

University of Anbar
College of Engineering
Department of
Electrical Engineering



Laboratory Manual of Fundamentals of Electrical Circuits

اعداد

مصطفى احمد نايف

مدرس مساعد

ياسر عبدالحافظ احمد

مدرس

ناظم منير محيسن

مهندس

March 2022

Contents

Subject	Page no.
How to Write a Lab. Report	2
Experiment No.1: Using and Employment Laboratory Instrument	5
Experiment No.2: Ohm's Law for Linear and Nonlinear Circuit	10
Experiment No.3: Kirchhoff's laws	15
Experiment no. 4: Resistors in Series & Parallel	19
Experiment No.5: Series-Parallel (Combination) Networks	23
Experiment No. 6: Delta – Star connection	26
Experiment No.7: Superposition Theorems	30
Experiment No.8: Thevenin's Theorems	34
Experiment No.9: Norton's Theorem	38
Experiment No.10: Impedance Elements Characteristics	43
Experiment No.11: RL and RC Circuits	50
Experiment No.12: The Series RLC Resonance Circuit	61
Experiment No.13: The Parallel RLC Resonance Circuit	66
Experiment N0. 14: Power Factor Correction	69
MultiSim	73

How to Write a Lab. Report

University of Anbar
College of Engineering
Department of.....



Lab. of: Fundamental of
Electrical Circuits

Stage:

Group:

Experiment No.

Title of the experiment (هنا يكتب عنوان التجربة)

اسم الطالب

(يكتب الاسم باللغة العربية)

Date of Experiment: / /

Title of experiment

1- Object:

.....

2- Theory:

هنا يكتب ما تم فهمه من الموضوع وليس واجبا ان تنقل ما موجود في الكراسة ويمكنك نقل الاشكال الضرورية فقط

.....

3- Procedures:

3.1 run-1:.....

تتم كتابة الاجراءات الخاصة بالقسم الاول ورسم الدائرة الكهربائية الخاصة بالقسم وانزال جدول النتائج الخاص بنفس القسم والذي حصلت عليه عمليا حيث يتم نقله بصورة مرتبة من ورقة النتائج التي حصلت عليها اثناء التجربة ثم يتم رسم العلاقة المطلوبة على ورقة بيانية

3.2 run-2:.....

نفس الاجراءات

3.3 run-3:.....

وهكذا جميعا بنفس التسلسل والاجراءات

4- Discussion:

هنا يتم الاجابة عن اسئلة المناقشة

ملاحظات عامة

- 1- يكتب التقرير باليد (عدا الواجهة فيمكن ان تكون طباعة او كتابة باليد) وباللغة الانكليزية ويكس في فايل
- 2- يكتب التقرير على ورق ابيض او مخطط حجم A4 ولايجوز استخدام ورق الدفتر
- 3- تكون الكتابة بالقلم الجاف الازرق او الاسود فقط ولا يجوز استخدام الالوان
- 4- ترسم العلاقات المطلوبة على ورق بياني وتكس مع التقرير
- 5- ترقم الاشكال والجدول من رقم (1) صعودا وحسب ورودها في التقرير ولا يشترط ترقيمها بارقام الكراسة
- 6- لايجوز ادخال اشكال كمالية او اطارات او اي شي خارج الموضوع
- 7- ترفق ورقة القراءات الموقعة في المحنبر مع التقرير
- 8- امنياتي لكم بالموفقية

Experiment No.1

Using and Employment Laboratory Instruments

Object

The students be familiar with Using and Employment laboratory instruments impact for their significant importance in their field of specialty.

- **D.C power supply**

A DC Power Supply Unit (commonly called a PSU) deriving power from the AC mains (line) supply performs a number of tasks:

1. It changes the level of supply to a value suitable for driving the load circuit.
2. It produces a DC supply from a pure AC wave.
3. It prevents any AC from appearing at the supply output.
4. It will ensure that the output voltage is kept at a constant level, independent of changes in the AC supply voltage at the supply input and the Load current drawn from the supply output.

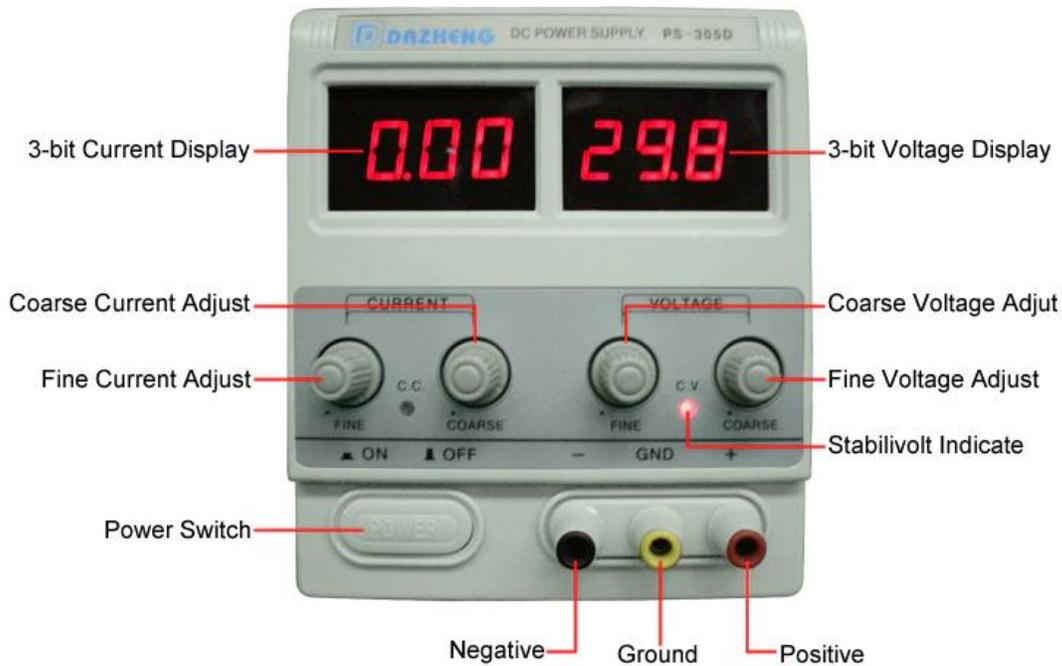


Fig 1: DC power supply

- **Multimeter**

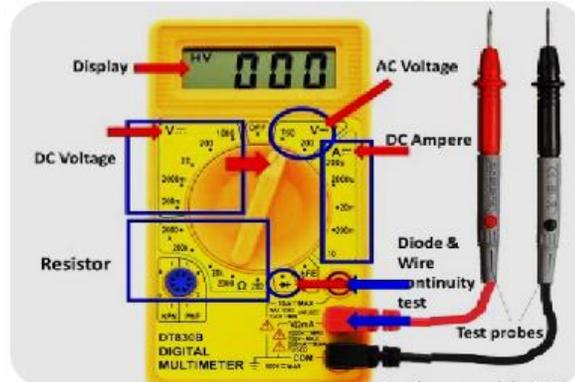


Fig 2: multimeter

- **Oscilloscope**

The oscilloscope is an electronic test instrument that displays electrical signals graphically, usually as a voltage (vertical or Y axis) versus time (horizontal or X axis). Oscilloscopes are commonly used for measurement applications such as:

- observing the wave shape of a signal
- measuring the amplitude of a signal
- measuring the frequency of a signal
- measuring the time between two events
- observing whether the signal is direct current (DC) or alternating current (AC)
- observing noise on a signal

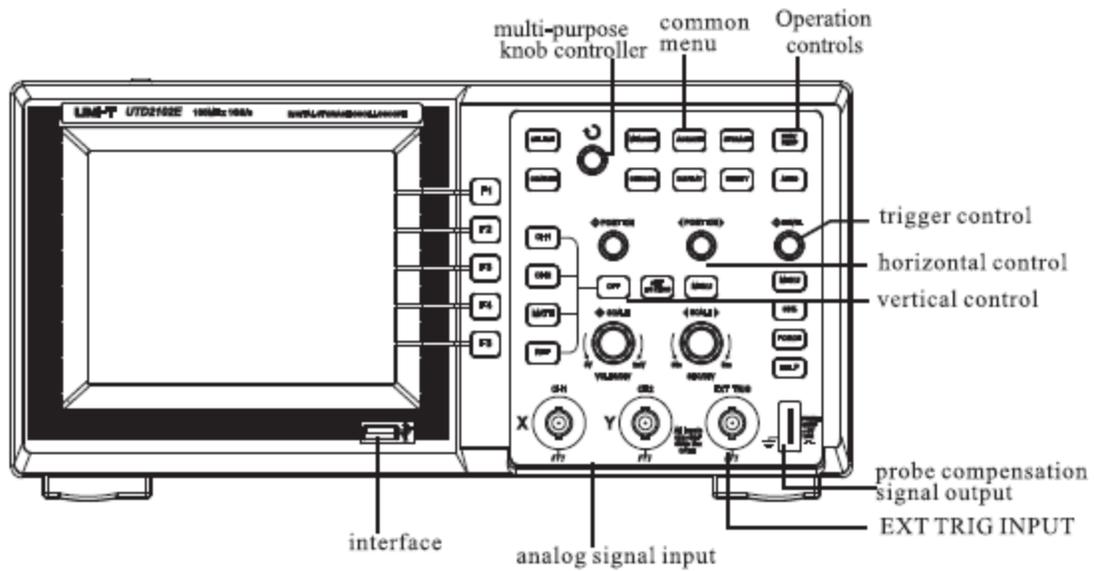


Fig 3: Oscilloscope

- **function generator**

A function generator shown in figure below is usually a piece of electronic test equipment or software used to generate different types of electrical waveforms over a wide range of frequencies. Some of the most common waveforms produced by the function generator are the sine wave, square wave, triangular wave and saw tooth shapes.

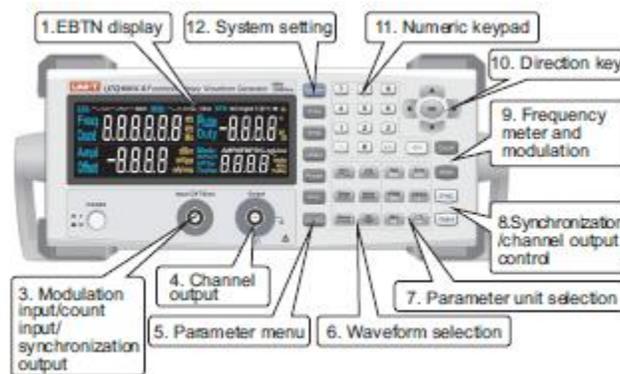


Fig 4: function generator

- **Color Coding and Standard Resistor Values**

A wide variety of resistors, fixed or variable, are large enough to have their resistance in ohms printed on the casing. Some, however, are too small to have numbers printed on them, so a system of color coding is used. For the thin-film resistor, four, five, or six bands may be used. The four-band scheme is described. Later in this section the purpose of the fifth and sixth bands will be described. For the four-band scheme, the bands are always read from the end that has a band closest to it, as shown in figure below. The bands are numbered as shown for reference in the discussion to follow. The first two bands represent the first and second digits, respectively. They are the actual first two numbers that define the numerical value of the resistor. The third band determines the power-of-ten multiplier for the first two digits (actually the number of zeros that follow the second digit for resistors greater than 10 Ω). The fourth band is the manufacturer's tolerance, which is an indication of the precision by which the resistor was made. If the fourth band is omitted, the tolerance is assumed to be 20%. The number corresponding to each color is defined in Fig. 1 The fourth band will be either 5% or 10% as defined by gold and silver, respectively.



Color coding for fixed resistors.

Number	Color
0	Black
1	Brown
2	Red
3	Orange
4	Yellow
5	Green
6	Blue
7	Violet
8	Gray
9	White
±5% (0.1 multiplier if 3rd band)	Gold
±10% (0.01 multiplier if 3rd band)	Silver
20%	non

$AB \times 10^C$

$20 \times 10^1 = 200 \Omega$

Tolerance

$5/100 \times 200 = \pm 10 \Omega$

$R = 200 + 10 = 210$

$R = 200 - 10 = 190$

$R = (190 - 210)$

example

$$AB \times 10^C$$

$$20 \times 10^5 = 2000000 \Omega = 2000 K\Omega = 2 M\Omega$$

Tolerance

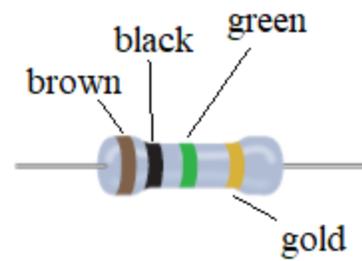
$$5/100 \times 20 \times 10^5 = \pm 100000 \Omega = 100 K\Omega$$

Tolerance

$$R = 2000 + 100 = 2100 K\Omega$$

$$R = 2000 - 100 = 1900 K\Omega$$

$$R = (1900 - 2100) K\Omega$$



Experiment No.2

Ohm's Law for Linear and Nonlinear Circuit

Object

To study the relationship between voltages and currents for linear elements.

Theory

In any electrical circuit, the charges flow (current) through its elements. large amount of energy will convert to heat due to the resistance of the material to the flow of electrical charges. The symbols of a resistors are shown in Fig.1

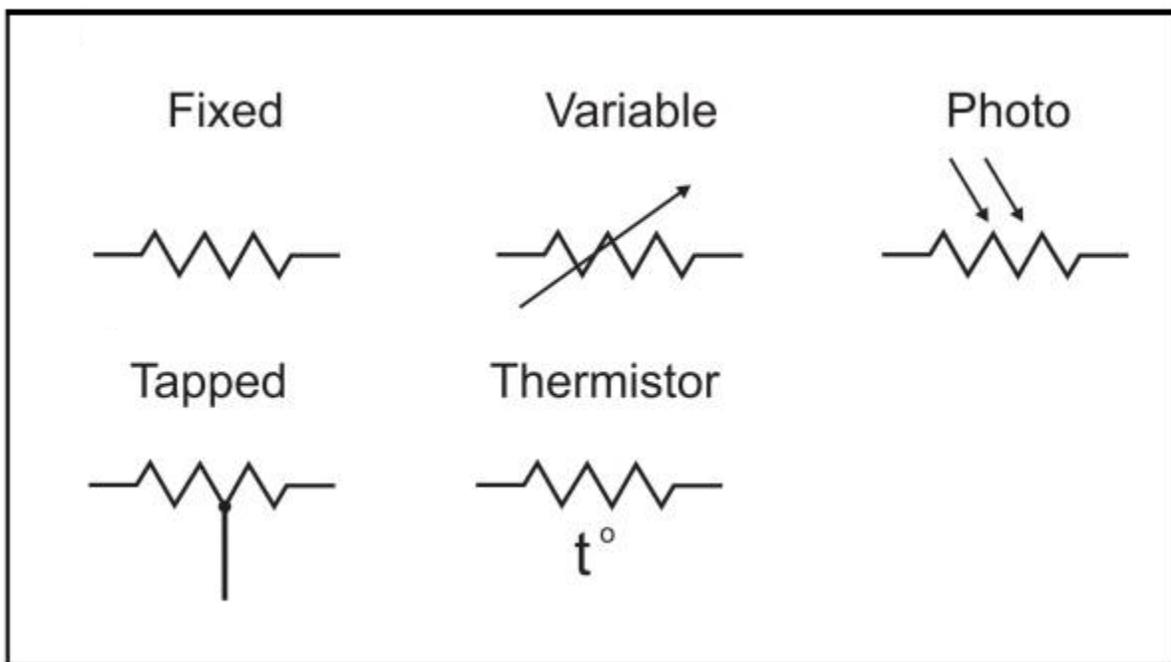


Fig.1: symbols of resistors

- Fixed Resistor: These are the predominant type of resistor configuration, and as the name suggests, they have a fixed resistance value.
- Variable resistors can be used to adjust circuit elements (such as a volume control or a lamp dimmer),
- A photoresistor: (also known as a Photocell, or light-dependent resistor, LDR, or photo-conductive cell) is a passive component that

decreases resistance with respect to receiving light on the component's sensitive surface.

- Thermistor: The word thermistor means a thermal resistor. Its resistance value changes with the change in the temperature.
- Tapped resistor: They are commonly used in audio/visual equipment and in transducers. these variable resistors use a sliding contact to regulate voltage.

Ohm's law describes the relationship between the terminal voltage and current of a resistor, and stated as follows: -

The ratio of potential difference (V) between any two points on a conductor to the current (I) flowing between them is constant when the temperature of the conductor does not change. this can be represented as in the formula:

$$R = V/I$$

- V: is the potential difference and measured in volt (V).
- I: is the current flowing through the conductor and measured in Ampere (A).
- R: is the resistance of the conductor and measured in Ohm (Ω).

Ohm's Law for Linear Elements

Fig.2 shows the linear (straight-line) relationship between the potential difference and the current flow through two fixed resistances: ($R=20 \Omega$) & ($R=40 \Omega$).

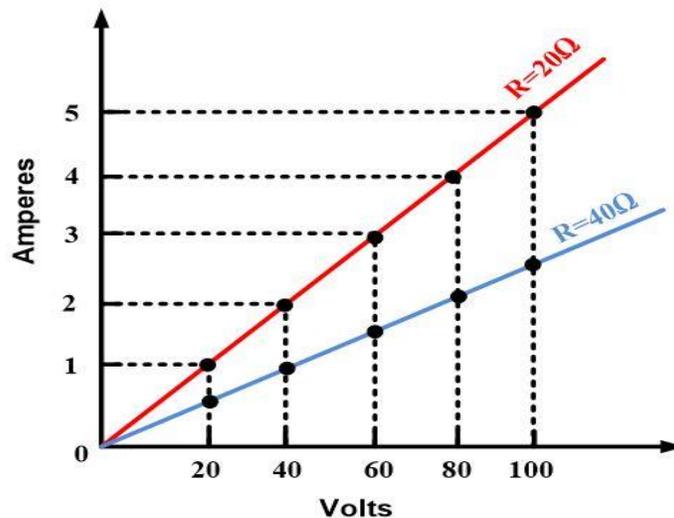


Fig.2: relationship between voltage and current through resistors.

Non-linear resistance (the incandescent lamp)

The V/I characteristic shown in Fig.2 is so-called ideal resistor, which is an ideal device keeps a constant and unchanged resistance in all physical conditions.

In practice all materials have a V/I characteristic which is really non-linear due to many parameters such as temperature, mechanical stress, age, etc....

In most case it is desirable that the resistors used in electronics circuits have values which are as constant as possible. In certain other cases a non-linear V/I characteristic may be useful for specific purposes.

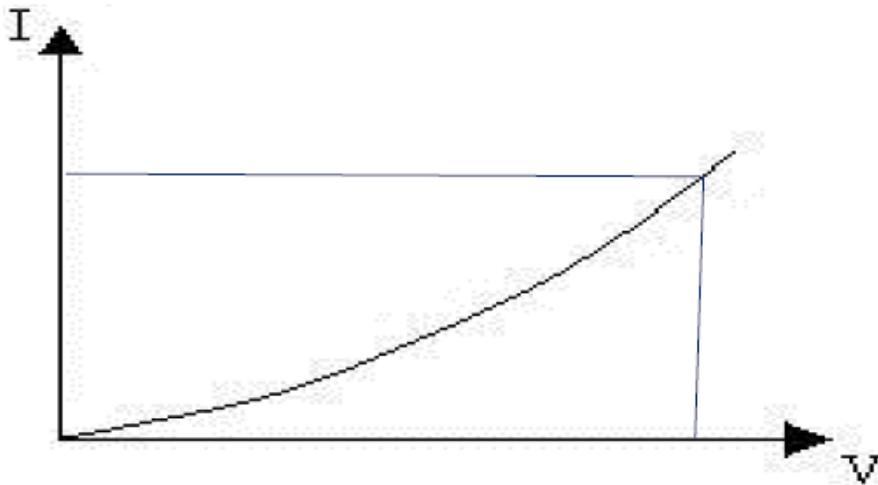


Fig.3: characteristics of lamp circuit

The bending of the characteristics is due to the TEMPERATURE COEFFICIENT of the material of which the filament is made.

The temperature coefficient in this case is positive:

- If the supply voltage is increased, the filament receives more power and therefore its temperature rises.
- The resistance of the filament increases for rising temperature and the filament drains less current than if the resistance remained constant.

So, positive temperature coefficient for an incandescent lamp is very useful for the lamp itself, comparison with the case of a hypothetical negative temperature coefficient lamp connected to the mains. An accidental rise in the line voltage would raise the temperature of the lamp draw more current and power from the line. This would in turn, further heat the lamp and so on, up to a rapid burning-out of the filament.

Note the following:

- The current rises for growing voltages.
- If the voltage doubles (e.g. 5V to 10V), the current also doubles (5mA to 10mA).

Procedures

- **Part 1:**

We shall start from a common incandescent lamp, which allows us understand the meaning of non-linear V/I characteristic.

1- Connect the circuit as shown in Fig.4

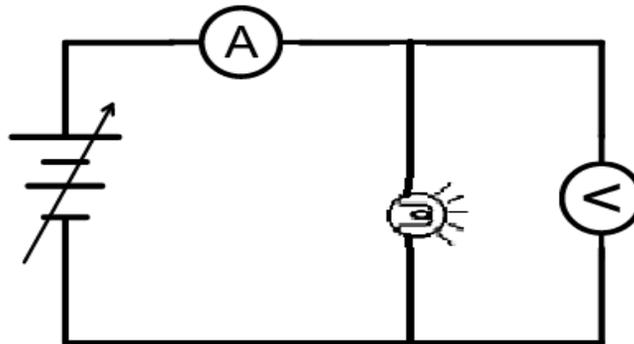


Fig.4: incandescent lamp circuit

2- Increase the applied voltage from the D.C power supply from (0 – 10) V, in step of (2)V, measure the current through the (1K Ω) resistor in each step and record the measured result in the second column of table.1.

Table 1

Power supply (Volt)	Current (I) measured (mA)	Resistance (R) calculated (Ω)
0		
2		
4		
6		
8		
10		

• **Part2:**

- 1- Connect the circuit shown in Fig.5
- 2- Increase the applied voltage from the D.C power supply from (0 – 10) V, in step of (2)V, measure the current through the (1K Ω) resistor in each step and record the measured result in the second column of table.2.

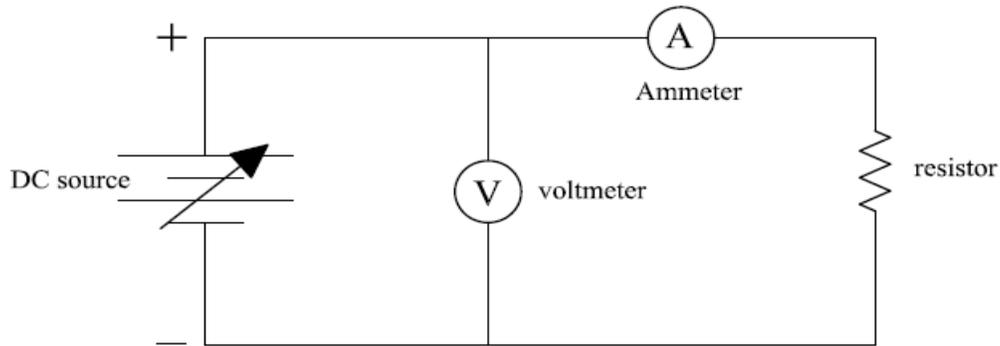


Fig.5: experimental circuit of part 2

Table 2

Power supply (Volt)	Current (I) measured (mA)	Current (I) calculated (mA)	Error %
0			
2			
4			
6			
8			
10			

Discussion

1. Plot a graph from the measured data of table.1, to show the relationship between V & I_{measured} .
2. Plot a graph from the measured data of table.2, to show the relationship between V & I_{measured} .
3. Calculate the slope and resistance value from the graph above.
4. From the graphs above, is the relationship between V&I linear or non-linear?

Experiment No.3

Kirchhoff's laws

Object

To verify Kirchhoff's voltage and current laws for simple DC circuits.

