

**TABLE OF CONTENTS**

<b>Exp No</b>	<b>Title</b>	<b>Page No</b>
<b>1</b>	<b>Electrical Circuits Simulation using Multisim Electronics Workbench</b>	<b>3</b>
<b>2</b>	<b>Resistors &amp; OHM'S Law</b>	<b>12</b>
<b>3</b>	<b>Kirchhoff's Law</b>	<b>18</b>
<b>4</b>	<b>Current and Voltage Divider Rules</b>	<b>21</b>
<b>5</b>	<b>Superposition Theorem</b>	<b>26</b>
<b>6</b>	<b>THEVENIN / NORTON Theorems and Maximum Power Transfer</b>	<b>29</b>
<b>7</b>	<b>The Oscilloscope and Function Generator</b>	<b>34</b>
<b>8</b>	<b>Frequency Domain Analysis</b>	<b>38</b>
<b>9</b>	<b>Maximum Power Transfer</b>	<b>43</b>
<b>10</b>	<b>Average and RMS Values</b>	<b>47</b>
	<b>Appendix I</b>	<b>51</b>
	<b>Appendix II</b>	<b>52</b>
	<b>LABORATORY REGULATIONS AND SAFETY RULES</b>	<b>53</b>

## PREFACE

The **EE 204: Fundamentals of Electric Circuits** Lab is intended to teach the basics of Electrical Engineering to undergraduates of other engineering departments. The main aim is to provide hands-on experience to the students so that they are able to put theoretical concepts to practice.

The manual starts off with the basic laws such as Ohm's Law and Kirchhoff's Current and Voltage Laws. The two experiments augment students' understanding of the relations of voltage and current how they are implemented in practical life.

Computer simulation is also stressed upon as it is a key analysis tool of engineering design. *Multisim* is used for simulation of electric circuits and is a standard tool at numerous universities and industries of the world. The simulated parameters are then verified through actual experiment. Use of oscilloscopes is also stressed upon as analysis tool. The important theorems of Thevenin and Norton are also provided along with the frequency domain analysis of circuits. They greatly simplify the complex electrical networks for analysis purposes.

At the end, the students should be able to grasp the concepts thoroughly the electric circuits and able to apply them further in their field of study.

## Experiment No. 1

### Electrical Circuits Simulation using Multisim Electronics Workbench: An Introduction

Simulation is a mathematical way of representing the actual behavior of a circuit. With simulation, you can determine a circuit's performance without physically constructing the circuit or using actual test instruments. *Multisim* is a complete system design tool that offers a very large component database, schematic entry, full analog/digital SPICE simulation, etc. It also offers a single easy to use graphical interface for all design needs.

#### Introduction

Go to **Start**→**Programs**→**Multisim** and click on **Multisim**. This will open the main window as shown in Fig. 1. In Fig. 1 important toolbars and menu are labeled. In addition to toolbars shown in Fig. 1, there may be other toolbars appearing on your screen concentrate on the labeled items in Fig. 1 at this time.

You can always open and close a toolbar from Main Menu. For example if you want to open or close (select/unselect) the Design Toolbar, select **View**→**Toolbars**→**Design**. If any toolbar is not appearing on your screen then use the above procedure to bring the toolbar.

Most of the analysis can be performed turning on-off the simulate switch. If the Simulation Switch shown in Fig. 1 is not appearing on your screen then select **View**→**Show Simulate Switch** in the Main Menu. This will open the Simulation Switch.

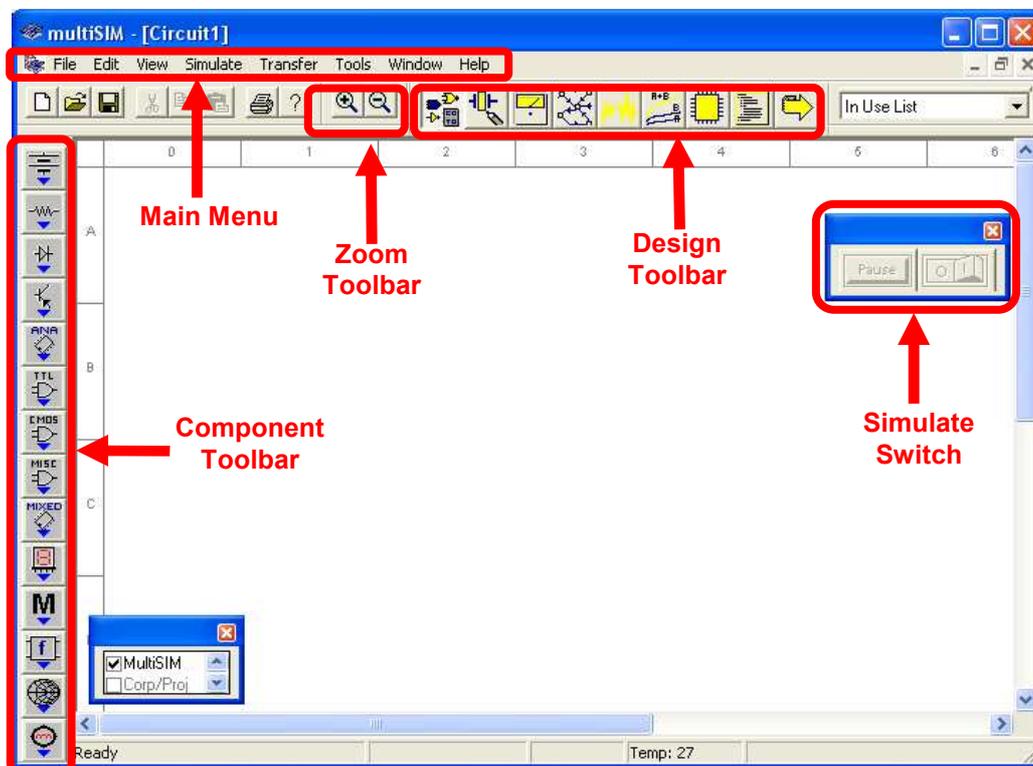


Figure. 1: Main Window of *Multisim* Simulation Software

We will now try to learn about *Multisim* simulation techniques by solving a simple example.

**Example**

Build the circuit shown in Fig. 2 using *Multisim Electronics Workbench*.

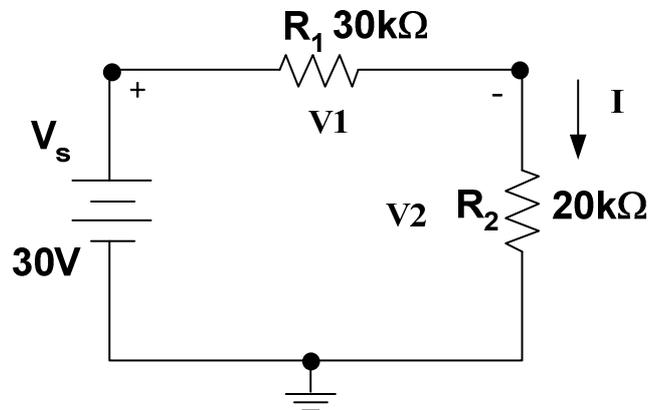


Figure. 2: Circuit for *Multisim* Simulation

### STEP A: Placing the Components

#### 1. Place a Battery (DC Source)

##### a. Bring a DC source in the *Multisim* workspace:

Open the *Multisim* program if it is not open. In the Component Toolbar, select **Sources** icon (refer to Fig. 1 to find the Component Toolbar). This will open another window with several types of DC sources and other components as shown below in Fig. 3. Click on “DC Voltage Source” in this new window.

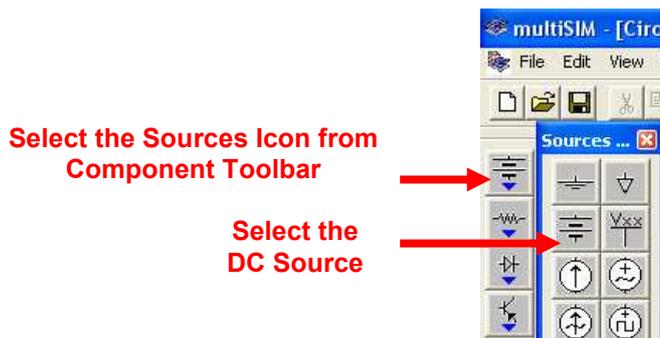


Figure. 3: Selection of DC Voltage Source in *Multisim*

Now bring your cursor in the workspace area and notice the change in the shape of cursor to



Click at any point in the workspace. This will put the voltage source as



**b. Change the value and name of voltage source:**

Double click on the voltage source that you just placed in the workspace, a new window with the name **Battery** will appear, as shown in Fig. 4. Select **Value** in the Battery menu, if it is not already selected. Change the value from 12 to 30. Keep the unit as Volts in this menu. Now select **Label** in this menu and change the **Reference ID** to Vs. Click on **OK**.

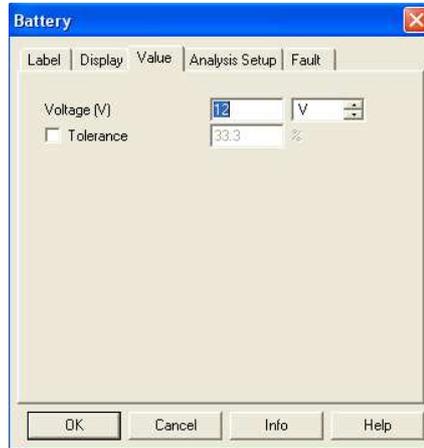


Figure. 4: Battery Window for setup of DC voltage source

**2. Place a Resistor:**

**a. Bring a resistor in the Multisim workspace:**

In the Component toolbar, select **Basic** icon as shown. This will open another window with several basic components as shown below in Fig. 5.

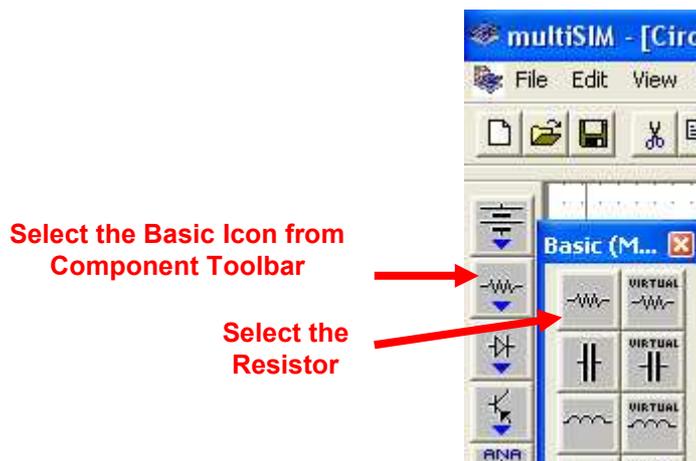
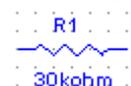


Figure. 5: Selection of Resistor in *Multisim*

Click on “Resistor”, this will open the **Browser-Basic** window, as shown in Fig. 6. Scroll through the **Component List**, select 30 kOhm, and click OK. The cursor shape will change again. Click in the workspace and this will put the resistor as,



**Tip** To make your scroll through the Browser's **Component List** faster, simply type the first few characters of the component's name. For example, type 30k to move directly to the area of 30 kOhm list.

