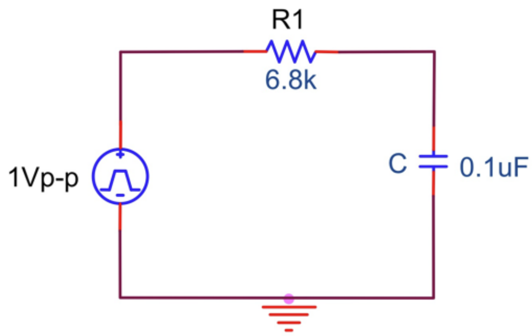


Figure 10.10

	τ	5τ	v_c
Calculated			
Measured			
Percent Error			

4. Using the procedure described in part 3 of the pre-lab, calculate R_{dc} for a 68mH inductor.

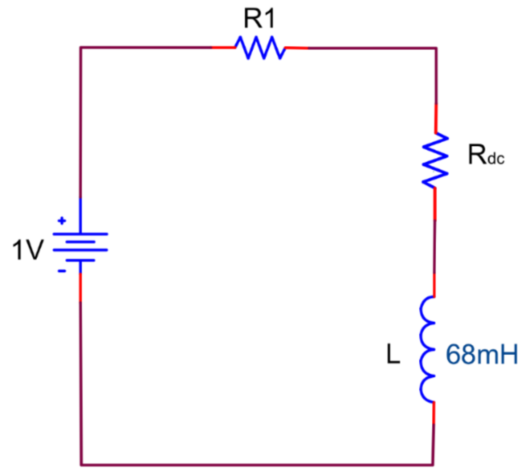


Figure 10.11

5. Construct the circuit of Figure 10.12. Adjust the frequency of the function generator for an to obtain a 5τ display. Compare this value to the calculated time constant. Be sure to include the effects of R_{int} and Rdc into your calculations.

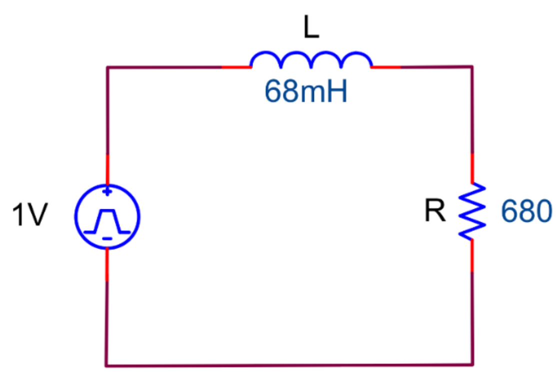


Figure 10.12

	τ	5τ	v_r
Calculated			
Measured			
Percent Error			

6. Using the above procedure, find (experimentally) the dc resistance and the inductance of the unknown inductor supplied by your lab instructor.

Finding Components:

1. The Pulse source can be found by going to “PSpice Component”→”Source”→”Voltage Sources”→”Pulse”.

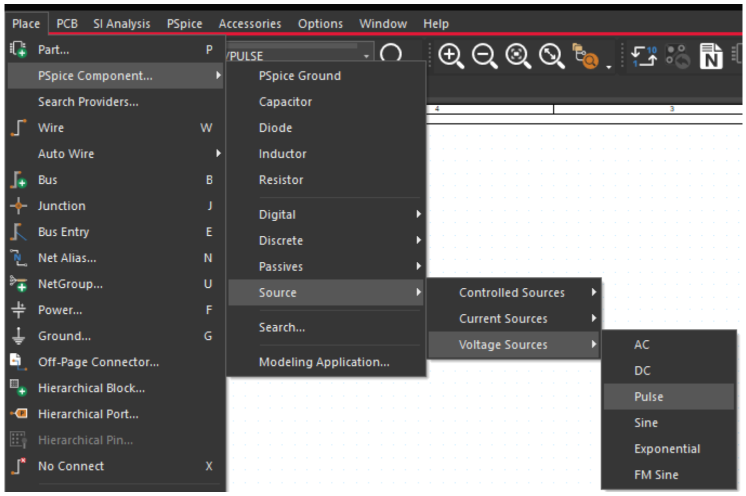


Figure 10.13

Adjusting Source Parameters:

The Pulse component requires seven parameters to be filled before running the simulation. V1,TD,TR,TF can have the same values shown in Figure 10.14. V2 needs to be the voltage that was used during the lab, PW is going to be half of the period, and PER will be calculated from the frequency used in the lab. As a reminder $f = \frac{1}{T}$.

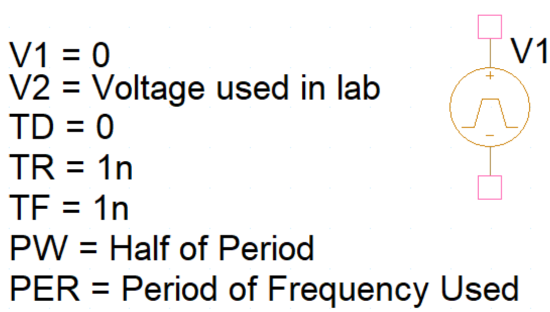


Figure 10.14

1. Simulate Figure 10.9 and obtain the same output as your oscilloscope.
2. Simulate Figure 10.10 and obtain the same output as your oscilloscope.
3. Simulate Figure 10.12 and obtain the same output as your oscilloscope.

EXPERIMENT 11 : CIRCUIT DESIGN II

OBJECTIVE

Leverage previous understanding of tau and frequency to design an RC circuit to output a specified wave form.

EQUIPMENT NEEDED

- Oscilloscope
- Function Generator
- Resistors and Capacitors

THEORY

In previous experiments, you have discovered that the frequency (f) of a signal is equal to the inverse of its time period (T), and that the time constant (τ) of a circuit with resistance (R) and capacitance (C) is equal to the product of R and C. By using these two equations, it is possible to determine the value of R that is required to achieve a specific signal with a known time interval. As a designer, it is important to recognize that you have control over certain variables in the circuit, such as R and C, and by manipulating these values, you can adjust the behavior of the signal. Therefore, understanding the relationship between frequency, time period, time constant, resistance, and capacitance is critical in designing circuits that perform the desired function.

$$f = \frac{1}{T} \qquad\qquad \tau = RC$$

PRELIMINARY CALCULATIONS

1. Using the circuit shown in Figure 11.1, choose values for R and C that will duplicate the waveform shown in Figure 11.2.

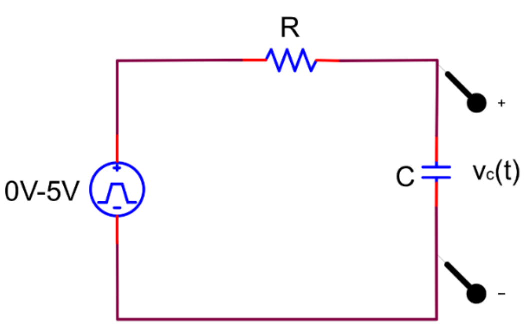


Figure 11.1

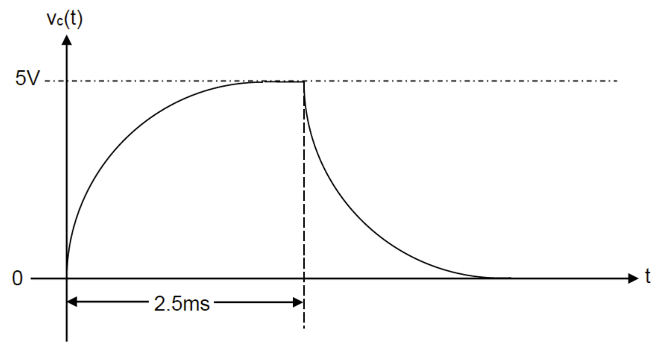


Figure 11.2

PROCEDURE

1. Construct Figure 11.3and measure the voltage across the capacitor at which the wave form on the oscilloscope matches Figure 11.2.

$$X_c = \frac{1}{2\pi FC} \qquad Z = \sqrt{R^2 + {X_c}^2} \qquad I = \frac{5V_{pp}}{Z} \qquad V_c = IX_c$$

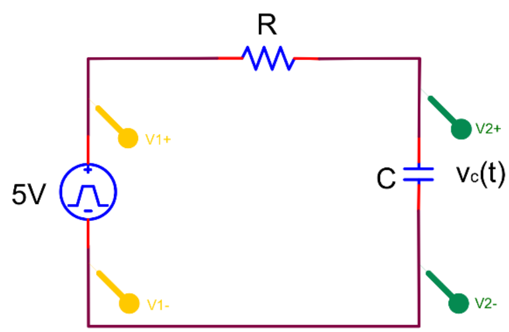


Figure 11.3

	τ	5τ	v_c
Calculated			
Measured			
Percent Error			

Using the Cursor:

1. Each version of Cadence might be slightly diffenent in the symbol used for the cursor but it should generally look the same as indicated below, it will be named “Toggle cursor” when the mouse hovers over the button.

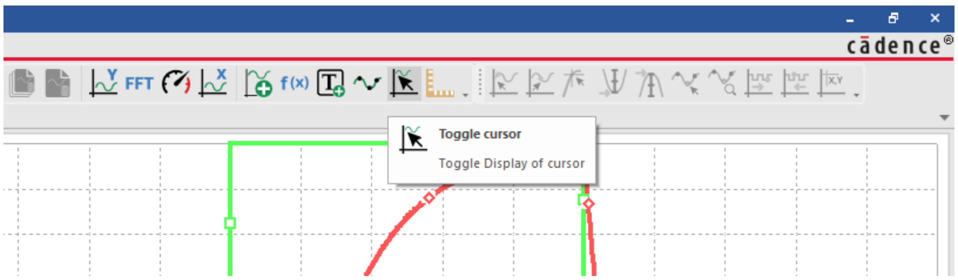
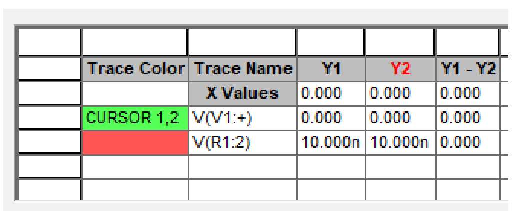


Figure 11.4

2. After the cursor has been toggled a window will appear at the bottom left of the screen. This window is akin to the cursor parameters on the oscilloscope. The image on the left is from Cadence and the image on the right is from the oscilloscope.