Theory

- **Kirchhoff's current law**

States that the sum of the currents flowing into a junction or node is equal to the sum of the current 's flowing out of that junction.

$$\sum I_{leaving} = \sum I_{in}$$

Example: in the fig 1-a:

For node A $I_{A1} + I_{A2} + I_{A3} = 0$

For node B $I_{B1} + I_{B2} + I_{B3} = 0$

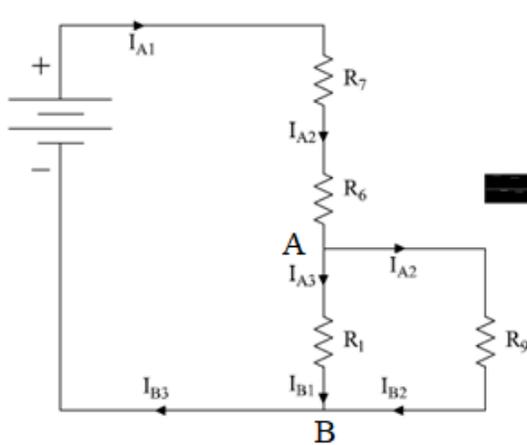


Fig.1a Circuit to study the 1st Kirchhoff law

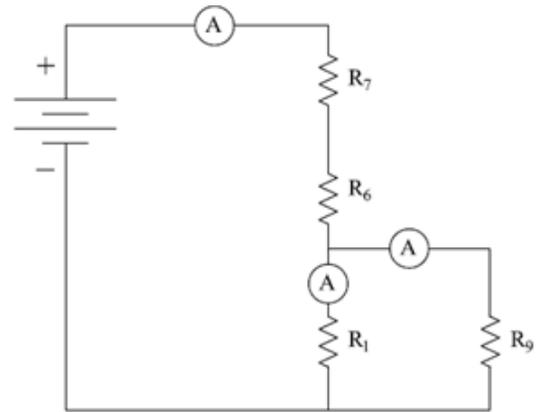


Fig.1b Circuit to study the 1st Kirchhoff law

- **Kirchhoff's voltage law**

State that the algebraic sum at all voltages around path or loop is zero.

Example

The circuit shown in Fig.2a. now with clockwise direction.

For Loop A

$$V_a + V_b + V_c + V_d = 0$$

For Loop B

$$V_e + V_d = 0$$

For Loop C

$$V_a + V_b + V_c + V_e = 0$$

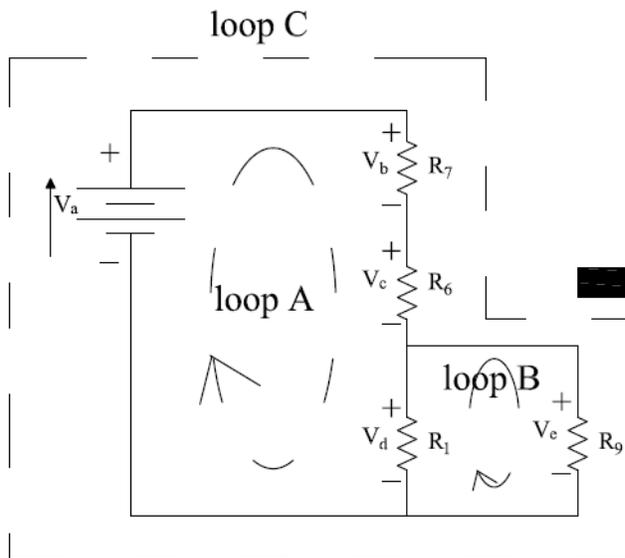


Fig.2a Circuit to study the 2^{ed} Kirchhoff law

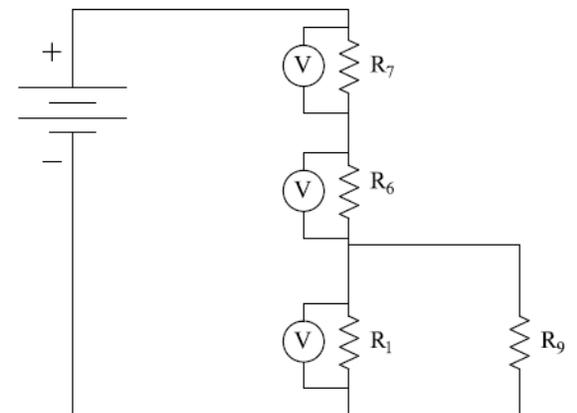


Fig.2b: Lab. Circuit for 2^{ed} Kirchhoff law

Procedures

• Part 1

1. Construct the circuit shown in Fig. 2b, and adjust the power supply voltage at (15V)
2. Measure the voltage across each resistor, take in consideration the polarity across each resistor. Record your measured result in the Third column of table (1). Determine the sum of these measured voltages. In
3. this stage Kirchhoff's voltage law should be stratified.

Table.1

Parameter	Calculated	Measured	Error %
Source	15V	15V
V_b			
V_c			
V_d			
Total			

• Part 2

1. Construct the circuit shown in Fig.1b and adjust power supply voltage at (10V)
2. Measure the three branch currents. Record your measured results in the third column Table (2). In Kirchhoff's current law should be satisfied.

Table. 2

Parameter	Calculated	Measured	Error %
I_{A1}			
I_{A2}			
I_{A3}			

Discussion

1. In your own words, state Kirchhoff's Voltage and Current law?
2. For the circuit shown in Fig.3. If $I_1=18$ mA and $I_4=4.5$ mA, then I_2 is:
a. 5.3mA b. 13.1mA c. 7.8mA d. 9 mA
3. If other resistors values are used for the circuit of Fig.2a in loop A, so that:

$V_b=0.54V$ and $V_d=3.58V$, Then V_c should be:

- a. 1016V b. 10.88V c. 107V d. 4.74V

4. The current i_o in the circuit Fig .4 is 4A find i_1, i_2, i_3, i_4 and i_g, V_1 and V_2 .

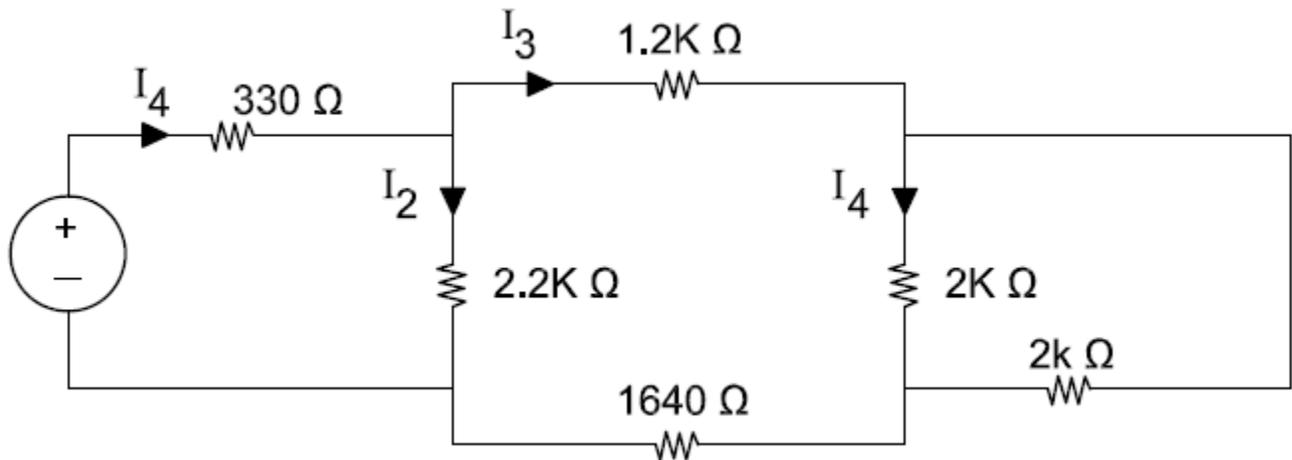


Fig.3

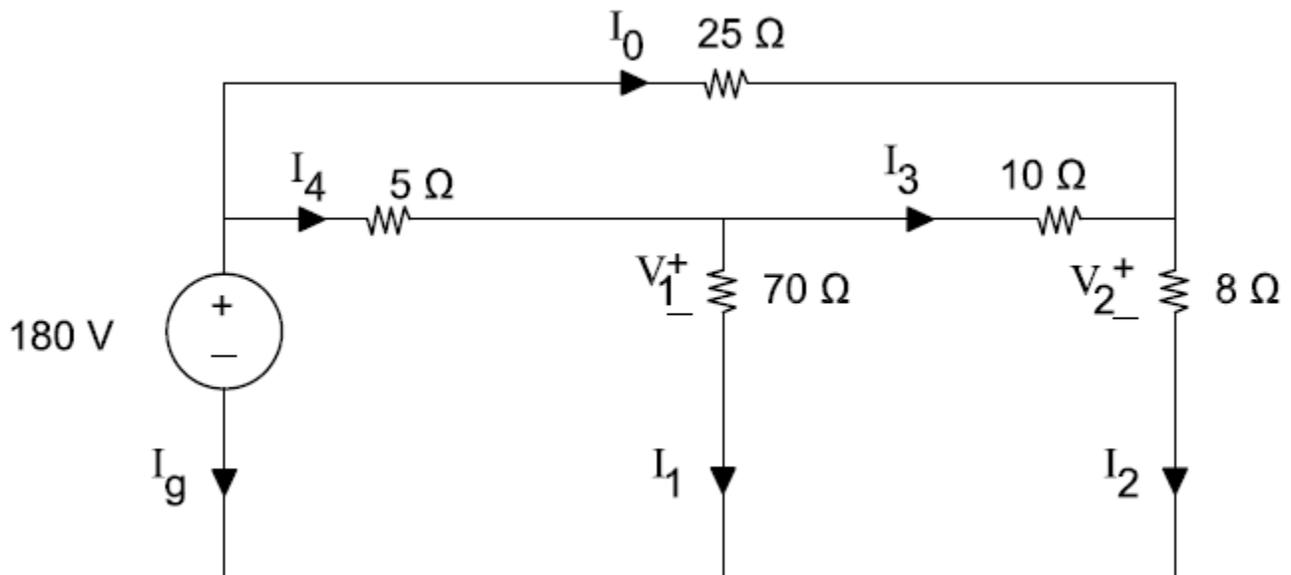


Fig.4

Experiment no. 4

Resistors in Series & Parallel

Object

To show you the difference between resistors in series and parallel.

Theory

For the purpose of the following discussion, assume that the terminals of each resistor are labelled Terminal 1 at one end and Terminal 2 at the other end.

A “series” connection is when Terminal 2 of one resistor is connected to Terminal 1 of the next resistor and so on. A power supply is connected between Terminal 1 of the first resistor in the circuit and Terminal 2 of the last resistor in the. the same electrical current (charge flow) flows through each of the resistors connected in series. It is important to note that in series connections, no other electrical connections can be made anywhere along the circuit to add more current or take some away. If extra connections are present, even though the resistors may appear to be in a circuit, our assumptions are invalid and the circuit is no longer a simple series combination. It is straightforward to show that resistances connected in series can be summed together to get the total resistance of the whole circuit. In other words:

$$R_{total} = R_1 + R_2 + R_3 + R_4 + \dots ..$$

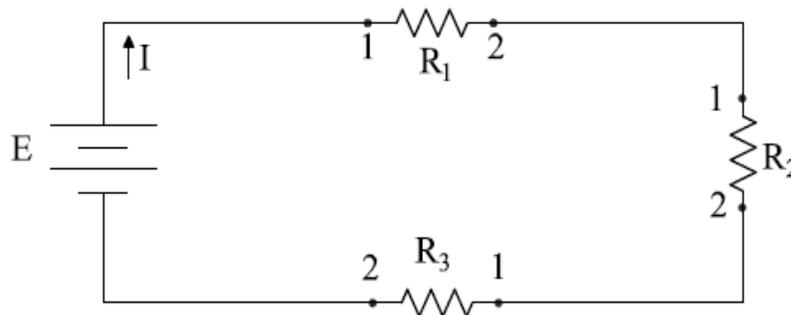


Fig1: series connection of resistors

A “parallel” connection is when all of the terminals “1” of several resistors are connected together. Likewise, all of the Terminals “2” are connected together. A power supply is then connected between the combined terminals 1 and the

combined terminals 2. In this case the applied voltage across each resistor is the same. Using this observation, it again is straightforward to show that the total resistance of such a parallel combination is:

$$\frac{1}{R_{total}} = \frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} + \frac{1}{R_4} + \dots \dots \dots$$

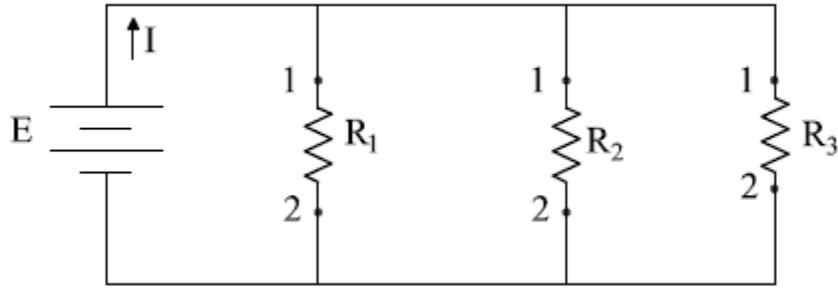


Fig2: parallel connection of resistors

Procedures

Part 1

1. Connect the circuit as in fig 3
2. Set the power supply at 20V
3. vary the value of potentiometer (200, 400,800, 1000, and 1200 Ω) and measure the values of current and voltages of each voltmeter for each step
4. Register the results in table 1

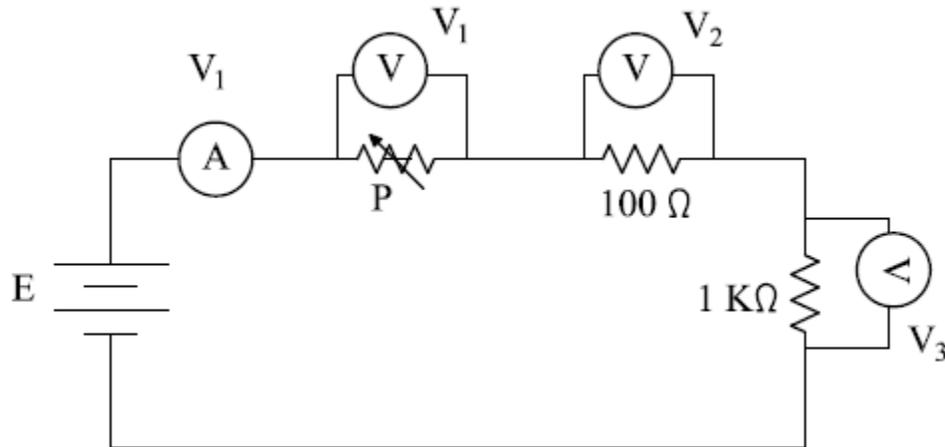


Fig 3: circuit of part 1

Table1

R (Ω)	I	V ₁	V ₂	V ₃
200				
400				
800				
1000				
1200				

Part 2

1. Connect the circuit as in fig 4
2. Set the power supply at 20V
3. Vary the value of potentiometer (200, 400, 800, 1000, and 1200 Ω) and measure the values of currents of each ammeter for each step
4. Register the results in table 2

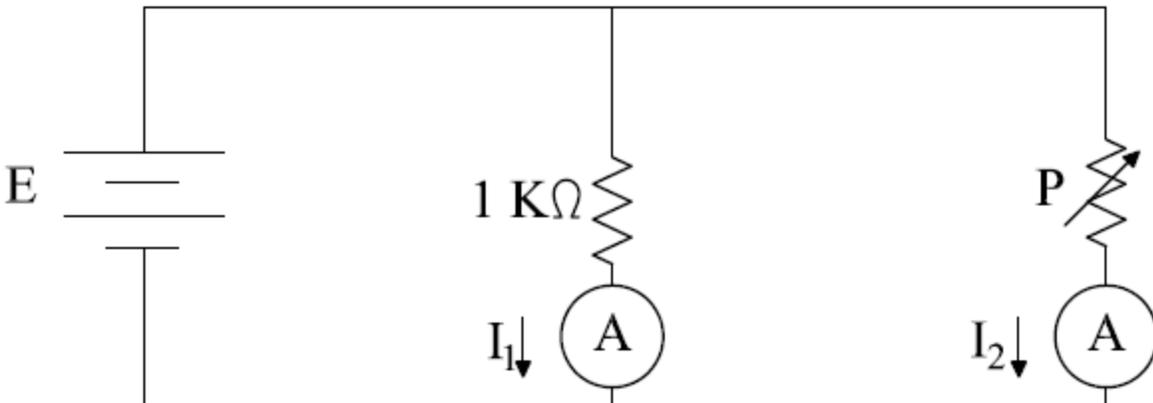


Fig4: circuit of part 2

Table 2

R (Ω)	I ₁ (measured)	I ₁ (calculated)	I ₂ (measured)	I ₂ (calculated)
200				
400				
800				
1000				
1200				

Discussion

1. In part 1: Are the voltages V_1 , V_2 and V_3 equal to each other? Why or why not?
2. In part 1: Calculate the total voltage $V = V_1 + V_2 + V_3$. Explain why it has the value which performed.
3. In part 2: Are I_1 and I_2 equal to each other? Explain.
4. In part 2: Compare I_{total} to the I_1 and I_2 .

Experiment No.5

Series-Parallel (Combination) Networks

Object

To become familiar with connect the series-parallel circuits and examine the series-parallel combination.

Theory

A series-parallel configuration is one that is formed by a combination of series and parallel elements Fig.1.

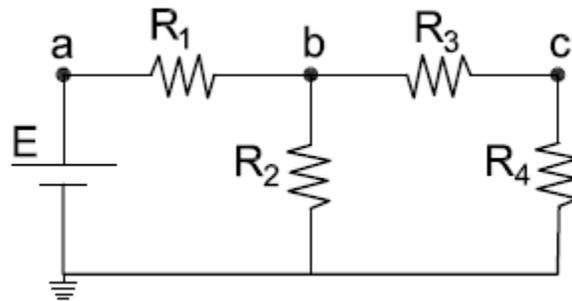


Fig.1

The network in Fig.1 is a series-parallel network. At first, you must be very careful to determine which elements are in series and which are in parallel. For instance, resistors R_1 and R_2 are not in series due to resistor R_3 connected to the common point b between R_1 and R_2 . Resistors R_2 and R_4 are not in parallel because they are not connected at both ends. They are separated at one end by resistor R_3 . Obvious If we look carefully enough at Fig.1, we do find that the two resistors R_3 and R_4 are in series because they share only point c , and no other element is connected to that point. The voltage source E and resistor R_1 are in series because they share point a , with no other elements connected to the same point. In the entire configuration, there are no two elements in parallel. However, resistors R_3 and R_4 can be combined to form a single resistor. The total resistance of the two is their sum as defined by series circuits. The resulting resistance is then in parallel with resistor R_2 , and they can be combined using the laws for parallel elements. So, the process steps: We are slowly reducing the network to one that will be represented by a single resistor equal to the total resistance “seen” from the source.

Equipments

- Several resistors.
- D.C power supply
- Multimeter.

Procedure

Connect the circuit shown in Fig.2.

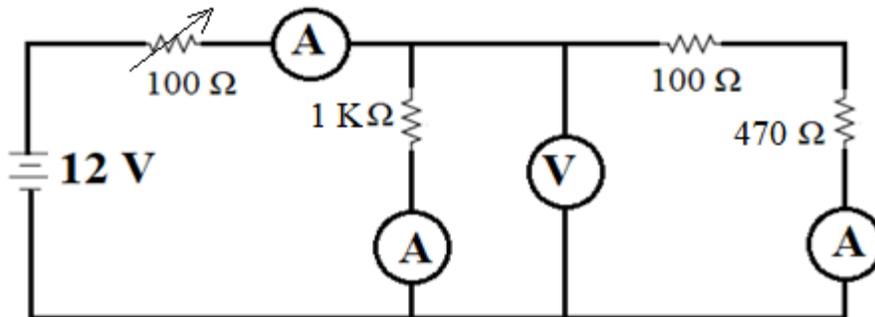


Fig.2

Part 1

Increase the applied voltage from the D.C power supply from (0 – 10) V, measure the current and voltage through the (R_3) resistor in each step and record the measured results in table.1.

Table 1

Power supply (volt)	I measured (mA)	V measured	I calculated (mA)	V calculated	Error %
0					
2					
4					
6					
8					
10					

Part 2

Set power supply at 24V. Increase the Value of R_1 . let R_2 100Ω and record the measured results in Table 2.

Table 2

Potentiometer R1 (Ohm)	I measured (mA)	V measured	I calculated (mA)	V calculated	Error %
200					
400					
800					
1000					
1500					
2000					

Part 3

Set power supply at 24V. Increase the Value of R_2 in same value of potentiometer R_1 as in table 2 and record the measured result in table 3. let R_1 100 Ω .

Table 3

Potentiometer R2 (Ohm)	I measured (mA)	V measured	I calculated (mA)	V calculated	Error %
200					
400					
800					
1000					
1500					
2000					

Discussion

1. What happened for the voltage and current in each branch when we Increase the applied voltage.
- 3 What happened for the voltage and current in each branch when we Increase the value of R_1
- 4 What happened for the voltage and current in each branch when we Increase the value of R_2
- 5 What is the difference between number (3) and number (4)?

Experiment No. 6

Delta – Star connection

Object

To study the properties of delta-star connection.

Theory

In solving networks (having considerable number of branches) by the application of Kirchhoff's Laws, one sometimes experiences great difficulty due to a large number of simultaneous equations that have to be solve. However, such complicated networks can simplify by successively replacing delta meshes by equivalent star systems and vice versa.

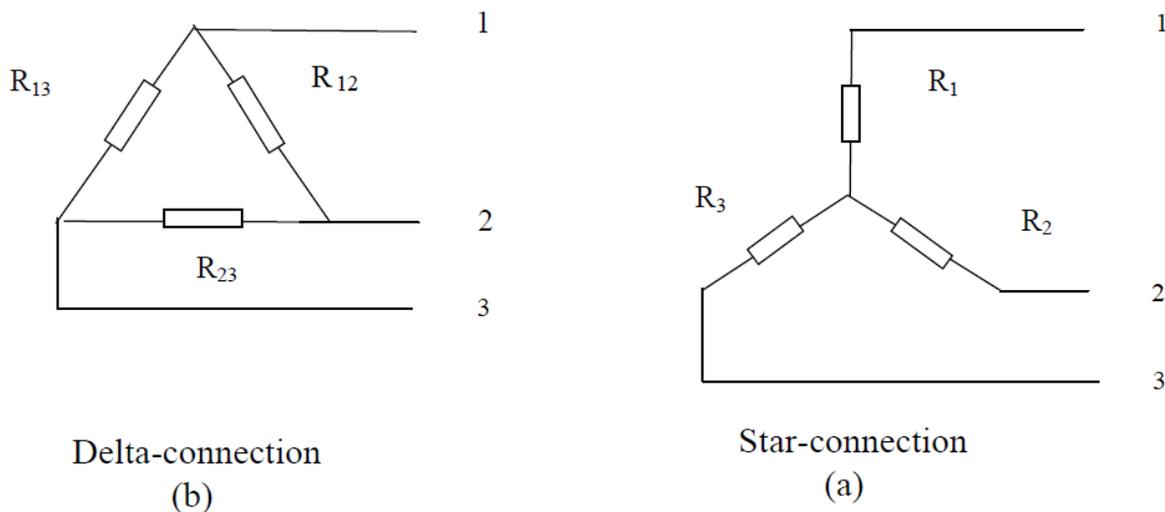


Fig 1: delta and star connection

Suppose we are given three resistance R_{12} , R_{23} and R_{13} connected in delta fashion between terminals 1,2 and 3 as in Fig.(1-a). So far as the respective terminals are concerned, these given three resistances can be replaced by the three resistances R_1 , R_2 and R_3 connected in star as shown in Fig.(1-b). These two arrangements will be electrically equivalent if the resistance as measured between any pair of terminals is the same in both arrangements.

1. To convert from delta connection to star connection

$$R_1 = \frac{R_{12} \times R_{13}}{R_{12} + R_{23} + R_{13}}$$

$$R_2 = \frac{R_{12} \times R_{23}}{R_{12} + R_{23} + R_{13}}$$

$$R_3 = \frac{R_{23} \times R_{13}}{R_{12} + R_{23} + R_{13}}$$

2. To convert from star connection to delta connection

$$R_{12} = R_1 + R_2 + \frac{R_1 \times R_2}{R_3}$$

$$R_{23} = R_2 + R_3 + \frac{R_3 \times R_2}{R_1}$$

$$R_{13} = R_1 + R_3 + \frac{R_1 \times R_3}{R_2}$$

Procedures

Part1

1. connect the circuit of fig. 2.