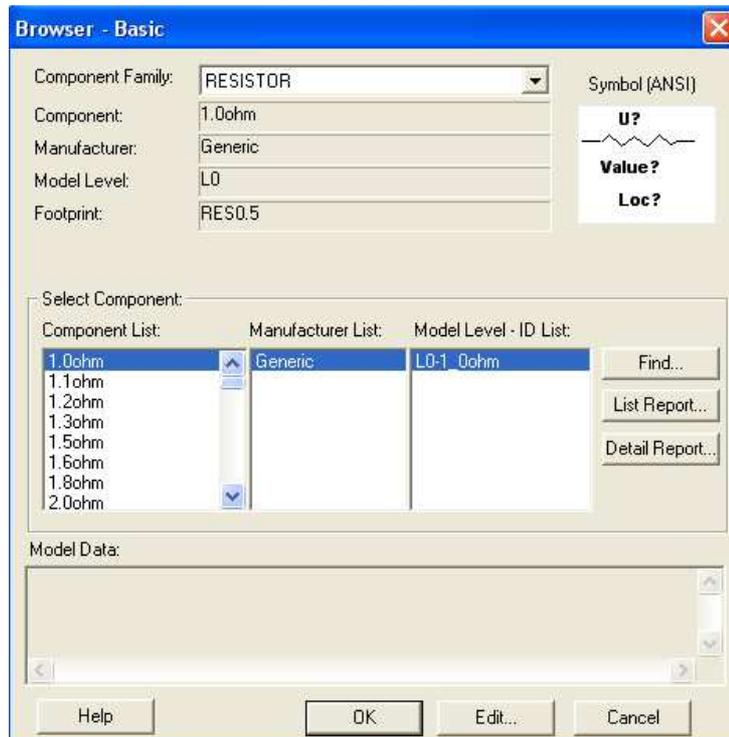


Figure. 6: Setup window for Resistor values

**b. Change the name of resistor:**

Double click on the resistor, a new window with the name **Resistor** will open as shown in Fig. 7. Select Label from the menu of this window. Change the **Reference ID** to R<sub>1</sub> (if it is not) and press **OK**. This will change the name of the resistor to R<sub>1</sub>.



Figure. 7: Battery Window for Label of Resistor

**d. Add other resistor  $R_2$**

Place resistor  $R_2$  of value 20 k $\Omega$  in the workspace through the same procedure.

**e. Rotate the resistor:**

Select resistor ' $R_2$ ' and press **Ctrl-R** to rotate the resistor or select **Edit→90 Clockwise** from the Main Menu. This will make the resistor vertical. Labels and values of all the components can be dragged individually. Drag the label ' $R_2$ ' and value '20 kOhm' individually to put them at a proper place.

**3. Place Ground:**

In the Component Toolbar, select **Sources** icon. Now click on **Ground** icon in the new window as shown in Fig. 8. Click in the workspace to put the Ground symbol as

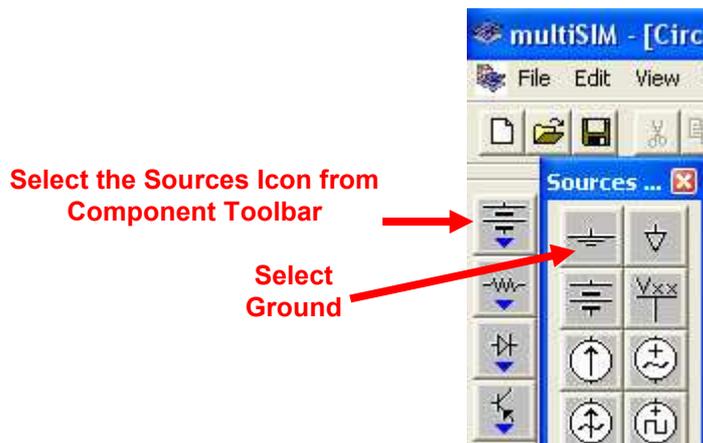


Figure. 8: Selection of Ground in *Multisim*

**STEP B: Connecting the Components**

**1. Arrange the components properly:**

Arrange the components according to the circuit given in Fig. 2. You can select and drag the component to any place in the workspace. Select the components and drag them one by one to proper places as shown in Fig. 9.

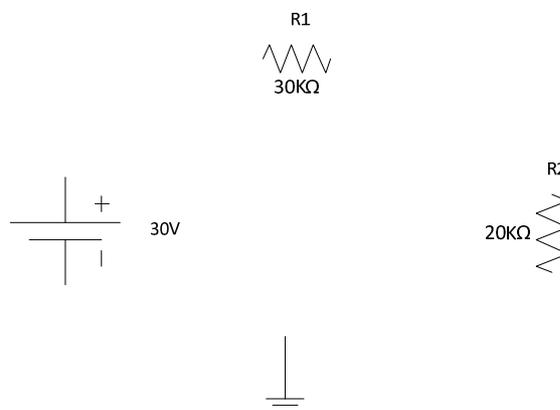


Figure. 9: Arranging the Components in proper order

## 2. Show Grid in the workspace:

You may show grid for ease of drawing the connections. Select **View**→**Grid Visible** in the Main Menu if it is not visible.

## 3. Connect DC Voltage Source “Vs” to “R<sub>1</sub>”:

Bring the cursor close to upper pin of “Vs”; cursor shape will change to a plus sign. Click and move a little upward. A wire appears, attached to the cursor. Click again at a small distance above the “Vs” source. Notice that the line will change direction. Control the flow of the wire by clicking on points as you drag. Each click fixes the wire to that point as shown in Fig. 10. In this way, when the cursor reaches the pin of R<sub>1</sub> click again, this will connect “Vs” to “R<sub>1</sub>” in a nice manner. Notice that a node number is automatically given.

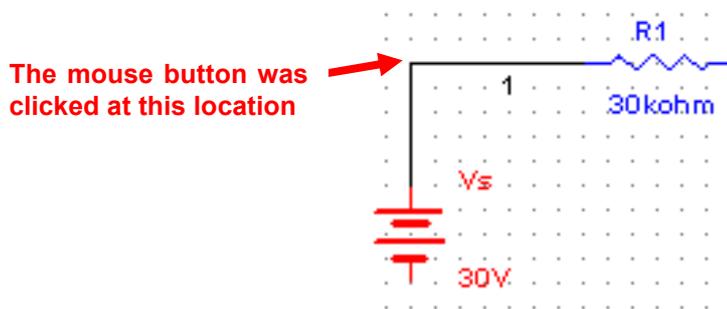


Figure. 10: Manual connection of components

## 4. Connect “R<sub>1</sub>” to “R<sub>2</sub>”:

In the same way connect R<sub>1</sub> to R<sub>2</sub> through the same procedure.

## 5. Making use of Junction to connect Ground:

In the similar manner connect ground with V<sub>s</sub> and R<sub>2</sub>. Notice that a small black circle appears just above the ground, this is called junction. When 2 or more components are connected at one point, a junction is created. A junction can also be placed manually by pressing Ctrl+J or selecting **Edit**→**Place Junction**. This can be used to control the connection points manually. Also notice the ground node is automatically given node number 0. Do not alter it.

This completes the connection and the complete circuit is shown in Fig. 11.

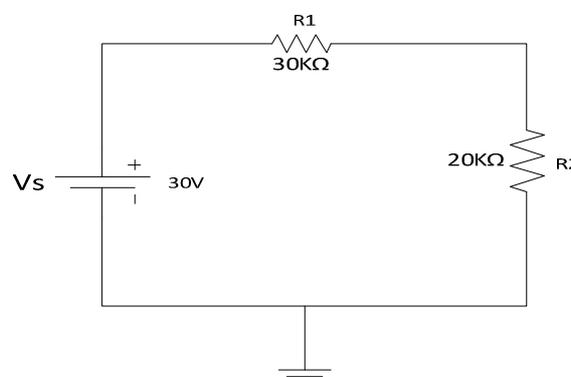


Figure. 11: Complete Circuit in *Multisim*

## 6. Wire paths can be modified using drag points. Click on a wire. A number of drag points will appear on the wire as shown in Fig. 12. Click any of these and drag to modify the

shape. You can also add or remove drag points to give you even more control over the wire shape. To add or remove drag points, press CTRL and click on the location where you want the drag point added or removed.

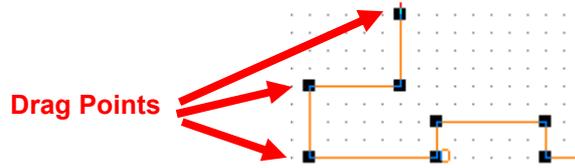


Figure. 12: Drag points for connecting wire

### STEP C: Placing Multimeter or Voltmeter in parallel to measure voltage

#### 1. To connect a Multimeter:

- a. Select **View**→**Toolbar**→**Instruments**. The Instruments toolbar will open as shown in Fig. 13.

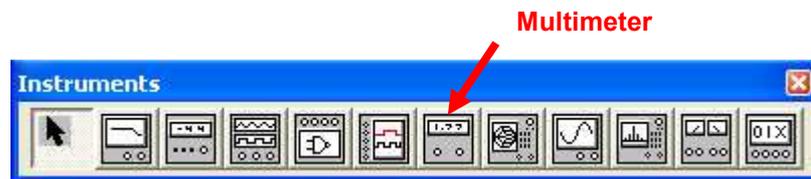


Figure. 13: Instruments Toolbar

- b. Click on **Multimeter** icon. Now click in the workspace to place the **Multimeter**. Drag it and place it near resistor  $R_1$  as shown in Fig. 14. Make a connection from '+' terminal of **Multimeter** to the left pin of  $R_1$  and from '-' terminal to right pin of  $R_1$ . Note that reversal of + and - terminals will give opposite readings.

