LABORATORY MANUAL

DEPARTMENT OF
ELECTRONICS & COMMUNICATION ENGINEERING

JAYPEE UNIVERSITY OF INFORMATION TECHNOLOGY
WAKNAGHAT, SOLAN, (H.P)-173234

18B17EC271

Electrical Science Lab

Revised
January 2020

Electrical Science Lab

COURSE CODE: 18B17EC271

COURSE CREDITS: 1

CORE/ELECTIVE: CORE

L-T-P: 0-0-2

Pre-requisite: None

Course Objectives:

1. The primary objective of this course is to provide a thorough understanding of circuit analysis and measurement of various electrical parameters.
2. Analysis of a given circuit depending on types of elements - DC analysis, Transient analysis and Frequency analysis.
3. To acquire hands on experience of conducting various experiments on electrical machines.

Course Outcomes:

S .No.	Course Outcomes	Level of Attainment
CO1	Understanding basic electrical sources and measuring devices: Power supply, Multimeter, CRO/DSO and Function Generator.	Familiarity
CO2	Understand the basic working principle of a transformer and the operation of electrical machines.	Usages
CO3	Practical implementation of the fundamental electrical theorems and modeling of simple electrical systems.	Usages
CO4	Accurate measurement of voltage, current, power and impedance of any circuit.	Usages
CO5	DC analysis, Transient analysis and Frequency analysis of a given circuit depending on types of elements.	Assessment
CO6	Teamwork skills for working effectively in groups and develop analytical skills to compare experimental results with theoretical concepts.	Assessment

List of Experiments

S. No	Description	Hours
1	Introduction to Power supply & Multimeter.	2
2	To determine the equivalent resistance of a circuit using color code and to verify it using a multimeter. To verify Voltage divider and Current divider.	2
3	To verify Delta to Star and Star to Delta conversion.	2
4	Introduction to DSO& Function Generator.	2
5	To verify Kirchoff’s voltage law (KVL) and Kirchoff’s Current Law (KCL).	2
6	To verify Superposition Theorem.	2

7	To verify Norton's Theorem.	2
8	To verify Thevenin's Theorem and Maximum Power Transfer Theorem.	2
9	To study the transient response of series RC circuits using different values and R and C.	2
10	Determination of frequency response of current in RLC circuit with sinusoidal ac input.	2
11	To determine the turns ratio and polarities of transformer windings.	2
12	To obtain the equivalent circuit parameters from OC and SC tests, and to estimate efficiency & regulation at various loads.	2
Total Lab hours		24

Suggested Resources:

1. W.H. Hayt, J. E. Kemerlay & S.M. Durbin, "Engineering Circuit Analysis", Eighth Edition, McGraw Hill, 2012.
2. Van Valkenburg, "Network Analysis", Prentice-Hall India, 2001.
3. D.C. Kulshreshtha, "Basic Electrical Engineering", First Edition, McGraw Hill, 2011.

Evaluation Scheme:

1	Mid Sem. Evaluation	20 Marks
2	End Sem. Evaluation	20 Marks
3	Attendance	15 Marks
4	Lab Assessment	45 Marks
	Total	100 marks

Course Outcomes (COs) contribution to the Program Outcomes (POs)

CO/PO	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	Average
CO1	3	3	3	2	2	2	3	1	1	1	1	1	1.92
CO2	3	3	3	2	3	2	1	1	1	1	1	1	1.83
CO3	3	3	2	3	2	3	2	1	1	1	2	1	2.00
CO4	3	3	3	2	3	2	1	1	1	1	1	1	1.83
CO5	3	2	3	3	3	2	1	1	1	1	1	1	1.83
C06	3	3	3	3	2	2	2	3	2	2	2	2	2.42
Average	3.00	2.83	2.83	2.50	2.50	2.17	1.67	1.33	1.17	1.17	1.33	1.17	

EXPERIMENT 1

Aim: Introduction to Power supply & Multimeter.

Theory:

DC Power Supply:

It is easy to use, low power, multi output general purpose laboratory supply (Figure 1.1). It is suitable for experimental set-ups, circuit development and low voltage applications. The power supply delivers three dc outputs:

- 0-32V dc output, continuously variable, with 2 Amps capacity. This section is provided with panel meters to monitor output voltage and current separately.
- 5V preset dc output with 5Amps capacity. A built-in over voltage protection circuit offers protection to the loads against over voltage.
- ±15V preset dc output with 500mA capacity.

Figure 1.1: Power Supply

All the outputs are floating (i.e. neither any of + ve or – ve output terminals nor any point within the regulator circuitry is connected to ground). The power supply is designed to operate satisfactorily in ambient temperature of up to 50°C and full power can be drawn if free air circulation is allowed. The unit works on main supply of 230V ac at 50 Hz.

Regulated Power Supply – Working, Circuit Diagram and Applications

We know that there are different types of electrical & electronic circuits which use a DC power supply. Universally, we cannot use the DC batteries due to expensive as well as require replacement when discharged. In this situation, we require a circuit which can change AC supply to DC supply. A rectifier filter circuit includes a normal DC power supply**.** The normal DC power supply o/p remains stable if the load is constant. Although in several electronic circuits it is extremely significant to maintain the DC power supply constant irrespective of alternative AC supply. Otherwise, the circuit will get damage. To overcome this problem, voltage regulating devices can be used. So, the blend of the voltage regulating devices by the normal dc power supply is named as DC regulated power supply. This is an electrical device, used to generate the steady DC supply irrespective of alternative AC supply.