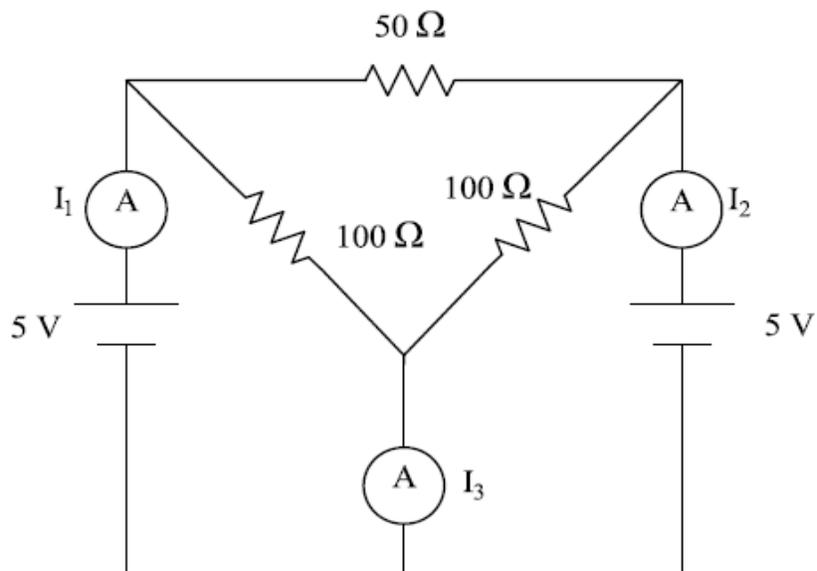


Fig 2 : experimental circuit of part 1

2. Measure " I_1, I_2, I_3 " practically.
3. Record your results in the table 1
4. By using delta-star conversion, find the star resistance R_1, R_2, R_3 theoretically.

Table 1

connection type	I_1	I_2	I_3
Part 1(delta)			
Part 2 (star)			

Part2

1. connect the circuit of fig 3

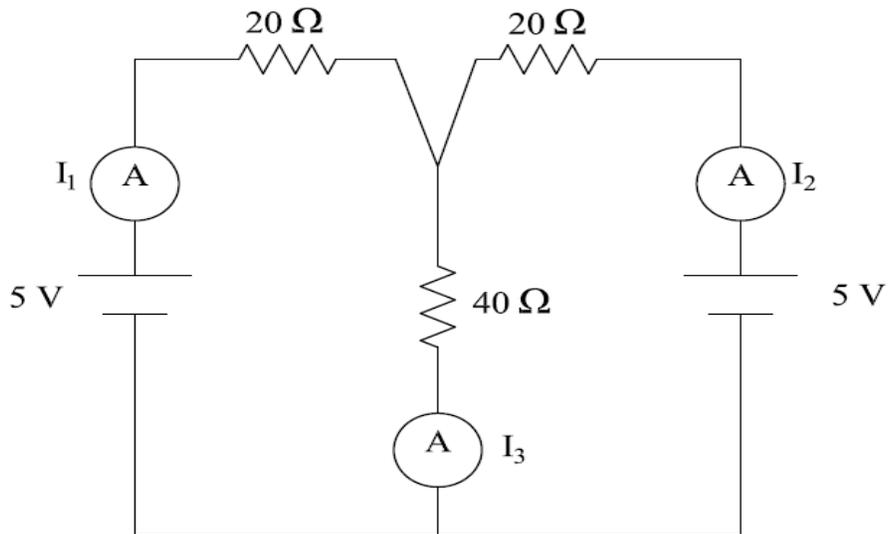


Fig 3 : experimental circuit of part 2

2. Measure " I_1, I_2, I_3 " practically.
3. Record your results in the table 1

Part 3

1. connect the circuit of fig 4
2. measure the equivalent resistance between any two terminals "1-2, 2-3, 1-3" by using AVO meter.

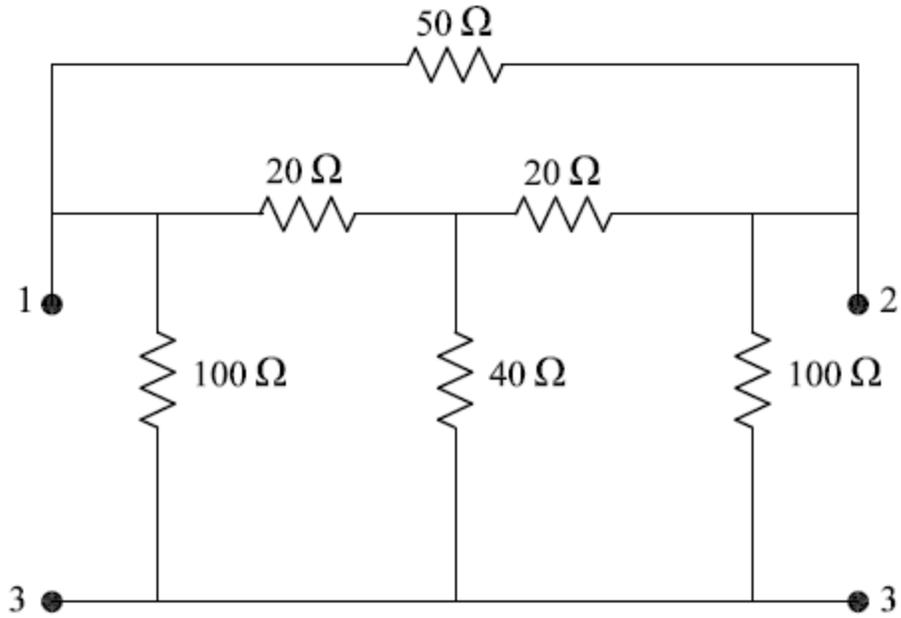


Fig 4 : experimental circuit of part 3

part 4

1. Connect circuit the circuit shown in Figure 5.
2. measure the equivalent resistance between any two terminals "1-2, 2-3, 1-3" by using AVO meter.

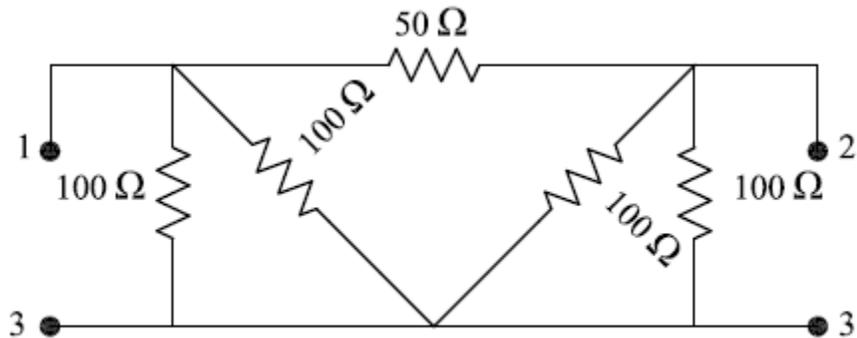


Fig 5 : experimental circuit of part 4

Discussion

1. Comment on your results.
2. Compare between the practical and theoretical results.
3. Comment on the results of part 3

Experiment No.7

Superposition Theorems

Object

To apply superposition theorem to a D.C electric circuit.

Theory

In any linear circuit containing multiple independent sources, the current through an element in this circuit (or voltage across) is the algebraic sum of the currents through (or voltages across) that element due to each independent source acting alone. So, if we have an electric circuit contains of (n – number) of sources then to calculate I_1 (for example) using superposition theorem, it will be $I_1 = (\pm I_{11}) + (\pm I_{12}) + (\pm I_{13}) + \dots \dots \dots + (\pm I_{1n})$ Where n = number of sources"

For example: the network shown in Figure (1) contains two voltage sources E_1 and E_2 . It is required to find the branch currents I_1 , I_2 and I_3 with the directions shown in Figure (1.a).

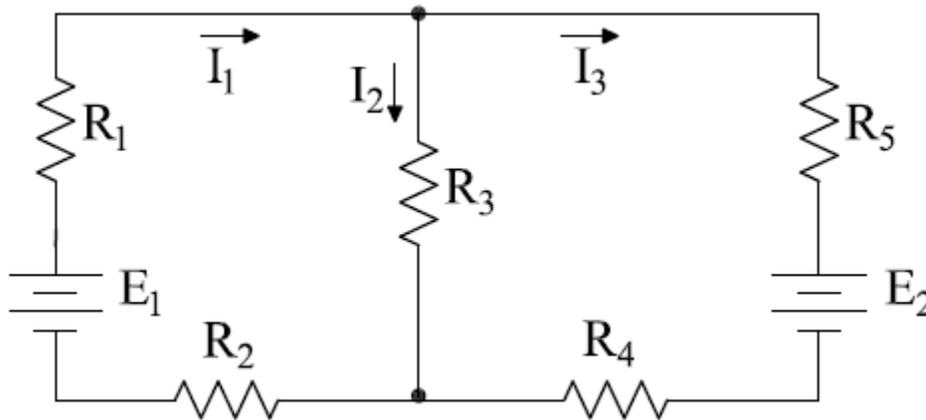


Fig.1.a

- 1- Deactivate all sources except one source. voltage source is replaced with a "Short Circuit" ($R = 0 \Omega$) and the current source is replaced with an open circuit ($R = \infty$). Then find the corresponding (I) due to that active source.

Figure (1.b) shows the branch currents I_1 , I_2 and I_3 when the voltage source E_1 is replaced by short circuit.

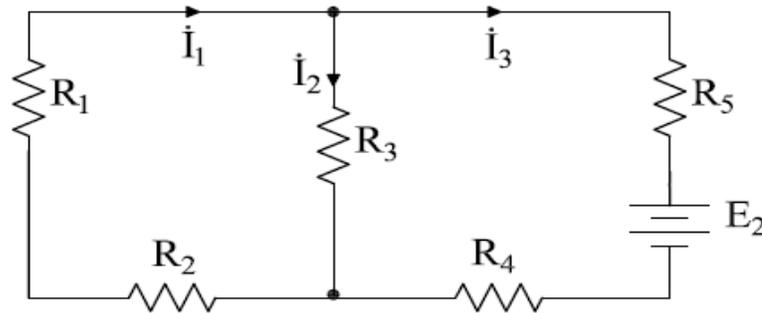


Fig.1.b

2- Repeat step (1) for each of the other sources.

Figure (1.c) shows the branch currents I_1 , I_2 and I_3 when the voltage source E_2 is replaced by short circuit.

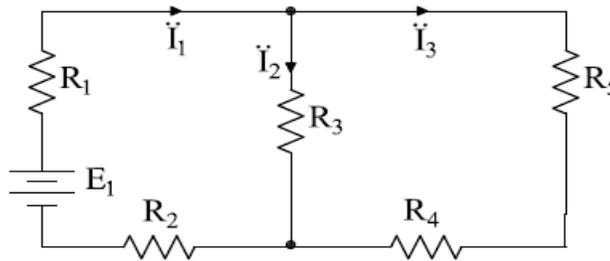


Fig.1.c

3- Find the total current by adding algebraically all the currents due to the active sources.

Now, according to the superposition theorem, if the branch currents according to the effect of each voltage source are in the same direction, then the resulted current is sum of two currents in the direction of another currents. But if the individual currents are in opposite direction through the same branch, the resulting currents is the difference of the two and has the direction of the larger.

So

$$\begin{aligned}
 I_1 &= \dot{I}_1 - \ddot{I}_1 \\
 I_2 &= \dot{I}_2 + \ddot{I}_2 \\
 I_3 &= \dot{I}_3 - \ddot{I}_3
 \end{aligned}$$

4- The above steps apply for currents and voltages

Equipments

1. Set of resistors
2. Two power supply units.
3. Voltmeter
4. Ammeter

Procedure

1. Connect the circuit as shown in Figure (2).
2. Measure the voltage across ($2.3\text{k}\Omega$, $1\text{k}\Omega$, and $2.2\text{k}\Omega$). resistors respectively, record your results in the second column of Table (1).
Note: consider the polarity of the voltages
3. Leave E_1 and replace E_2 by short circuit, repeat step 1 and record. your results in the third column of Table (1).
4. Leave E_2 and replace E_1 by short circuit, repeat step 1 and record your results in the fourth column of Table (1).
5. Find the value of V which is equal to ($\dot{V} = \ddot{V}$) record the results in the fifth column of Table (1) and compare them with the first column of Table (1).

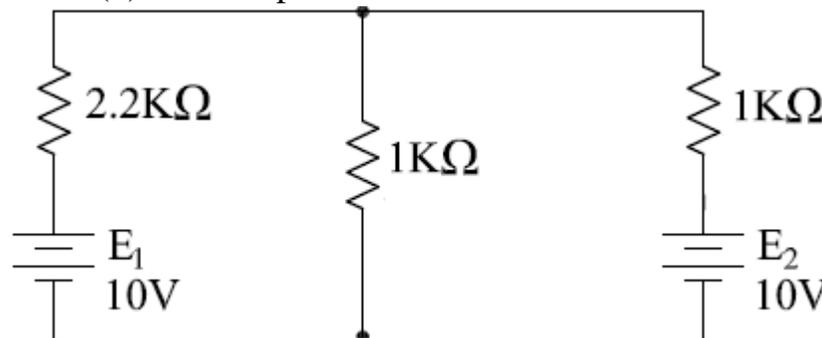


Fig.2

Table (1) Practical Results

R KΩ	(E1, E2) active (Volt)	E_1 active (Volt)	E_2 active (Volt)	$\dot{V} + \ddot{V}$ (Volt)
1				
1				
2.2				

Discussion

1. Calculate theoretically using superposition theorem the voltage across each resistor in Figure (2) record your results in Table (2).
2. Compare between the practical and theoretical results.
3. For the network shown in Figure (3), Find the voltage across the points A and B using superposition theorem.

Table (2) Theoretical Results

R Ω	E_1 active (Volt)	E_2 active (Volt)	$\dot{V} + \ddot{V}$ (Volt)
2.3k			
1k			
2.2k			

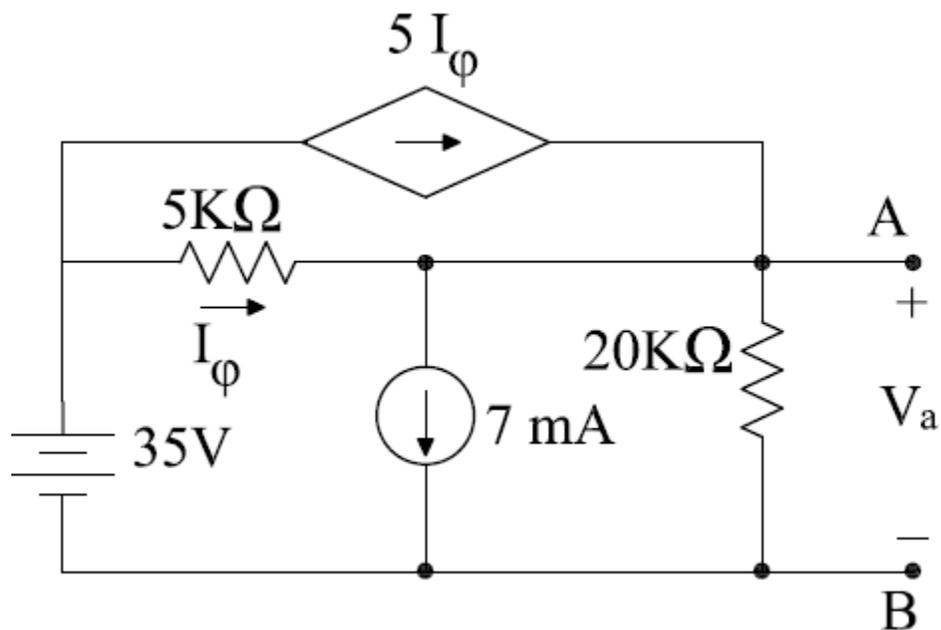


Fig.3

Experiment No.8

Thevenin's Theorems

Object

To study and apply Thevenin theorem to Eclectic circuit.

Theory

Thevenin's theorem states the following: “Any Two-terminals of linear D.C network can be replaced by a single voltage source E_{th} With a series resistance (R_{th}). It makes the solution of a complicated electric network quite quick and easy. The application of this theorem will be explained with the help of the following simple example.

If it is required to find the current following through load resistance R_L as shown in Fig.1(a), the following steps will be performed:

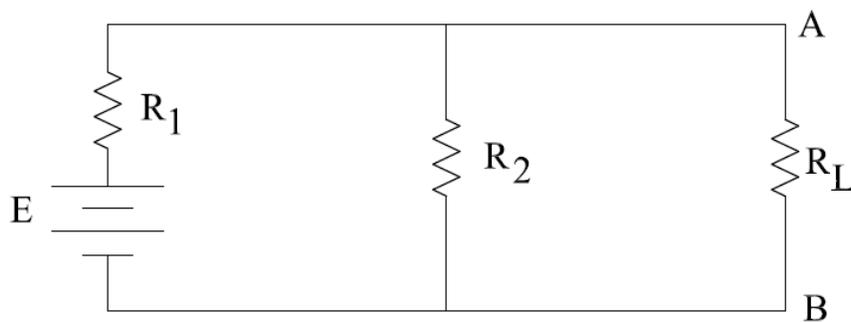


Fig.1(a) The active network with R_L

1. Remove R_L from the circuit terminals A and B and redraw the circuit as shown in Fig1(b). Observably the terminal A-B has become an Open circuit.
2. Calculate the open circuit voltage ($V_{oc} = E_{th}$) which viewed from terminals A and B.

As shown in Fig.1(b), $V_{oc} = E_{th} =$ voltage drop across R_2

Where:

$$V_{R2} = I R_2 \quad \text{where } (I) \text{ is the circuit current}$$

$$I = E / (R_1 + R_2)$$

$$V_{R2} = V_{oc} = E_{th} = I R_2 = E * R_2 / (R_1 + R_2)$$

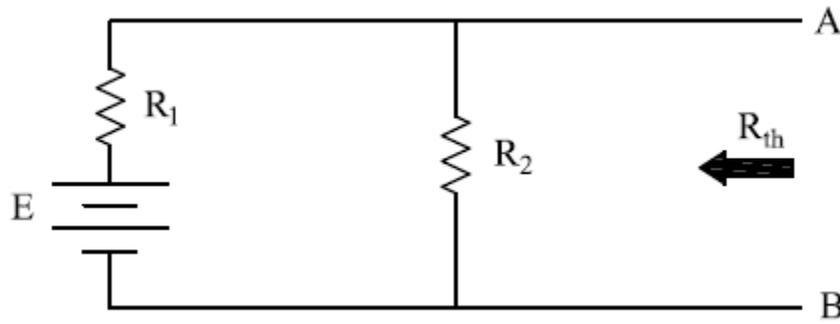


Fig.1(b) The equivalent circuit according to the Thevenin Theorem

3. Replace the voltage source with short circuit and redraw circuit as shown in Fig.1(c). When viewed from terminals A and B, the circuit will consist of two parallel resistances (R_1 and R_2). The equivalent resistance of the network is called THEVENIN RESISTANCE (R_{th}) or open circuit resistance (R_0).

$$R_{th} = R_1 \parallel R_2 = (R_1 * R_2) / (R_1 + R_2)$$

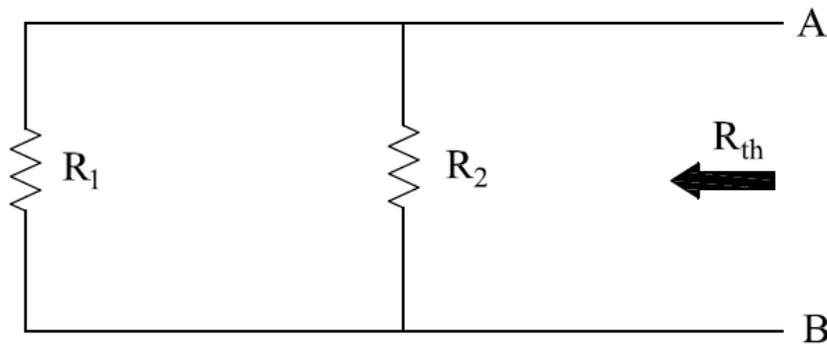


Fig.1(c) The equivalent circuit according to the Thevenin Theorem

4. R_L is now connected across terminals A and B from where it was temporarily removed earlier. As shown in Fig.2, the current flowing through R_L is given by: $I = E_{th} / (R_{th} + R_L)$

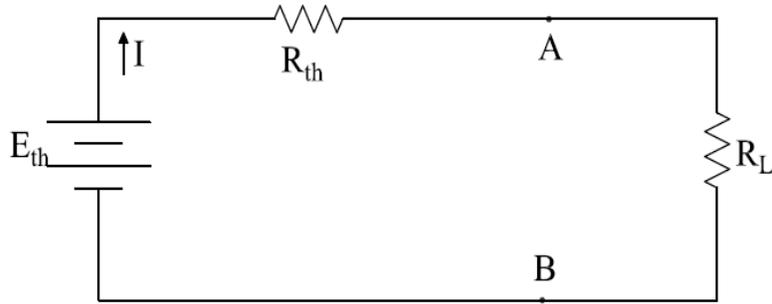


Fig.2 The equivalent Thevenin circuit with R_L

Procedure

1. Connect the circuit as shown in Fig.3

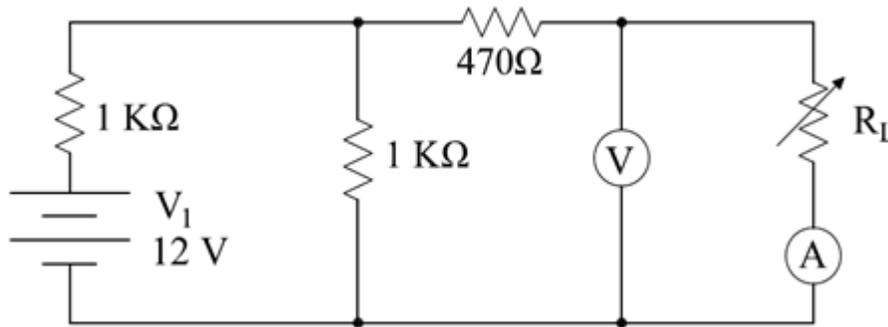


Fig.3

2. Vary R_L resistance as shown in Table (1), measured I_L and V_L in each step. Record your results in the second and third column of Table (1).
3. Disconnect R_L , then measure the open loop voltage (E_{th}).
4. measure R_{th} practically and connect Thevenin equivalent circuit as shown in Figure (4).

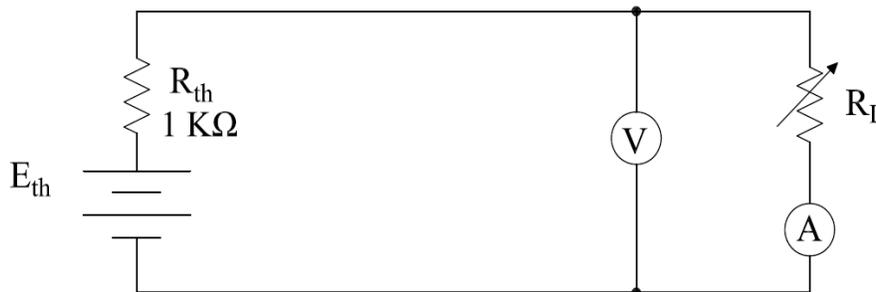


Fig.4

5. Repeat step (2) and record your results in the fourth and fifth column of Table (1).

Table.1 Practical Result

R_L Ω	Fig.3		Fig.4	
	I_L (mA)	V_L (Volt)	I_L (mA)	V_L (Volt)
400				
500				
600				
700				
800				

Table.2 Theoretical Result

R_L Ω	Fig.3		Fig.4	
	I_L (mA)	V_L (Volt)	I_L (mA)	V_L (Volt)
400				
500				
600				
700				
800				

Discussion

1. Calculate I_L and V_L theoretically from Fig.3 and Fig.4 then record your results in Table (2).
2. Compare briefly between the practical and theoretical results.
3. Using Thevenin's Theorem, determine the voltage across R_L , of the network shown in Fig.5

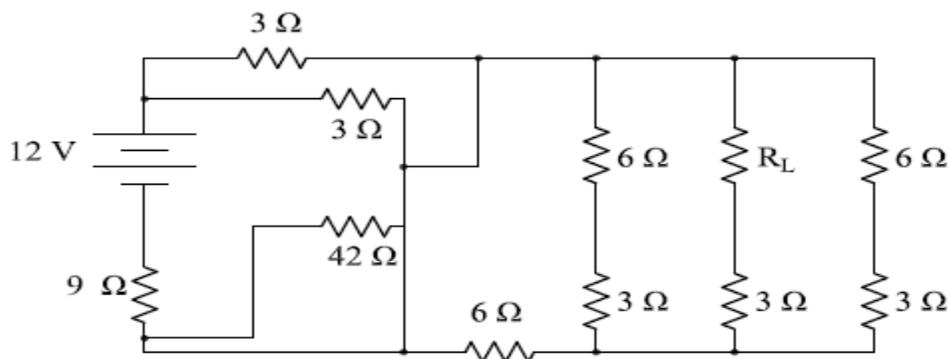


Fig.5

Experiment No.9

Norton's Theorem

Object

To prove Norton's theorem practically.