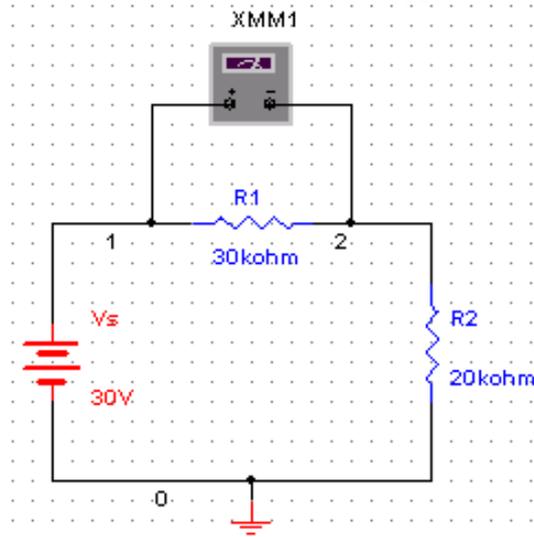


Figure. 14: **Multimeter** connection for voltage measurement

- c. **Set the Multimeter to measure DC voltage:**

Double click on **Multimeter** to open the properties window shown in Fig. 15. Select 'V' to measure voltage. Select the DC wave shape. (Notice that the meter can also

measure current 'A' and resistance 'Ω'. It can measure AC as well as DC values. Leave the window open for viewing the measurements.



Figure. 15: **Multimeter** properties window

#### STEP D: Placing a Multimeter or Ammeter in series to measure current

1. Place a second **Multimeter** in the workspace as we did in Step C. Remove the connection between  $R_1$  and  $R_2$ . Connect the '+' terminal of the **Multimeter** towards  $R_1$  and the '-' terminal towards  $R_2$  as shown in Fig. 16.
2. **Set the Multimeter to measure current:**  
Double click on this **multimeter** and select 'A' in the **multimeter** properties window. Set the wave shape to DC. If current flows from 3 to zero, the meter will read positive.

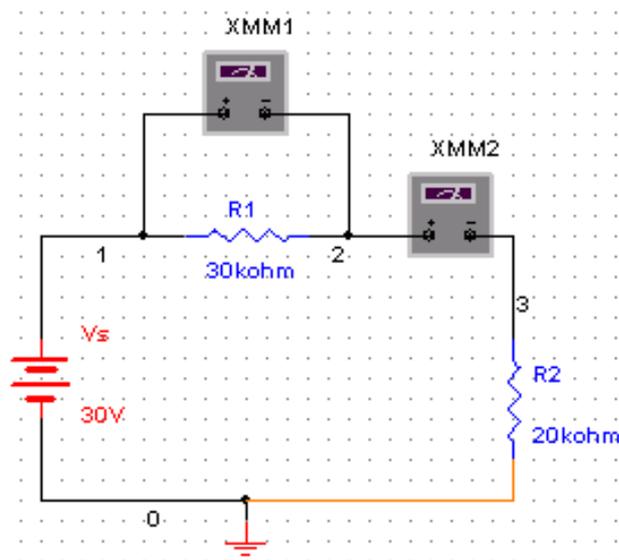


Figure. 16: **Multimeter** connection for current measurement

<b>NAME</b> _____	<b>SEC #</b> _____
<b>ID #</b> _____	<b>SERIAL#</b> _____

**Example:**

V1	V2	IT

**Exercise:**

IT	I1	I2	V1	V2

**Conclusion:**

## Experiment No. 2

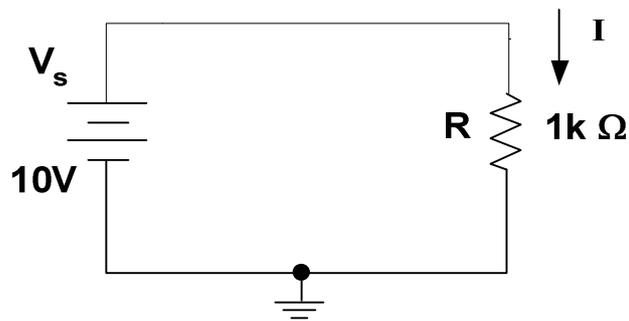
### Resistors & OHM'S Law

#### Objective:

- 1- To determine the value of a selection of resistors using three different methods:
  - a. Using the color codes (to give the nominal value)
  - b. Using the digital Ohmmeter.
  - c. Using Ohm's Law.
- 2- To determine qualitatively the effect of increased power dissipation on carbon resistor.

#### Pre-Lab Assignment:

For the circuit shown in the Figure below, calculate V, and I.



#### Apparatus:

DC Power Supply  
 Digital Multimeter  
 Resistor: 100Ω , 470Ω , 1 KΩ.

#### Theory:

- 1- The basic relationship between voltage, current, and resistance is determined by Ohm's law:

$$V = I R \quad (1)$$

Where: V = Voltage across the resistor (in Volts)

I = Current through the resistor (in Amperes)

R = Resistance of the resistor (in Ohms)

2- Given any resistor, its resistance can be found by one of the four methods:

- a- Using the color codes. This value is called the nominal value, and it is only approximate.
- b- Direct measurement using an Ohmmeter.
- c- Measuring the voltage across the resistor and the current through the resistor, then applying Ohm's law.

3- The product of the voltage across a resistor and the current through the resistor gives power P (Watt), namely:

$$P = V I = I^2 R = V^2 / R \quad (2)$$

Where equation (1) has been used to express P in terms of  $I^2$  and in terms of  $V^2$ .

The power absorbed by the resistor appears in the form of heat. The physical size of the resistor determines the amount of power that it can safely dissipate. This amount is referred to as the power rating. The dissipation of power that exceeds the power rating can damage the resistor physically.

When the resistor gets heated due to excessive power dissipation, its resistance changes. It will either increase or decrease depending on the temperature coefficient. A carbon resistance is expected to increase as the temperature increases.

Resistors that are operated above the power rating will deviate from the straight line relationship between V and I. The resistor in this case is operating in the non-linear region. In such a case, the resistance is no longer equal the slope of the V versus I graphs. It may however, be calculated using the ratio V/I.

### Procedure:

You will be supplied with sets of 5 resistors.

1. Find the nominal value and the tolerance of each resistance using the Resistor colour code chart shown below. Note the relation between the power rating and the physical size of the resistance.
2. Using the digital multimeter as an ohmmeter, measure and record the resistance of each resistor.
3. Connect the circuit as shown in Figure. 2 for  $R = 100 \Omega$  and perform the following :
  - a- Set the source voltage  $V_s$  to 10 V.
  - b- Measure V and I.
  - c- Repeat steps 3a and 3b for remaining resistors.
  - d- Record your results in Table 1.

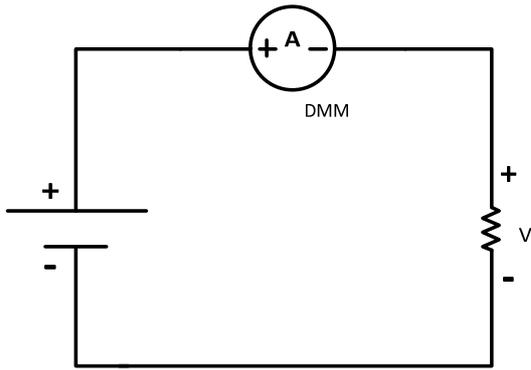


Fig. 2a : Series Connection

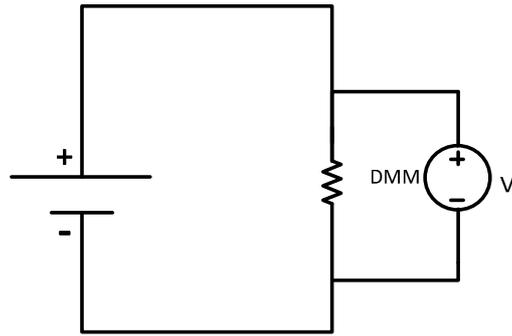
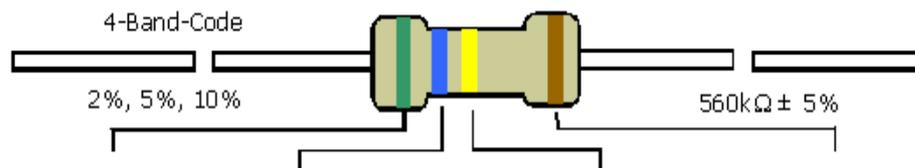


Fig. 2b : Parallel Connection

- Using a 1 W resistor ( $R = 470\Omega$ ), vary the input voltage from 6 to 20 volts. Measure  $V$  and  $I$  and calculate the resistance  $R$  (by calculating the ratio  $V/I$ ) and the power dissipated in the resistance (by calculating the product  $VI$ ). As the measurement proceeds, touch the resistor from time to time to observe the temperature rise. Record the results in Table 2.



COLOR	1st BAND	2nd BAND	3rd BAND	MULTIPLIER	TOLERANCE
Black	0	0	0	1Ω	
Brown	1	1	1	10Ω	± 1% (F)
Red	2	2	2	100Ω	± 2% (G)
Orange	3	3	3	1KΩ	
Yellow	4	4	4	10KΩ	
Green	5	5	5	100KΩ	±0.5% (D)
Blue	6	6	6	1MΩ	±0.25% (C)
Violet	7	7	7	10MΩ	±0.10% (B)
Grey	8	8	8		±0.05%
White	9	9	9		
Gold				0.1	± 5% (J)
Silver				0.01	± 10% (K)



**Questions:**

1. Plot P versus R in Fig. 3.

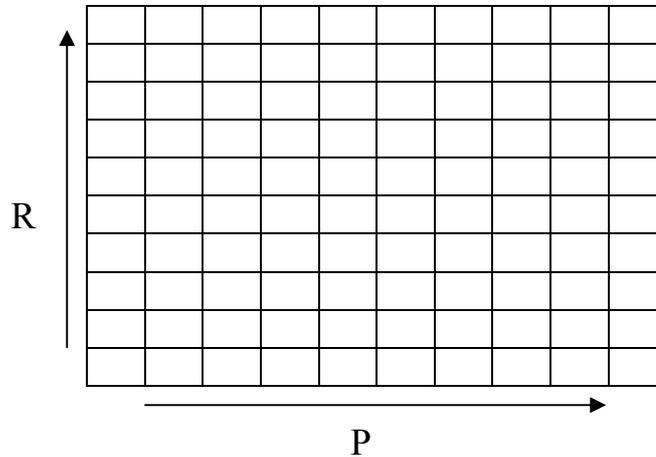


Figure. 3: R versus P

Comment on the linearity of R as P increases.

2. Does the resistor in step 4 operate in the linear region or non-linear region? Explain by considering the power rating of the resistor.
3. An electric heater takes 1.48 kW from a voltage source of 220 V. Find the resistance of the heater?
4. If the current in a resistor doubles, what happens to the dissipated power? (Assume the resistor operates in the linear region).
5. A  $4\ \Omega$  resistor is needed to be used in circuit where the voltage across the resistor is 3V .If two  $4\ \Omega$  resistors with 2 W and 3 W power rating are available, which will you use and why?