Trace Color	Trace Name	Y1	Y2	Y1 - Y2
	X Values	0.000	0.000	0.000
CURSOR 1,2	V(V1:+)	0.000	0.000	0.000
	V(R1:2)	10.000n	10.000n	0.000

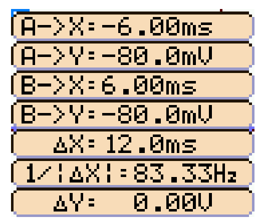
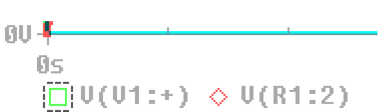
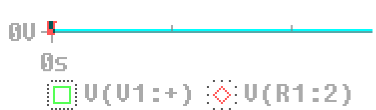


Figure 11.5

Figure 11.6

3. While in cursor mode there will also be the name of the traces that are on the graph, in this example trace one is “V(V1:+)” and trace two is “V(R1:2)”. There are two cursors that can be used, one is activated with a left click on the mouse and the other is a right click on the mouse. Figure 11.7 denotes that both cursors are on the green trace, this is indicated by the solid dashed lines. Figure 11.8 denotes one cursor on the green trace and one cursor on the red trace, the boarders have switched from dashed lines to dotted lines. Figure 11.9 denotes both cursors are on the red trace, again it is denoted with solid dashed lines.





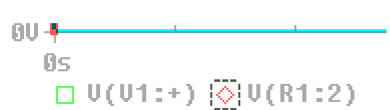


Figure 11.7

Figure 11.8

Figure 11.9

4. Now that both cursors are on the red trace, left click for the first cursor and right click for the second cursor. Ensure that you are measuring the same points as in the lab (5τ).

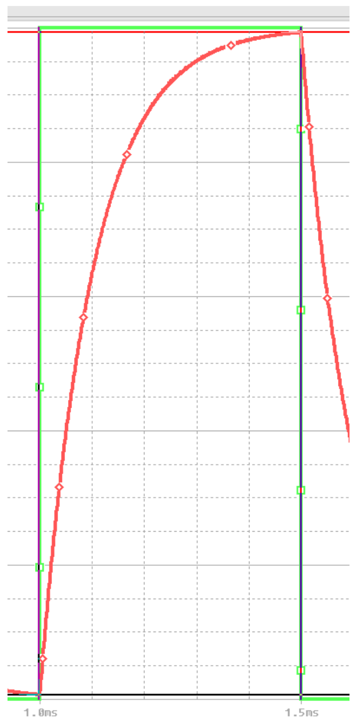


Figure 11.10

5. With the cursors placed in the appropriate locations reference the table at the bottom left of the screen (Figure 11.11). The values of interest are the "X Values", for the example below they are as follows "Y1=0.9983m", "Y2=1.500m" and "Y1-Y2=-501.622u". The 5τ for this circuit is the ΔY value, 501.662u.

	Trace Color	Trace Name	Y1	Y2	Y1 - Y2		Y1(Cursor1) - Y2(Cursor2)		-9.865		
		X Values	0.9983m	1.5000m	-501.662u		Y1 - Y1(Cursor1)	Y2 - Y2(Cursor2)	Max Y	Min Y	Avg Y
		V(V1:+)	0.000	10.000	-10.000		-68.055m	66.866m	10.000	0.000	5.0000
	CURSOR 1,2	V(R1:2)	68.055m	9.933	-9.865		0.000	0.000	9.933	68.055m	5.0006

Figure 11.11

1. Build and simulate Figure 11.3. On the graph display the V_{pp} and 5τ of the wave form.

NOTE: To measure τ in Cadence you must use the cursors just like the oscilloscope in the lab.

EXPERIMENT 12 : IMPEDANCE AND ADMITTANCE CIRCUITS

EQUIPMENT NEEDED

- Oscilloscope
- Function Generator
- Resistors, Capacitors, Inductors

THEORY

A simple way of solving for steady state solution of circuits with sinusoidal input is to convert the voltages and currents to phasor notation. The voltage v(t) and current i(t) can be converted to phasor notation as follows:

$$v(t) = V cos(\omega t + \theta)$$

$$i(t) = I cos(\omega t + \phi)$$

The ratio of the phasor voltage to the phasor current is defined as the impedance.

$$Z = \frac{V}{I} \ \angle Z = \theta - \phi$$

In general impedance is complex. The real part of impedance is known as resistance and the imaginary part as reactance.

$$Z = R + jX$$

Reactance for an inductance and a capacitor are given below in and respectively.

$$X_L = \omega L$$

$$X_C = \frac{1}{\omega C}$$

The ratio of the phasor current to the phasor voltage is defined as the admittance.

$$Y = \frac{I}{V} \ \angle Y = \phi - \theta$$

Like impedance, admittance is also complex. The real part of admittance is known as conductance and the imaginary part as susceptance.

$$Y = G + jS$$

Susceptance for an inductance and a capacitor are given below in and respectively.

$$S_L = -\frac{1}{\omega L}$$

$$S_C = \omega C$$

All the rules of circuit analysis that were covered for the case of pure resistive circuits apply here. The exception is that all voltages and currents are phasors, and resistors are replaced by the impedances.

Kirchoff's Laws:

Kirchhoff's Voltage Law: The algebraic sum of the voltages around any closed path is zero.

$$\sum_{n=1}^{N} V_n = 0$$

Kirchhoff's Current Law: The algebraic sum of the currents at any node is zero.

$$\sum_{n=1}^{N} I_n = 0$$

Series Circuit:

A series circuit in terms of phasors and impedance is shown in Figure 12.1. The current I in the series circuit is the same through all elements and other rules are like those of resistive circuit.

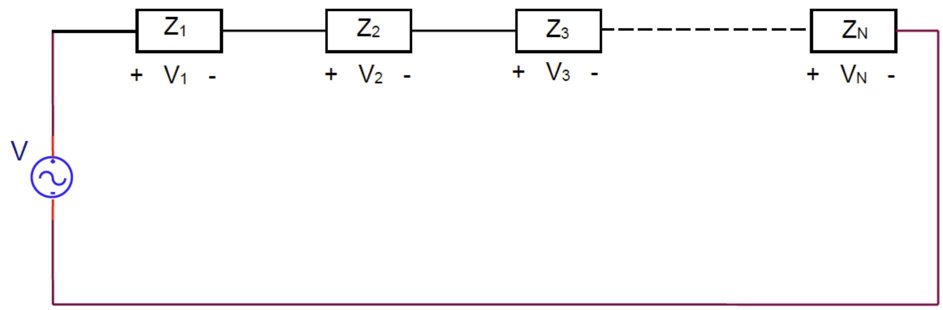


Figure 12.1

KVL: $V = V_1 + V_2 + V_3 + \cdots + V_N$

Total Impedance: $Z_s = Z_1 + Z_2 + Z_3 + \cdots + Z_N$

Voltage Divider Rule:

$$V_1 = \left(\frac{Z_1}{Z_S}\right)V \qquad V_2 = \left(\frac{Z_2}{Z_S}\right)V \qquad V_3 = \left(\frac{Z_3}{Z_S}\right)V \quad \ldots \quad V_N = \left(\frac{Z_N}{Z_S}\right)V$$

Parallel Circuit:

A parallel circuit in terms of phasors and admittances is shown in Figure 12.2. The voltage V across parallel elements is the same and other rules are like those of resistive circuit.

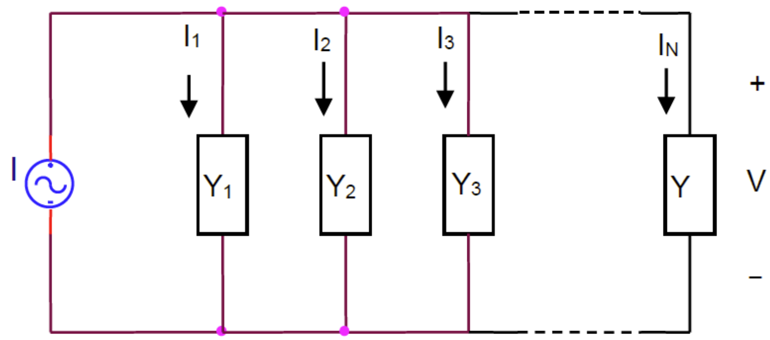


Figure 12.2

$$KCL: I = I_1 + I_2 + I_3 + \cdots I_N$$

Total Admittance: $Y_P = Y_1 + Y_2 + Y_3 + \cdots Y_N$

Current Divider Rule:

$$I_1 = \left(\frac{Y_1}{Y_P}\right)V \qquad I_2 = \left(\frac{Y_2}{Y_P}\right)V \qquad I_3 = \left(\frac{Y_3}{Y_P}\right)V \quad \ldots \quad I_N = \left(\frac{Y_N}{Y_S}\right)V$$

Phase Measurement

In general phasor voltages and current as well as impedances are complex. Hence measuring the phase becomes as important as the amplitude. The input voltage source can be considered as having zero-degree phase shift and the phase of other voltages and/or currents in a circuit are measured with respect to the input. There are two methods of measuring the phase difference between two sinusoids and these methods are explained below.

Method 1 – Dual Trace

Consider two sinusoids having frequency = 2πf. For the t axis, the difference between peaks is ϕ, measured in radians, and this is the difference in phase between v1 and v2.

(Note that V_2 lags V_1) However, on an oscilloscope, the axis is time t.