What is Regulated Power Supply?

The IC Regulated power supply (RPS) is one kind of electronic circuit, designed to provide the stable DC voltage of fixed value across load terminals irrespective of load variations. The main function of the regulated power supply is to convert an unregulated alternating current (AC) to a steady direct current (DC). The RPS is used to confirm that if the input changes then the output will be stable. This power supply is also called a linear power supply, and this will allow an AC input as well as provides steady DC output. (Figure 1.2)

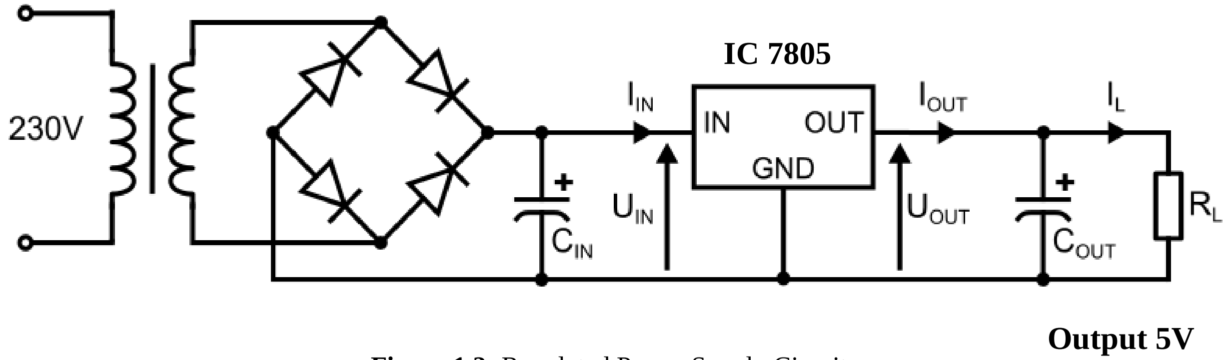


Figure 1.2: Regulated Power Supply Circuit

Block Diagram of Regulated Power Supply

The block diagram of a regulated power supply mainly includes a step-down transformer**,** a rectifier, a DC filter, and a regulator (Figure 1.3). The Construction & working of a regulated power supply is discussed below.

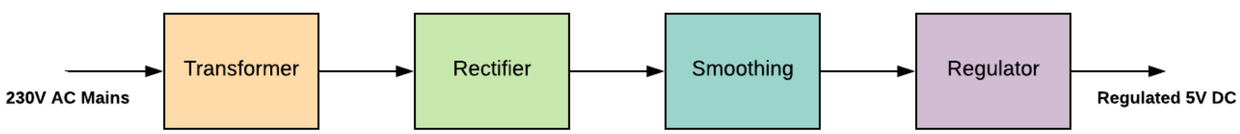


Figure 1.3: Regulated Power Supply Block Diagram

Transformer and AC Supply

A power supply can be used for providing the necessary amount of power at the precise voltage from the main source like a battery. A transformer alters the AC mains voltage toward a necessary value and the main function of this is to step up and step down the voltage. For instance, a step-down transformer is used in a transistor radio, and a step-up transformer is used in a CRT**.** Transformer gives separation from the power-line, and must be used even as any modify within voltage is not required.

Rectifier

A rectifier is a device that converts an oscillating two-directional alternating current (AC) into a single-directional direct current (DC).

Filter

The purpose of power supply filters is to smooth out the ripple contained in the pulses of DC obtained from the rectifier circuit while increasing the average output voltage or current.

Voltage Regulator

A voltage regulator in the regulated power supply is essential for keeping a steady DC output voltage by supplying load regulation as well as line regulation. For this reason, we can employ regulators like a Zener, transistorized, otherwise 3-terminal integrated regulators. An SMPS-switched mode power supply can be used for supplying huge load current by small power dissipation within the series pass transistor.

Applications of Regulated Power Supply

The applications of the regulated power supply include the following.

A regulated power supply (RPS) is an embedded circuit, used to convert unregulated alternating current into a stable direct current by using a rectifier. The main function of this is to supply a constant voltage to a circuit that should be functioned in a particular power supply limit.

- Mobile phone chargers
- Regulated power supplies in different appliances
- Various oscillators & amplifiers

Thus, this is all about a regulated power supply (RPS)**.** From the above information, finally, we can conclude that an RPS changes unregulated alternating current to a stable direct current. A regulated DC power supply is also named as a linear power supply. This supply will allow an AC input as well as provides a stable DC o/p.

Digital Multi-Meter:

The digital multi-meter measures voltage, current and resistance. There are separate settings for measuring AC and DC values. If you examine the soft-key pad on the lower left of the multi-meter, you will find primary functions in black and alternate functions in blue. The alternate functions are accessed by pressing the shift soft-key and then the function soft-key. For example, to measure DC voltage, you press the DC V soft-key. To measure DC current, press the shift soft-key followed by DC, which is the same soft-key used for DC voltage. (Figure 1.4)

Figure 1.4: Digital Multi-Meter

The multi-meter also has the capability of measuring other quantities such as the frequency of periodic waveforms. There are three vertically aligned holes (banana jacks) in the multi-meter located at the top right of the meter front panel. To measure voltage, place the positive terminal in the top hole, with the negative terminal in the center hole. The top hole is labeled Hi in red and V in black. To measure current, place the positive terminal (the terminal at which current enters the multi-meter) in the bottom hole (labeled in red as I), and the other terminal in the enter hole. Other than that, the operation of the digital multi-meter is almost entirely automatic, simply set the multi-meter to the type of measurement you wish to make by pressing the button labeled voltage, current, etc. and read the value from the display**.**

Precautions:

1. Be careful with the DC power supply leads.
2. Avoid letting them touch at all times. When they touch, a short circuit is formed.
3. Consider what would happen if you shorted the wall socket, or a car battery!
4. Short circuits can be dangerous; care should be taken to avoid them.