**Table 1. Resistor values:**

Resistor	R1	R2	R3
Nominal value / Tolerance			
Ohmmeter reading			
V / I			
% Deviation from nominal value			
Actual value within tolerance?			

**Table 2. Power Rating: (R = 470Ω, 1W)**

V (volt)	6	10	15	20
I (mA)				
R (Ohm) = V/I				
P (Watt)				

NAME _____	SEC # _____
ID # _____	SERIAL# _____

**Table. 1: Resistors Values**

Resistor	R1	R2	R3
Nominal value / Tolerance			
Ohmmeter reading			
V / I			
% Deviation from nominal value			
Actual value within tolerance?			

**Table .2: Power Rating**

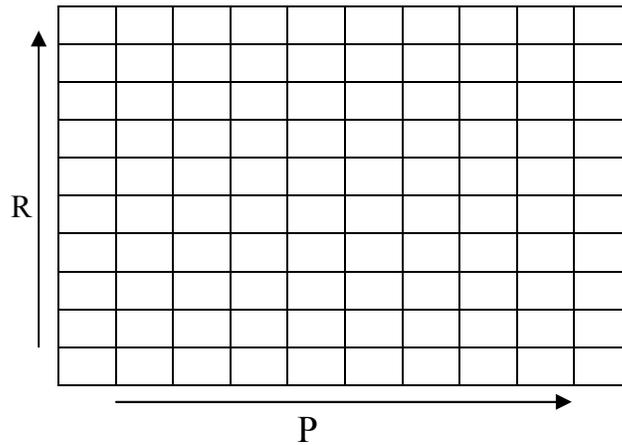


Figure. 3: R versus P

**Answer to Questions:**

- 1.
- 2.
- 3.
- 4.

### Experiment No. 3

#### Kirchhoff's Law

**Objective:**

To verify **Kirchhoff's Voltage** and **Current Law** experimentally.

**Pre-Lab Assignment:**

For the circuit shown in Figure 1, calculate:

- 1-  $V_{AB}$ ,  $V_{BC}$ ,  $V_{AD}$ ,  $V_{DC}$ ,  $V_{BD}$ , and  $V_{AC}$ .
- 2-  $I_1$ ,  $I_2$ ,  $I_3$ ,  $I_4$  and  $I_5$ .

**Apparatus:** DC Power Supply  
Digital Multimeter  
Carbon Resistors: 100 $\Omega$ , 150 $\Omega$ , 220 $\Omega$ , and 330 $\Omega$

**Theory:****•Kirchhoff's Voltage Law ( KVL ):**

The algebraic sum of all voltages around any closed path is equal to zero.

**•Kirchhoff's current Law ( KCL ):**

The algebraic sum of all currents at a junction point is equal to zero.

**Procedure:**

- 1- Check the values of the resistors, used in the circuit of Figure.1, using a multimeter. Record the values in Table 1.
- 2- Connect the circuit as shown, and have it checked by the instructor. Adjust the supply voltage  $V_s$  to 10 V, using a DC voltmeter (DMM).
- 3- Measure the voltages  $V_{AB}$ ,  $V_{BC}$ ,  $V_{AD}$ ,  $V_{DC}$ ,  $V_{BD}$ , and  $V_{AC}$ . Record their values (including the signs) in Table 2
- 4- Measure the currents  $I_1$ ,  $I_2$ ,  $I_3$ ,  $I_4$  and  $I_5$  and record their values (including the signs) in Table. 3

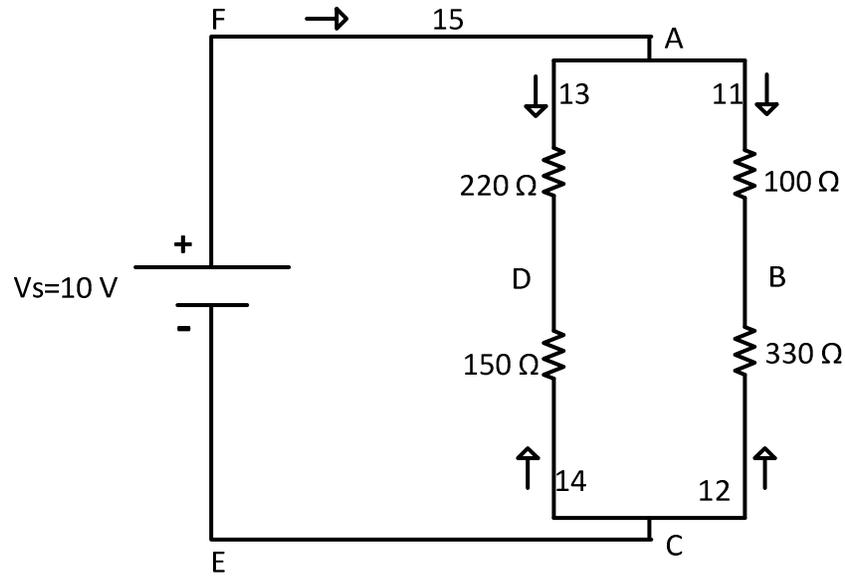


Figure. 1

**Report:**

- 1- Verify KVL by adding the experimental values of voltages around the loops :
  - a) ABCEFA
  - b) CDAC
- 2- Verify KCL by adding the experimental values of current at nodes:
  - a) A
  - b) B
  - c) C

**Questions:**

- 1- Do the experimental and theoretical values of voltages and currents agree? Indicate the percentage of differences.
- 2- Give possible reasons for any discrepancies.

NAME _____	SEC # _____
ID # _____	SERIAL# _____

**Table 1 Resistor Values:**

Resistor	R1	R2	R3	R4
Nominal value (Ohm )	100	150	220	330
Ohmmeter reading				

**Table 2 Voltages:**

Voltage	$V_{AB}$	$V_{BC}$	$V_{AD}$	$V_{DC}$	$V_{BD}$	$V_{AC}$
Theory						
Experiment						
% Error						

**Table 3: Currents:**

Current	$I_1$	$I_2$	$I_3$	$I_4$	$I_5$
Theory					
Experiment					
% Error					

**Report:**

1.

2.

**Answers to Questions:**

1.

2.

## Experiment No. 4

### Current and Voltage Divider Rules

#### Objective:

To experimentally verify the **Current Divider Rule (CDR)** for parallel circuits and the voltage divider rule for series circuits.

#### Pre-Lab Assignment:

For the circuit shown in Figure 1 and Figure 2, calculate:

- 1- The unknown voltages and currents shown.  $V_s=10V$ .
- 2- The equivalent resistance seen by  $V_s$ .

Hint: Read through this experiment.

**Apparatus:** DC Power Supply  
Digital Multimeter  
Carbon Resistors:  $100\Omega$  ,  $150\Omega$  ,  $220\Omega$  and  $330\Omega$  .

#### Objectives:

1. To study the voltage current relationships of series and parallel circuits
2. To verify the voltage current divider and voltage divider rules.

#### Introduction:

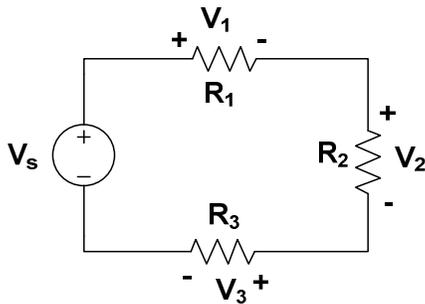


Figure. 1: Series circuit

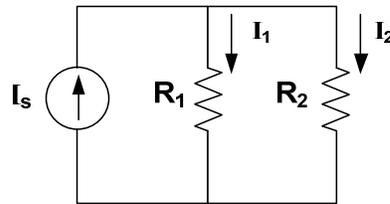


Figure. 2: Parallel circuit

For a series circuit shown in Fig. 1, the voltages across resistors  $R_1$ ,  $R_2$  and  $R_3$  can be written as,

$$\begin{aligned} V_1 &= \frac{R_1}{R_1 + R_2 + R_3} V_s \\ V_2 &= \frac{R_2}{R_1 + R_2 + R_3} V_s \\ V_3 &= \frac{R_3}{R_1 + R_2 + R_3} V_s \end{aligned} \quad (1)$$

This is the **Voltage Divider Rule (VDR)**.

For a parallel circuit given in Fig. 2, the branch currents can be written in terms of the total current as,

$$\begin{aligned} I_1 &= \frac{R_2}{R_1 + R_2} I_s \\ I_2 &= \frac{R_1}{R_1 + R_2} I_s \end{aligned} \quad (2)$$

This is termed as the **Current Divider Rule (CDR)**.

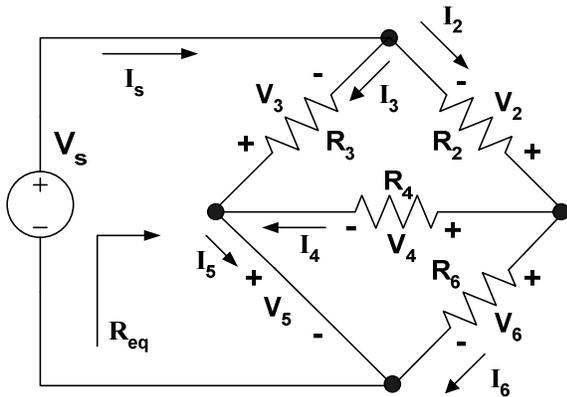


Figure. 3: Series-parallel circuit I

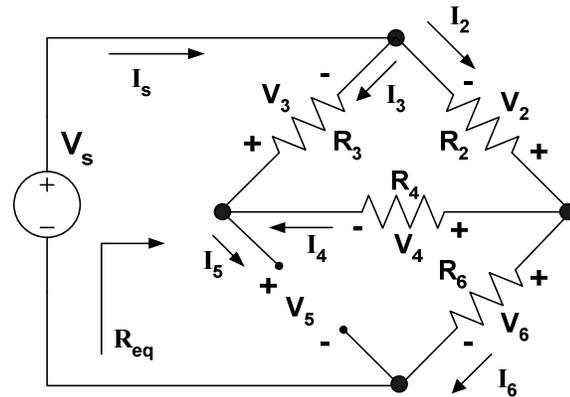


Figure. 4: Series-parallel circuit II

$$V_s = 10V, R_2 = 100\Omega, R_3 = 150\Omega, R_4 = 220\Omega, R_6 = 330\Omega$$

**Procedure:****Simulation**

1. Build the circuit given in Fig. 3 on *Multisim Electronics Workbench*.
2. Connect **voltmeters**, ammeters (or **multimeters**) at appropriate positions to measure voltages and currents shown in Table 1.
3. Disconnect the voltage source. Connect a mutimeter and measure the total resistance and record the value in Table 1. (Remember resistance is always measured without any source connected to the circuit)
4. Repeat steps 2 and 3 for the circuit given in Fig. 4 and record the values in Table 2.