Theory

Norton's theorem states the following: "Any two terminal linear D.C network can be replaced by any equivalent circuit consisting of a constant current source (I_N) and a parallel resistance (R_N), as shown in Fig.1(a).

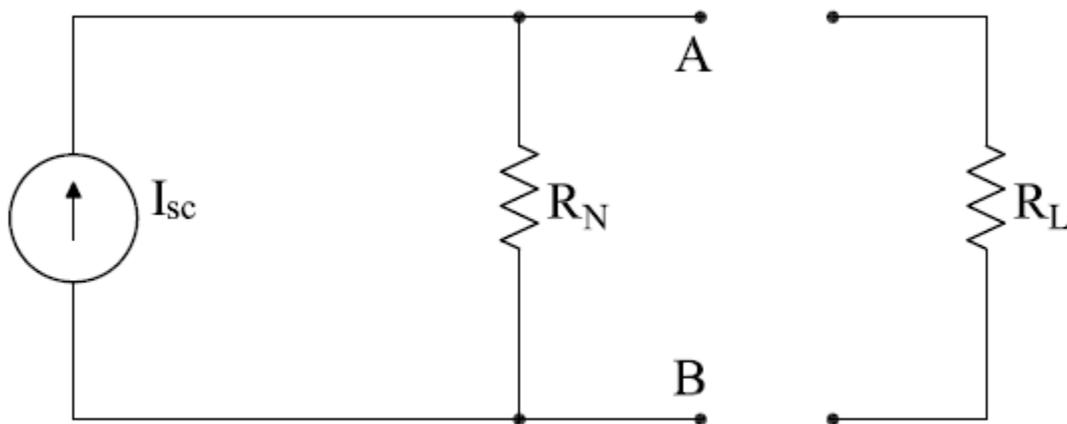


Fig.1(a) Equivalent circuit according to Norton theorem

The constant current is equal to the current which would flow in a short circuit placed across the terminals A and B as shown in Fig.1(b), and is called ($I_{SC} = I_N$).

Where:

I_{SC} : is the short circuit current.

I_N : is the Norton current.

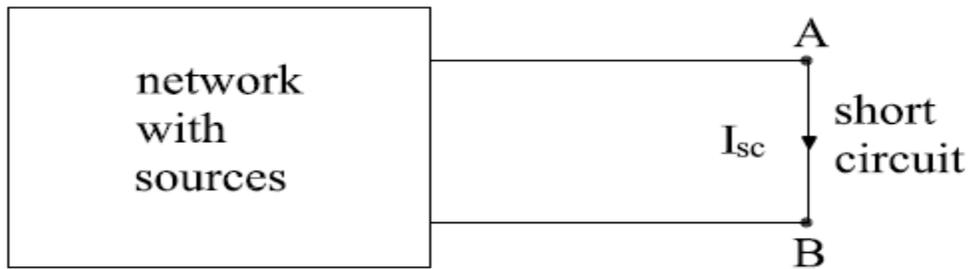


Fig.1(b) Calculating the constant current source $I_{SC} = I_N$

The parallel resistance is the resistance of the network when viewed from A-B open terminals after all voltage and current has been removed and replaced by short or open circuits respectively, as shown in Fig.1(c)

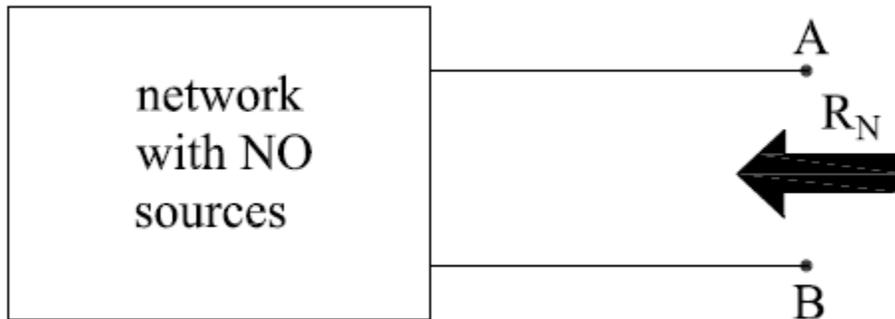


Fig.1(c): Calculating the equivalent parallel resistance R_N

Then according to Fig.2, the current through R_L (when R_L is connected to Norton equivalent circuit) will be:

$$I_L = I_N * R_N / (R_N + R_L)$$

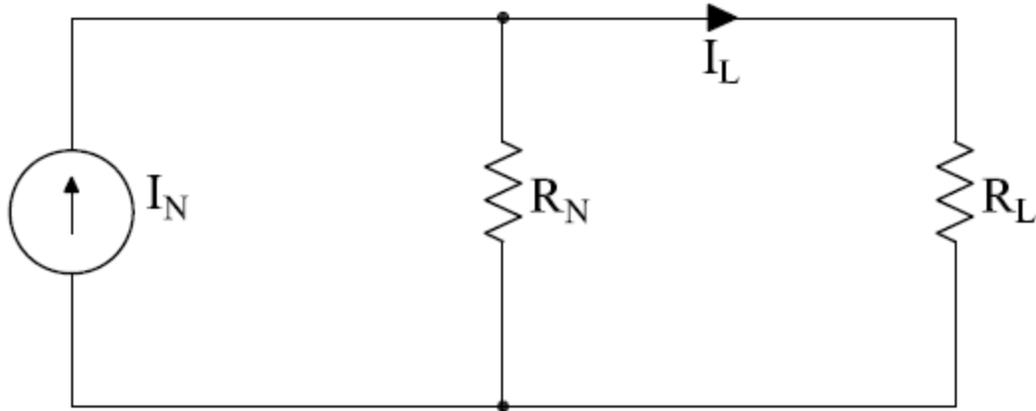


Fig.2 The equivalent Norton circuit with R_L

Equipments

- Power supply
- AVO meter
- Four resistors

Procedures

1. Connect the circuit as shown in Fig.3.a

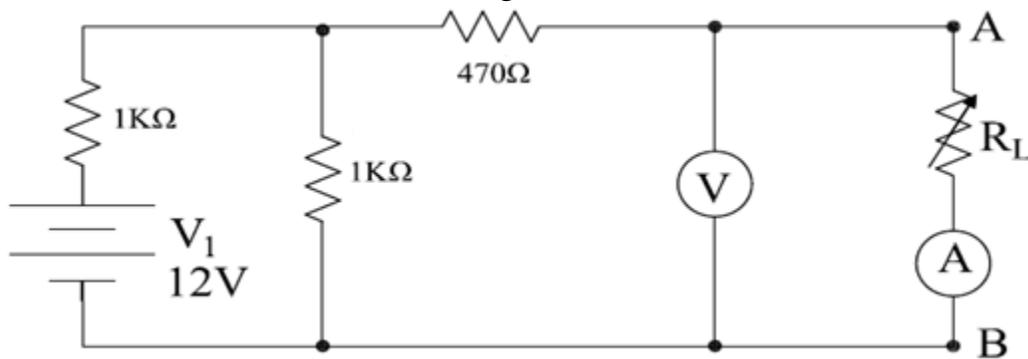


Fig.3.a: first lab. circuit

2. Vary R_L resistance as the values in first column of table (1), measure I_L and V_L in each step, Record your results in the second and third column.

Table.1 Practical Result

R_L Ω	Fig.3.a		Fig.3.b	
	I_L (mA)	V_L (Volt)	I_L (mA)	V_L (Volt)
400				
500				
600				
700				
800				

3. Disconnect R_L , then measure the short circuit current (I_{SC}) between A and B terminals.
4. measure R_N practically and connect Norton equivalent circuit as shown in Fig.3.b.

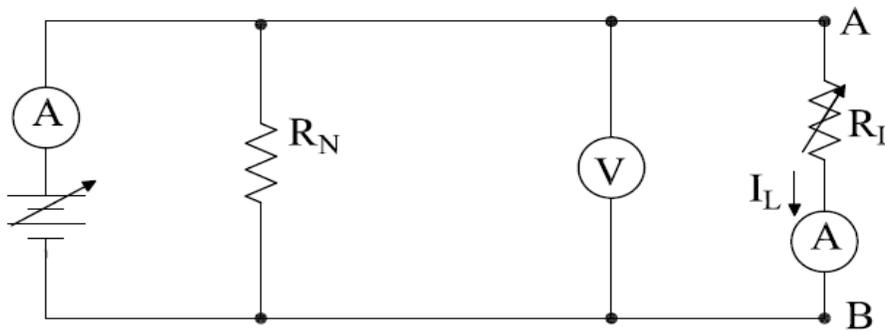


Fig.3.b: second lab. circuit

5. Repeat step (2) (Make sure that the constant current source is remains constant in each step of values of R_L , by means of varying the D.C power supply). record your results in the fourth and fifth column of Table (1).

Discussion

1. Calculate I_{total} and V_i theoretically from Fig.3.a and Fig.3.b, then record your results in Table (2)
2. Compare briefly between the practical and theoretical results.
3. Find the voltage between the open terminals A and B for the network shown in Fig.4 using Norton theorem. And the value of R_L is one half the value of R_N , find the current through R_L

Table 2: theoretical Results

R_L Ω	Fig.3.a		Fig.3.b	
	I_L (mA)	V_L (Volt)	I_L (mA)	V_L (Volt)
400				
500				
600				
700				
800				

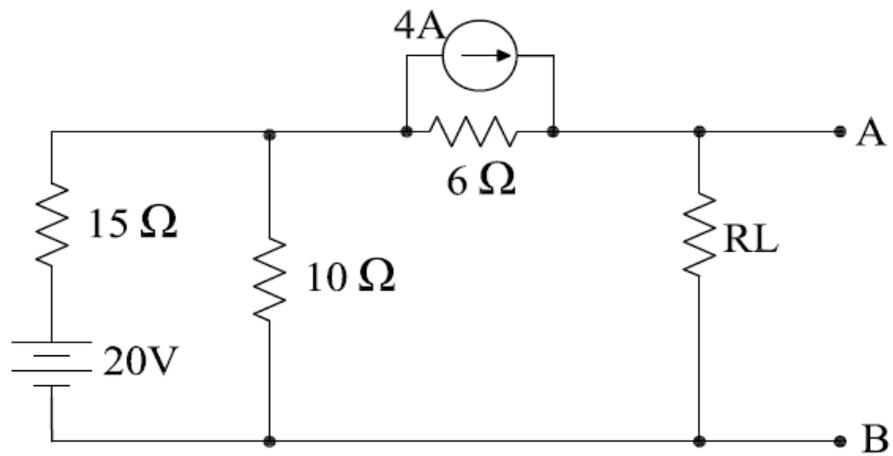


Fig.4

Experiment No.10

Impedance Elements Characteristics

Object

in this experiment we shall investigate the behavior of resistive, inductive, and capacitive components when operated in AC circuits

Theory

The study of alternating current (AC) is very important as it has practical applications in our daily lives. As the name implies, the current and voltage change directions periodically at a fixed frequency. In the Iraq, this frequency is usually 50 Hz. The alternating current is transmitted at high voltage and low ampere on the main transmission lines so that power loss may be kept to a minimum. The voltage is stepped down several times before it enters your home. Most electric appliances need a voltage of 220 V, though some appliances like heating and air conditioning equipment, water heaters, and ovens. In this lab you will analyze the relationship between current and voltage when the elements (resistance, inductance, and capacitance) are connected in a circuit with an ac source of emf. The symbol for an AC voltage source is shown in fig.1.



fig.1.: The symbol for an AC voltage source

The voltage of an AC source varies sinusoidally with time or with a frequency f given by

$$V(t) = V_0 \cos(\omega t) \dots\dots\dots(1)$$

$$\omega = 2\pi f$$

where V_0 : is the maximum or peak voltage

ω : the angular frequency measured in radians/second.

t : the time.

The maximum voltage varies between V_0 and $-V_0$. A graph of voltage as a function of time is shown in Figure 2.

(The current delivered by an AC source is also sinusoidal).

$$I(t) = I_0 \cos(2\pi ft) \dots \dots \dots (2)$$

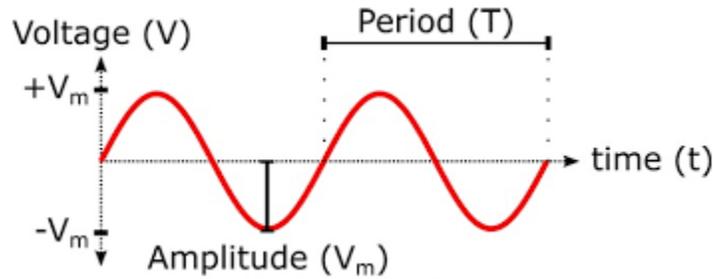


Figure 2. Sinusoidal voltage source

• **Purely Resistive load**

Consider a purely resistive circuit with a resistor connected to an AC generator, as shown in Figure 3 a.

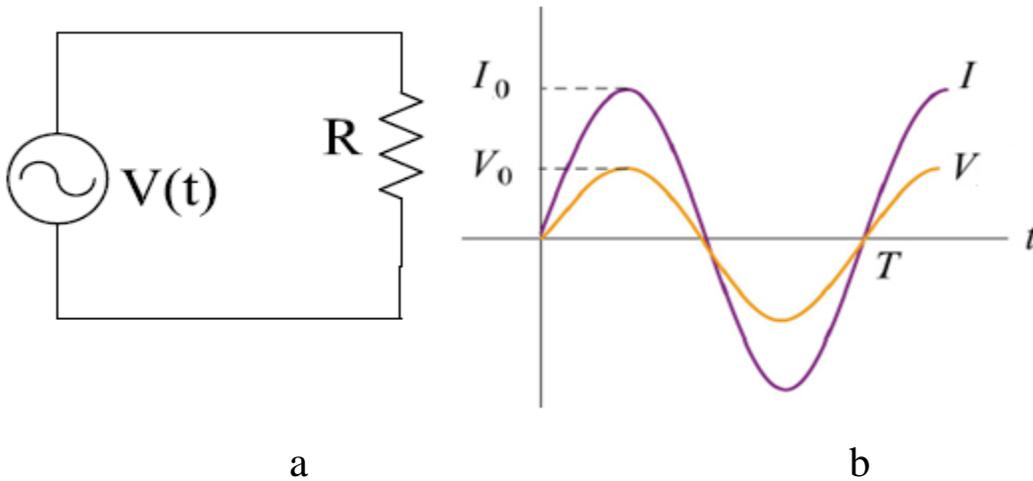


Figure 3: Current and voltage in a resistor connected to an AC source.

The voltage across the resistor obeys Ohm's Law.

$$V = IR = I_0 \cos(2\pi ft)R = V_0 \cos(2\pi ft) \dots \dots \dots (3)$$

where the peak voltage across the resistor is

$$V_0 = I_0R.$$

Fig. 3(b) shows the plots of current and voltage as a function of time t . Note that both the current and voltage reach peak values at the same instant of time and they are both zero at the same instant of time. Therefore, we say that the current and voltage are in phase or the phase difference is zero.

• **Purely Inductive Load**

Consider now a purely inductive circuit with an inductor connected to an AC source, as shown in Fig. 4(a).

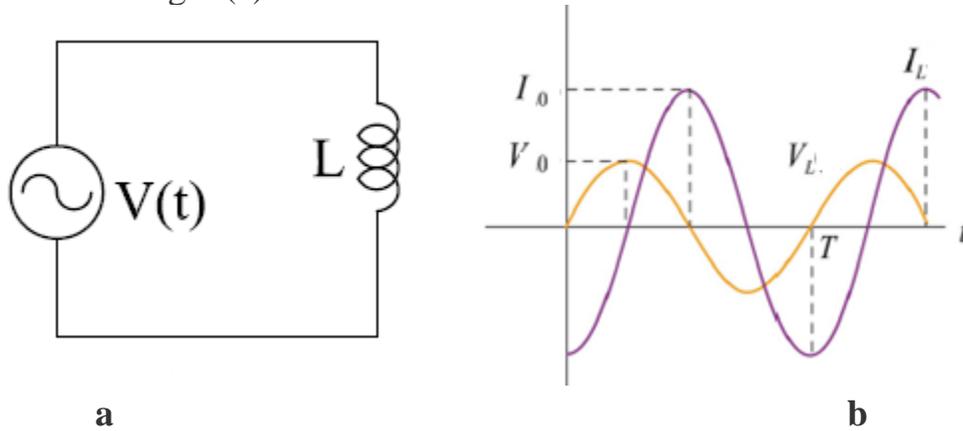


Figure 4: Current and voltage in an inductor connected to an AC source.

the voltage across the inductor is given by

$$V_L = \frac{dI}{dt} \dots\dots\dots(4)$$

Substitute eq.(2)in (4)

$$V_L = -V_0 \cos(2\pi ft) \dots\dots\dots(5)$$

Fig. 4(b) shows the plots of current and voltage as a function of time t . In this case, when the voltage is positive maximum, the current is zero and in the process of increasing. When the voltage is negative maximum, the current is decreasing. The current reaches its maximum value 1/4 cycle after the voltage reaches its maximum. Therefore, we say that the voltage leads the current by 90° or the current lags behind the voltage by 90° .

Eq. (5) can be written as

$$V_L = V_0 \cos(2\pi ft + \varphi) \dots\dots\dots(6)$$

where the phase difference $\phi = \pi/2$.

The current and voltage are related by an equation similar to Ohm's Law with

$$V_L = I X_L \dots\dots\dots(7)$$

where X_L is known as the **inductive reactance**, is measured in units of ohms, and is given by

$$X_L = 2\pi fL = \omega L \dots\dots\dots(8)$$

• **Purely Capacitive Load**

In the purely capacitive case, both resistance R and inductance L are zero. The circuit diagram is shown in Figure 5 a.

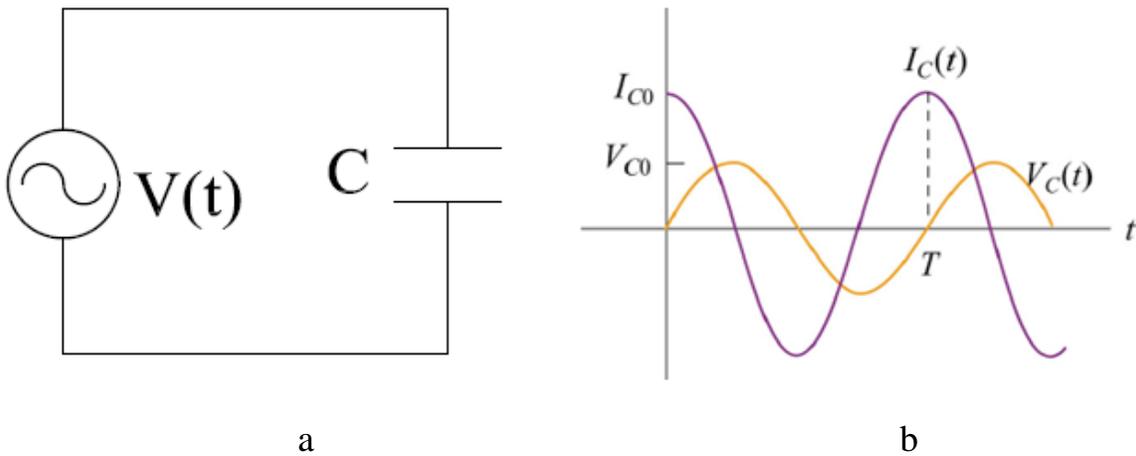


Figure 5: Current and voltage in a capacitor connected to an AC source

the voltage across the capacitor is given by

$$V_C = \frac{Q}{C} \dots\dots\dots(9)$$

Where, Q = charge,

$$I = \frac{dQ}{dt} \dots\dots\dots(10)$$

Substitute eq.(1) and (9) in (10)

$$V_C = V_0 \sin(2\pi ft) \dots\dots\dots(11)$$

Fig. 4(b) shows the plots of current and voltage as a function of time t . In this case, the current reaches its maximum value 1/4 cycle before the voltage reaches its

maximum. Therefore, we say that the voltage lags behind the current by 90° or the current leads the voltage by 90° . The voltage can be written as

$$V_C = V_0 \cos(2\pi ft - \varphi) \dots \dots \dots (12)$$

where the phase difference $\varphi = \pi/2$.

The current and voltage are related by an equation similar to Ohm's Law with

$$V_C = I X_C \dots \dots \dots (13)$$

where X_C is known as the capacitive reactance, measured by ohm and given by

$$X_C = \frac{1}{2\pi f c} = \frac{1}{\omega c} \dots \dots \dots (14)$$

Procedures

Part 1:

- 1- Connect the circuit in fig 1
- 2- Adjust the frequency of the signal generators to the values as in table 1
- 3- The amplitude voltage shall be kept at 2Vp-p, re-adjusted at each step if needed.
- 4- Measure the voltage and current at each step.
- 5- Graph the relationship between R and frequency as in fig 4a.

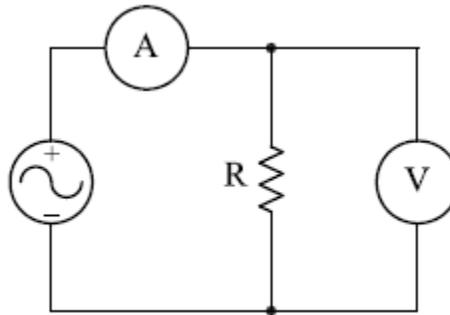


Fig1: experimental circuit of part 1

Table 1: results of part 1,2,3

Frequency (Hz)	R (ohm)	Voltage (V)	Current (A)
50			
100			
200			
400			
800			
1600			
3200			
6400			

Part 2:

- 1- Connect the circuit in fig 2
- 2- Repeat the steps as in 1 (replace R in table 1 by X_L)
- 3- Graph the relationship between X_L and frequency as in fig 4b.

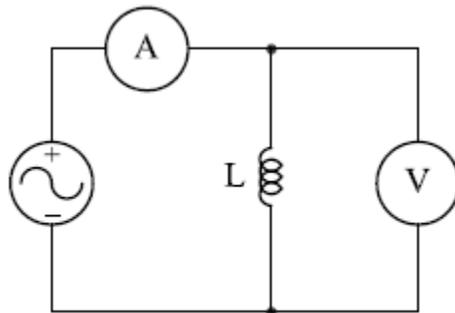


Fig2: experimental circuit of part 2

Part 3 :

- 1- Connect the circuit in fig 3
- 2- Repeat the steps as in 1 (replace R in table 1 by X_C)
- 3- Graph the relationship between X_C and frequency as in fig 4c.

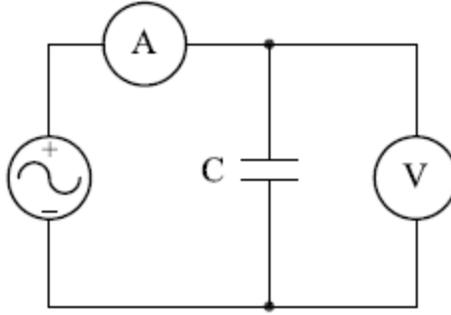


Fig3: experimental circuit of part 3

Discussion:

- 1- From your graphs, read the values of X_L and X_c at selected frequency values, calculate the values of L and C using the following formula:

$$X_L = j\omega L, \quad X_C = \frac{1}{j\omega c}$$

where $\omega = 2\pi f$ and $j = \sqrt{-1}$

- 2- match the values obtained for L and C from measurements of X_L and X_c define performed at different frequencies.