Project Board (Bread Board):

Product Description: This project board has unique features; the bread boards are mounted on a laminated bakelite sheet fixed on a wooden frame for convenient working. In the following configuration.

1. 384, Groups of 5, connected terminals.

2. 24, Bus of 5, connected terminals.

These project boards are specially designed for assembly experimental circuit in the laboratories,during practicals. These are proved excellent device for solderless assembly of Electronics circuit.(Figure 1.5)

Figure 1.5: Project Board

Bread board:

The breadboard, used for circuit assembly, appears as a symmetrical arrangement of holes (see Figure 1.6). Unseen are metal strips located beneath the proto-board holes, which connect rows or columns of these holes.

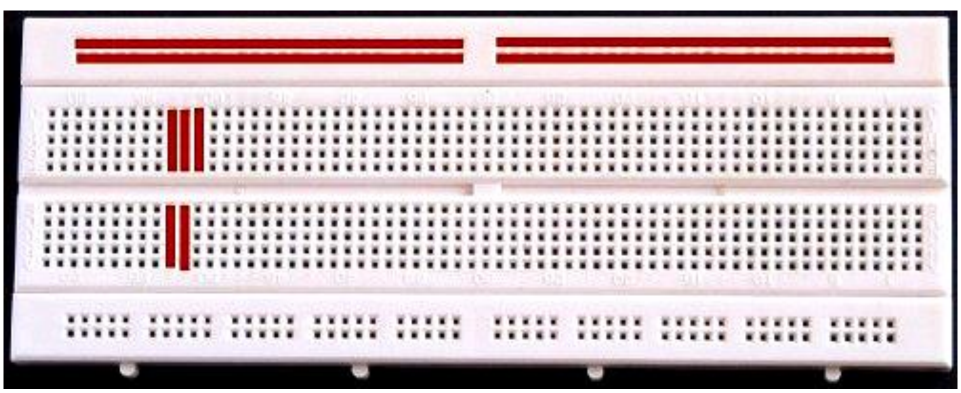


Figure 1.6: Breadboard

Electrical components are mounted on the proto-board. Wires are used to connect sources and decade boxes with the internal wiring of the proto-board to make a complete circuit. Only no. 22 wires may be inserted into the bread-board. Wires should be stripped no more than ¼" before insertion into the board to avoid the possibility of short circuits during circuit construction. Likewise, resistors should be inserted no more than ¼" into the bread-board. Since resistors will be used again, the ends of resistors should not be cut off.

EXPERIMENT 2

Aim: To determine the equivalent resistance of a circuit using colour code and to verify it using a multi-meter. To verify Voltage divider and Current divider rule.

Components Required: Resistors, Multi-meter, Bread Board, DC Power Supply, Connecting Wires.

Activity-1: To determine the equivalent resistance of a circuit using colour code and to verify it using a multi-meter.

Resistors: Resistors are cylindrical shaped components with leads at either end. The resistance in ohms (Ω) associated with the resistor is specified by a colour code (see Figure 2.1) in the form of bands painted on the body of the resistor.

1. The first band is located nearest the end of the resistor, and specifies the first significant digit of the resistance.
2. The second band specifies the second significant digit.
3. The third band tells the power of the ten by which the two-digit number is multiplied to obtain the resistor value.
4. The fourth band indicates the tolerance.

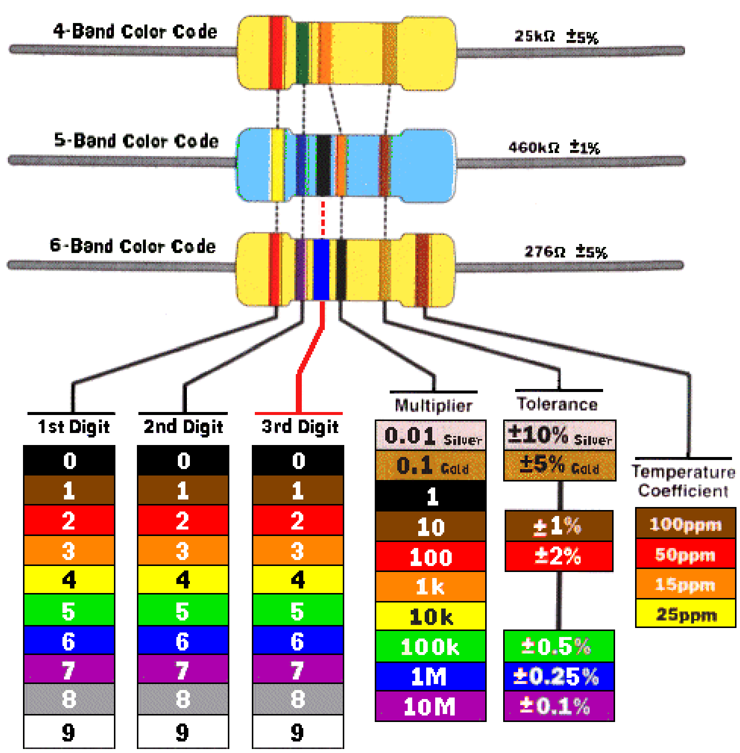


Figure 2.1: Resistance Colour Code

Circuit Diagram:

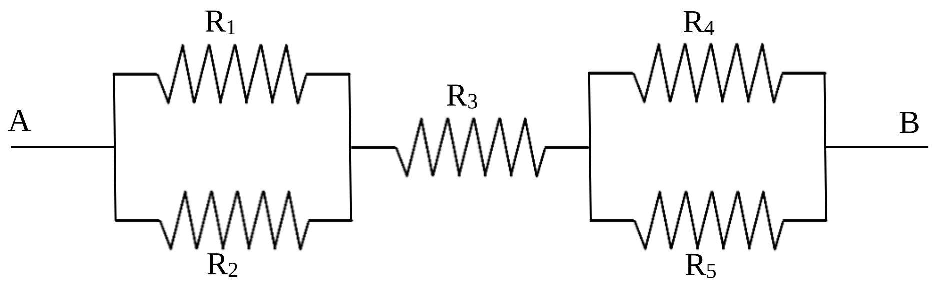


Figure 2.2: Parallel-Series Resistance Circuit

Procedure:

1. Take 5 different resistances. Note down the colours of the bands on these resistances.
2. Using Figure 2.1, determine the values of these resistances.
3. Measure the values of these resistances using a multi-meter.
4. Mount the circuit as shown in figure 2.2on the bread board.
5. Determine the equivalent resistance of the circuit between terminals A and B theoretically and verify using a multi-meter.

Observation:

Resistance	**Colour code**	**Value as per colour code**	**Value measured using Multi-meter**
R_1			
R_2			
R_3			
R_4			
R_5			

Results& Discussion:

Activity-2: **To verify Voltage divider and Current divider rule.**

Voltage divider Rule: In a series circuit, in which the same current flows through all of the components, the total resistance is equal to the sum of the resistance of each of the resistors. In addition, the sum of individual voltage drops across each resistor is equal to the total voltage applied to the circuit. This is often referred to as Kirchoff's Voltage Law.

According to Ohm's law, if the voltage is constant and the resistance is changed, the current must change as well.

To derive the Voltage Divider Rule for, series circuit, we begin with Ohm's law:

$$V = IR \qquad (1)$$

According to Kirchoff's Voltage Law-in accordance with Ohm's Law-there are two distinct voltages that vary according to two distinct resistances in series. Equations 2 and 3 show this relationship.

$$V_{R1} = IR_1 \qquad (2)$$

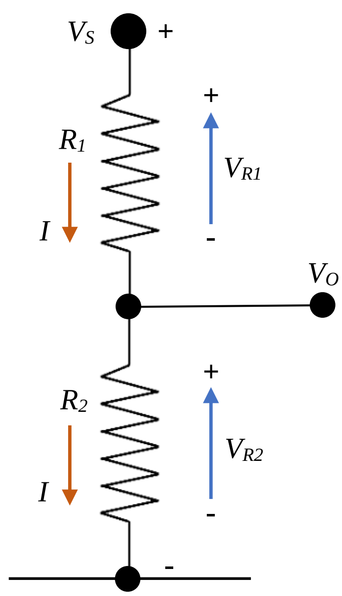


Figure 2.3: Voltage Divider

$$V_{R2} = IR_2 \qquad (3)$$

From Kirchoff's Voltage Law, we can know that the sum of each individual voltage is equivalent to the total voltage of the circuit (Equation 4). From these three equations, we can then make a series of algebraic calculations to determine the Voltage divider rule.

$$V_S = V_{R1} + V_{R2} \qquad (4)$$

$$V_S = (R_1 + R_2)I \tag{5}$$

$$I = \frac{V_S}{R_1+R_2} \tag{6}$$

Thus,

$$V_{out} = V_{R2} = R_2 \frac{V_S}{R_1+R_2} \tag{7}$$

Similarly,

$$V_{out} = V_{R1} = R_1 \frac{V_S}{R_1+R_2} \tag{8}$$

By taking the suitable values of R_1 and R_2, voltage divider rule can be easily verified. V_{R1}

Observation Table:

S. No.	R_1	R_2	V_{R1},(using Multi-meter)	V_{R2}(using Multi-meter)	V_{R1}(Theoretically using voltage divider rule)	V_{R2}(Theoretically using voltage divider rule)
1						
2						
3						
4						

Current Divider Rule:

When current flows through more than one parallel path, each of the paths shares a definite portion of the total current depending upon the impedance of that path. The definite portion of total current shared by any of the parallel paths can easily be calculated if the impedance of that path and the equivalent resistance of the parallel system are known to us. The rule or formula derived from these known impedances to know the portion of total current through any parallel path is known as current division rule**.**

This rule finds application when we have to find the current passing through each impedance when these are connected in parallel. Let us say, two impedances Z_1 and Z_2 are connected in parallel as shown in figure 2.4.