**Hardware Experiment**

5. Build the circuit of Fig. 3 with the hardwired components. Take the voltage current measurements and  $R_{eq}$  and record in Table 1. Considering the Workbench results as the base compute the percentage errors.
6. Build the circuit of Fig. 4 with the hardwired components. Take the voltage current measurements and  $R_{eq}$  and record in Table 2. Considering the Workbench results as the base compute the percentage errors.

Check the values of resistors used in the circuit of Figure. 2, using the Multimeter. Record the values in Table 1.

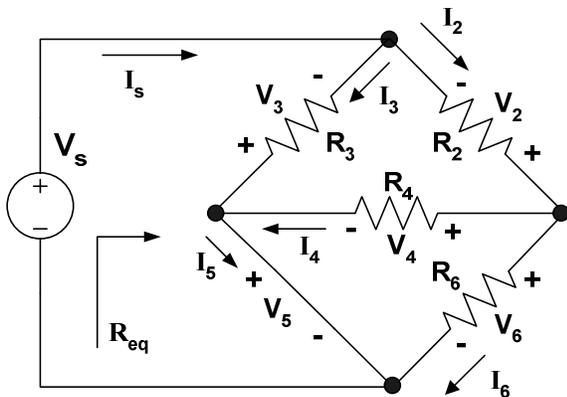


Figure. 3: Series-parallel circuit I

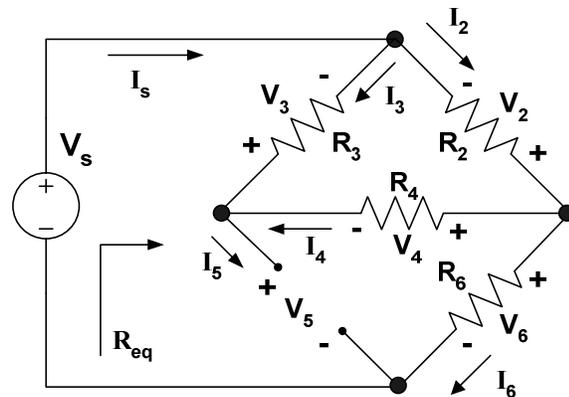


Figure. 4: Series-parallel circuit II

$$R_2 = 100\Omega, R_3 = 220\Omega, R_4 = 150\Omega, R_6 = 330\Omega$$

7. Connect the circuit of Figure. 2a and adjust the supply voltage  $V_s$  to 10 V, using the DC voltmeter.
8. Measure the entire unknown voltages and currents shown. Record their values in Table 2.
9. Measure  $R_{eq}$  using an Ohmmeter and record its values in Table 2.
10. Connect the circuit of Figure. 2b and adjust the supply voltage  $V_s$  to 10 V, using the DC voltmeter.
11. Measure the entire unknown voltages and currents shown. Record their values in Table 3 (recall that when measuring current by an ammeter, the ammeter should be placed in series with the element in which the current passes. Keep this fact in mind when measuring  $I_5$ ).
12. Measure  $R_{eq}$  and record its value in Table 3.

### Questions

1. Give reasons for any discrepancies between the theoretical and experimental values of voltages and currents and the equivalent resistance of both circuits.

By referring to the circuit of Figure.3

2. Are  $R_4$  and  $R_6$  in parallel or in series?
3. Are  $R_3$  and  $R_4$  in parallel or in series?
4. Are  $V_s$  and  $R_6$  in series or in parallel?
5. Is VDR applicable for applicable for  $R_3$  and  $R_4$ ?
6. Is CDR applicable for  $R_4$  and  $R_6$ ?
7. Is the parallel combination of  $R_4$  and  $R_6$  in series or in parallel with  $R_2$ ?

**TABLE 1**

#### Resistor Values:

Resistor	R2	R3	R4	R6
Nominal Value (Ohm)	100	220	150	330
Ohmmeter reading				

**TABLE 2**

#### Circuit of Figure. 2a:

Unknown	$I_1$	$I_2$	$I_3$	$I_4$	$I_5$	$I_6$	$V_2$	$V_3$	$V_4$	$V_5$	$V_6$	$R_{eq}$
Theory												
Experiment												
% Error												

**TABLE 3**

#### Circuit of Figure. 2b

Unknown	$I_1$	$I_2$	$I_3$	$I_4$	$I_5$	$I_6$	$V_2$	$V_3$	$V_4$	$V_5$	$V_6$	$R_{eq}$
Theory												
Experiment												
% Error												

**NAME** \_\_\_\_\_

**SEC #** \_\_\_\_\_

**ID #** \_\_\_\_\_

**SERIAL#** \_\_\_\_\_

**Answers to Questions:**

1.

2.

3.

4.

5.

6.

7.

## Experiment No. 5

### Superposition Theorem

#### Objective:

The objective of this experiment is to verify superposition theorem experimentally and using simulation.

#### Pre- Lab Assignment:

For the circuit shown Figure 1:

1. Find the  $V_L$  &  $I_L$  superposition.
2. Record your values in Table 2 and 3

Note:

- $V_L'$ ,  $I_L'$  are due to 5 V Source
- $V_L''$ ,  $I_L''$  are due to 10 V Source
- $V_L$  &  $I_L$  are due to both sources

#### Apparatus:

DC Power Supply (Two)  
 Digital Multimeter.  
 Carbon Resistors: 10 k $\Omega$ , 22 k $\Omega$ , 33 k $\Omega$ , 47 k $\Omega$ , and 1 k $\Omega$ .  
 PC for simulation with *multisim*

#### Introduction

The voltage and current responses in a network from two or more sources acting simultaneously can be obtained as the sum of the responses from each source acting alone with other sources deactivated. A deactivated current source is an open circuit. A deactivated voltage source is a short circuit.

#### Procedure:

**Note: do all the steps using computer simulation and experimentally.**

**Step 1 is experimental only.**

1. Check the values of the resistors using the multimeter. Record the values in Table.1.
2. Connect the circuit of Figure.1 and measure  $V_L$  &  $I_L$
3. Deactivate the 10 V source and measure  $V_L'$  &  $I_L'$
4. Reactivate the 10 V sources and deactivate the 5 V source. Measure  $V_L''$ , and  $I_L''$
5. Record the results in Tables 2 and 3

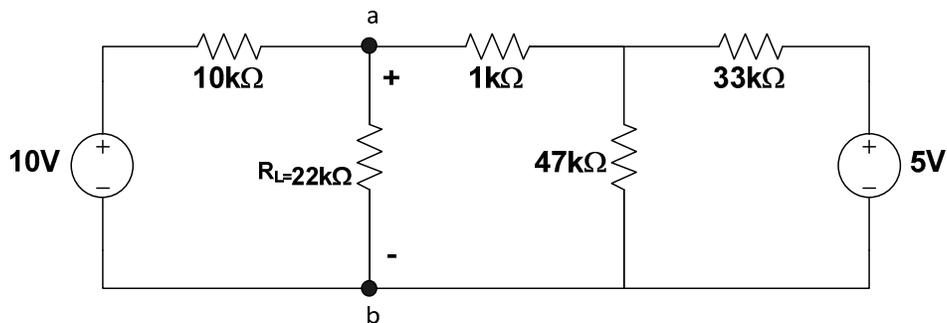


Figure. 1

**Questions:**

- 1- Discuss the reasons for any discrepancies between the theoretical and experimental values for all cases.
- 2- Does computer simulation represent actual circuit behavior? Discuss your answer by referring to your results



## Experiment No. 6

## THEVENIN / NORTON Theorems and Maximum Power Transfer

**Objective:**

- 1- To experimentally verify the Thevenin and Norton Theorems.
- 2- To experimentally verify the Maximum Power Transfer Theorem for resistive circuits.

**Pre- Lab Assignment:**

For the circuit shown Figure 1:

- 1- Find the  $V_{TH}$ ,  $I_{SC}$  and  $R_{th}$  seen by  $R_L$ .
- 2- Use the Thevenin's equivalent circuit you obtained to find the values of  $V_L$  when  $R_L$  is varied from 2.5 K to 10.5 K in steps of 1 K Ohms. Calculate  $V_L$  in each case.
- 3- Calculate the power  $P_L$  absorbed by  $R_L$  in each case of step 2.
- 4- Find the value of  $R_L$  for *Maximum Power Transfer* and the value of the *Maximum Power*.

**Apparatus:** DC Power Supply (Two)  
Digital Multimeter.  
Carbon Resistors: 10 k $\Omega$ , 22 k $\Omega$ , 33 k $\Omega$ , 47 k $\Omega$ , and 1 k $\Omega$ .  
Decade Resistor Box.

**Theory:****• Thevenin's Theorem:**

A two-terminal network can be replaced by a voltage source with the value equal the open circuit voltage across its terminals, in series with a resistor with the value equal to the equivalent resistance of the network.

**• Norton's Theorem:**

A two terminal network can be replaced by a current source with the value equal to the **short circuit** current at its terminal, in parallel with a resistor with the value equal to the equivalent resistance of the network. The equivalent resistance of a two-terminal network is equal to the **open circuit** voltage divided by the short circuit current.

• **Maximum Power Transfer Theorem:**

In a resistive circuit, a resistive load receives maximum power when the load resistance is equal to the Thevenin's equivalent resistance of the circuit (i.e.  $R_L = R_{TH}$ ). The maximum power can be calculated using the expression:

$$P = \frac{V_{OC}^2}{4R_{TH}}$$

where  $V_{TH}$  is the open circuit voltage.

**Procedure:**

Connect the circuit as shown in figure 1.

1. Check the values of the resistors using the multimeter. Record the values in Table.1.
2. Remove  $R_L$  from the original circuit and measure the open circuit voltage  $V_{oc}$  (or  $V_{TH}$ ).
3. Measure the short circuit current  $I_{sc}$ . This is accomplished by placing an Ammeter between A and B. In this manner, the Ammeter will act as a short circuit.
4. Replace the voltage sources with short circuits. With  $R_L$  removed from the circuit, measure  $R_{th}$  using a multimeter.
5. Record the results in Table 2.