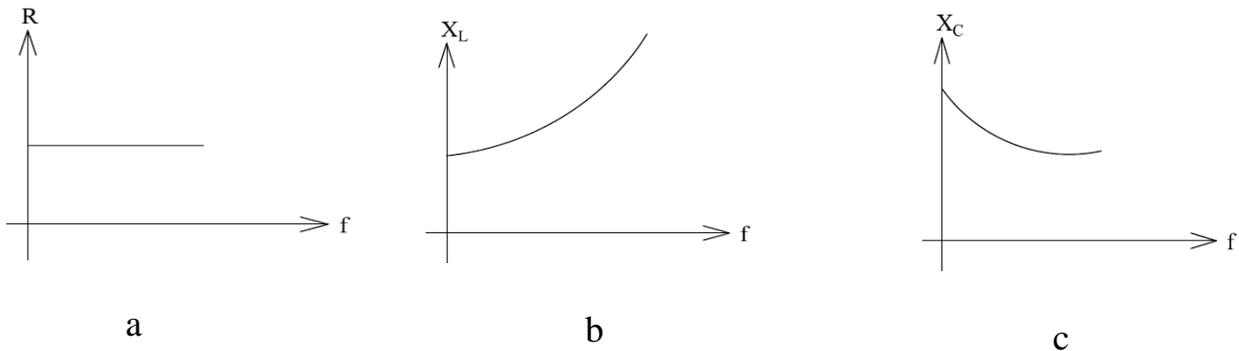


fig 4: relationship between frequency and (a: resistor, b: X_L , c: X_c)

Experiment No.11

RL and RC Circuits

Object

in this experiment we shall investigate the behavior of RL and RC circuits when operated in AC circuits

Theory

1- RL series circuit

A circuit that contains a pure resistance R ohm connected in series with a coil having a pure inductance of L (Henry) is known as **RL Series Circuit**. When an AC supply voltage V is applied, the current, I flows in the circuit.

So, I_R and I_L will be the current flowing in the resistor and inductor respectively, but the amount of current flowing through both the elements will be same as they are connected in series with each other. The circuit diagram of RL Series Circuit is shown in fig 1:

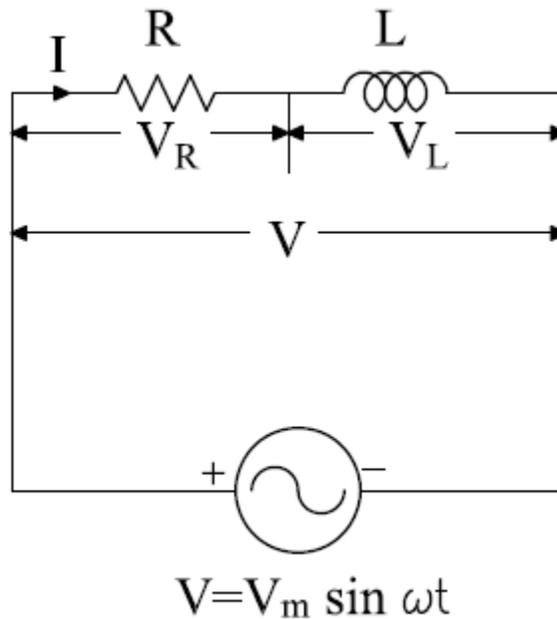


Fig 1: RL series circuit

The phasor diagram of the RL Series circuit is shown in fig 2:

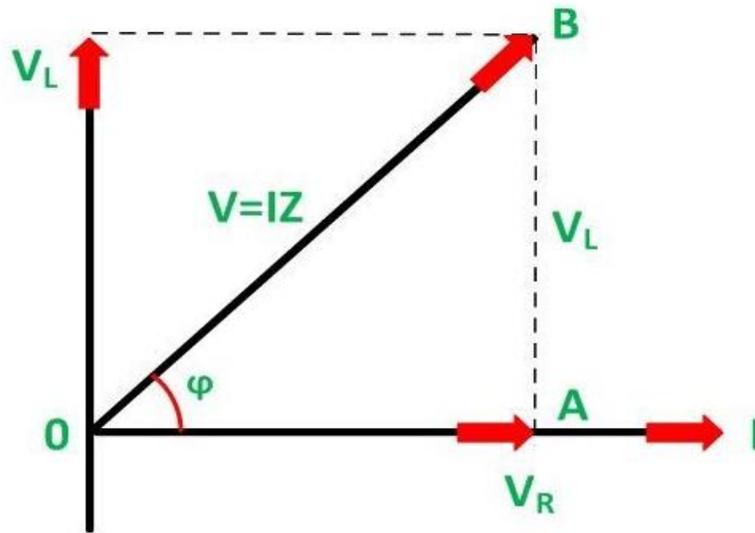


Fig 2: The phasor diagram of the RL Series circuit in right-angle triangle OAB

$$V_R = I_R \quad \text{and} \quad V_L = IX_L \quad \text{where} \quad X_L = 2\pi fL$$

$$V = \sqrt{(V_R)^2 + (V_L)^2} = \sqrt{(IR)^2 + (IX_L)^2}$$

$$V = I\sqrt{R^2 + X_L^2}$$

OR

$$I = \frac{V}{Z}$$

$$Z = \sqrt{R^2 + X_L^2}$$

Where,

Z is the total opposition offered to the flow of alternating current by an RL Series circuit and is called impedance of the circuit. It is measured in ohms (Ω).

In RL Series circuit the current lags the voltage by 90 degrees angle known as phase angle. It is given by the equation:

$$\tan\phi = \frac{V_L}{V_R} = \frac{IX_L}{IR} = \frac{X_L}{R}$$

OR

$$\phi = \tan^{-1} \frac{X_L}{R}$$

The waveform of the RL series circuit is shown below:

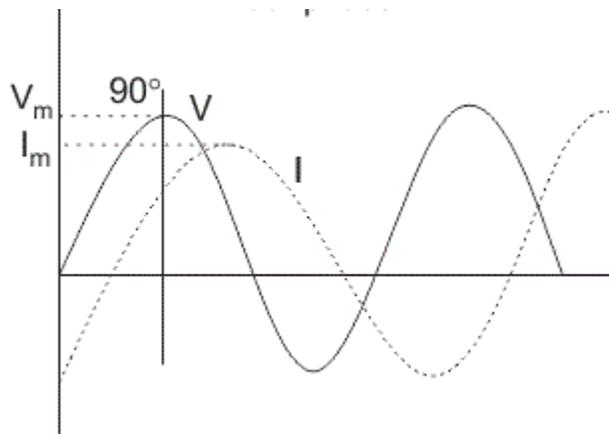
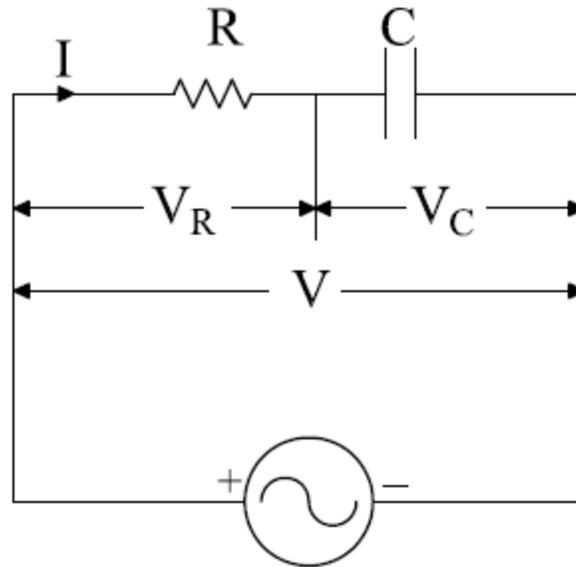


Fig 3: The waveform of the RL series circuit

2- RL series circuit

A circuit that contains pure resistance R ohms connected in series with a pure capacitor of capacitance C farads is known as **RC Series Circuit**. A sinusoidal voltage is applied and current I flow through the resistance (R) and the capacitance (C) of the circuit.

The RC Series circuit is shown in the figure 4:



$$V = V_m \sin \omega t$$

Fig 4: RC series circuit

The phasor diagram of the RC series circuit is shown in fig 5:

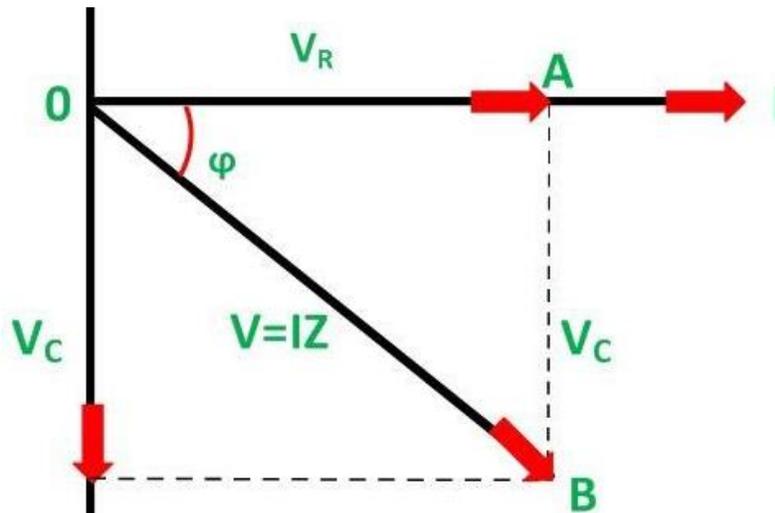


Fig 5: The phasor diagram of the RC Series circuit

$$V_R = I_R \quad \text{and} \quad V_C = IX_C \quad \text{where} \quad X_C = 1/2\pi fC$$

In right triangle OAB

$$V = \sqrt{(V_R)^2 + (V_C)^2} = \sqrt{(IR)^2 + (IX_C)^2}$$

$$V = I\sqrt{R^2 + X_C^2}$$

OR

$$I = \frac{V}{\sqrt{R^2 + X_C^2}} = \frac{V}{Z}$$

where

$$Z = \sqrt{R^2 + X_C^2}$$

Z is the total opposition offered to the flow of alternating current by an RC series circuit and is called **impedance** of the circuit. It is measured in ohms (Ω).

From the phasor diagram shown above, it is clear that the current in the circuit leads the applied voltage by an angle ϕ and this angle is called the **phase angle**.

$$\tan\phi = \frac{V_C}{V_R} = \frac{IX_C}{IR} = \frac{X_C}{R}$$

OR

$$\phi = \tan^{-1} \frac{X_C}{R}$$

The waveform and power curve of the RC circuit is shown in fig 6:

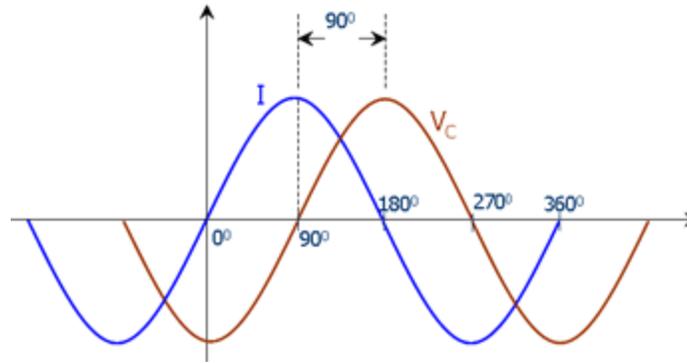


Fig 6: The waveform of the RC series circuit

3- Parallel RC Circuits

When R and C are in parallel, each branch has its own value of current depending upon the branch value of R or X_c . Also, because they are in parallel, the voltage (magnitude and phase) across the parallel branches is the same. Therefore, there is no phase difference between the applied voltage and the voltage across R and C in parallel. The current through the resistive branch is in phase with the applied signal. But the current through the capacitive branch leads its voltage V_c by 90 degrees. Therefore, the two branch currents (I_r and I_c) are out of phase (90 degrees apart). With equal values of R of R and X_c , the total current in the main line is 45° apart from either branch' Figure 23-3 shows the relationships. When a sine wave is applied to an RC parallel circuit, the total opposition to current flow is a combination of R and X_c . This total opposition is known as impedance, symbolized by Z. However, unlike series RC circuits, the combination of R and X_c , in parallel, is not a vector sum. Therefore, solve for the total circuit current and then use Ohm's law to find the equivalent circuit impedance:

To solve for I_T , use individual branch currents:

$$I_T = \sqrt{(I_R)^2 + (I_C)^2}$$

To solve for equivalent Z, use Ohm's law:

$$Z = V_T / I_T$$

For parallel RC circuits, the impedance angle is based on the ratio between the branch currents and is not solved in a straightforward manner like series

RC circuits. However, a parallel RC circuit can still be characterized as resistive or capacitive, but in a different manner. When R is 10 times greater than X_c the circuit is capacitive, because most of the current is flowing throughout the capacitive side. When X_c is 10 times greater than R , the circuit is resistive because most of the current is flowing through the resistor.

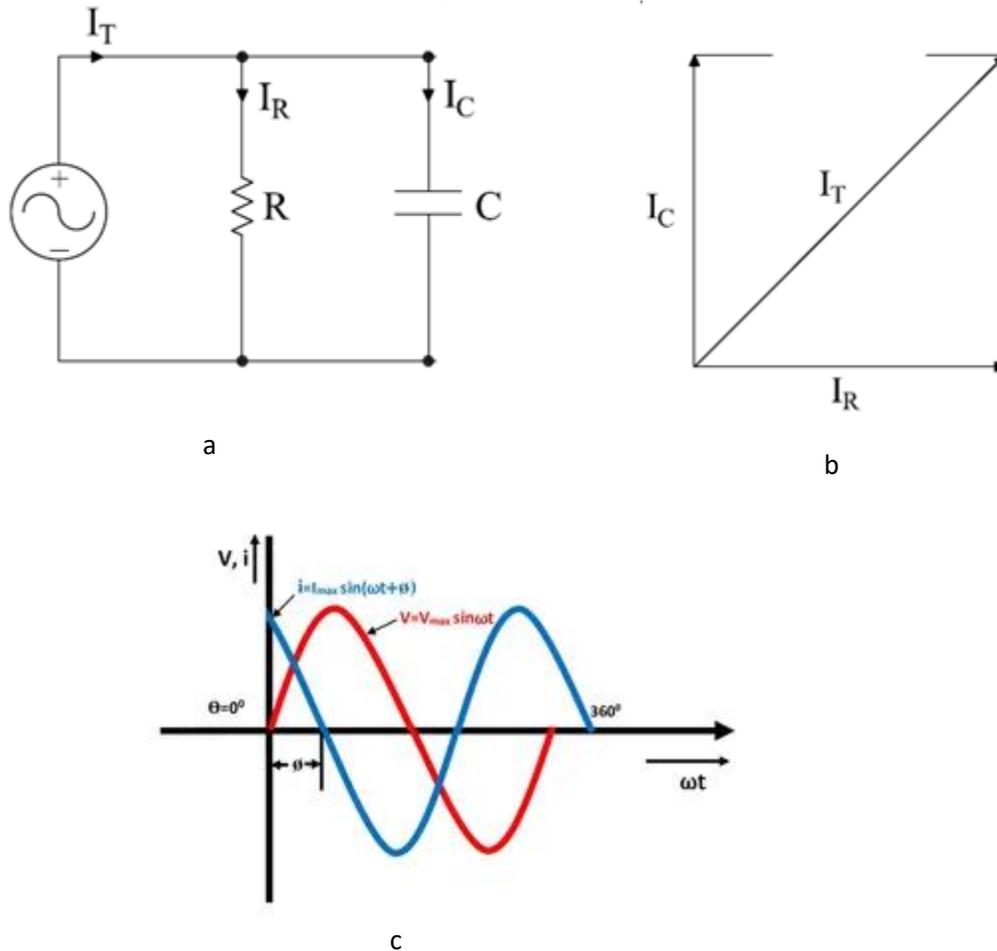


Fig 7: a: RC parallel circuit, b: phasor diagram, c: waveform

4- Parallel RL Circuits

When components R and L are in parallel, each branch has its own value of current that depends on the branch value of R or X_L . Also, because they are in parallel, the voltage (magnitude and phase) across the parallel bank of L and R is the same. This means there is no phase difference between the applied voltage and the voltage across R and L in the parallel bank.

The current through the resistive branch is in phase with the applied signal. However, the current through X_L , in the inductive branch, lags the self-induced voltage V_L by 90 degrees and also lags the applied voltage. Therefore, the two currents (I_R and I_L) are out of phase and their phasor sum is equal to the total current I_T . Figure 24-3 shows the relationships. When a sine wave is applied to an RL parallel circuit, the total opposition to Current flow is a combination of R and X_L . This total opposition is also known as Impedance, symbolized by Z . However, unlike series RL circuits, the combination of R and X_L is not a vector sum. Therefore, a practical solution is used to find the total circuit current

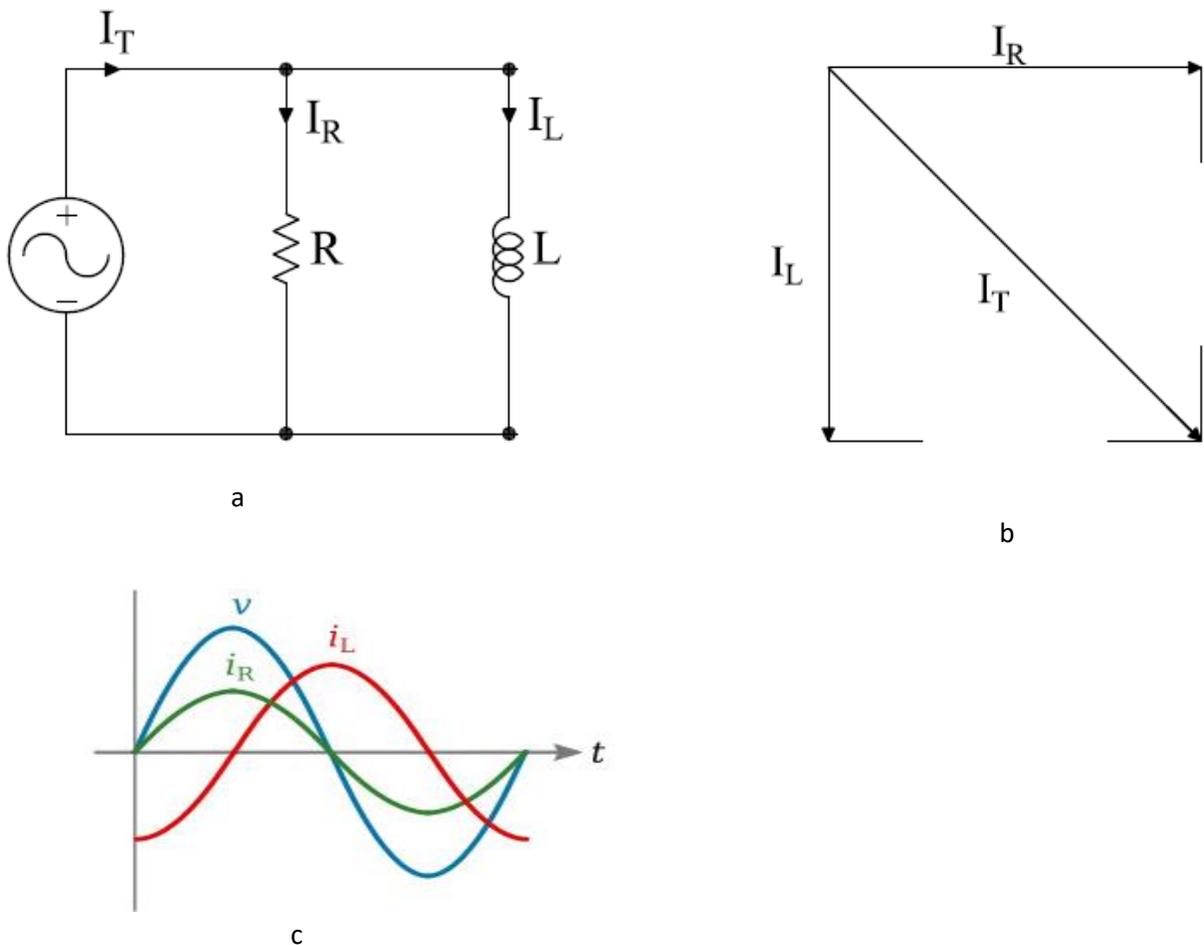


Fig 8: a: RL parallel circuit, b: phasor diagram, c: waveform

and then Ohm's law can be used to find the equivalent circuit impedance: To solve for I_T , use individual branch currents:

$$I_T = \sqrt{(I_R)^2 + (I_L)^2}$$

To solve for equivalent Z , use Ohm's law:

$$Z = V_T/I_T$$

Unlike series RL circuits, the impedance angle in parallel RL circuits is not solved in a straightforward manner. This is because the impedance angle is based on the ratio between the branch currents. However, a parallel RL circuit can still be characterized as resistive or inductive. When R is 10 times greater than X_L the circuit is inductive because most of the current is flowing through the less resistive coil and into the main line. When X_L is 10 times greater than R , the circuit is resistive because most of the current is flowing through the resistor

Procedures:

Part 1

1. Connect the circuit as shown in Figure (9)
2. Measure I_T , V_1 , V_2 , and V_T .
3. record your results in the first row of Table (1).

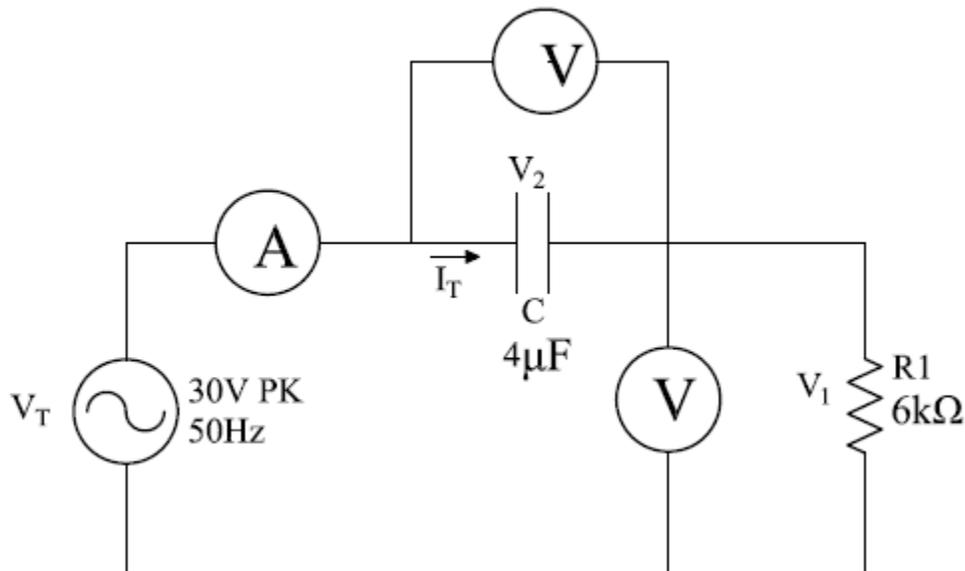


Figure 9: experimental circuit of part1

Part 2

1. Connect the circuit as shown in Figure (10)
2. Measure I_T , V_1 , V_2 , and V_T .
3. record your results in the second row of Table (2).