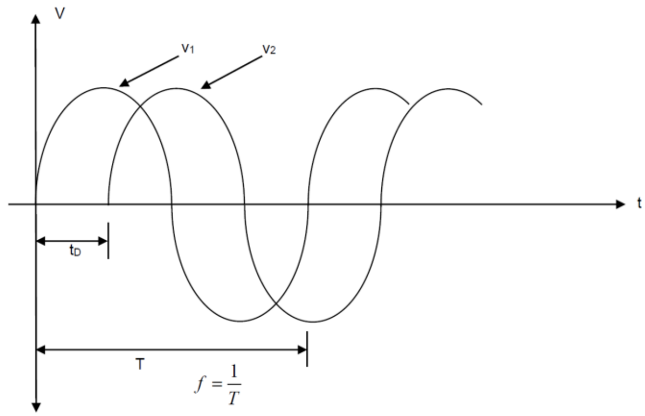


Figure 12.3

Since: $t_D = \frac{\phi}{\omega}$

Then $\phi = \omega t_D \longrightarrow \phi = \frac{2\pi t_D}{T} radians$

The phase angle can also be expressed in degrees:

$$\phi = 2\pi\left(\frac{t_D}{T}\right)\left(\frac{360°}{2\pi}\right) \longrightarrow \phi = \left(\frac{t_D}{T}\right)(360°)$$

Method 2 – Lissajous Pattern

An alternative procedure for measuring phase is to apply one sinusoid to the vertical axis y of a scope and one to the horizontal axis x. The resulting ellipse, called a Lissajous pattern, is shown. (Figure 12.4)

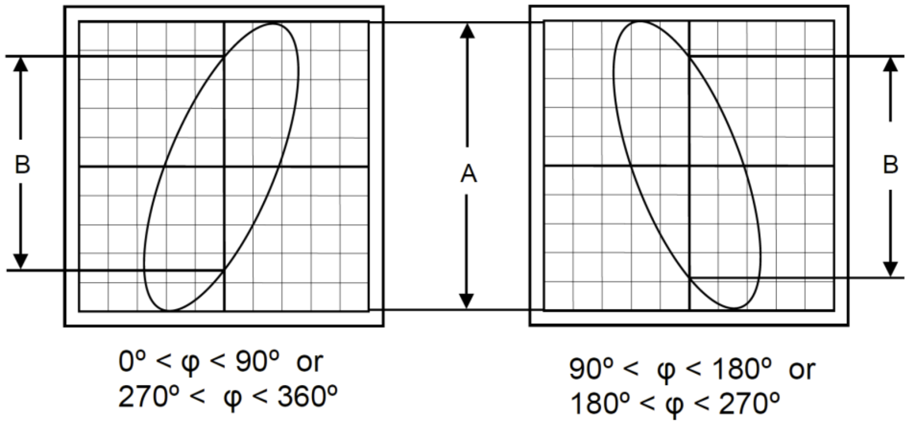


Figure 12.4

$$\phi = \sin^{-1}\frac{B}{A}$$

1. For the circuit of Figure 12.5, calculate Z_1, Z_2, andZ_s, the impedances of the series R-L, the series R-C, and the entire circuit, respectively.

   a. $f = 1kHz \quad C = 0.1\mu F$

   b. $f = 1kHz \quad C = 1\mu F$

   c. $f = 500Hz \quad C = 0.1\mu F$

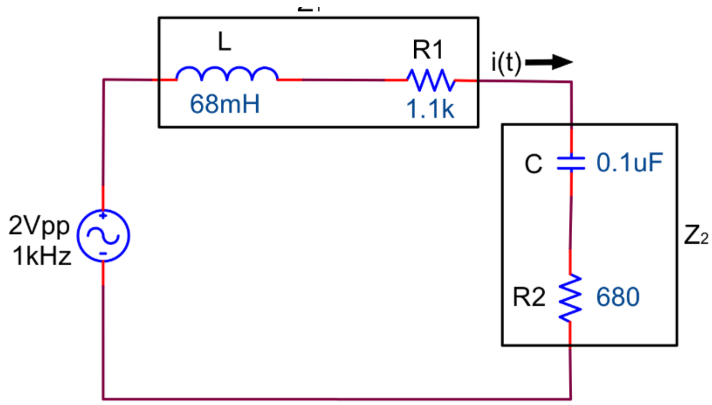


Figure 12.5

	V_1	V_2	I	θ_1	θ_2
$F = 1kHz, C = 0.1\mu F$					
$F = 1kHz, C = 1\mu F$					
$F = 500Hz, C = 0.1\mu F$					

2. For the three cases in part 1, calculate the phasors V_1, V_2, and I corresponding to $V_1(t)$, $V_2(t)$, and i(t), respectively.

PROCEDURE

1. Construct the circuit of **Error! Reference source not found.**. Use the oscilloscope to measure the magnitude of the phasor V_2 and the voltage across the 680Ω resistor. While measuring Figure 12.6 ensure the oscilloscope is configured to measure θ_1.

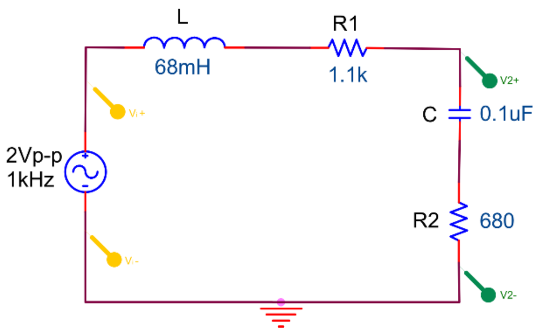


Figure 12.6

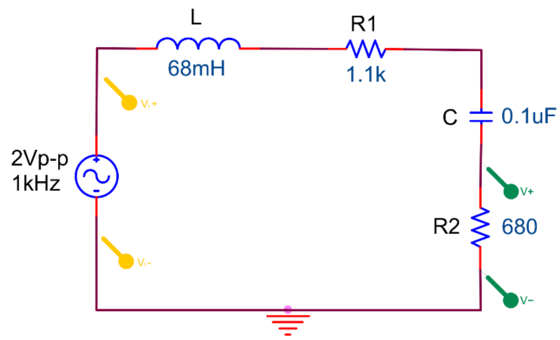


Figure 12.7

2. Construct Figure 12.8 and use the oscilloscope to measure the magnitude of the phasor V_1 and record the θ_2.

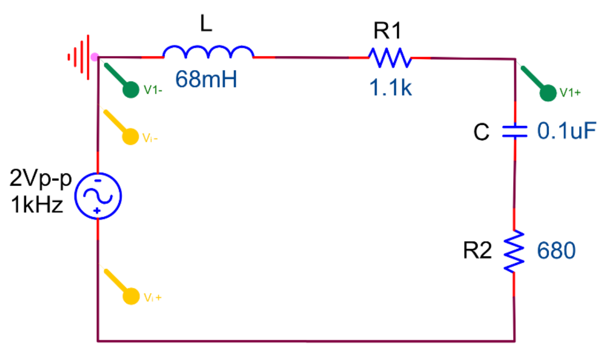


Figure 12.8

	V_1	V_2	$V_{680\Omega}$	I	θ_1	θ_2
Calculated						
Measured						

3. Using the measured values of V_1, V_2, V_S, and I, calculate the impedances (mag. and phase) Z_1, Z_2, and Z_S (as defined in pre-lab). Show that $Z_S = Z_1 + Z_2$. Compare the calculated (experimental) values to those obtained in the pre-lab.

1. Construct the circuit of Figure 12.9. Use the oscilloscope to measure the magnitude of the phasor V_2 and θ_1. After, measure the voltage across the 680Ω resistor.

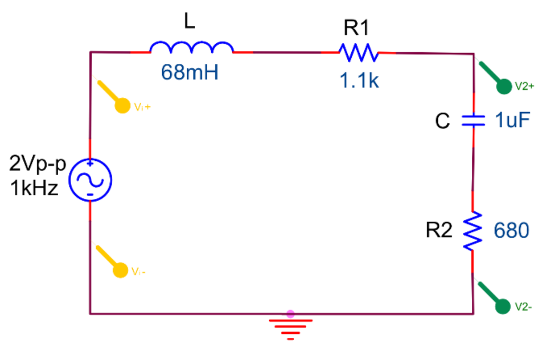


Figure 12.9

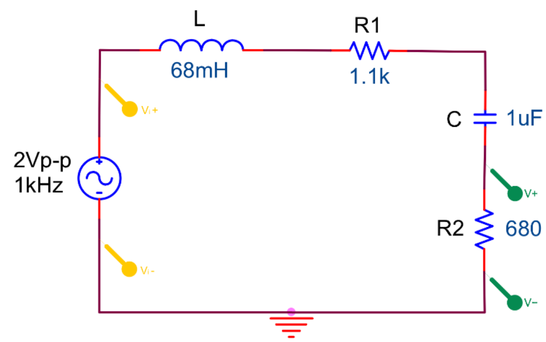


Figure 12.10

2. Construct Figure 12.8 and use the oscilloscope to measure the magnitude of the phasor V_1 and record the θ_2.

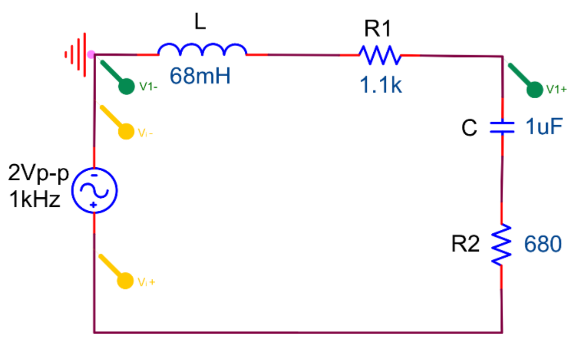


Figure 12.11

	V_1	V_2	$V_{680\Omega}$	I	θ_1	θ_2
$Calculated$						
$Measured$						

3. Using the measured values of V_1, V_2, V_S, and I, calculate the impedances (mag. and phase) Z_1, Z_2, and Z_S (as defined in pre-lab). Show that $Z_S = Z_1 + Z_2$. Compare the calculated (experimental) values to those obtained in the pre-lab.

4. Construct the circuit of Figure 12.9. Use the oscilloscope to measure the magnitude of the phasor V_2 and θ_1. After, measure the voltage across the 680Ω resistor.