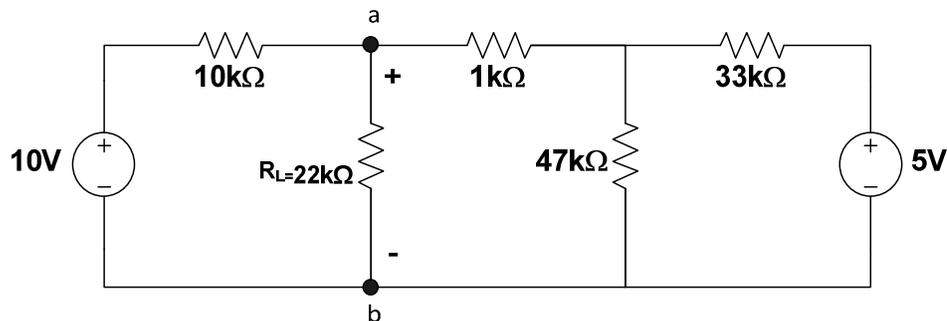


Fig. 1

**Maximum Power Transfer:**

6. Reconnect the circuit as shown in Figure 1, but replace the  $22\text{ K}\Omega$  resistor between A and B and by a variable resistor (i.e.  $R_L$  in this case is the variable resistor).
7. Vary  $R_L$  from  $2.5\text{ K}\Omega$  to  $10.5\text{ K}\Omega$  in steps of  $1\text{ K}\Omega$  and measure  $V_L$  in each case.
8. Record the results in Table 3.
9. Calculate  $P_L$  from Step. 7 above and record the results in Table 4.

**Questions:**

1. Draw the Thevenin's and Norton's equivalent circuit obtained experimentally.
2. Plot the theoretical and experimental values of  $P_L$  versus  $R_L$  (on the same graph) and compare the two graphs.
3. Discuss the reasons for any discrepancies between the theoretical and experimental values for all cases.
4. Thevenin's and Norton's Theorem are very useful. List at least two reasons for it
5. Is the Maximum Power Theorem verified experimentally? Explain.

<b>NAME</b> _____	<b>SEC #</b> _____
<b>ID #</b> _____	<b>SERIAL#</b> _____

**TABLE 1**

Resistor	R1	R2	R3	R4	R5
Nominal value (Ohm)	10K	22K	1K	47K	33K
Ohmmeter Reading					

**TABLE 2**

	$V_{TH}$ ( V )	$I_{sc}$ ( mA )	$R_{TH}$ ( K Ohm )
Theory			
Experiment			
Error			

**TABLE 3**

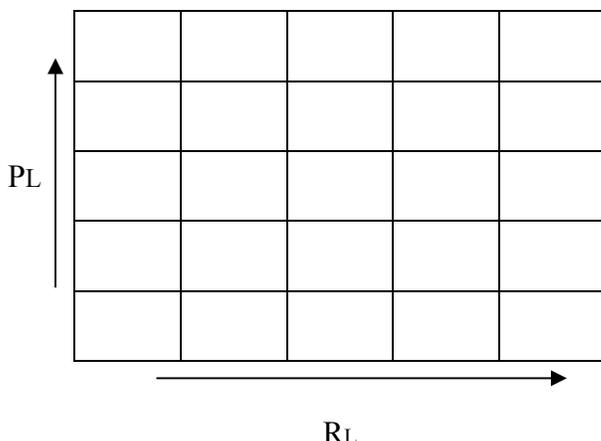
$R_L$ (K Ohm )	2.5	3.5	4.5	5.5	6.5	7.5	8.5	9.5	10.5
$V_L$ (Theoretical)									
$V_L$ (Experimental)									

**TABLE 4**

$R_L$ (K Ohm )	2.5	3.5	4.5	5.5	6.5	7.5	8.5	9.5	10.5
$P_L$ (Theoretical )									
$P_L$ (Experimental)									
% Error									

**Answers to Questions:**

Thevenin's Equivalent Circuit	Norton's Equivalent Circuit
1.	

<p>2.</p>  <p>The diagram shows a 5x5 grid of squares. To the left of the grid is a vertical arrow pointing upwards, labeled 'PL'. Below the grid is a horizontal arrow pointing to the right, labeled 'RL'.</p>
3.
4.
5.

## Experiment No. 7

### The Oscilloscope and Function Generator

---

#### Introduction

The **Oscilloscope** is one of the most important electronic instruments available for making circuit measurements. It displays a curve plot of time-varying voltage on the **Oscilloscope** screen.

The controls on the **Oscilloscope** are as follows:

1. The TIME/DIV control adjusts the time scale on the horizontal axis in time per division. The X\_POS control determines the horizontal position where the curve plot begins.
2. The CHANNEL A control adjusts the volts per division on the vertical axis for the channel A curve plot. The POSITION control located in top of CHANNEL A determines the vertical position of the channel A curve plot relative to the horizontal axis. Selecting AC places a capacitance between the channel A vertical input and the circuit testing point. Selecting “GND” connects channel A vertical input to ground.
3. Same thing applies to CHANNEL B.

#### The Function Generator

The **Function Generator** is a voltage source that supplies different time-varying voltage functions. The **Function Generator** can supply *Sine Wave*, *Square Wave*, and *Triangular Wave* voltage functions. The waveshape, frequency, amplitude, and DC offset can be easily changed. The controls on the **Function Generator** are as follows:

1. You can select a waveshape by selecting the appropriate waveshape (**Sine wave**, **Square Wave** and **Triangular Wave**) on the top of the **Function Generator**.
2. The frequency control allows you adjust the frequency of the output voltage. Select the frequency button and select the frequency with the appropriate scale.
3. The AMPLITUDE buttons allows you to adjust the peak to peak value or RMS of the output voltage. The peak to peak value is twice the amplitude setting.
4. The OFFSET buttons adjusts the DC level of the voltage curve generated by the **Function Generator**. When selecting the DC option on the Oscilloscope, an offset of 0 positions the curve plot along the x-axis with an equal positive and negative voltage setting. A positive offset raises the curve plot above the x-axis and a negative offset lowers the curve plot below the x-axis.

**Procedure**

1. Connect the output of the Function Generator to CHANNEL A on the Oscilloscope.
2. Turn the Function Generator and select the *Sine Wave* button. Set frequency to 1 kHz and 2 Vp-p. Set the DC offset to 0 V DC.
3. Turn on the Oscilloscope and select the GROUND button. Using the POSITON control on the top, bring your line to the center of the screen. Now select the AC position.
4. Select the TIME/DIV to 0.2ms and CHANNEL A to 0.5 V/DIV.

**Question:** What was the time period (T) and the frequency of your signal?

5. Select the “GROUND” on the **Oscilloscope** channel A.

**Question:** What change occurred on the **Oscilloscope** Channel A curve plot? Explain.

6. Change the **Oscilloscope** channel A to 1V/div.

**Question:** What change occurred on the **Oscilloscope** Channel A curve plot? Explain.

7. Change the **Oscilloscope** Time Base to 0.1ms/div.

**Question:** What change occurred on the **Oscilloscope** Channel A curve plot? What is the period and the frequency of your signal? Compare the results to that of step 4.

8. Return the **Oscilloscope** time base to 0.2m/DIV and CHANNEL A to 0.5V/DIV. Select the *Triangular Wave* shape on the **Function Generator**.

**Question:** What change occurred on the **Oscilloscope** curve plot?

9. Select the *Square Wave* on the **Function Generator** and run the analysis again.

**Question:** What change occurred on the **Oscilloscope** curve plot?

10. Change the AMPLITUDE on the **Function Generator** to 2Vp-p

**Question:** What change occurred on the **Oscilloscope** curve plot? Explain.

11. Change the frequency on the **Function Generator** to 2 kHz.

**Question:** What change occurred on the **Oscilloscope** curve plot? Explain.

12. Change the offset on the **Function Generator** to 1V.

**Question:** What change occurred on the **Oscilloscope** curve plot? Explain.

13. Select the DC setting on CHANNEL A of the oscilloscope.

**Question:** What change occurred on the **Oscilloscope** curve plot? Explain.

NAME \_\_\_\_\_

SEC # \_\_\_\_\_

ID # \_\_\_\_\_

SERIAL# \_\_\_\_\_

**Answers to Questions in Procedures****Question of Step 4****Question of Step 5****Question of Step 6****Question of Step 7****Question of Step 8**

**Question of Step 9**

**Question of Step 10**

**Question of Step 11**

**Question of Step 12**

**Question of Step 13**

**Question of Step 13**

## Experiment No. 8

### Frequency Domain Analysis

**Objective:**

1. To experimentally verify frequency domain calculations using the phasor method.
2. Experimentally measure the phase difference between two sinusoidal signal.

**Pre-Lab Assignment:**

For the circuit shown in Figure 2:

- 1- Assume that the inductor has an internal resistance of 4 Ohms.
- 2- Add 1.5 K $\Omega$  resistor with the inductor.
- 3- Let the source voltage  $V_s$  have a peak amplitude of 5 V and frequency  $f = 20$  kHz. Assume that  $V_s$  has a zero phase angle.
- 4- Use phasor method to calculate all voltages and currents shown.

Record your results in Table 2.

**Apparatus:**

Signal Generator.  
 Digital Multimeter.  
 Oscilloscope.  
 Capacitor 0.022  $\mu$ F.  
 Inductor 10 mH  
 Resistor: 1.5 K $\Omega$ .

**Theory:**

In the frequency domain (phasor domain), the currents and voltages are represented by complex numbers whose magnitudes are equal to the maximum values of the sinusoidal time-domain quantities, and whose angles are equal to the phase angles of the time-domain functions expressed as cosines.

In terms of voltage and current phasors and the complex impedance  $\bar{Z}$ , Ohm's law becomes:

$$\bar{V} = \bar{I} \bar{Z} \quad (1)$$

Where

$$\begin{aligned} \bar{Z} &= R && \text{for a resistance} \\ &= j\omega L && \text{for an inductance} \\ &= \frac{1}{j\omega C} && \text{for a capacitance} \end{aligned}$$

where  $\omega = 2\pi f$  is the angular frequency of the source. The bar indicates a complex quantity.

In general, for  $\bar{V} = V\angle\theta_1$  and  $\bar{I} = I\angle\theta_2$ , the impedance  $\bar{Z} = \frac{V}{I}\angle\theta_1 - \theta_2$

Analytically, frequency-domain circuits are treated by the same method as used in DC circuits, except that the algebra of complex numbers is used.

Experimentally, the frequency-domain phasors can be measured on the oscilloscope. The magnitudes can be measured by means of calibrated vertical scales. Phase difference can be measured by using the dual traces and measuring the time difference between two waveforms, as illustrated in Figure. 2.

### IMPORTANT NOTE:

When measuring the phase difference between two signals, make sure that the control knobs in the oscilloscope are set properly such that the signals are not relatively inverted.