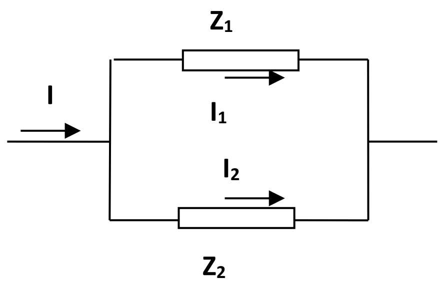


Figure 2.4: Current Divider

A current I pass and is being divided into I_1 and I_2 at the junction of these two impedances as shown. I_1 and I_2 pass through Z_1 and Z_2 respectively. Our aim is to determine I_1 and I_2 in terms of I, Z_1 and Z_2. As Z_1 and Z_2 are connected in parallel, voltage drop across each will be same.

Hence, we can write

$$I_1 Z_1 = Z_2 I_2 \tag{9}$$

Also applying Kirchoff's current law at junction, we get

$$I = I_1 + I_2 \tag{10}$$

We have two equations and can determine I_1 and I_2.

From (9), we have

$$I_2 = \frac{I_1 Z_1}{Z_2} \tag{11}$$

Putting this in (10), we get

$$I = I_1 \left[1 + \frac{Z_1}{Z_2}\right] \tag{12}$$

Or,

$$I_1 = \frac{I}{1+\frac{Z_1}{Z_2}} \tag{13}$$

Or,

$$I_1 = \frac{I Z_2}{Z_1+Z_2} \tag{14}$$

We have,

$$I_2 = \frac{I_1 Z_1}{Z_2} \tag{15}$$

Putting value of I_1, we get

$$I_2 = \frac{I Z_2}{Z_1+Z_2} \times \frac{Z_1}{Z_2} = \frac{I Z_1}{Z_1+Z_2} \tag{16}$$

Thus, we can determine I_1 and I_2 in terms of I, Z_1, Z_2. Therefore, the current division rule can be stated as follows: Suppose, we have to determine the current I_1.

The rule is:

$$I_1 = \frac{Total\ Current}{sum\ of\ impedances} \times Impedance\ through\ which\ current\ is\ not\ determind$$

Applying this rule, we get

$$I_1 = \frac{IZ_2}{Z_1+Z_2}\ \ and\ I_2 = \frac{IZ_1}{Z_1+Z_2}$$

By taking the suitable values of Z_1 and Z_2, current divider rule can be easily verified.

Observation Table:

S. No.	Z_1	Z_2	$I_1(Z_1)$, $I_2(Z_2)$, (using Multi-meter)	$I_1(Z_1)$, $I_2(Z_2)$ (practically using current divider rule)
1				
2				
3				
4				

Procedure:

1. Take resistances. Note down the colours of the bands on these resistances.
2. Using figure 2.1, determine the values of these resistances.
3. Measure the values of these resistances using a multi-meter.
4. Mount the circuit as shown in figure 2.4on the bread board.

Precautions:

1. Care should be taken that low value resistances are not connected across the circuit.
2. Ammeter should be connected in series and voltmeter should be connected in parallel.
3. Take care to use the proper polarity when measuring voltage and current.

Results& Discussion:

EXPERIMENT 3

Aim: To verify Delta to Star and Star to Delta conversion.

Components Required: Resistor, Digital Multi-meter, Regulated DC Power Supply (0-32V), Bread Board.

Theory: Star-Delta Transformations and Delta-Star Transformations allow us to convert impedances connected together in a 3-phase configuration from one type of connection to another. We can now solve simple series, parallel or bridge type resistive networks using Kirchhoff´s Circuit Laws, mesh current analysis or nodal voltage analysis techniques, however in a balanced 3-phase circuit we can use different mathematical techniques to simplify the analysis of the circuit and thereby reduce the amount of math's involved. Standard 3-phase circuits or networks take on two major forms with names that represent the way in which the resistances are connected, a Star connected network which has the symbol of the letter, Y (wye) and a Delta connected network which has the symbol of a triangle, Δ (delta).

If a 3-phase, 3-wire supply or even a 3-phase load is connected in one type of configuration, it can be easily transformed or changed it into an equivalent configuration of the other type by using either the Star Delta Transformation or Delta Star Transformation process. A resistive network consisting of three impedances can be connected together to form a T or "Tee" configuration but the network can also be redrawn to form a Star or Y type network as shown below.

Delta Star Transformation:

To convert a delta network to an equivalent star network, we need to derive a transformation formula for equating the various resistors to each other between the various terminals.

Delta to Star Network:

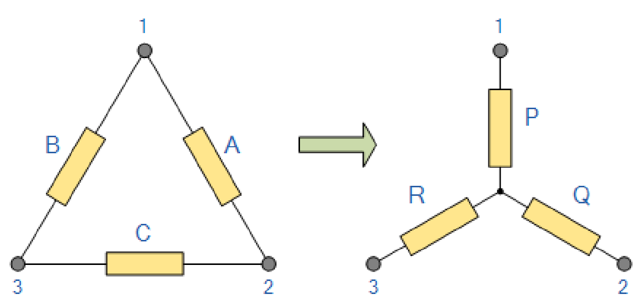


Figure3.1: Delta to Star Conversion

Compare the resistances between terminals1 and 2.

P+Q = *A* in Parallel with *(B+C)*

$$P + Q = \frac{A(B+C)}{A+B+C} \tag{1}$$

Resistance between the terminals 2 and 3.