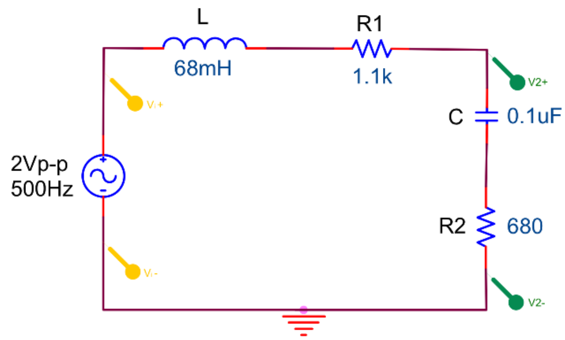


Figure 12.12

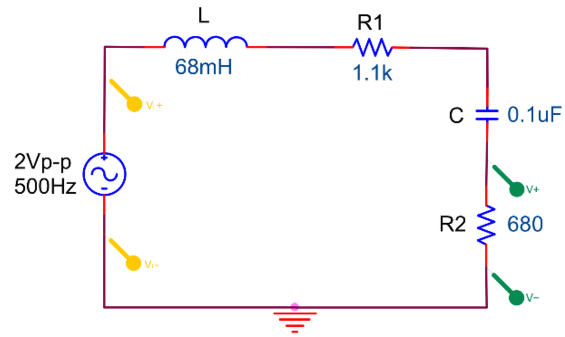


Figure 12.13

5. Construct Figure 12.8 and use the oscilloscope to measure the magnitude of the phasor V_1 and record the θ_2.

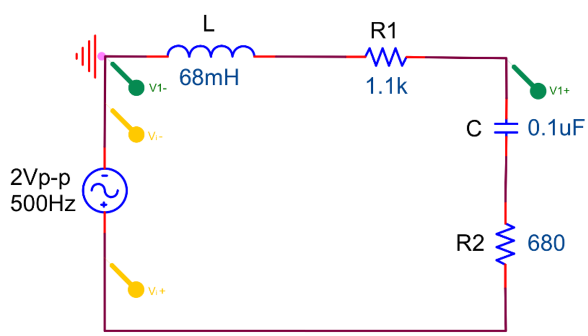


Figure 12.14

6. Obtain the magnitude and phase angle of the current phasor, I, by observing the voltage across the 680Ω resistor.

	V_1	V_2	$V_{680\Omega}$	I	θ_1	θ_2
Calculated						
Measured						

7. Using the measured values of V_1, V_2, V_S, and I, calculate the impedances (mag. and phase) Z_1, Z_2, and Z_S (as defined in pre-lab). Show that $Z_S = Z_1 + Z_2$. Compare the calculated (experimental) values to those obtained in the pre-lab.

EXPERIMENT 13 : FREQUENCY RESPONSE

OBJECTIVE

Learning how to calculate and measure the half power point.

EQUIPMENT NEEDED

- Oscilloscope
- Function Generator
- Resistors, Capacitors, Inductor

THEORY

In the previous experiment, phasor voltages and currents were computed and measured for a sinusoidal input at frequencies of 500Hz and 1kHz. Since impedances of inductors and capacitors are dependent on frequency of the input, the phasor voltages and currents are also frequency dependent.

The frequency response considers the input-output relation in terms of amplitude and phase for a range of desired frequencies. As an example, consider a phasor series circuit shown in Figure 13.1.

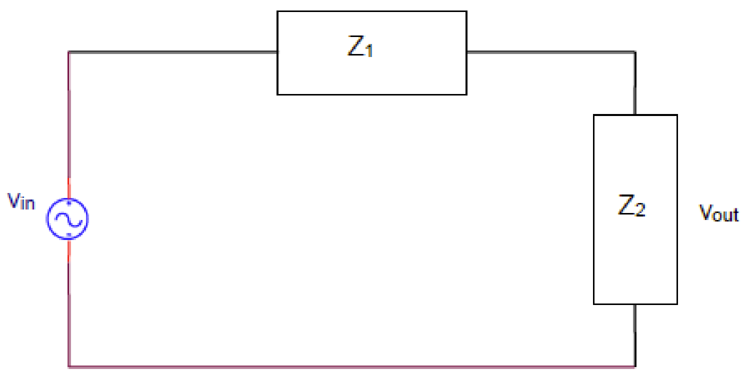


Figure 13.1

The system transfer function $H(j\omega)$ is defined as

$$H(j\omega) = \frac{v_{out}}{v_{in}}$$

Using voltage division: $V_{out} = \left(\frac{Z_2}{Z_1+Z_2}\right) v_{in} \longrightarrow H(j\omega) = \frac{Z_2}{Z_1+Z_2}$

In general, $H(j\omega)$ is complex, that is

$$H(j\omega) = |H(j\omega)|\angle\phi(j\omega)$$

The term $|H(j\omega)|$ is known as the amplitude response and $\phi(j\omega)$ is known as the phase response. Both $|H(j\omega)|$ and $\phi(j\omega)$ are dependent on the input frequency ω.

1. For the circuit in Figure 13.2:

   a. Calculate an expression for $H(j\omega) = \frac{V_2}{V_1}$, [V2 and V1 are the phasors for the voltages v2(t) and v1(t) respectively].

   b. Sketch $|H(j\omega)|$ versus ω , indicating the peak value and half power point.
   Note: the half power point is the frequency at which $|H(j\omega)|$ is reduced to $\frac{1}{\sqrt{2}}$ of its peak value.

   c. Plot arg $|H(j\omega)|$ versus ω.

   d. Calculate 10, 1k, 100k

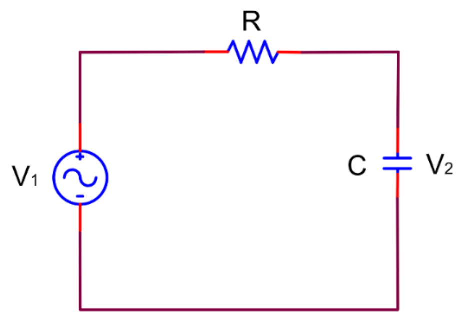


Figure 13.2

2. For the circuit in Figure 13.3:

   a. Calculate an expression for $H(j\omega) = \frac{V_2}{V_1}$, [V2 and V1 are the phasors for the voltages v2(t) and v1(t) respectively].

   b. Sketch $|H(j\omega)|$ versus ω , indicating the peak value and half power point.
   Note: the half power point is the frequency at which $|H(j\omega)|$ is reduced to $\frac{1}{\sqrt{2}}$ of its peak value.

   c. Plot arg $|H(j\omega)|$ versus ω.

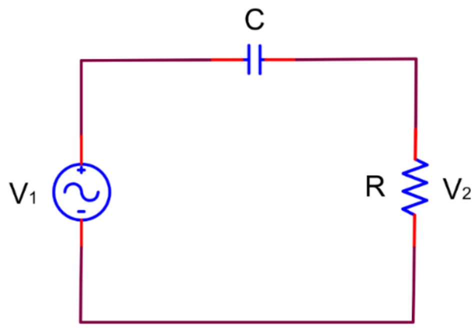


Figure 13.3

3. For the circuit in Figure 13.4:

a. Calculate an expression for $H(j\omega) = \frac{V_2}{V_1}$, [V2 and V1 are the phasors for the voltages v2(t) and v1(t) respectively].

b. Sketch $|H(j\omega)|$ versus ω , indicating the peak value and half power point.

Note: the half power point is the frequency at which $|H(j\omega)|$ is reduced to $\frac{1}{\sqrt{2}}$ of its peak value.

c. Plot arg $|H(j\omega)|$ versus ω.

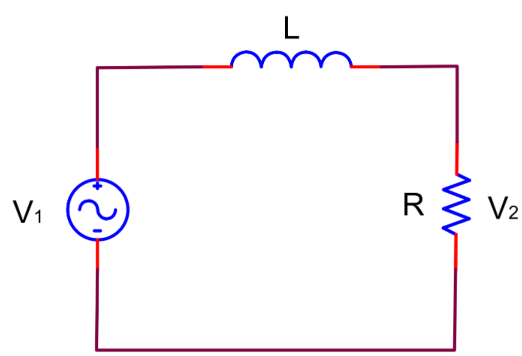


Figure 13.4

1. Construct the circuit of Figure 13.5. Use the oscilloscope to measure the magnitude and phase angle (with respect to the input) of v2. Make sure that the input remains at 1V as you vary the frequency. Use your measurements to plot the magnitude and phase angle versus frequency.
   Note: Since v1 = 1V and φ1=0º, observing v2 is the same as observing $H(j\omega)$

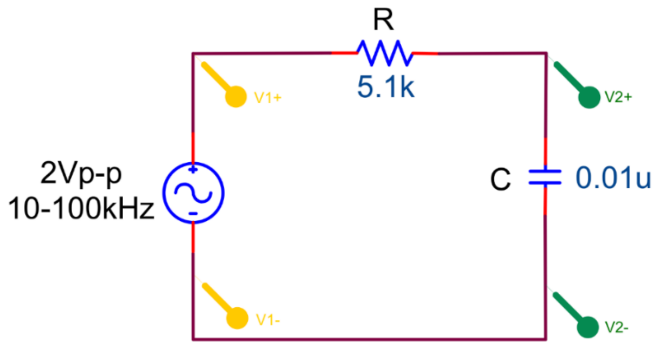


Figure 13.5

2. Determine the experimental half power point from your magnitude plot. Compare it to the theoretical value obtained in the pre-lab.