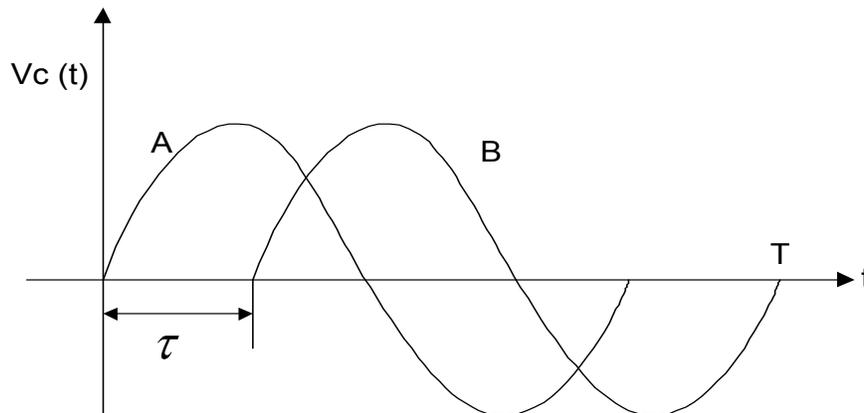


Figure. 1

Time shift =  $\tau$

Phase shift =  $360 \left( \frac{\tau}{T} \right)$  degrees

Signal B lags signal A.

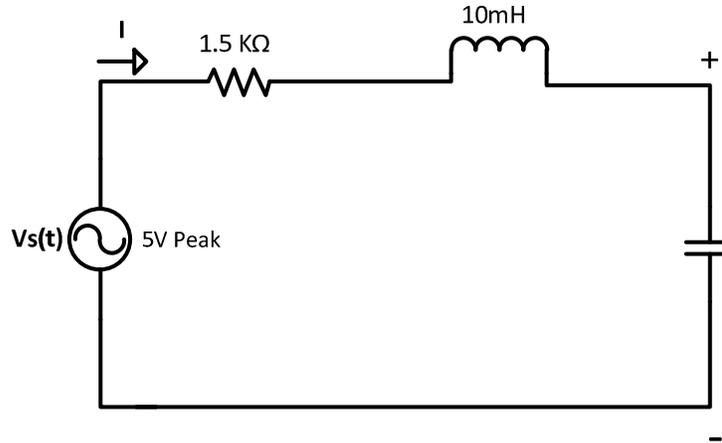


Figure. 2

**Procedure:**

- 1- Measure the resistor value and the internal resistance of the inductor, using an Ohmmeter.
- 2- Connect the circuit of Figure. 2. Adjust the source voltage to 5 V peak at 20 kHz, **while it is connected to the circuit.**
- 3- Use the oscilloscope to measure the magnitudes and phases of all voltages and record the values in Table 2.
- 4- Find the Value of the current using the following equation:

$$I = V_R / R.$$

**Important Note:**

1. When measuring the phase difference between two signals, make sure that the control knob of the oscilloscope are set properly such that the signals are not relatively inverted.
2. Whenever two signals are to be displayed simultaneously on the oscilloscope, they should have one common node as a reference. Therefore, you may have to change the position of some elements to be able to measure two signals simultaneously.
3. To find  $I_2$ , measure the voltage on the 1.5KΩ resistor.

**Report:**

- 1- Draw the circuit of Figure. 2 in the frequency domain (in the phasor representation). Use the space available in the report sheet at the end of the experiment.
- 2- Record the theoretical and experimental values in Table 2.
- 3- Draw the phasor diagram, showing all the voltages and currents, based on the experimental values. Use the space available in the report sheet at the end of the experiment.
- 4- Discuss the sources of discrepancies between the theoretical and experimental values.

**Questions:**

1. For a resistance and capacitance in series with a voltage source, show that it is possible to draw a phasor diagram for the current and all voltages from magnitude measurement of these quantities only. Illustrate your answer graphically.
2. The equivalent impedance of a capacitor in series with an inductor is equivalent to a short circuit (i.e. equal to zero) at a certain frequency. Derive an expression for this frequency.
3. The equivalent impedance of a capacitor in parallel with an inductor is equivalent to an open circuit (i.e. equal to infinity) at a certain frequency. Derive an expression for this frequency.

<b>NAME</b> _____	<b>SEC #</b> _____
<b>ID #</b> _____	<b>SERIAL#</b> _____

**TABLE 1**

Resistor	$R_1$	Internal resistance of Inductor
Nominal Value ( Ohm )	1.5K $\Omega$	
Ohmmeter Reading		

Internal resistance of the inductor = \_\_\_\_\_  $\Omega$

**TABLE 2**

		$V_R$	$V_L$	$V_C$	I
Theoretical Values	Magnitude ( Volts )				
	Phase Shift ( Degree )				
Experimental Values	Magnitude ( Volts )				
	Phase Shift ( degrees )				
% Error	Magnitude (Volts)				
	Phase Shift ( degrees )				

**CIRCUIT IN FREQUENCY DOMAIN**

**PHASOR DIAGRAM**

## Experiment No. 9

## Maximum Power Transfer

**Objective:**

1. To obtain maximum output power from a sinusoidal source with an internal impedance
2. To experimentally verify the theory of maximum power transfer.

**Pre- Lab Assignment:**

For the circuit shown Figure 1:

1. Assume that the inductor has an internal resistance of 4 ohms accounts for this resistance in this calculation
2. Find the value of  $R_L$  and  $C_L$  for maximum power transfer, and
3. Calculate the maximum power transferred to this load
4. Summarize your results in Table 4.

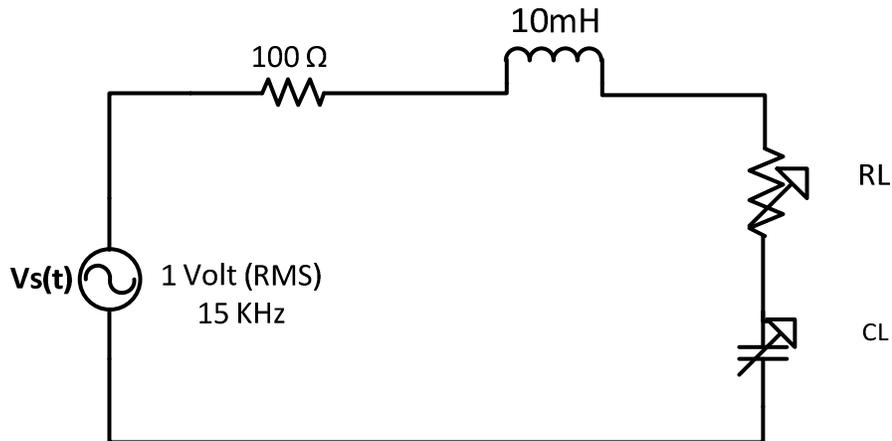


Fig. 1

**Apparatus:**

- Signal Generator
- Digital Multimeter
- Decade Resistance Boxes RDB and RDA
- Decade Capacitance Box CDA
- Inductor (10 mH)
- Resistance (100 Ohms)

**Theory:**

Consider Figure. 1. If the source impedance has a resistive and reactive parts, the maximum power is transferred to the load (actually to the resistive part of the load) when the load resistance equal the source resistance and the load reactance is opposite to the source reactance.

Namely

$$Z_L = Z_S^* \quad (1)$$

Equation (1) implies that if the source impedance is inductive the load impedance must be capacitive and vice versa. In order to compute the value of a capacitance, one can use the following relation

$$\begin{aligned} \omega &= 1/\sqrt{LC} \\ C &= 1/\omega^2 L \end{aligned} \quad (2)$$

Note that since  $Z_L=Z_S$ ,  $V_S$  sees a combined load with impedance  $Z_L+Z_S=Z_S+Z_S=2R_S$  which is purely resistive. This means, under this condition (the condition of maximum power transfer),  $V_S$  becomes in phase with the current through the load and thus  $V_S$  becomes in phase with the voltage across the load resistance.

The maximum power transferred to the load in this is

$$P_{\max} = \frac{V_s^2}{4R_s} \quad (3)$$

**Procedure:**

1. Consider the 10 mH inductor in the circuit of Fig. 1. as part of the source impedance. Measure its internal resistance  $R_i$ , Add 100 Ohm in addition to this resistance. This means  $R_S= R_i+100$ . Set the source voltage to a sinusoid of 1  $V_{RMS}$  and frequency of 15 KHz.

2. With  $R_L$  set the value of  $R_S$ , vary  $C$  from  $0.007 \mu\text{F}$  to  $0.015 \mu\text{F}$  in steps of  $0.001 \mu\text{F}$  (use two decade capacitors in parallel). Measure the voltage across  $R_L$  in each case, maintaining an input voltage of  $1 V_{\text{RMS}}$ . Record the values in table 2.
3. Display the source voltage and the voltage across  $R_L$  simultaneously on the oscilloscope. Vary  $C$  and notice the phase shift between the two signals. Record the value of  $C$  that makes the two signals in phase.
4. With  $C$  set at the value found in step 3, vary  $R_L$  from 80-120 and measure the voltage across  $R_L$  in each case, again maintaining an input voltage of  $1 V_{\text{RMS}}$ . Record the values in table 4.

**Note:** Record your results in Table No. 1, 2, 3 & 4 and answer the questions.

**Questions:**

- 1) What is the phase difference between the current and the voltage source when maximum power transfer is achieved?
- 2) If the frequency of the source is doubled, what change should be done to maintain maximum power transfer to the load? How does this change affect the value of the maximum power? Explain.
- 3) Comment on the causes of errors between the measure and calculated values.

NAME	_____	SEC #	_____
ID #	_____	SERIAL#	_____

**Table 1. Resistive value**

Resistor	R1
Nominal Value ( Ohm )	4
Ohmmeter Reading	

Internal Resistance of the inductance  $R_i =$

**Table 2. Results for C**

C ( $\mu$ F )	0.007	0.008	0.009	0.010	0.011	0.012	0.013	0.0147	0.0153
$V_L$ ( V )									
$P_L$ ( W )									

**Table 3. Results for  $R_L$**

$R_L$ ( Ohm )	80	90	95	100	$R_s$	110	120
$V_L$ ( V )							
$P_L$ ( W )							

**Table 4. Comparison at Maximum Power Transfer**

	$R_L$	C	$P_{max}$
Theory			
Experiment			
% Error			

## Experiment No. 10

### Average and RMS Values

#### Objective:

- 1- To measure the average and **Root Mean Square (RMS)** values of some electrical signals.
- 2- To compare the calculated and experimental values.

#### Pre-Lab Assignment:

- 1- For the periodic signals shown in Figure 1, find:
  - a) The average value of each signal.
  - b) The RMS value of each signal.
- 2- For the periodic signals shown in Figure 2, find the average value of each signal.