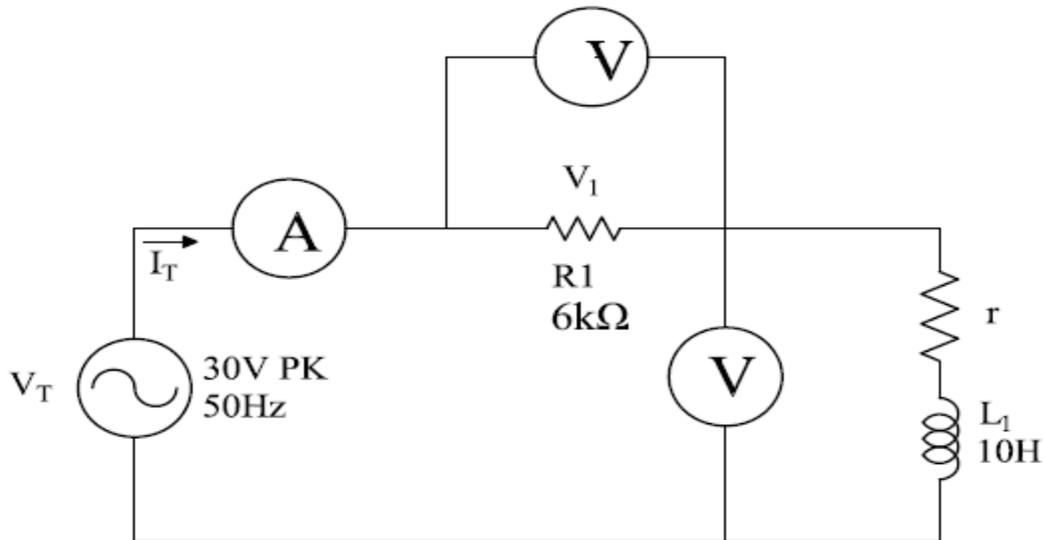


Figure 10: experimental circuit of part 2

Part 3

1. Connect the circuit as shown in Figure (11)
2. Measure V_T , I_1 , I_2 and I_T .
3. record your results in Table (2).

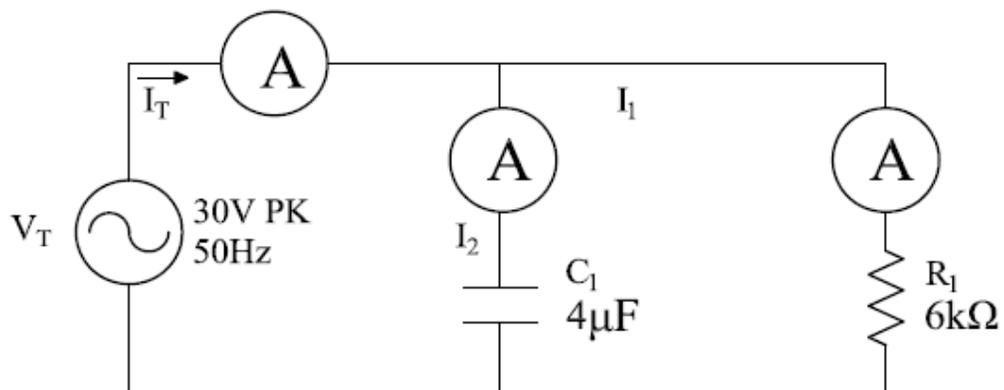


Figure 11: experimental circuit of part 3

Part 4

1. Connect the circuit as shown in Figure (12)
2. Measure V_T , I_1 , I_2 and I_T .
3. record your results in Table (2).

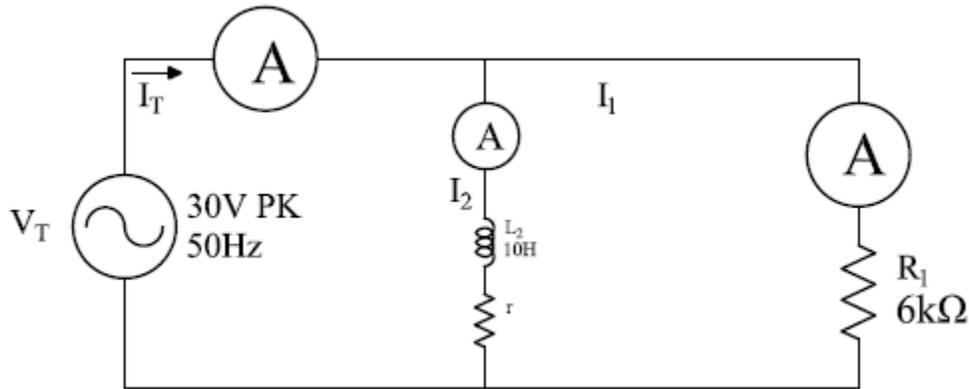


Figure 12: experimental circuit of part 4

Table (1)

Figure No.	I_T	V_1	V_2	V_T
5				
6				

Table (2)

Figure No.	V_T	V_1	V_2	I_T
7				
8				

Discussion

1. For the Figure (5), find theoretically the value of V_c and V_R in polar form and compare them with the practical results.
2. For the Figure (7), find theoretically the value of I_c and I_R in polar form and compare them with the practical results.
3. Draw the voltage phasor diagram of the capacitive in Figure (5) and the inductive in Figure (6). Calculate the phasor angle θ .
4. Draw the current phasor diagram of the capacitive in Figure (7) and the inductive in Figure (8). Calculate the phasor angle θ .

Experiment No.12

The Series RLC Resonance Circuit

Object

to study resonance in a series RLC circuit by measuring and plotting amplitude of the voltage versus frequency as well as to observe the variation in phase angle for values of the frequency on either side of the resonant value.

Theory

An RLC circuit (tuned circuit) is an electrical circuit consisting of a resistor of resistance R , a coil of inductance L , a capacitor of capacitance C , and a voltage source V arranged in series or parallel. When these elements are connected in series (Fig. 1), the circuit presents low electrical impedance to alternating current of the same frequency as the resonance frequency of the circuit and high impedance to current of other frequencies.

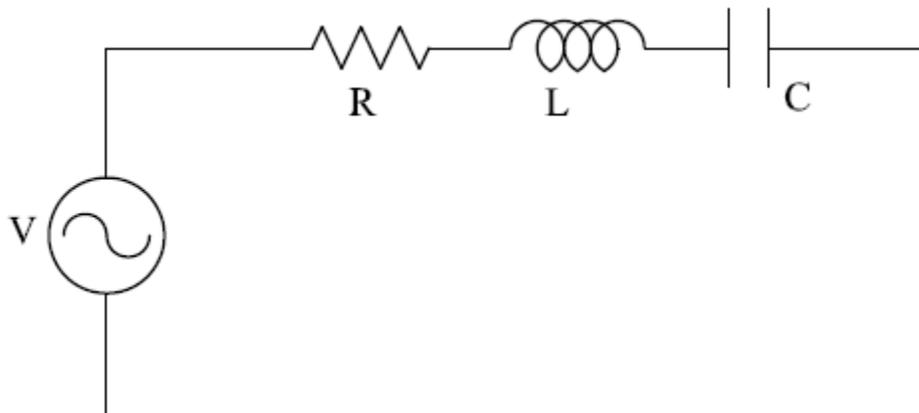


Figure 1: A simple RLC circuit

The voltage from the AC source is given by

$$V = V_R + V_C + V_L \dots \dots \dots (1).$$

Note that the peak voltage V_0 of the source will not equal the sum of the individual peak voltages of the three elements. This is due to the fact that the voltages across the resistor, capacitor, and inductor do not reach their peak values at the same time. The total impedance Z of the RLC circuit is given by

$$Z = \sqrt{(X_C + X_L)^2 + R^2} \dots\dots\dots(2)$$

Ohm's Law for this circuit as well as for each component is then the following.

$$V_0 = I_0 Z \dots\dots\dots(3)$$

$$V_R = IR \dots\dots\dots(4)$$

$$V_C = IX_C \dots\dots\dots(5)$$

$$V_L = IX_L \dots\dots\dots(6)$$

The applied voltage and the current are generally *out of phase*, and the phase difference is given by

$$\varphi = \tan^{-1} \left[\frac{X_C - X_L}{R} \right] \dots\dots\dots(7)$$

Since the impedance depends on frequency, the current varies with frequency as well, even when the voltage amplitude is constant. According to Eq. (3), the current will be at a maximum when the impedance is at a minimum. This occurs when

$X_L - X_C = 0$, i.e., when

$$\omega = \frac{1}{\sqrt{LC}} \dots\dots\dots(8)$$

This frequency at which the current is maximum is known as the ***resonant frequency*** of the circuit. Fig. 2 is a graph of the current versus the frequency for a particular value of R .

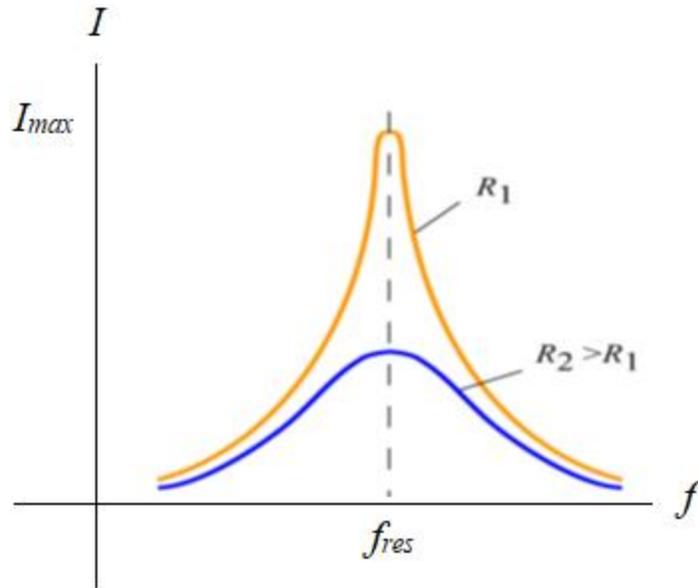


Figure 2: Plot of current versus frequency

At resonant frequency, the impedance is purely resistive. For very small values of R the circuit is referred to as an LC circuit and the energy oscillates between the inductor and the capacitor. Of course, a little bit of energy will be dissipated through the resistor. Applications of the phenomenon of resonant frequency include radio and TV sets where a varying L or C value is used for tuning to a particular station.

Procedures

Part 1

1. Connect the resistance, capacitance and inductance in series as shown in Fig.3

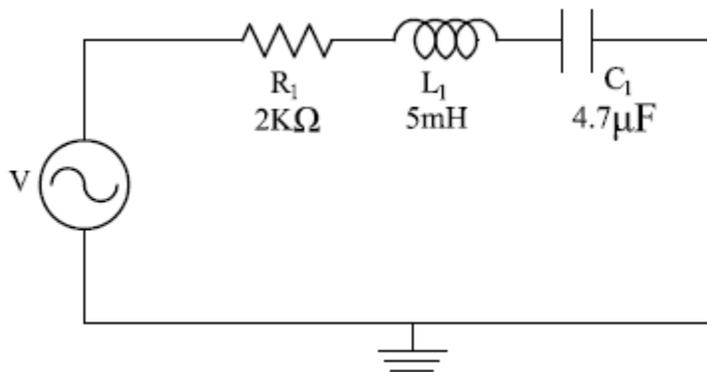


Figure 3: experimental circuit of part 1

2. Adjust the function generator to 200 Hz, 10 V and 50% duty cycle.

3. Select sine, waveform, turn the circuit ON for about 50 ms then turn it OFF.
4. Measure the phase shift between the current I and the input voltage V using oscilloscope.
5. Draw the result of the display graph.
6. Draw the Phasor diagram.

Part 2

Resonance

1. Connect the circuit as shown in Fig. (4).
2. Set the voltmeter to $6 V_{r.m.s}$.
3. Select sine waveform; vary the oscillator from 14 kHz to 17 KHz in steps of 0.5 kHz.
4. Record the reading of the voltmeter at each step as in table (1)

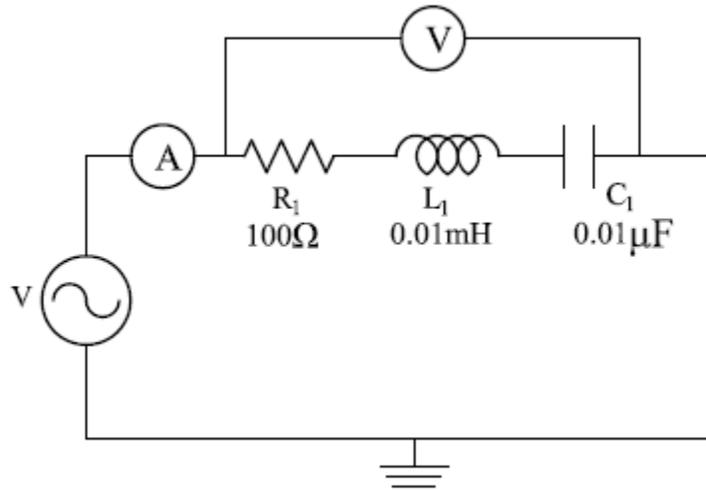


Figure 4: experimental circuit of part 2

table (1)

Frequency(kHz)	$I_{r.m.s}$ (A)

5. Evaluate the impedance of the circuit at each step.
6. Plot a graph of X_L , and X_C w.r.t. of frequency
7. Plot a graph of impedance w.r.t. of frequency
8. Determine the value of the impedance at the resonant frequency.
9. Compare the value of the resonant frequency to the theoretical value.

Discussion

1. Can we obtain a plot of X_L , against frequency f experimentally?
2. Explain why Phasor and impedance diagrams have the same angles.
3. What is the value of Phasor shift if $R= 300\Omega$ $L= 400\text{mH}$ with $f=50\text{Hz}$. Discuss the increase or decrease in the phase shift.
4. Comment on the result you have obtained

Experiment No.13

The Parallel RLC Resonance Circuit

Object

To perform be familiar with The Parallel RLC Resonance Circuit and their laws.

Theory

This experiment is about studying resonance in RLC parallel circuits. This experiment will be used to examine the sinusoidal frequency response of the series and parallel to see at what frequency the current through an RLC parallel circuit reaches maximum value. A network is in resonance when the voltage and current at the network input terminals are in phase and the input impedance of the network is purely resistive.

Consider the Parallel RLC circuit of figure 1. The steady-state admittance offered by the circuit is:

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

where $Y = \frac{I}{Z}$: Admittance

Resonance occurs when the voltage and current at the input terminals are in phase. This corresponds to a purely real admittance, so that the necessary condition is given by

$$\omega C - \frac{1}{\omega L} \dots \dots \dots (2)$$

The resonant condition may be achieved by adjusting L, C, or ω . Keeping L and C constant, the resonant frequency ω_r

is given by:

$$\omega_r = \frac{1}{\sqrt{LC}} \quad \frac{rad}{sec} \dots \dots \dots (3)$$

Or

$$f_r = \frac{1}{2\pi\sqrt{LC}} \quad Hz \dots \dots \dots (4)$$

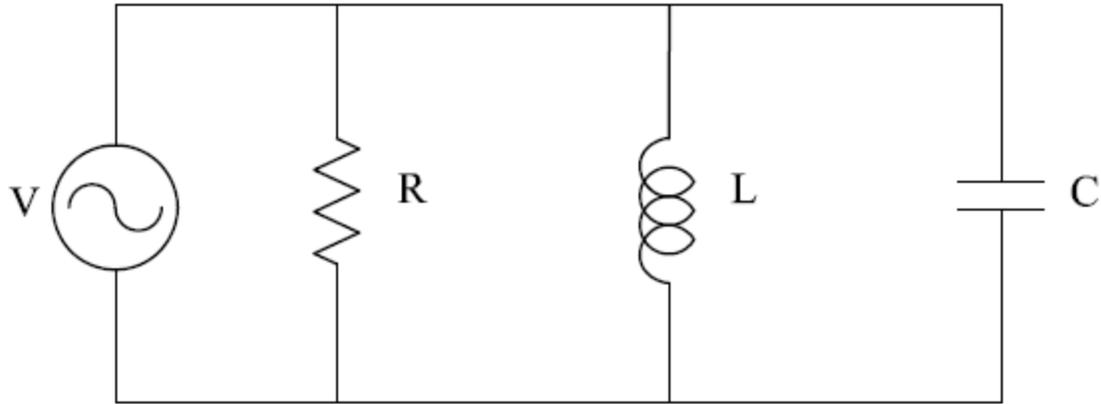


Figure 1. Parallel RLC circuit.

Procedures

Part 1

1. Connect the resistance and inductance in series as shown in Fig.2

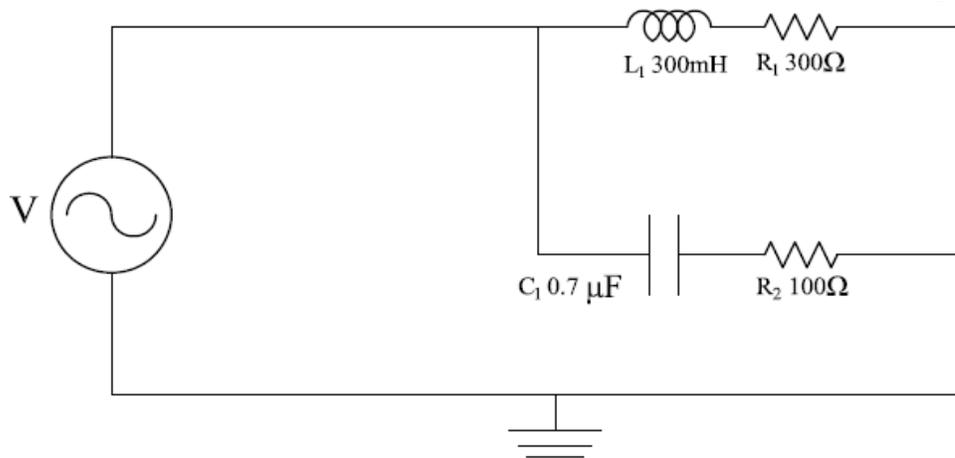


Fig.2: experimental circuit of part 1

2. Adjust the function generator to 200 Hz, 10 V and 50% duty cycle.
3. Select sine, waveform, turn the circuit ON for about 50 ms then turn it OFF.
4. Measure the phase shift between the current I and the input voltage V using oscilloscope.
5. Draw the result of the display graph.
6. Draw the Phasor diagram

Part 2

Resonance

1. Connect the circuit as shown in Fig. (3).
2. Set the voltmeter to 6 Vrms.
3. Select sine wave; vary the oscillator from (14-17) KHz, steps of 0.5 kHz.
4. Record the reading of the voltmeter at each step as in table (1)
5. Evaluate the impedance of the circuit at each step.
6. Plot a graph of X_L , and X_C w.r.t. of frequency
7. Plot a graph of impedance w.r.t. of frequency
8. Determine the value of the impedance at the resonant frequency.
9. Compare the value of the resonant frequency to the theoretical value.

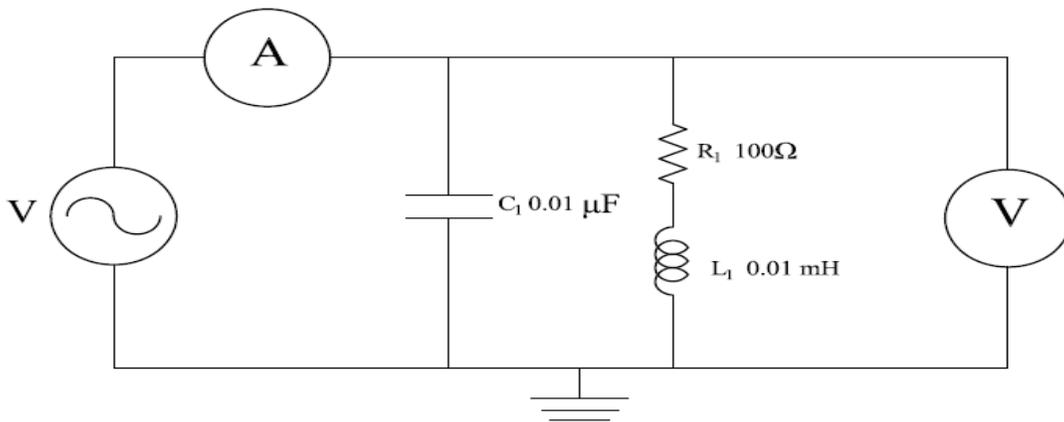


Fig.2: experimental circuit of part 1

table (1)

Frequency(kHz)	$I_{r.m.s}$ (A)

Discussion

1. Can we obtain a plot of X_L , against frequency f experimentally?
2. Explain why Phasor and impedance diagrams have the same angles.
3. What is the value of Phasor shift if $R= 300\Omega$, $L= 400\text{mH}$ with $f=50\text{Hz}$.
Discuss the increase or decrease in the phase shift.

Experiment 14

Power Factor Correction

Object

To calculate and measure complex power, observe lagging power factor, and correct lagging power factor by adding capacitive compensation

Theory

The apparent power or voltamperes (VA) in a given circuit is the product of the rms voltage and rms current magnitudes. The real power (also referred to as average power or active power) is the apparent power times the cosine of the angle between the voltage and the current waveforms or phasors. This cosine term is known as the power factor, and it is desirable to operate at or near unity power factor. This is the case since equipment costs are largely proportional to conductor size and insulation, which are determined by the voltamperes required. Also, low power factor operation

implies high current with resulting small useful work. This is clear since a purely inductive load may draw a large current and thus large voltamperes demand. The real power lost in the transmission or distribution line feeding this load could be substantial. These I^2R losses are costly to utilities and large energy consumers. It is common to add capacitive reactance to an inductive circuit in order to bring the voltage and current in phase (and thus bring the power factor to unity). This practice is known as power factor correction. Clearly, power factor correction can reduce the cost of electric power system operation and can permit generators to produce more active power at rated kVA. In order to maintain a higher voltage at the load, and reduce line losses, the correction should always be made at the load rather than the source.

Without the shunt capacitor in parallel with the load, the source must supply both real power P_L and reactive power Q_L to the load, thus resulting in high current, high voltage drop and high losses in the transmission line. When the capacitor is added as shown, the capacitor supplies reactive power to the load and thus relieves

the source and line of this unnecessary load. As a load, a capacitor has zero power factor leading, thus making its reactive power consumption negative (i.e., it generates reactive power). If the capacitors were shunted at point 1 instead of point 2, the compensation would still relieve the source of the unnecessary reactive power Q_L of the load, but the line would still carry the full $P_L + jQ_L$, thus still causing substantial voltage drop and line losses.

The convention used for leading and lagging power factor is standard. When computing the power factor of a load (assuming load notation with the current entering the + terminal), the power factor is lagging if $0^\circ < \phi < 180^\circ$ (i.e. current lags voltage), and leading if $-180^\circ < \phi < 0^\circ$ (i.e. current leads voltage). Note that for passive RLC loads, $-90^\circ \leq \phi \leq 90^\circ$.

Procedures

Part 1

1. Consider the circuit of Figure 1.
 - a) Calculate the impedance Z_L of the “load”.
 - b) Given $V_G = 5V$, calculate the phasor current (I), V_{RM} (maximum voltage across the $100\ \Omega$ resistor), V_{LM} (maximum load voltage), and pf (load power factor).
2. Use a multimeter to obtain the rms voltages for V_G , V_R , and V_L . Convert the calculated (maximum) values from part 1 to rms values and use a phasor diagram to graphically determine the pf.
3. Calculate the capacitance which, when placed in parallel with the load, will raise the pf to 1
4. Use LTSpice to determine the pf of the (original) load. Determine the phase difference between the load voltage and current. Then, include the capacitor (step 3) and determine the new pf.

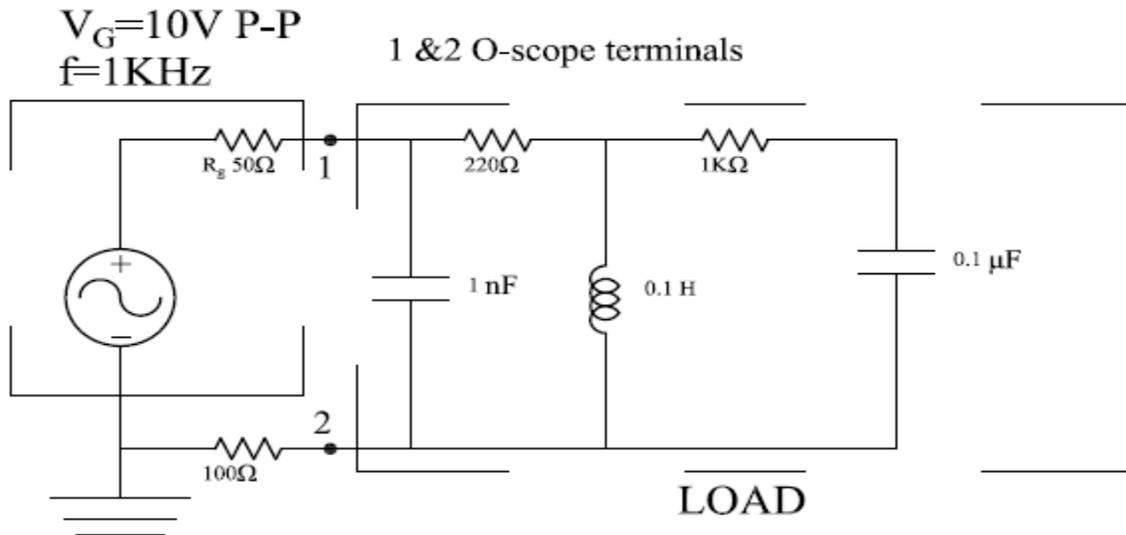


Fig 1: experimental circuit

Part 2

1. Use the inductance/capacitance meter to measure the exact component values for the inductor and capacitors used in this experiment. Use a multimeter to measure the exact values for the resistors used.
2. Use the oscilloscope to measure the phase angle between the load voltage and load current (measured through the 100Ω resistor). Connect the O-scope and circuit as in Figure 1. (Note: since the signal generator and O-scope are both internally grounded to the power line, measuring the phase angle between the load voltage and the current is not straightforward).
 - a) Connect channels 1 and 2 as shown. Set ch1 to be the trigger, and set the same vertical amplification for each channel. (Now “ch1-ch2” is the load voltage and “ch2/100Ω” is the load current).
 - b) Observe ch2 and adjust so a positive slope zero crossing aligns with a graticule (line on the O-scope face).
 - c) Switch to ADD mode (with ch2 inverted) to get the load voltage signal
 - d) Measure phase shift by measuring the time shift between positive zero crossings of “ch1- ch2” and “ch2” (the screen graticule set in part B).
 - e) Make sure the waveforms are centered about the zero-volt axis (they tend to shift when going to/from the “ch1-ch2” mode).