3. Construct the circuit of Figure 13.6. Use the oscilloscope to measure the magnitude and phase angle (with respect to the input) of v2. Make sure that the input remains at 1V as you vary the frequency. Use your measurements to plot the magnitude and phase angle versus frequency.
   Note: Since v1 = 1V and φ1=0º, observing v2 is the same as observing $H(j\omega)$

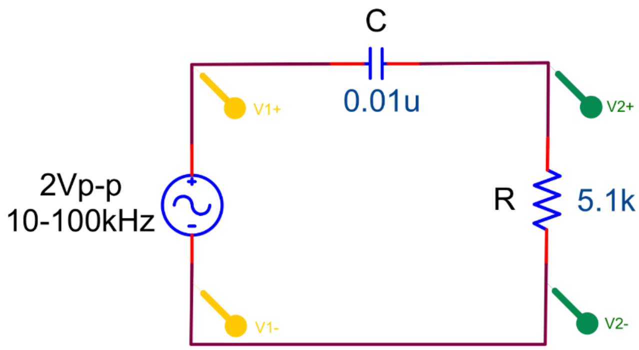


Figure 13.6

4. Determine the experimental half power point from your magnitude plot. Compare it to the theoretical value obtained in the pre-lab.

5. Construct the circuit of Figure 13.7. Use the oscilloscope to measure the magnitude and phase angle (with respect to the input) of v2. Make sure that the input remains at 1V as you vary the frequency. Use your measurements to plot the magnitude and phase angle versus frequency.
Note: Since v1 = 1V and φ1=0º, observing v2 is the same as observing $H(j\omega)$

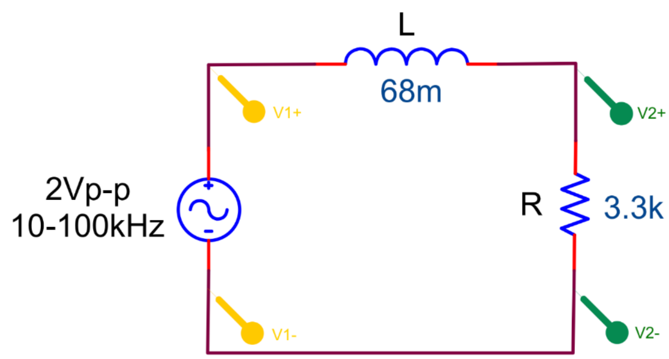


Figure 13.7

6. Determine the experimental half power point from your magnitude plot. Compare it to the theoretical value obtained in the pre-lab.

PSPICE

1. Use the SPICE AC analysis to plot the magnitude and phase angle of V2 versus frequency for Figure 13.5. Compare these to your experimental results.
2. Use the SPICE AC analysis to plot the magnitude and phase angle of V2 versus frequency for Figure 13.6. Compare these to your experimental results.
3. Use the SPICE AC analysis to plot the magnitude and phase angle of V2 versus frequency for Figure 13.7. Compare these to your experimental results.

EXPERIMENT 14 : SECOND ORDER CIRCUITS

OBJECTIVE

Understanding the behavior of an RLC circuit and how to identify overdamped, critically damped, and underdamped on an oscilloscope.

EQUIPMENT NEEDED

- Oscilloscope
- Function Generator
- Resistor, Capacitors, Inductors

THEORY

The Differential Equation – 2nd Order

The differential equations of second order circuits are obtained by application of KVL and/or KCL and using equations that were discussed in the Theory of Experiment 9. If the application of KVL or KCL results in an integro-differential equation, the derivative of both sides of the equation should be taken to convert the equation into a differential equation.

Consider a general second order differential equation

$$a\frac{d^2y}{dt^2} + b\frac{dy}{dt} + cy = 0$$

With roots

$$r_{1,2} = -\frac{b}{2a} \pm \frac{\sqrt{b^2 - 4ac}}{2a}$$

Assume a, b, and c are positive. Three cases are considered here:

Case1 “Overdamped” $b^2 - 4ac > 0$

In this case, r_1 and r_2 are real, distinct, and negative. This results in solutions of (11.1) in the form of decaying exponentials: $y = A_1 e^{r_1 t} + A_2 e^{r_2 t}$. Coefficients A_1 and A_2 are determined using the initial conditions of the system or circuit.

Case 2 “Critically Damped” $b^2 - 4ac = 0$

In this case, r_1 and r_2 are real, equal, and negative ($r = r_1 = r_2$). This results in solutions of (11.1) in the form of following equation: $y = A_1 e^{rt} + A_2 e^{rt}$ or $y = e^{rt}(A_1 + A_2 t)$

Case 3 "Underdamped" $b^2 - 4ac < 0$

In this case, r_1 and r_2 are complex. This results in solutions of (11.1) in the form of the following equation: $y = e^{\propto t}(A_1 cos\omega_d t + A_2 sin\omega_d t)$ where:

$$\alpha = -\frac{b}{2a}$$

$$\omega_d = \sqrt{\frac{4ac - b^2}{2a}}$$

The term α is known as the damping factor and ω_d is the damped frequency. Consider the graph of an underdamped case shown in Figure 14.1 where:

$$v(t) = Ae^{\propto t}\cos(\omega_d t + \theta)$$

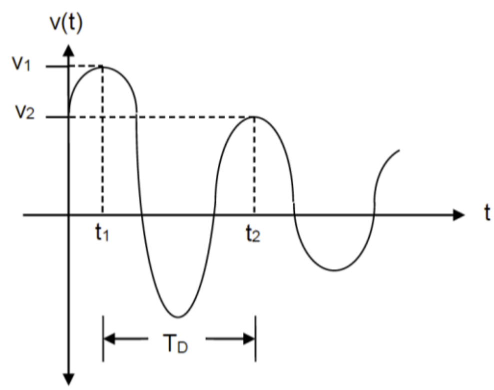


Figure 14.1

Using Figure 14.1, the following items can be determined:

$$v_1 = Ae^{\propto t_1}\cos(\omega_d t_1 + \theta)$$

$$v_2 = Ae^{\propto t_2}\cos(\omega_d t_2 + \theta)$$

$$\frac{v_1}{v_2} = e^{\propto(t_2 - t_1)}$$

where

$$t_2 - t_1 = T_D \rightarrow T_D = \frac{2\pi}{\omega_d}$$

Substituting and taking the natural logarithm of both sides results in:

$$\ln\left(\frac{v_1}{v_2}\right) = \frac{2\pi\alpha}{\omega_d}$$

$$\alpha = \frac{\omega_d}{2\pi}\ln\left(\frac{v_1}{v_2}\right)$$

1. For the circuit in Figure 14.2, write a differential equation for v(t) (in terms of R, L, and C). What range of values for R correspond to over-, under-, and critically damped cases?

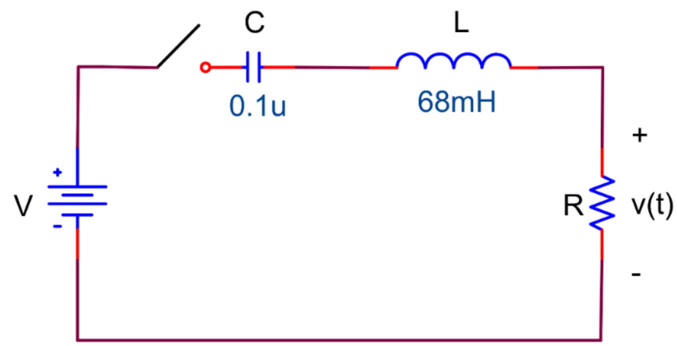


Figure 14.2

2. For the circuit in Figure 14.3, write a differential equation for v(t) (in terms of R, L, and C). What range of values for R correspond to over-, under-, and critically damped cases?

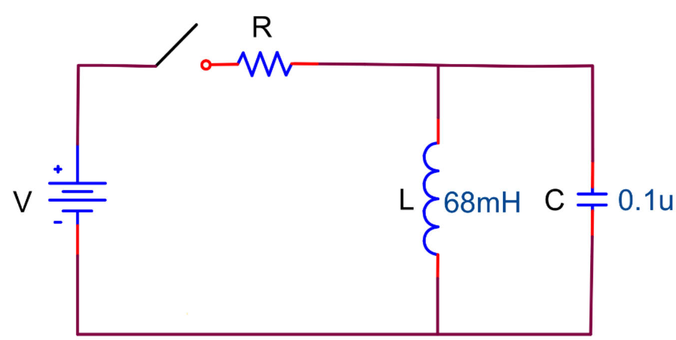


Figure 14.3

1. Use the oscilloscope to observe v(t) (while the capacitor is charging) for R = 6.8kΩ. Vary the frequency of the square wave so that you observe v(t) until it reaches its steady state value. State whether the circuit is over-, under-, or critically damped. For the underdamped case(s) calculate the theoretical damping factor, and the damped frequency. Compare these to your experimental values.

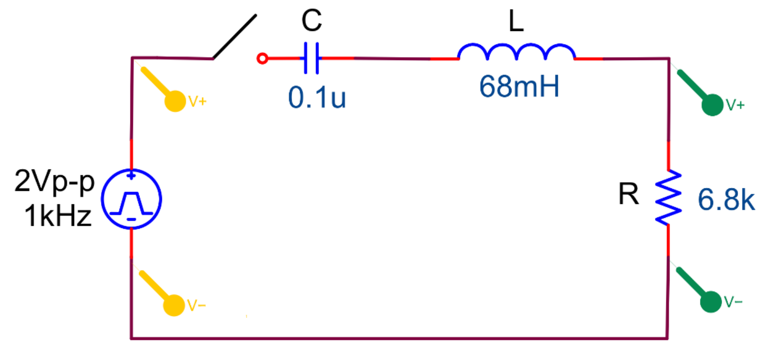


Figure 14.4

2. Use the oscilloscope to observe v(t) (while the capacitor is charging) for R = 560Ω. Vary the frequency of the square wave so that you observe v(t) until it reaches its steady state value. State whether the circuit is over-, under-, or critically damped. For the underdamped case(s) calculate the theoretical damping factor, and the damped frequency. Compare these to your experimental values.