Q+R = *C* in parallel with *(A+B)*

$$Q + R = \frac{C(A+B)}{A+B+C} \tag{2}$$

Resistance between the terminals 1 and 3

P+R =*B* in parallel with *(A+C)*

$$P + R = \frac{B(A+C)}{A+B+C} \tag{3}$$

Now, subtracting equation 2 from equation 3 and represented as follows:

$$P - Q = \frac{BA+CB}{A+B+C} - \frac{CA+CB}{A+B+C}$$

$$P - Q = \frac{BA-CA}{A+B+C} \tag{4}$$

With the help of equation (1) and equation (4)

$$(P+Q)+(P-Q) = \frac{AB+AC}{A+B+C} + \frac{BA-CA}{A+B+C}$$

Final equation for resistor P is given as:

$$P = \frac{AB}{A+B+C}$$

Then to summarize a little about the above maths, we can now say that resistor P in a Star network can be found as Equation 1 plus (Equation 3 minus Equation 2). Similarly, to find resistor Qin a star network, is equation 2 plus the result of (equation 1 minus equation 3) and this gives us the transformation of Q as:

$$Q = \frac{AC}{A+B+C}$$

and again, to find resistor R in a Star network, is equation 3 plus the result of equation 2 minus equation 1 and this gives us the transformation of R as:

$$R = \frac{BC}{A+B+C}$$

When converting a delta network into a star network the denominators of all of the transformation formulas are the same: $A + B + C$, and which is the sum of all the delta resistances. Then to convert any delta connected network to an equivalent star network we can summarize the above transformation equations as:

$$P = \frac{AB}{A+B+C}, Q = \frac{AC}{A+B+C}, R = \frac{BC}{A+B+C}$$

Star to Delta Transformation:

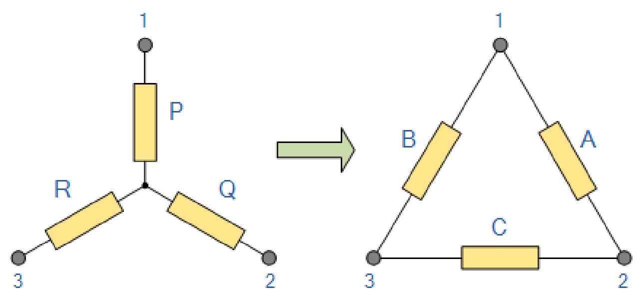


Figure 3.2: Star to Delta Conversion

The value of the resistor on any one side of the delta, Δ network is the sum of all the two-product combinations of resistors in the star network divide by the star resistor located "directly opposite" the delta resistor being found. For example, resistor A is given as:

$$A = \frac{PQ + QR + RP}{R}, \quad B = \frac{PQ + QR + RP}{Q}, \quad C = \frac{PQ + QR + RP}{P}$$

Procedure:

1. Take three resistances and determine their value using colour code and verify using Multi-meter
2. Connect the circuit on the bread board as shown in the figure 3.1 and 3.2.
3. Name the nodes and find the corresponding output values of resistance of the circuit.

Observations:

Delta to Star Conversion:

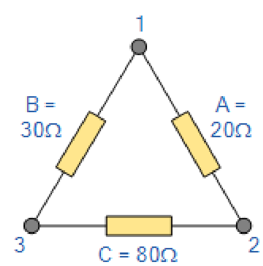
S. No.	Practical Resistance value b/w the Node	In Delta Network	In Star Network
1.	Resistance b/w 1-2		
2.	Resistance b/w 2-3		
3.	Resistance b/w 3-1		

Star to Delta Conversion:

S. No.	Practical Resistance value b/w the Node	In Star Network	In Delta Network
1.	Resistance b/w 1-2		
2.	Resistance b/w 2-3		
3.	Resistance b/w 3-1		

Examples: Delta – Star

Q.1 Convert the following Delta Resistive Network into an equivalent Star Network.



Answer:

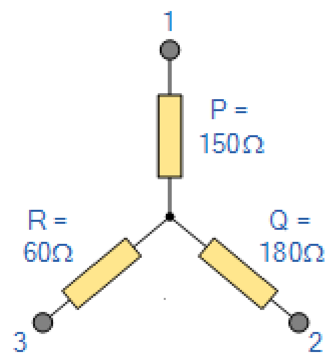
$$P = \frac{AB}{A+B+C} = \mathbf{4.61\ \Omega}$$

$$Q = \frac{AC}{A+B+C} = \mathbf{12.31\ \Omega}$$

$$R = \frac{BC}{A+B+C} = \mathbf{18.46\ \Omega}$$

Star – Delta:

Q.1 Convert the following Star Network into an equivalent Delta Resistive Network.

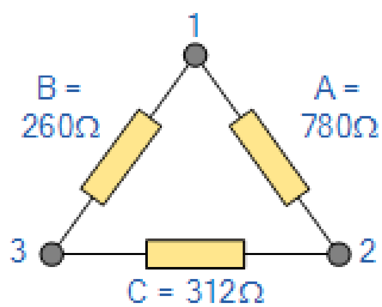


Answer:

$$A = \frac{PQ + QR + RP}{R} = 780\Omega$$

$$B = \frac{PQ + QR + RP}{Q} = 260\ \Omega$$

$$C = \frac{PQ + QR + RP}{P} = 312\ \Omega$$



EXPERIMENT4

Aim: Introduction to DSO & Function Generator.