Use the (rms) multimeter to measure the voltages across the load, generator, and the 100Ω resistor.

Use the measured phase angle and the rms voltage/current from the meter to calculate the power factor pf.

3.

- a) Calculate the capacitance needed to raise the pf to 1.
- b) Connect the calculated capacitance (from A), re-adjust the signal generator to $10V$ $p-p$, and again measure the phase angle between the load voltage and current.
- c) Again, use the rms multimeter to measure the voltages across the load, generator, and the 100Ω resistor (as in 2).

Discussion

1. Compare the pf determined
 - a) analytically (step 1 in part 1).
 - b) graphically (step 2 in part 1).
 - c) with LTSpice (step 4 of part 1).
 - d) with the O-scope in the lab. Discuss any variations in results.
2. What would you do to improve this experiment?

MultiSim

Simulation

Nowadays, modern electric circuits are composed by millions of elements. For example, an Intel microprocessor needs more than 500 million transistors to be able to handle data at a speed greater than 2.5 GHz and an iPhone with an A6 processor can have almost the same number of transistors. To test these devices before the actual manufacturing needs extensive simulation of the circuit to ensure proper performance once the chip is in the computer or phone. Failure to carry on a proper simulation procedure can cost a company losses in the millions of dollars.

Multisim

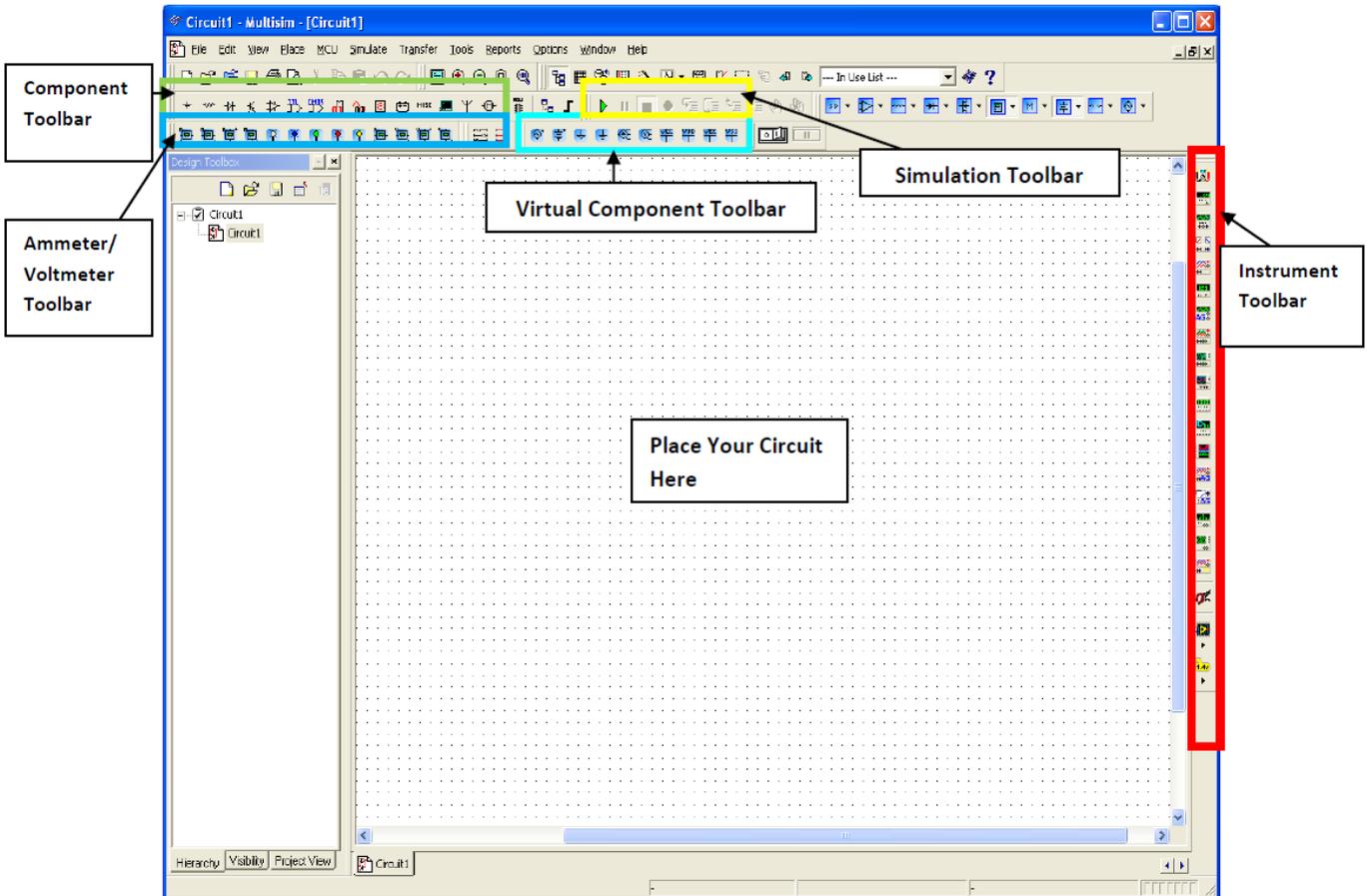
Multisim is a simulation tool that can be used to expedite the analysis and design of various circuits. Of the many simulators available, Multisim has emerged as the best for circuit simulation. Multisim is a SPICE-based simulator, produced by National Instruments Inc. with a schematic-capture interface that allows easy circuit topology input and specification of simulation data.

Circuit analysis using Multisim allows the user to:

- observe the circuit behavior before the actual manufacturing.
- use ideal components to isolate design and circuit limitations.
- make measurements that are hard to make in the real circuit because they might damage the circuit.

Start Simulation by Multisim

Here is window of multisim, as it appears first time when you start the software.



1. Open/Create Schematic

A blank schematic Circuit is automatically created. To create a new schematic; click on **File – New – Schematic Capture**. To save the schematic; click on **File /Save As**. To open an existing file; click on **File/ Open** in the toolbar.

2. Place Components

To Place Components, click on **Place/Components**. On the Select Component window, click on **Group** to select the components needed for the circuit. Click **OK** to place the component on the schematic.

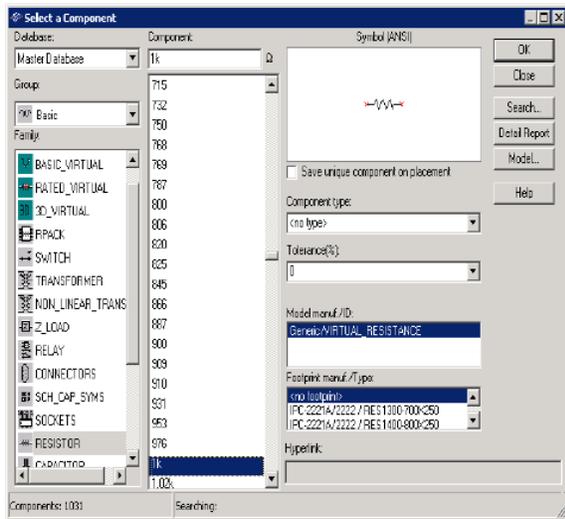


Figure 1: Select Resistor

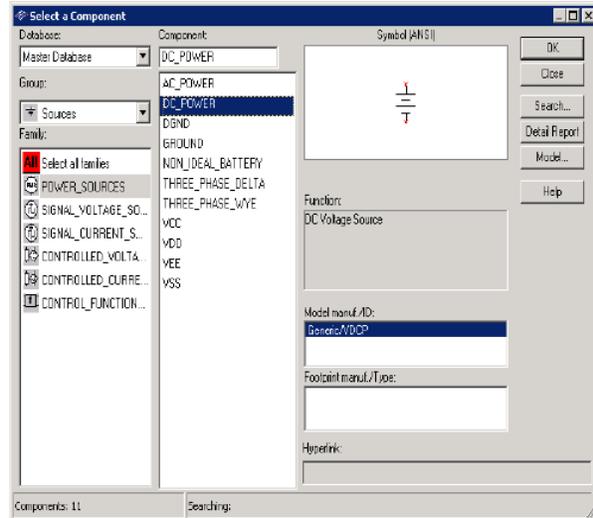


Figure 2: Select DC voltage

For example, to select resistors and the DC source shown in Figure 3; click on **Place/ Components**. In **Group**; select **Basic** scroll down to Resistors and select the value of the resistor needed to construct the circuit. for this example; select 1k. To place DC source; click on **Sources** in **Group** and select **DC Source**. As shown in Figure 1 and Figure 2 respectively.

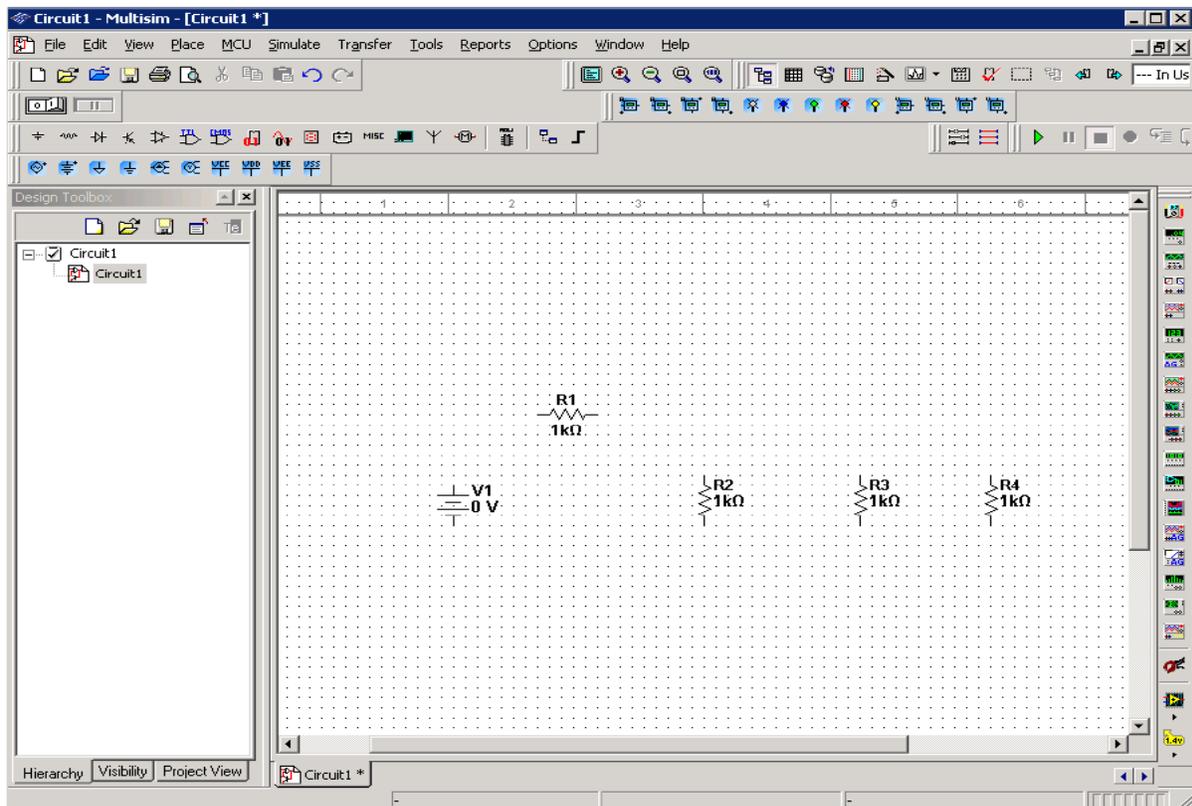


Figure 3: DC Source & Resistors

3. Virtual Components

Components can also be placed on the circuit using **Virtual components**. Click on **View – Toolbars** and select the toolbar needed for the circuit.

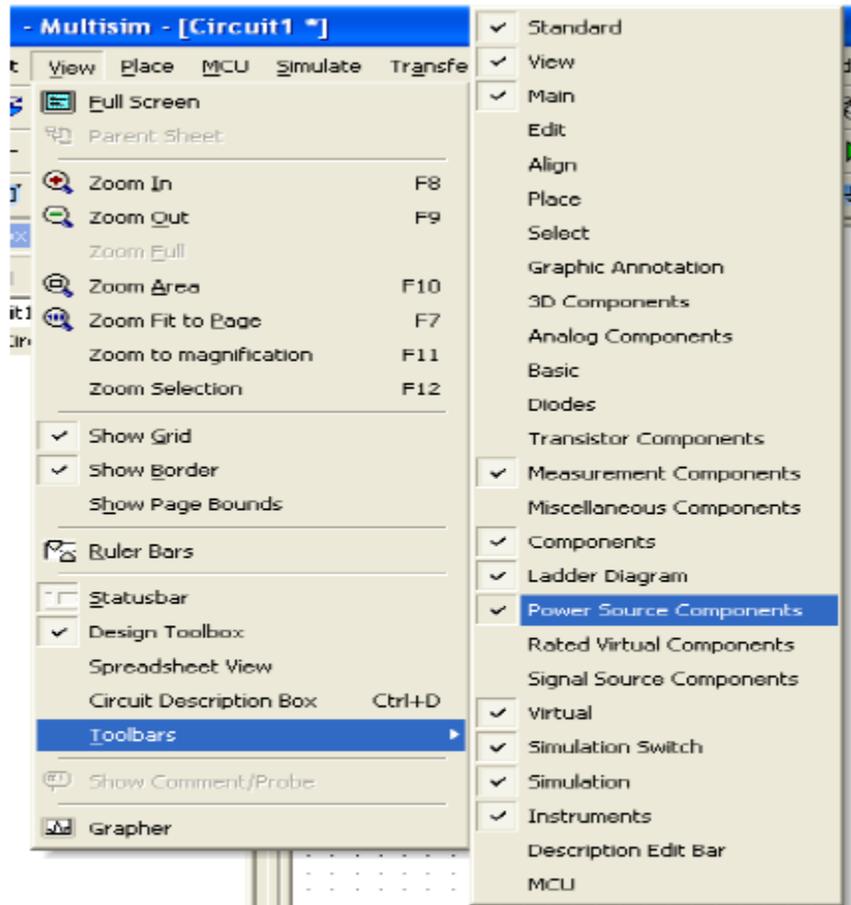


Figure 4: Virtual Components

4. Rotate Components

To rotate the components; **right click** on the Resistor. To flip the component on 90 Clockwise (**Ctrl +R**) and 90 Counter Clockwise (**Ctrl+Shift+R**). Figure 5

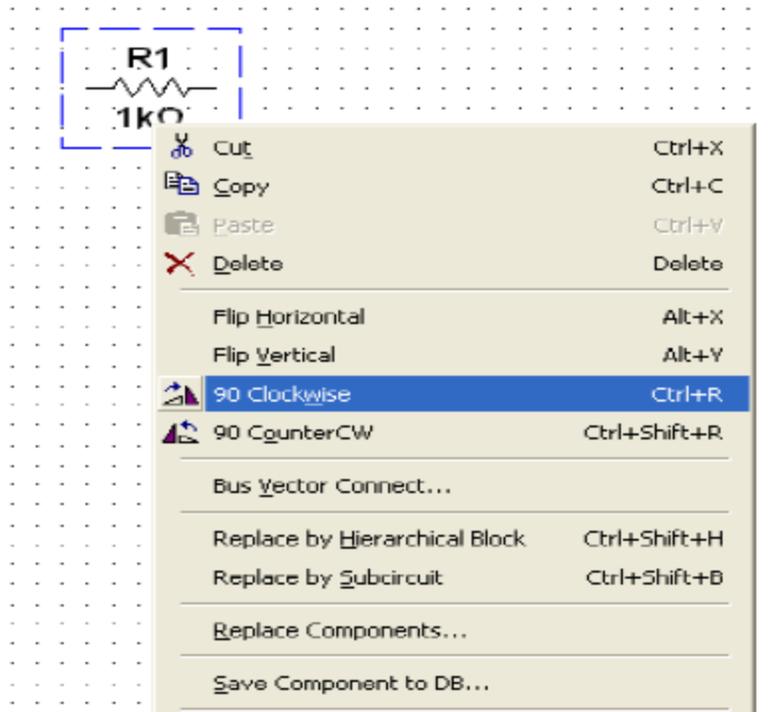


Figure 5: Rotate Components

5. Place Wire/Connect Components

To connect resistors; click on **Place/Wire** drag and place the wire. Components can also be connected by **clicking the mouse** over the terminal edge of one component and dragging to the edge of another component. Figure 6.

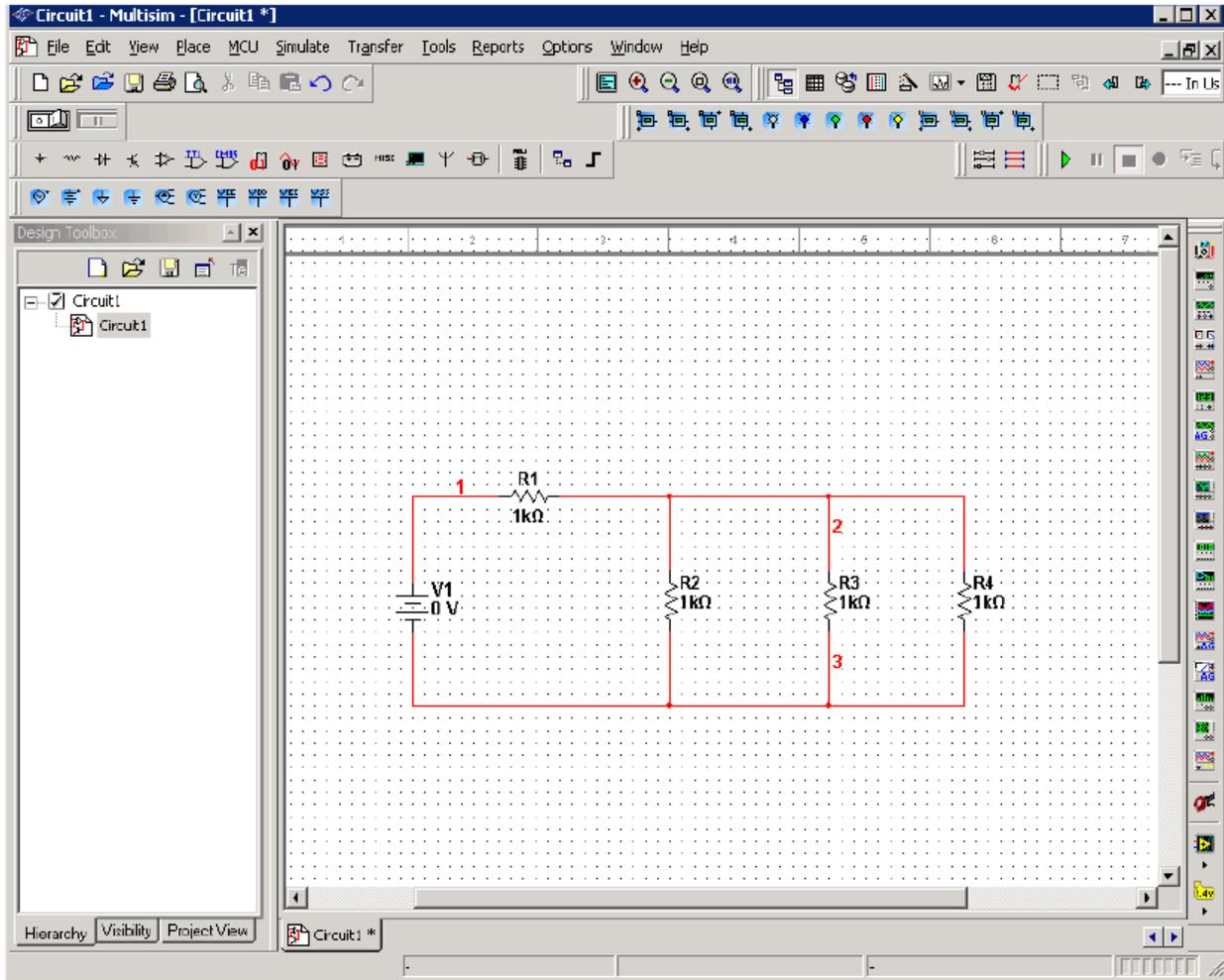


Figure 6: Place/ Wire

6. Change Component Values

To change component values; **double click** on the component this brings up a window that display the properties of the component (Figure 7). Change R_1 from 1k Ohm to 10 Ohms, R_2 to 20 Ohms, R_3 to 30 Ohms, and R_4 to 40 Ohms. Also change the **DC source** from 0 V to 20 V. Figure 8 shows the completed circuit