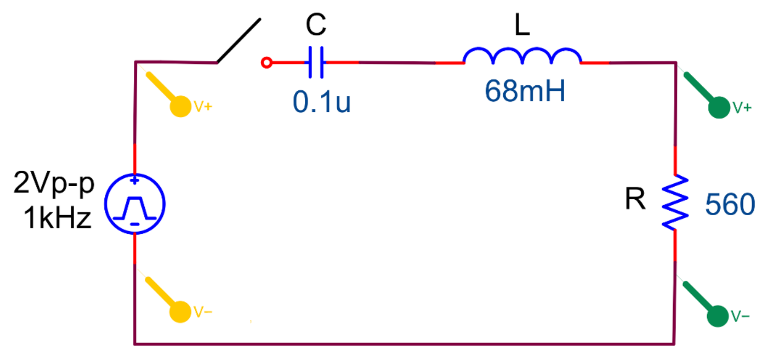


Figure 14.5

3. Use the oscilloscope to observe v(t) (while the capacitor is charging) for R = 6.8kΩ. Vary the frequency of the square wave so that you observe v(t) until it reaches its steady state value. State whether the circuit is over-, under-, or critically damped. For the underdamped case(s) calculate the theoretical damping factor, and the damped frequency. Compare these to your experimental values.

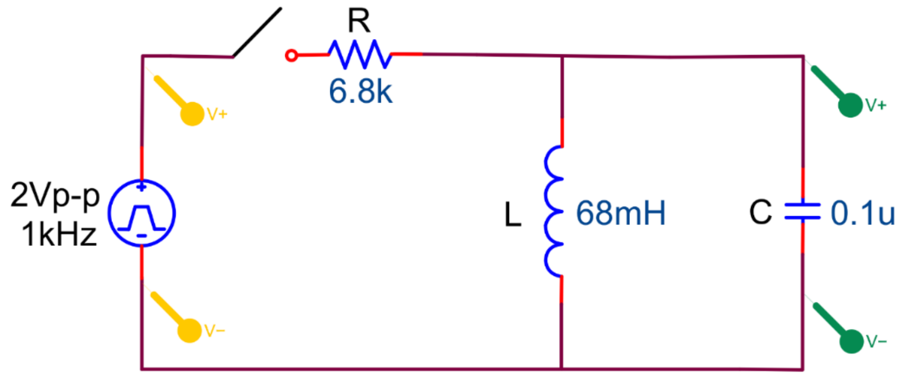


Figure 14.6

4. Use the oscilloscope to observe v(t) (while the capacitor is charging) for R = 6.8kΩ. Vary the frequency of the square wave so that you observe v(t) until it reaches its steady state value. State whether the circuit is over-, under-, or critically damped. For the underdamped case(s) calculate the theoretical damping factor, and the damped frequency. Compare these to your experimental values.

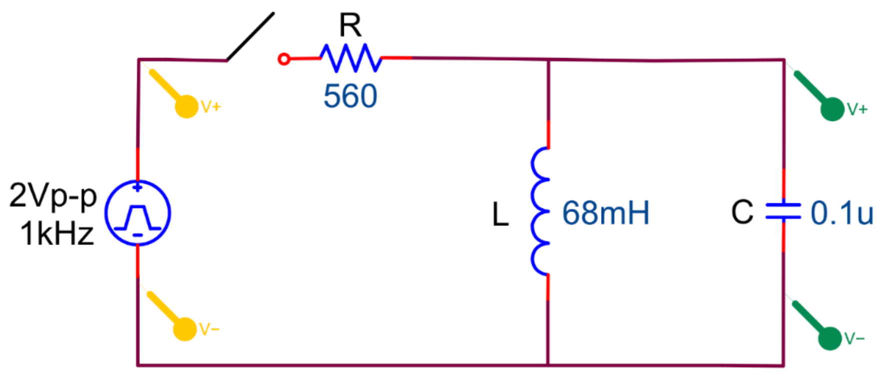


Figure 14.7

PSPICE

1. Using PSPICE plot v(t) versus t for each of the four circuits, compare them to your experimental results. Recall that the circuit you are analyzing is equivalent to that shown in Figure 14.2 (zero initial conditions). You do not need to use the pulse input for PSPICE.

EXPERIMENT 15 : PASSIVE FILTERS

OBJECTIVE

Learn how to measure and calculate bandwidth and frequency.

EQUIPMENT NEEDED

- Oscilloscope
- Function Generator
- Resistors, Capacitors, Inductors

THEORY

Frequency response and the system transfer function H(jω) were discussed in the theory of the previous experiment. The definitions of resonant frequency ω_r, bandwidth BW and quality factor Q are presented here for a second order RLC network. Consider the amplitude response of a network (Figure 14.1)

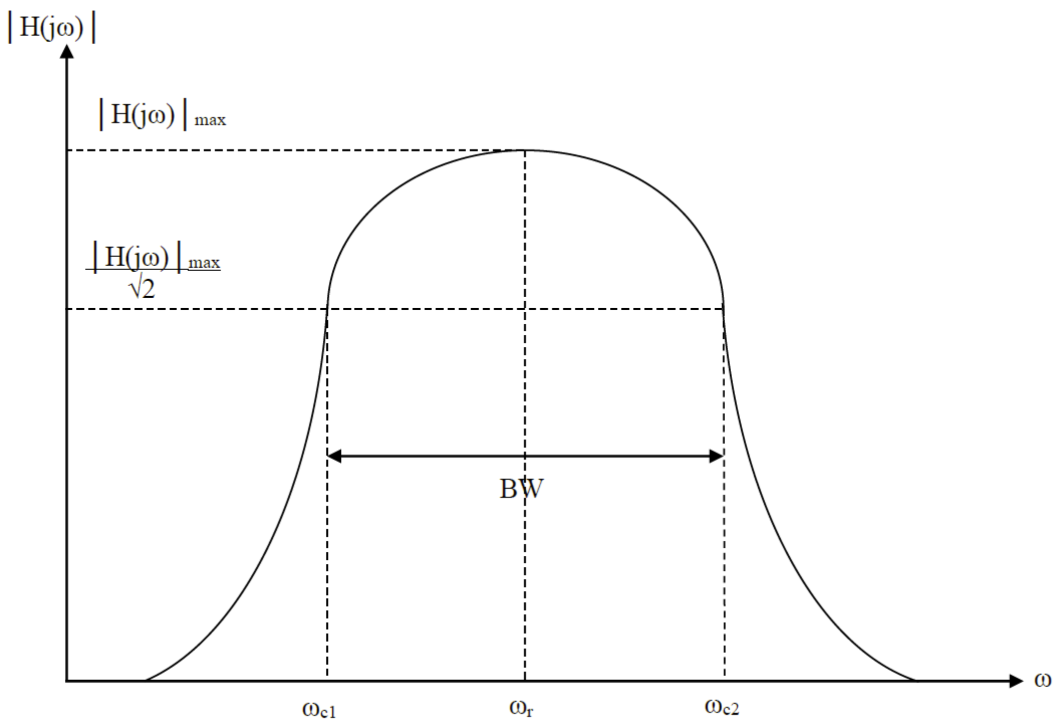


Figure 15.1

Resonant Frequency:

The frequency at which the response amplitude is maximum is known as the resonant frequency. This frequency is denoted by ω_r.

Cutoff Frequency:

The frequency (frequencies) at which the response amplitude is $\frac{1}{\sqrt{2}}$ of maximum is (are) known as the cutoff frequency (frequencies). Figure 14.1 shows two such frequencies denoted by ω_{c1} and ω_{c2}. Sometimes these frequencies are referred to as corner frequencies or half-power frequencies.

Bandwidth:

The width of the frequency between the cutoff frequencies ωc1 and ωc2 is known as the bandwidth. That is,

$$BW = \omega_{c2} - \omega_{c1}$$

Quality Factor:

The quality factor Q is a measure of the sharpness of peak in a resonant circuit, and is defined as:

$$Q = \frac{\omega_r}{BW}$$

Hence smaller Q means larger bandwidth and larger Q means smaller bandwidth. In general the amplitude response |H (jω)| is not symmetrical about the resonant frequency. Normally for a large value of Q (>5) the amplitude response |H (jω)| is symmetrical, in which case we can write

$$\omega_{c1} = \omega_r - \frac{BW}{2}$$

And

$$\omega_{c2} = \omega_r - \frac{BW}{2}$$

Example

Consider the parallel RLC network shown in Figure 15.2

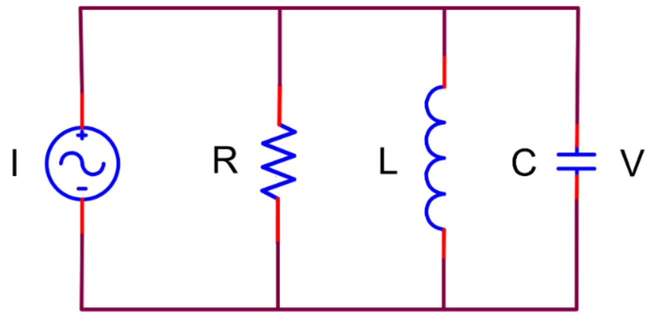


Figure 15.2

Let us define the transfer function H(jω) as $\frac{V(j\omega)}{I(j\omega)}$,which is the total impedance.

Hence:

$$H(j\omega) = \frac{1}{\frac{1}{R} + j\left(\omega C - \frac{1}{\omega L}\right)}$$

The amplitude response $|H(j\omega)|$ is maximum if:

$$\omega C - \frac{1}{\omega L} = 0$$

The cutoff frequencies occur when

$$\left|H\left(j\omega_{c1,c2}\right)\right| = \frac{1}{\sqrt{2}}|H(j\omega)|_{MAX}$$

Therefore:

$$\frac{R}{\sqrt{2}} = \frac{1}{\sqrt{\left(\frac{1}{R}\right)^2 + \left(\omega C - \frac{1}{\omega L}\right)^2}}$$

Solving (14.8) results in:

$$\omega_{c1} = -\frac{1}{2RC} + \sqrt{\frac{1}{LC} + \frac{1}{4R^2C^2}}$$

And

$$\omega_{c2} = \frac{1}{2RC} + \sqrt{\frac{1}{LC} + \frac{1}{4R^2C^2}}$$

The bandwidth BW is evaluated by:

$$\omega_{c2} - \omega_{c1} = \frac{1}{RC}$$

And the quality factor Q is

$$Q = \omega_r RC$$

The above equations were obtained for a parallel RLC circuit. Similar procedure should be used to obtain ω_r, BW and Q for any other second order RLC network with sinusoidal input.