#### Apparatus:

Oscilloscope  
 Signal Generator  
 Digital Multimeter (DMM)

#### Theory:

##### 1- Average Value:

The average value of a periodic signal  $f(t)$  is defined as:

$$f_{avg} = \frac{1}{T} \int_0^T f(t) dt = \frac{\text{area under one period}}{\text{period}} \quad (1)$$

##### 2- The Root Mean Square value:

The RMS value of a periodic signal  $f(t)$  is defined as :

$$f_{rms} = \sqrt{\frac{1}{T} \int_0^T f^2(t) dt} = \text{Square root of the average of } f^2(t). \quad (2)$$

For a general sinusoidal signal  $f(t) = A\cos(\omega t + \phi)$  ., it is easy to show that

$$f_{avg} = 0 \quad (3)$$

$$f_{max} = \frac{A}{\sqrt{2}} \quad (4)$$

For a signal that consists of a DC and AC parts such as,  $f(t) = B + A\cos(\omega t + \phi)$  where, B = constant, it is equally easy to show that the average value is equal to the DC part only, namely:

$$f_{avg} = B \quad (5)$$

### Procedure:

- 1- Set the oscilloscope controls as follows:

Time / Div.	0.1 ms	Volts / Div.	0.5 V
Vertical Display	A Coupling D.C.		

- 2- Connect the output of the signal generator to channel A of the oscilloscope.
- 3- Set the signal generator to a sinusoid of 2000 Hz and a peak to peak voltage  $V_{p-p} = 2$  V.
- 4- Set the DC offset of the signal generator to zero, so that the signal you obtain on the oscilloscope has no DC value as shown in figure 1.
- 5- Measure :
- The average value of the signal with a DMM (set the voltmeter to D.C.).
  - The RMS value of the signal with a DMM (set the voltmeter to AC).
  - The period of the signal from the oscilloscope. Then calculate the corresponding frequency from the relation  $f = 1/T$ .
- 6- Repeat steps 3-5 for a square signal with the same  $V_{p-p}$  and frequency.
- 7- Repeat steps 3-5 for a triangular signal with the same  $V_{p-p}$  and frequency.
- 8- Set the DC offset of the signal generator to 1 V so that the signal you obtain on the oscilloscope has no DC value of 1 V as shown in figure 1.
- 9- Repeat the measurement of the average value for the same signals as in previous steps with the following changes:
- Adjust the ground level to the bottom of the screen.
  - Adjust the D.C. offset of the signal generator, so that the signals start from zero and reach a peak value of 1 V as shown in Figure.2.

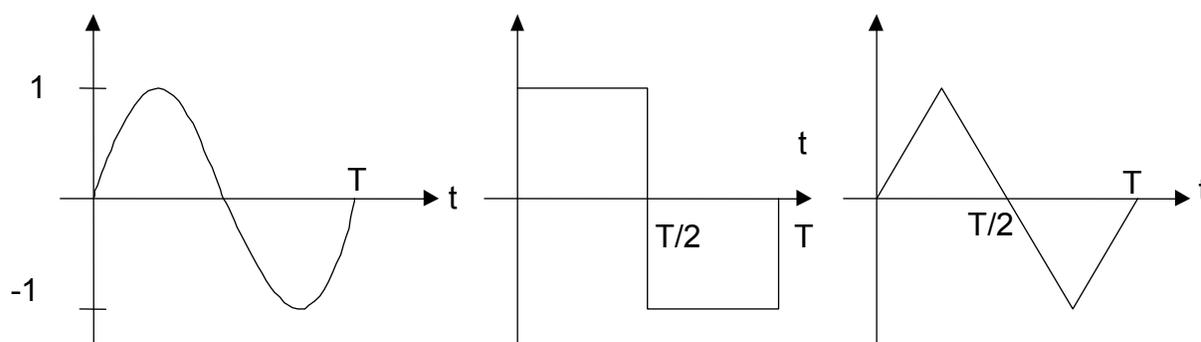


Figure. 1

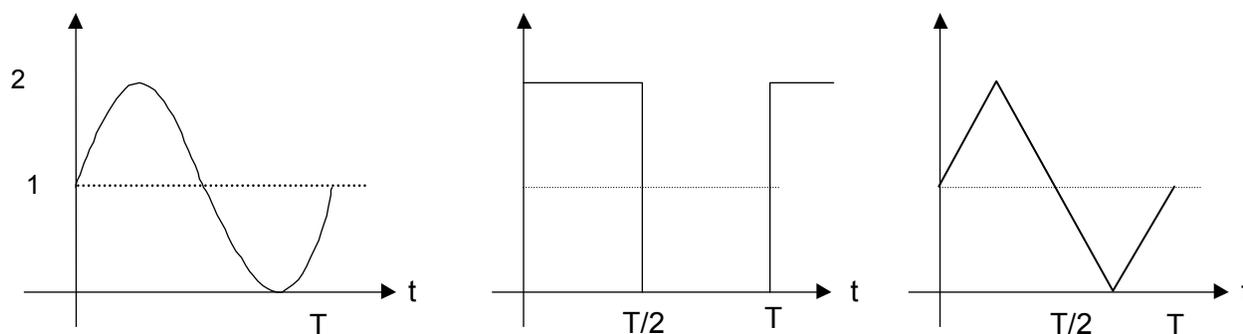


Figure.2

**Note:** Record your results in Table No. 1 & 2 and answer the questions.

### Questions:

- 1- The frequency of the voltage in your house is 60 Hz. How much time is required for the waveform to complete three cycles?
- 2- What is the difference between AC and DC coupling of the oscilloscope? Explain how to use them to measure the average value of any periodic signal.
- 3- Some meters are calibrated to read RMS. Value of sinusoidal waveforms from the basic unit that responds to the peak value of the waveform. In terms of the peak value,  $V_p$ , the meter will read  $\frac{V_p}{\sqrt{2}}$ , which is the correct RMS value for a sinusoidal signal. Can this meter be used to read the correct RMS value for other waveforms like square, triangular, etc? Explain.

<b>NAME</b>	_____	<b>SEC #</b>	_____
<b>ID #</b>	_____	<b>SERIAL#</b>	_____

**TABLE 1**

	Sinusoidal	Square	Triangular
T (ms)			
f (Hz)			
Average Value (Calculated)			
Average Value (Experimental)			
RMS Value ( Calculated )			
RMS Value (Experimental)			
% Error			

**TABLE 2**

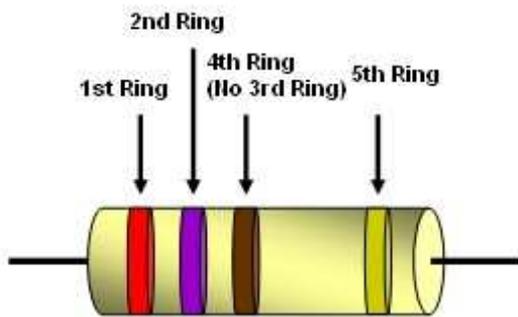
	Sinusoidal	Square	Triangular
Average Value (Calculated)			
Average value (Experimental)			
% Error			

**Answers to Questions:**

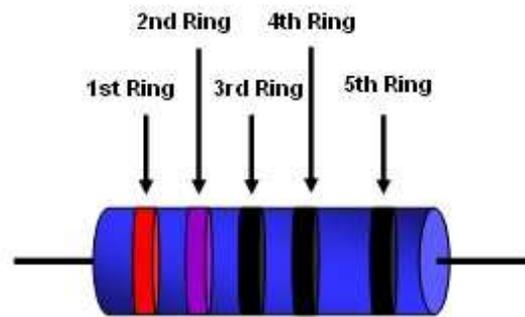
- 1.
  
- 2.
  
- 3.

Appendix I

4 COLOR BAND RESISTOR



5 COLOR BAND RESISTOR



Color	1st Ring	2nd Ring	3rd Ring	4th Ring (Multiplier)	5th Ring (Tolerance)
Black	0	0	0	1	1 %
Brown	1	1	1	10	
Red	2	2	2	100	
Orange	3	3	3	1000	
Yellow	4	4	4	10000	
Green	5	5	5	100000	
Blue	6	6	6	1000000	
Violet	7	7	7	10000000	
Gray	8	8	8	100000000	
White	9	9	9	1000000000	
Gold	-	-	-	-	5 %
Silver	-	-	-	-	10 %
No Color					20%

## Appendix II

**Capacitor conversion table for pico-Farads, nano-Farads, and micro-Farads**

<b>micro-Farads (<math>\mu\text{F}</math>)</b>	<b>Nano-Farads (nF)</b>	<b>Pico-Farads (pF)</b>
<b>0.000001</b>	<b>0.001</b>	<b>1</b>
<b>0.00001</b>	<b>0.01</b>	<b>10</b>
<b>0.0001</b>	<b>0.1</b>	<b>100</b>
<b>0.001</b>	<b>1</b>	<b>1000</b>
<b>0.01</b>	<b>10</b>	<b>10000</b>
<b>0.1</b>	<b>100</b>	<b>100000</b>
<b>1</b>	<b>1000</b>	<b>1000000</b>
<b>10</b>	<b>10000</b>	<b>10000000</b>
<b>100</b>	<b>100000</b>	<b>100000000</b>

**LABORATORY REGULATIONS AND SAFETY RULES**

The following Regulations and Safety Rules must be observed in all concerned laboratory location.

1. **It is the duty of all concerned who use any electrical laboratory to take all reasonable steps to safeguard the HEALTH and SAFETY of themselves and all other users and visitors.**
2. **Be sure that all equipment is properly working before using them for laboratory exercises. Any defective equipment must be reported immediately to the Lab. Instructors or Lab. Technical Staff.**
3. **Students are allowed to use only the equipment provided in the experiment manual or equipment used for senior project laboratory.**
4. **Power supply terminals connected to any circuit are only energized with the presence of the Instructor or Lab. Staff.**
5. **Students should keep a safety distance from the circuit breakers, electric circuits or any moving parts during the experiment.**
6. **Avoid any part of your body to be connected to the energized circuit and ground.**
7. **Switch off the equipment and disconnect the power supplies from the circuit before leaving the laboratory.**
8. **Observe cleanliness and proper laboratory housekeeping of the equipment and other related accessories.**
9. **Wear the proper clothes and safety gloves or goggles required in working areas that involves fabrications of printed circuit boards, chemicals process control system, antenna communication equipment and laser facility laboratories.**
10. **Double check your circuit connections specifically in handling electrical power machines, AC motors and generators before switching “ON” the power supply.**
11. **Make sure that the last connection to be made in your circuit is the power supply and first thing to be disconnected is also the power supply.**
12. **Equipment should not be removed, transferred to any location without permission from the laboratory staff.**
13. **Software installation in any computer laboratory is not allowed without the permission from the Laboratory Staff.**
14. **Computer games are strictly prohibited in the computer laboratory.**
15. **Students are not allowed to use any equipment without proper orientation and actual hands on equipment operation.**
16. **Smoking and drinking in the laboratory are not permitted.**

**All these rules and regulations are necessary precaution in Electrical Laboratory to safeguard the students, laboratory staff, the equipment and other laboratory users.**