Apparatus Required: DSO, Function Generator.

Theory: A digital storage oscilloscope (often abbreviated DSO) is an oscilloscope which stores and analyses the signal digitally rather than using analog techniques. It is now the most common type of oscilloscope in use because of the advanced trigger, storage, display and measurement features which it typically provides.

Digital Storage Oscilloscope:

Front Pane:

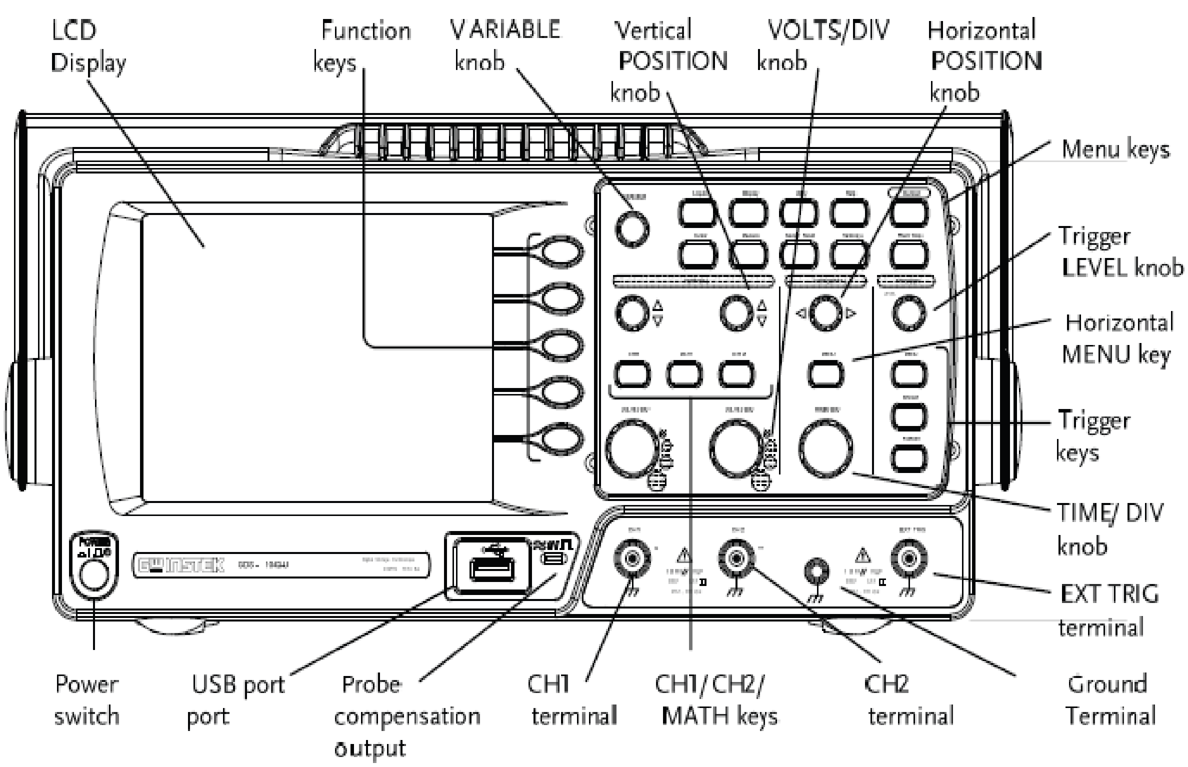


Figure 4.1: Front panel view of DSO

The functioning of various switches in the front panel of Figure 4.1 is as following:-

Table 4.1

Switch Name	Function of switch
Display	LCD display TFT colour, 320 x234 resolution, wide angle view LCD display
Function keys:F1 (top) to F5 (bottom)	Activates the functions which appear in the left side of the LCD display
Variable knob	Increases or decreases values and moves to the next or previous parameter
Display key	Configures the display settings
Acquire key	Configures the acquisition mode
Cursor key	Runs cursor measurements
Utility key	Configures the Hardcopy function, shows the system status, selects the menu language, runs the self-calibration, configures the probe compensation signal, and selects the USB host type
Help key	Shows the Help contents on the display
Auto-set key	Automatically configures the horizontal, vertical, and trigger settings according to the input signal
Measure key	Configures and runs automatic measurements
Save/Recall key	Saves and recalls images, waveforms, or panel settings
Hardcopy key	Stores images, waveforms, or panel settings to USB
Run/Stop key	Runs or stops triggering
Trigger level knob	Sets the trigger level
Trigger menu key	Configures the trigger settings
SINGLE TRIGGRING key	Selects the single triggering mode
Trigger force key	Acquires the input signal once regardless of the trigger conditional the time
Horizontal menu	Configures the horizontal view Key
Horizontal position knob	Moves the waveform horizontally
TIME/DIV knob	Selects the horizontal scale
Vertical position knob	Moves the waveform vertically
CH1/CH2 key	Configures the vertical scale and coupling mode for each channel
VOLTS/DIV knob	Selects the vertical scale

Input terminal	Accepts input signals: 1MΩ±2% input impedance, BNC terminal.
Ground terminal	Accepts the DUT ground lead to achieve a common ground
MATH key	Performs math operations
USB port	Facilitates transferring waveform data, display images, and panel settings
Probe Compensation output	Outputs a 2Vp-p, square signal for compensating the probe or demonstration
External trigger input	Accepts an external trigger signal
Power switch	Powers the oscilloscope on or off

Function Generator:

A function generator is usually a piece of electronic test equipment or software used to generate different types of electrical wave forms 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. These waveforms can be either repetitive or single-shot (which requires an internal or external trigger source).Integrated circuits used to generate waveforms may also be described as function generator ICs.