1. Conduct the following:

   a. Derive an expression for $H(j\omega) = \frac{v2(j\omega)}{v1(j\omega)}$ for the circuit of Figure 15.3.

   b. What is the resonant frequency of this circuit?

   c. What is the bandwidth of this circuit?

   d. What is the Q of this circuit?

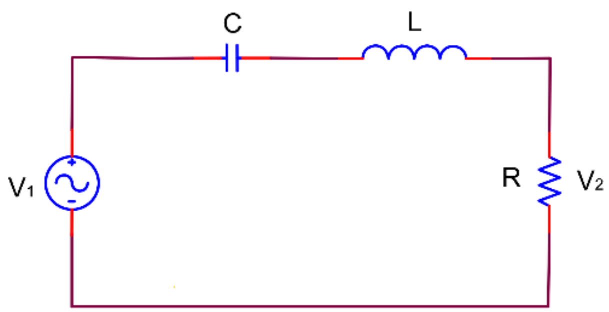


Figure 15.3

2. Conduct the following:

   a. Derive an expression for $H(j\omega) = \frac{v2(j\omega)}{v1(j\omega)}$ for the circuit of Figure 15.4.

   b. What is the resonant frequency of this circuit?

   c. What is the bandwidth of this circuit?

   d. What is the Q of this circuit?

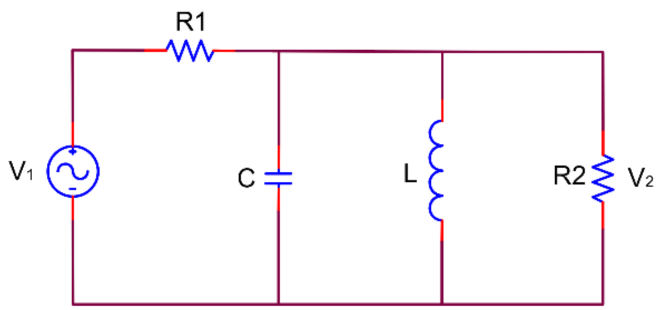


Figure 15.4

1. Conduct the following:

   a. Construct the circuit of Figure 15.5. Vary the frequency from 10Hz to 100kHz and use the oscilloscope to measure the magnitude of V2. Make sure that the input voltage remains constant throughout the measurements. Plot the magnitude of V2 vs. frequency on semi-log graph (frequency on log scale).

   b. Use the above plot to calculate the resonant frequency of the circuit. How does it compare to the theoretical value obtained in the pre-lab?

   c. Use your graph to determine the experimental bandwidth of the circuit. How does this value compare to the value in the pre-lab?

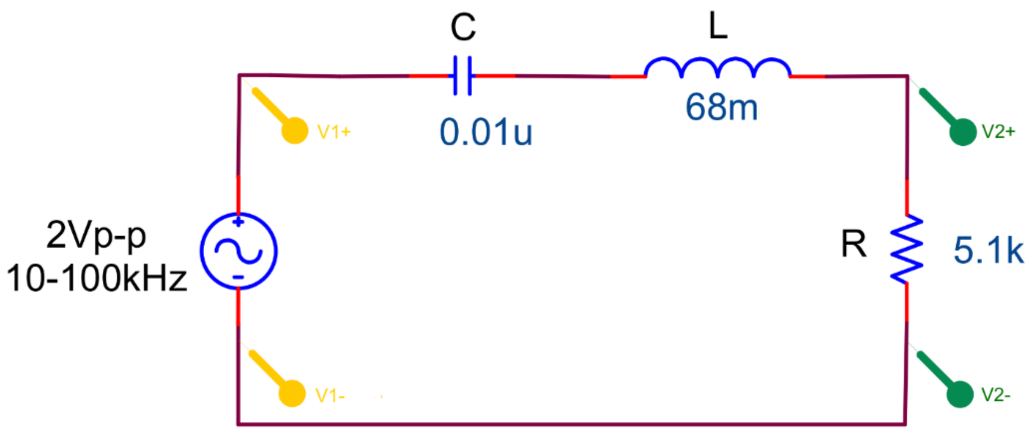


Figure 15.5

2. Conduct the following:

   a. Construct the circuit of Figure 15.6. Vary the frequency from 10Hz to 100kHz and use the oscilloscope to measure the magnitude of V2. Make sure that the input voltage remains constant throughout the measurements. Plot the magnitude of V2 vs. frequency on semi-log graph (frequency on log scale).

   b. Use the above plot to calculate the resonant frequency of the circuit. How does it compare to the theoretical value obtained in the pre-lab?

   c. Use your graph to determine the experimental bandwidth of the circuit. How does this value compare to the value in the pre-lab?

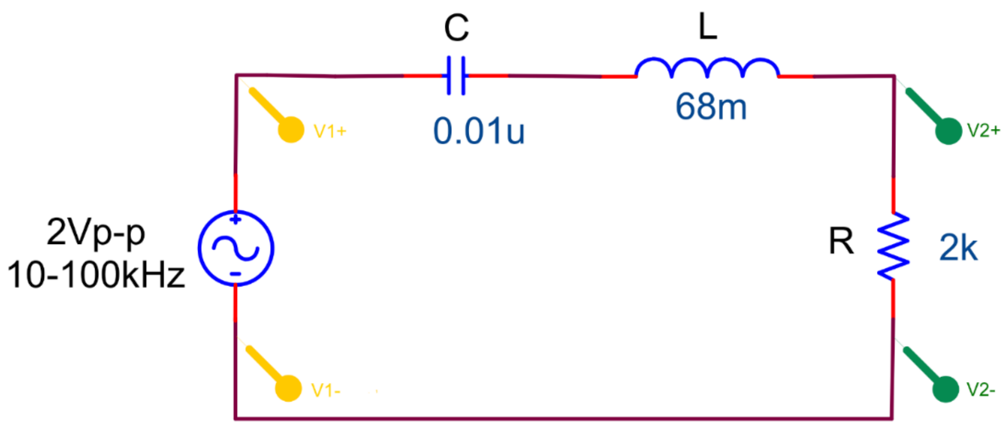


Figure 15.6

3. Conduct the following:

   a. Construct the circuit of Figure 15.7. Vary the frequency from 10Hz to 100kHz and use the oscilloscope to measure the magnitude of V2. Make sure that the input voltage remains constant throughout the measurements. Plot the magnitude of V2 vs. frequency on semi-log graph (frequency on log scale).

   b. Use the above plot to calculate the resonant frequency of the circuit. How does it compare to the theoretical value obtained in the pre-lab?

   c. Use your graph to determine the experimental bandwidth of the circuit. How does this value compare to the value in the pre-lab?

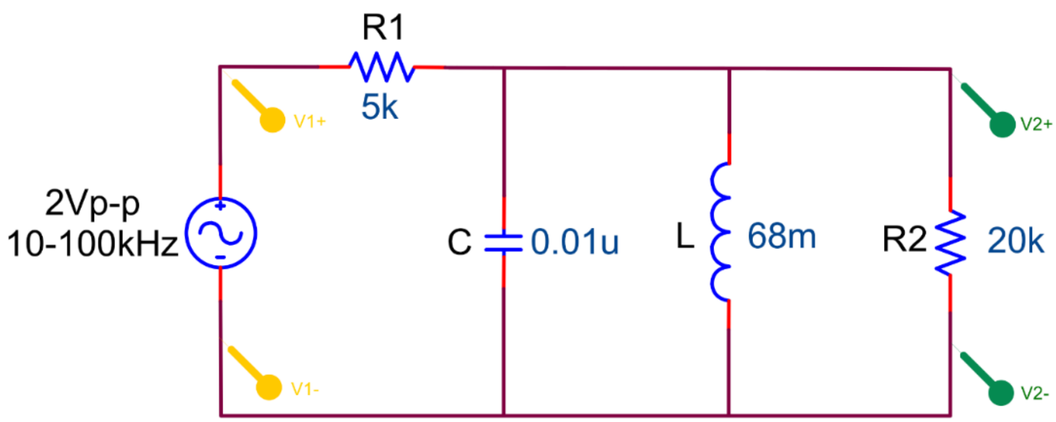


Figure 15.7

4. Conduct the following:

   a. Construct the circuit of Figure 15.8. Vary the frequency from 10Hz to 100kHz and use the oscilloscope to measure the magnitude of V2. Make sure that the input voltage remains constant throughout the measurements. Plot the magnitude of V2 vs. frequency on semi-log graph (frequency on log scale).

   b. Use the above plot to calculate the resonant frequency of the circuit. How does it compare to the theoretical value obtained in the pre-lab?

   c. Use your graph to determine the experimental bandwidth of the circuit. How does this value compare to the value in the pre-lab?

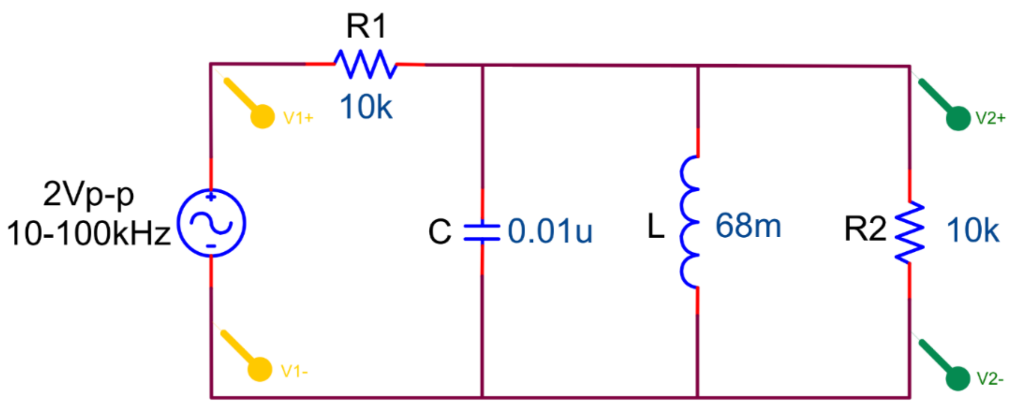


Figure 15.8

PSPICE

1. Use the AC analysis of SPICE to plot the magnitude of V2 versus frequency for Figure 15.5. How do your results compare to those obtained in part a?
2. Use the AC analysis of SPICE to plot the magnitude of V2 versus frequency for Figure 15.6. How do your results compare to those obtained in part a?
3. Use the AC analysis of SPICE to plot the magnitude of V2 versus frequency for Figure 15.7. How do your results compare to those obtained in part a?
4. Use the AC analysis of SPICE to plot the magnitude of V2 versus frequency for Figure 15.8. How do your results compare to those obtained in part a?