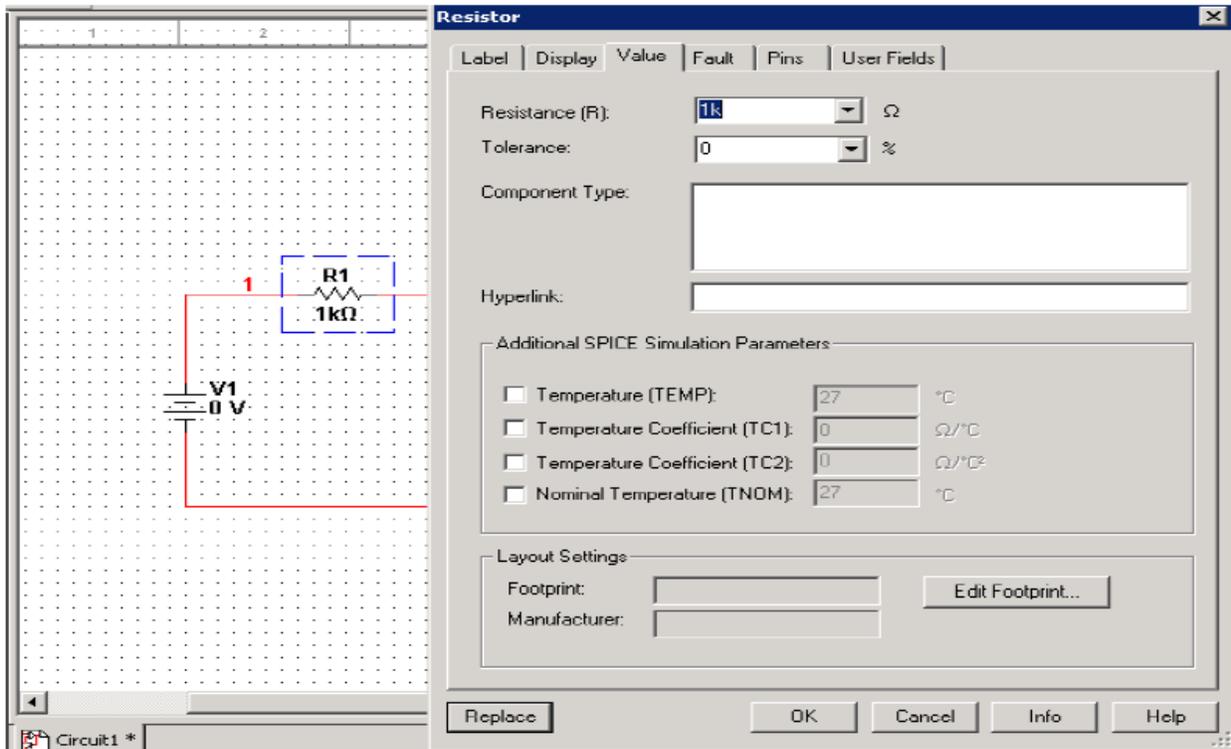


Figure 7: Change Component Values

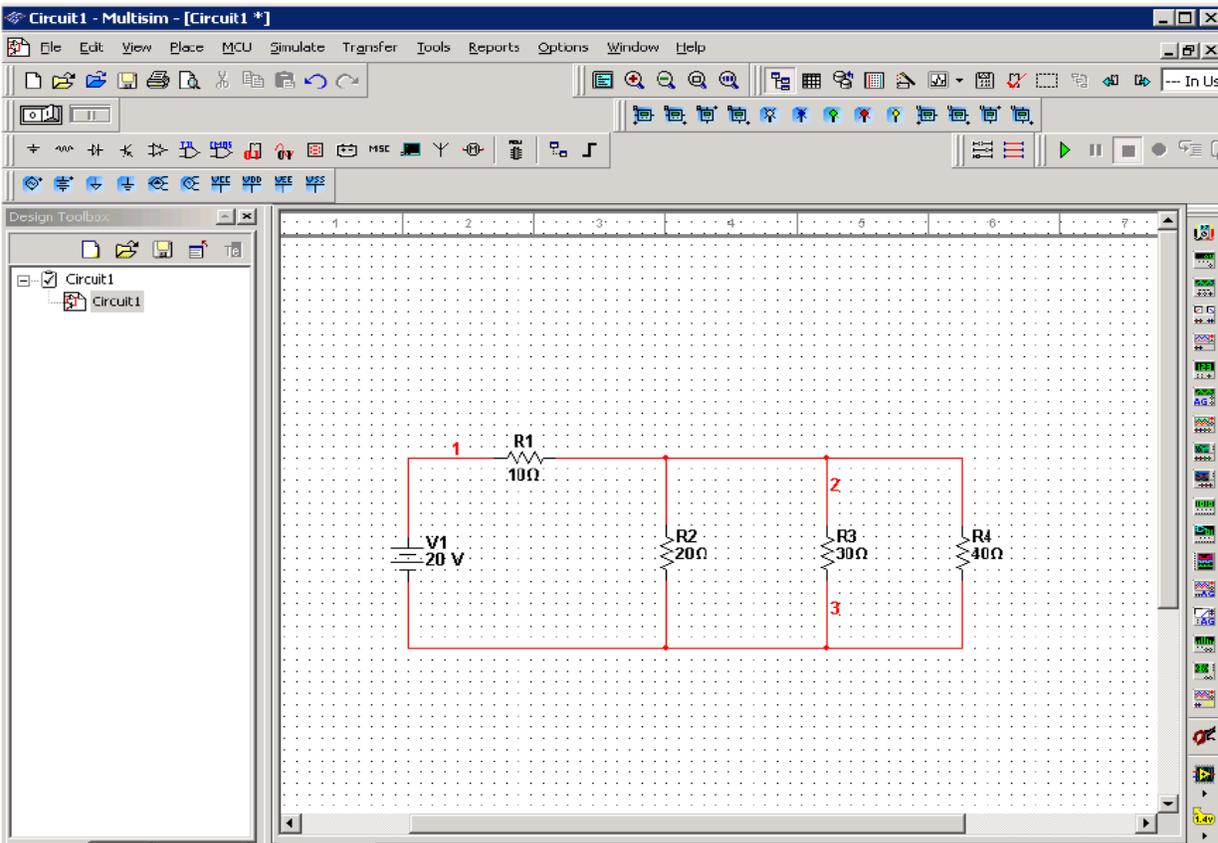


Figure 8: Completed Circuits

7. Grounding:

All circuits must be grounded before the circuit can be simulated. **Click on Ground** in the toolbar to ground the circuit. If the circuit is not grounded Multisim will not run the simulation.



Figure 9: Grounding

8. Simulation:

To simulate the completed circuit; Click on **Simulate/Run** or **F5**. This feature can also be accessed from the toolbar as shown in the Figure 10 below.

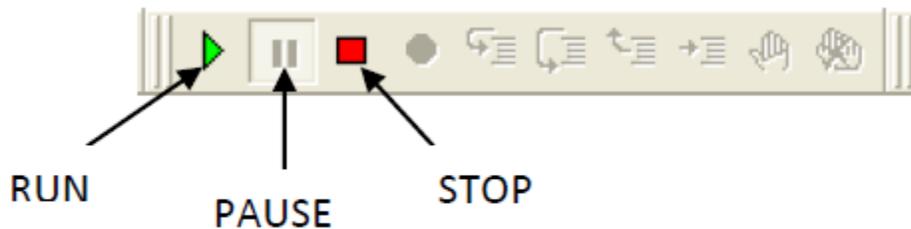


Figure 10: Simulation

Analyzing Components

Multisim offers multiple ways to analyze the circuit using virtual instruments. Some of the basic instruments needed for this lab are described below.

1. Multimeter

Use the Multimeter to measure AC or DC voltage or current, and resistance or decibel loss between two nodes in a circuit. To use the Multimeter; click on the **Multimeter** button in the **Instruments** toolbar and click to place its icon on the workspace. **Double-click** on the icon to open the instrument face, which is used to enter settings and view measurements.

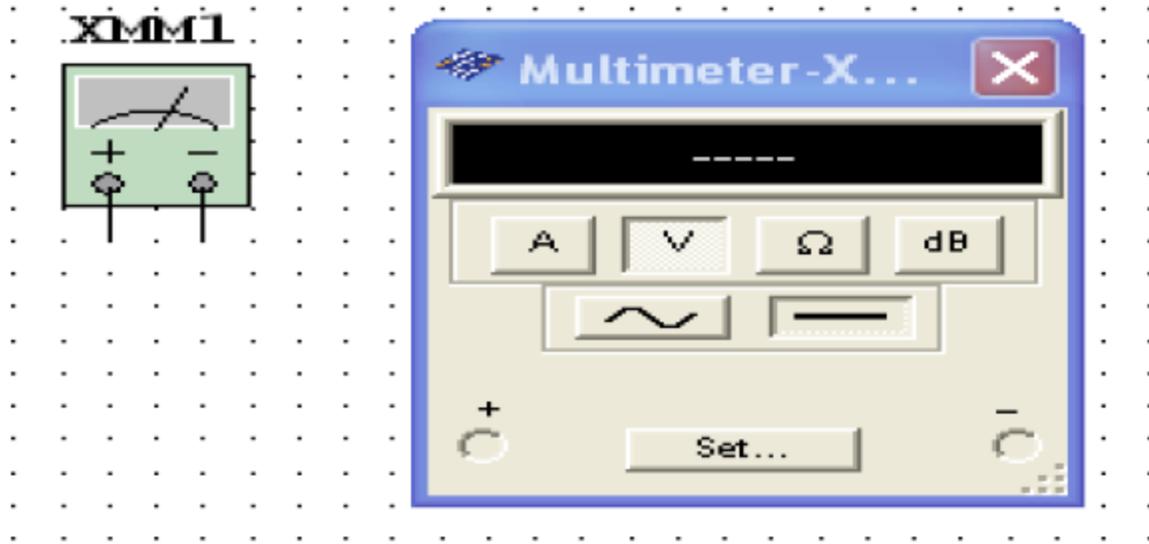


Figure 11: Multimeter

To measure Voltage; place multimeter in Parallel with the component (Resistor, Voltage etc). To measure Current place the multimeter in series with the component. Reference the Figure 12 and 13.

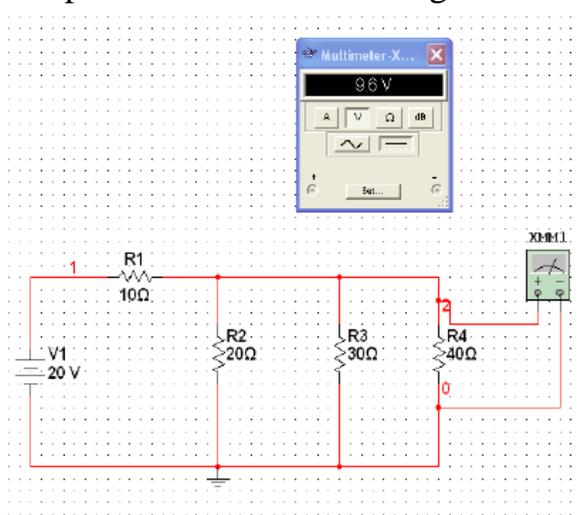


Figure 12: Measure Voltage

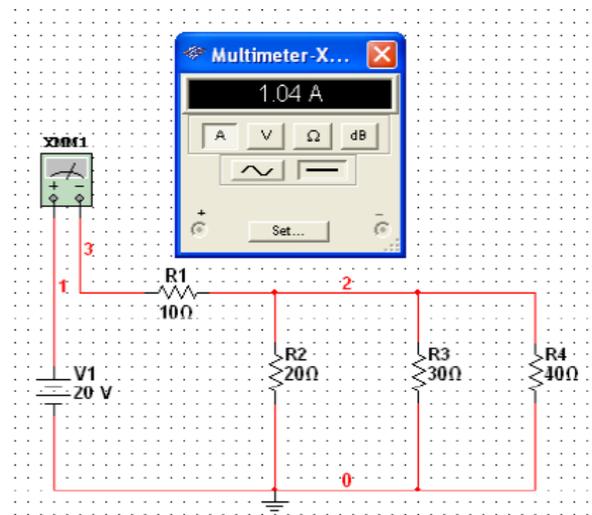


Figure 13: Measure Current

2. Wattmeter

The wattmeter measures power. It is used to measure the magnitude of the active power, that is, the product of the voltage difference and the current flowing through the current terminals in a circuit.

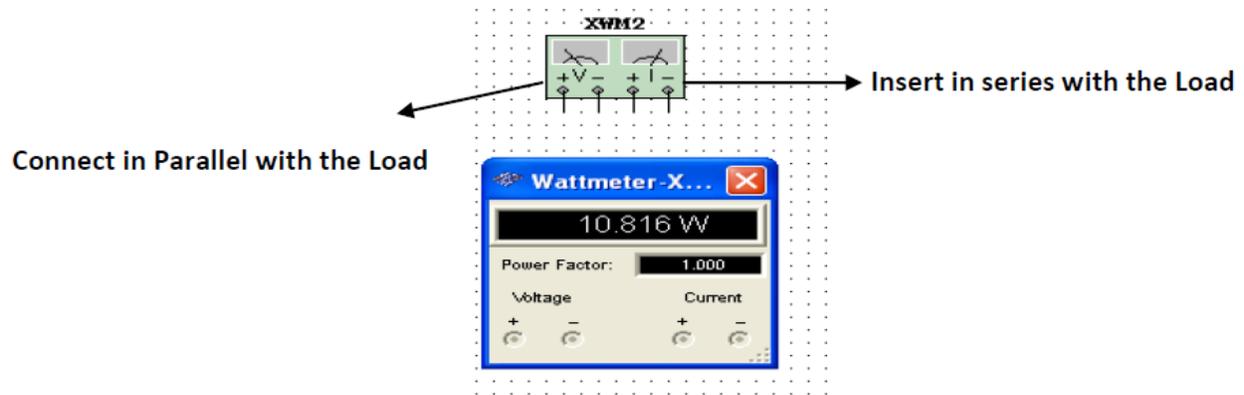


Figure 14: Wattmeter

To use the instrument; click on the **Wattmeter** button in the **Instruments** toolbar and click to place its icon on the workspace. The icon is used to wire the Wattmeter to the circuit. **Double-click** on the icon to open the instrument face, which is used to enter settings and view measurements (Figure 15).

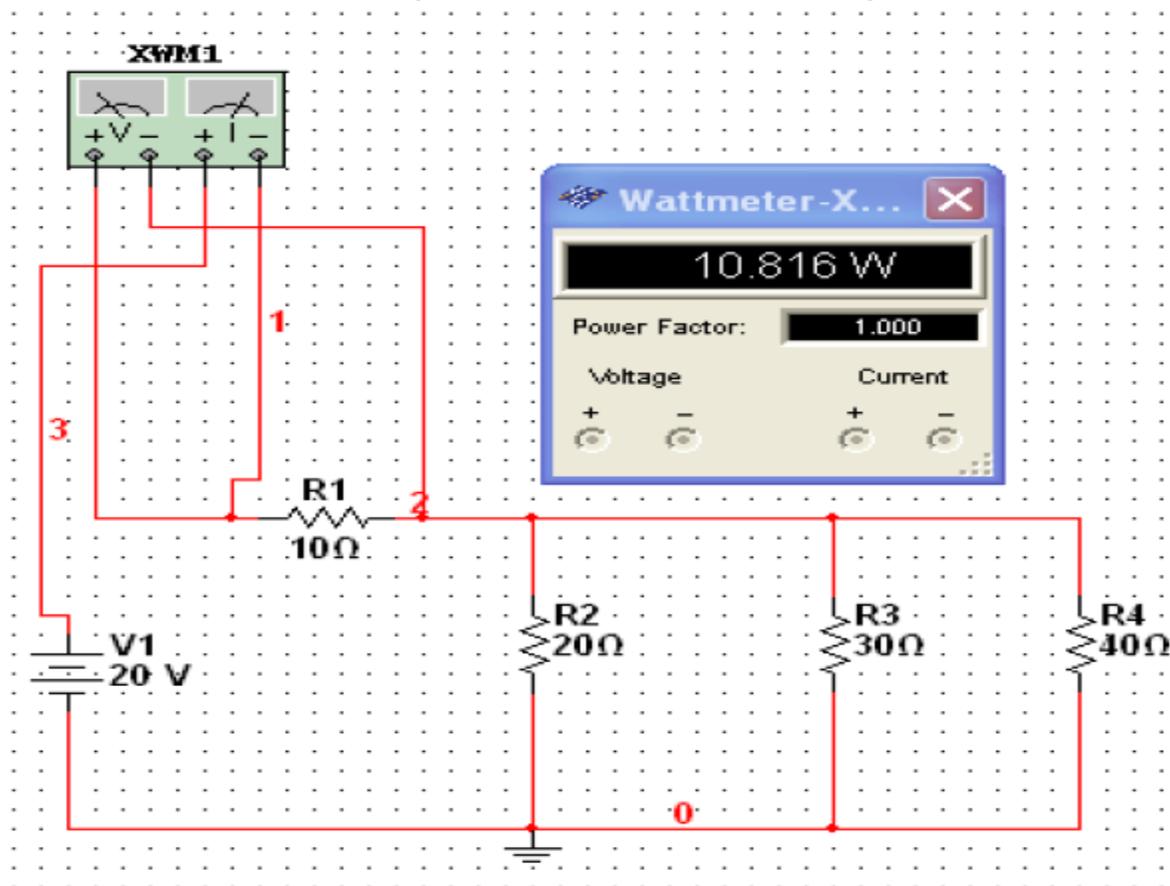


Figure 15: Wattmeter Connection

3. Agilent Multimeter

1. The Agilent Multimeter Instrument can also be used to measure and simulate circuits with more accuracy. To use the multimeter; click on the **Agilent Multimeter tool button**, place its icon on the workspace and **double-click** on the icon to open the instrument. Click on the **Power button** to switch on the instrument.

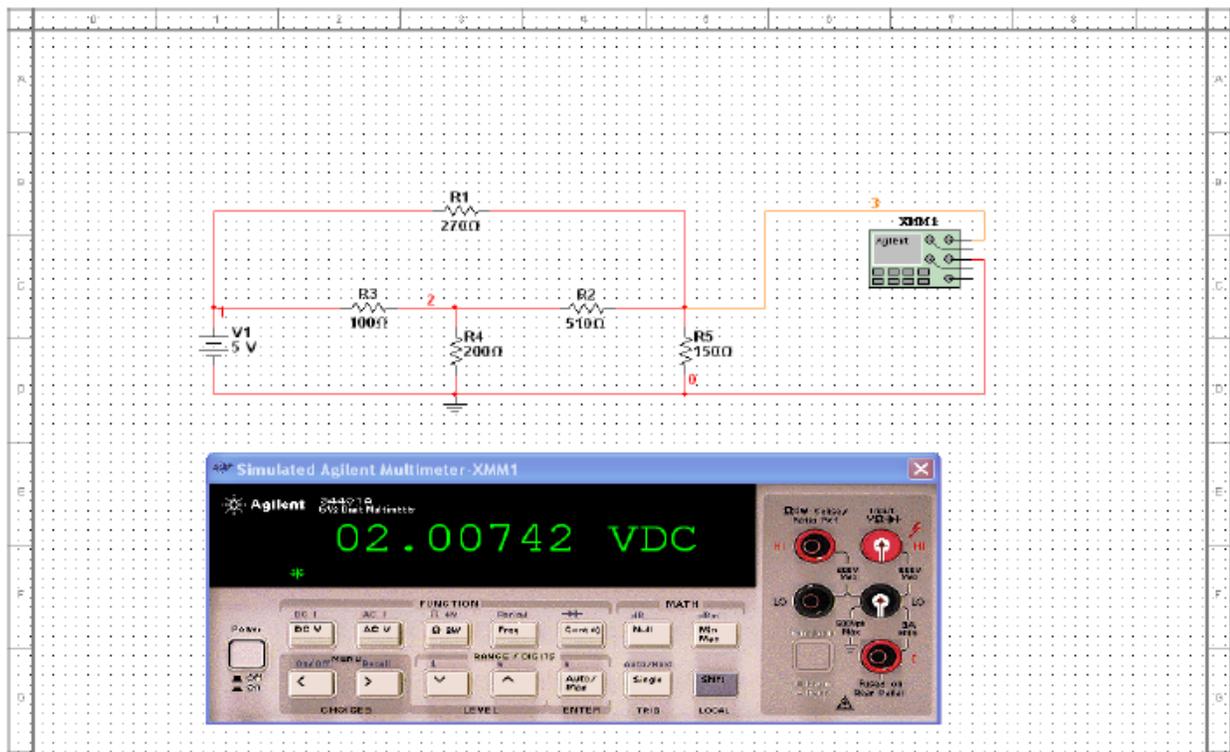


Figure16: Agilent Multimeter.

4. Ammeter:

The ammeter offers advantages over the multimeter for measuring current in a circuit. It takes up less space in a circuit and you can rotate its terminals to suit your layout. Always connect the ammeter in series with the load. To place **Ammeter** click on **View--- Toolbar --- Select Measurement Components** (Figure 17).

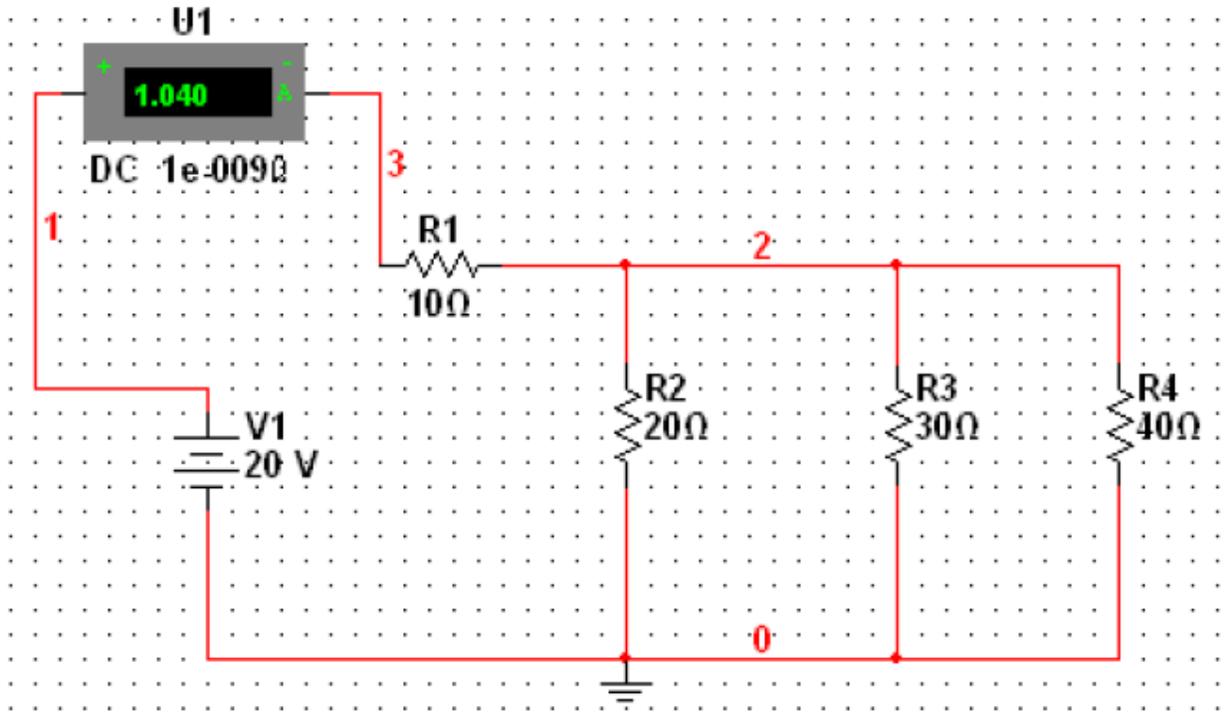


Figure 17: Ammeter

5. Voltmeter

The Voltmeter offers advantages over the multimeter for measuring voltage in a circuit. Always connect the voltmeter in parallel with the load. The voltmeter can be found in the **measurement toolbar**.



Ammeter

Voltmeter

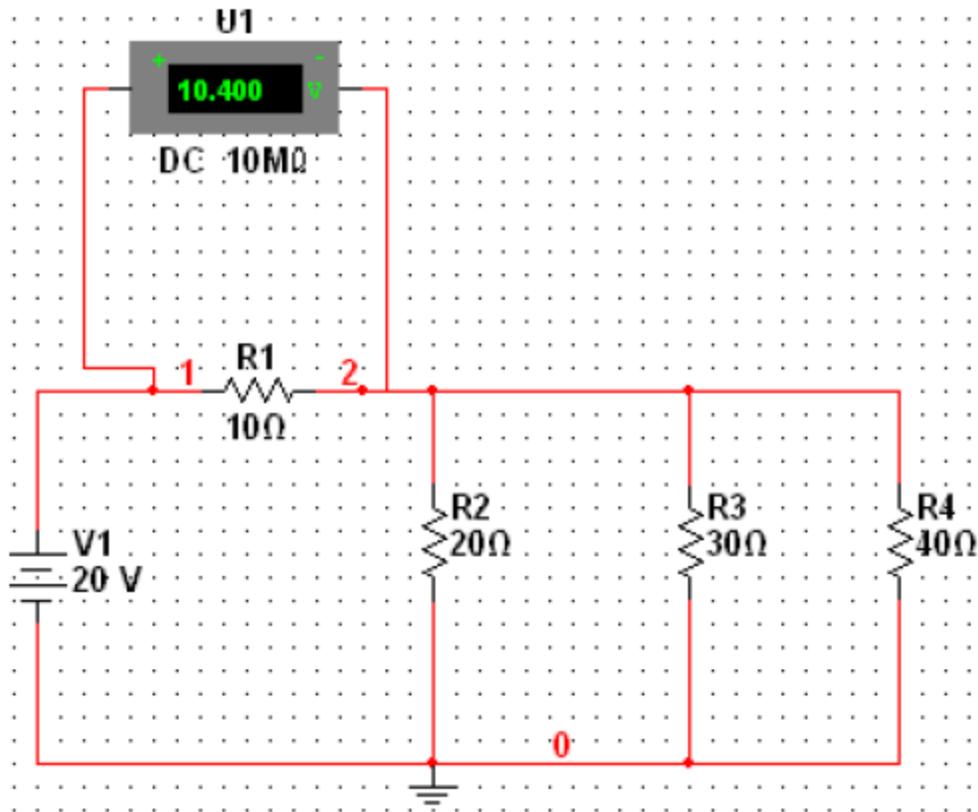


Figure 18: Voltmeter