EXPERIMENT 16 : DIODES

OBJECTIVE

Understand the basic characteristics of the common diode.

EQUIPMENT NEEDED

- Power supply
- Function Generator
- Digital Multimeter
- Oscilloscope
- Resistor
- Diode (1N4002)

THEORY

The diode acts like a switch and has two positions. If it is forward-biased, the switch is closed. If it is reversed-biased the switch is open. The ideal diode model states the voltage across the diode is zero. The practical model of a diode incorporates a forward voltage V_F of 0.7V.

PRELIMINARY CALCULATIONS

1. For Figure 16.1 calculate V_{R1}, V_{D1}, and I_F. Enter the data into table 1 in the procedure section.

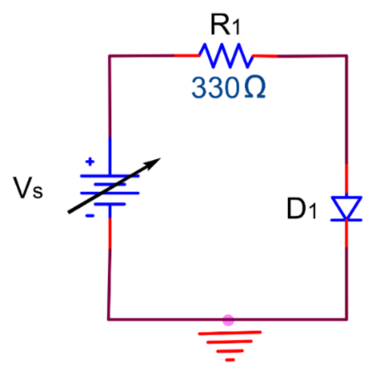


Figure 16.1

2. For Figure 16.2 calculate V_{R1}, V_{D1}, and I_F. Enter the data into table 2 in the procedure section.

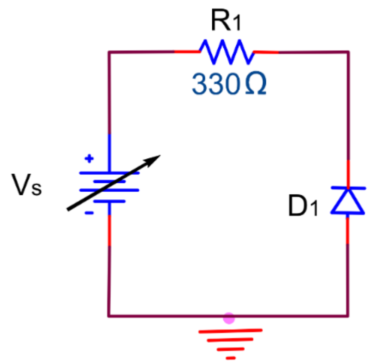


Figure 16.2

3. For Figure 16.3 calculate V_{R1}, V_{D1}, and I_f. Enter the data into table 3 in the procedure section.

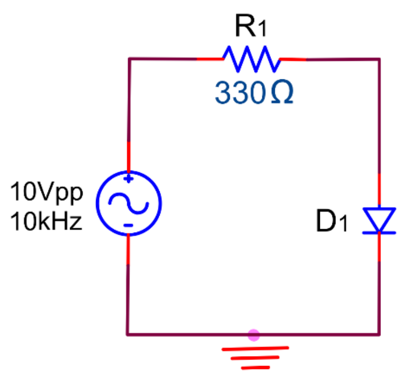


Figure 16.3

PROCEDURE

1. Measure V_{R1} and V_{D1} with the DMM, calculate the current based on the voltage obtained.

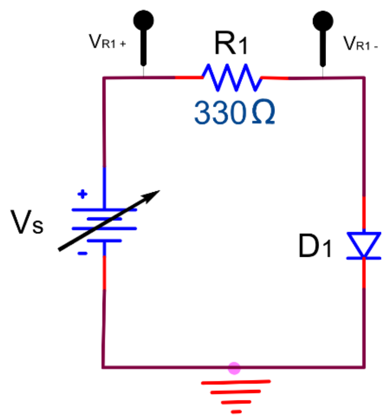


Figure 16.4

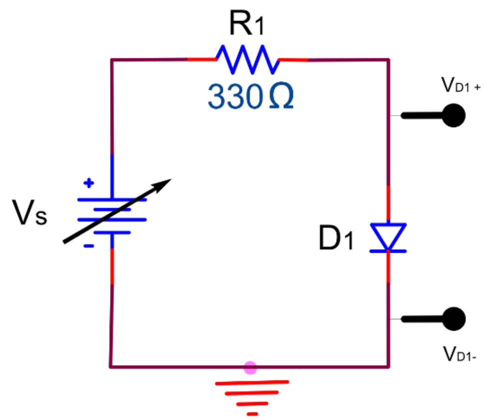


Figure 16.5

Table 1	Calculated			Measured		
V_S	V_{R1}	V_{D1}	I_F	V_{R1}	V_{D1}	I_F
0.50						
0.60						
0.70						
0.75						
0.80						
0.85						
0.90						
1.00						
1.25						
1.50						

2. Measure V_{R1} and V_{D1} with the DMM, calculate the current based on the voltage obtained.

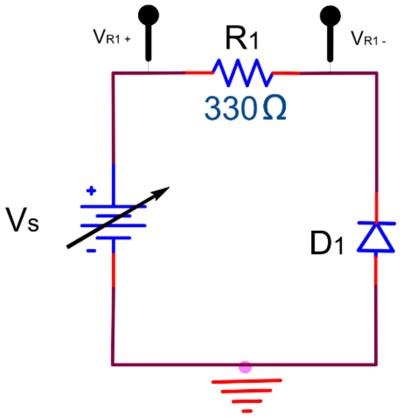


Figure 16.6

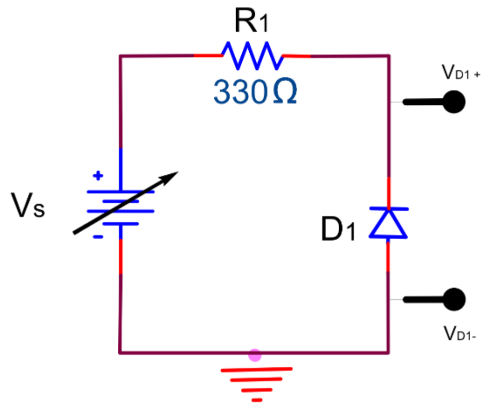


Figure 16.7

Table 2	Calculated			Measured		
V_S	V_{R1}	V_{D1}	I_F	V_{R1}	V_{D1}	I_F
5.0						
10.0						
15.0						

3. Measure V_{R1} and V_{D1} with the oscilloscope, calculate the current based on the voltage obtained.

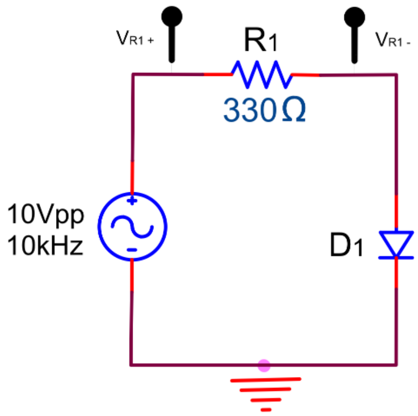


Figure 16.8

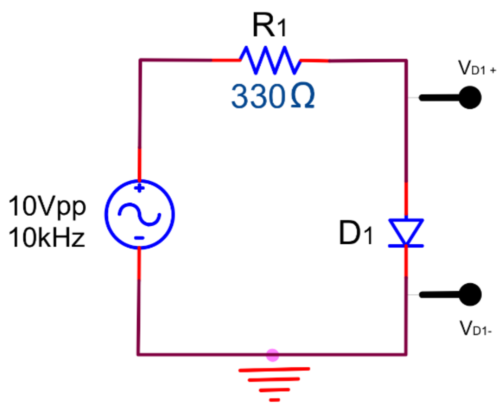


Figure 16.9

Table 3	Calculated			Measured		
V_S	V_{r1}	V_{d1}	I_f	V_{r1}	V_{d1}	I_f
5.0						
10.0						
15.0						

PSPICE

1. Simulate Figure 16.8 and Figure 16.9 and compare them to the results obtained in the laboratory.

EXPERIMENT 17 : TANSISTOR DC BIASING

OBJECTIVE

Obtain the rudimentary skills of biasing a transistor for future AC analysis.

EQUIPMENT NEEDED

- Power supply
- Digital Multimeter
- Resistor
- 2N2222A

THEORY

DC biasing is extremely important to the transistor's stability during operation and the first step towards designing amplifiers that are built with BJT's or MOSFET's. A properly biased transistor can maintain its stability as external parameters change relative to a poorly biased transistor. The general method for DC biasing is to place the Quiescent point halfway between zero and V_{CC} allowing the future AC signal plenty of leeway as to not clip or enter saturation.

PRELIMINARY CALCULATIONS

1. Calculate the resistors needed to obtain a $V_C = 11.30V$, $V_B = 3.46V$, and $V_E = 2.82V$.

2. Calculate the remaining voltages and currents listed in Table 1.

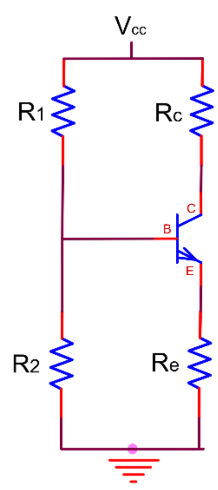


Figure 17.1

PROCEDURE

1. Construct the circuit below with the values calculated in the prelab and measure each node voltage to the BJT.

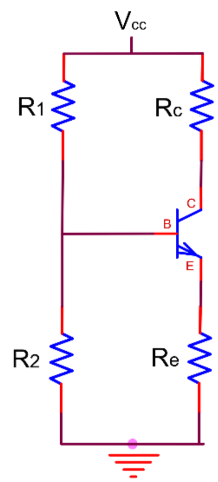


Figure 17.2

Table 1	V_C	V_B	V_E	V_{CE}	V_{BE}	V_{RC}	I_C	I_B	I_E
Theoretical									
Measured									
Percent Error									

PSPICE

1. Simulate the circuit that was designed in the prelab and compare these results to the ones taken in the laboratory.

   Note: The 2N2222A used in the laboratory can be found as 2N2222A/ZTX