In addition to producing sine waves, function generators may typically produce other repetitive waveforms including saw-tooth and triangular waveforms, square waves, and pulses. Another feature included on many function generators is the ability to add a DC offset.

Although function generators cover both audio and RF frequencies, they are usually not suitable for applications that need low distortion or stable frequency signals. When those traits are required, other signal generators would be more appropriate.

Some function generators can be phase-locked to an external signal source (which may be a frequency reference) or another function generator.

Function generators are used in the development, test and repair of electronic equipment. For example, they may be used as a signal source to test amplifiers or to introduce an error signal into a control loop. Function generators are primarily used for working with analog circuits, related pulse generators are primarily used for working with digital circuits. The view of function generator is shown in figure 2.

Front panel of Function Generator:

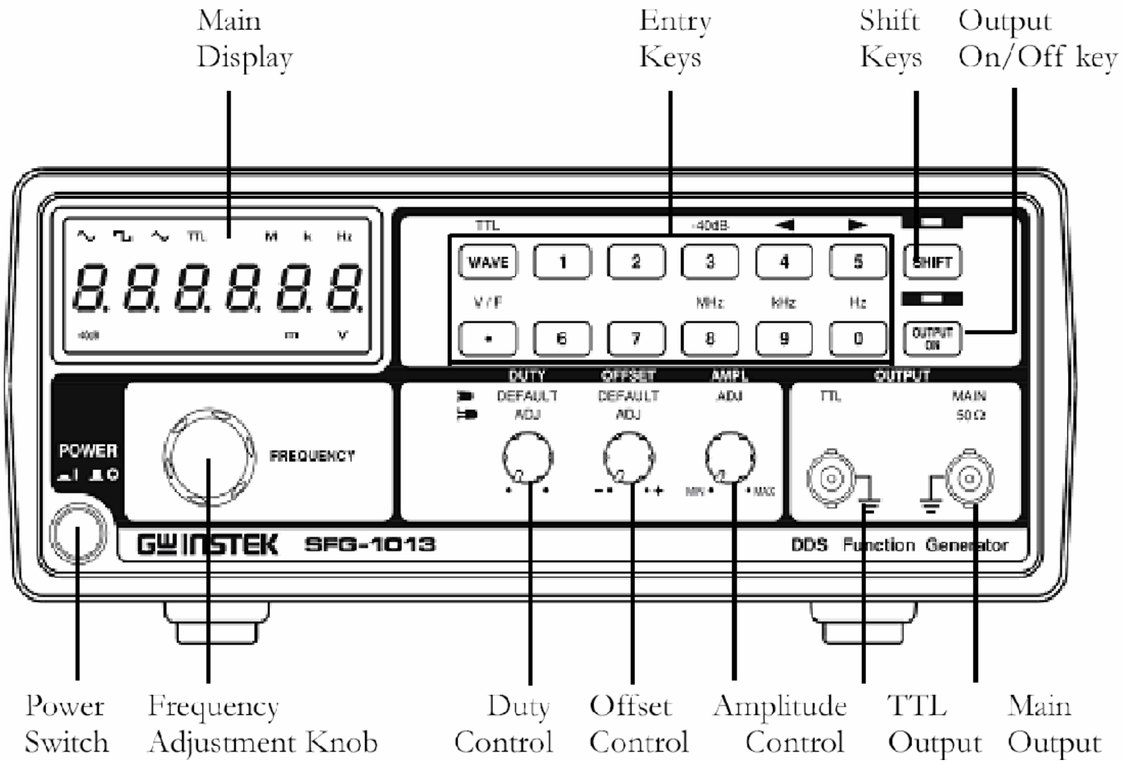


Figure 4.2: Front panel view of Function Generator

The functioning of various switches in the front panel of Figure 4.2 is as following:

Table 4.2

Switch Name	Function of switch
7 segment LED Main Display	Shows frequency and voltage
TTL indicator	Indicates that the TTL output is enabled.
Waveform indicator	Indicates the waveform shape: Sine, Square, and Triangle
Frequency indicator	Indicates the output frequency: MHz, kHz, or Hz
Voltage indicator	Indicates Voltage unit: mV, or V. For voltage Measurement detail.
-40dB indicator	Indicates −40dB attenuation is activated
Waveform key	Selects the waveform: sine, square, and triangle. For details,
TTL activation	Activates TTL output. For details

Numerical keys	Specifies frequency
Frequency unit selection	Specifies the frequency unit: MHz, kHz, or Hz.
Cursor selection	Moves the cursor (frequency editing point) left or right
−40dB attenuation	Attenuates amplitude by −40dB. For details
Frequency /Voltage display selection	Switches the display between frequency and voltage
Shift key	Selects the 2nd function associated to the entry keys. The LED lights when Shift is activated.
Output On/Off Key	Turns the output On/Off. The LED lights when the output is On.
Frequency editing knob	Increases (right turn) or decreases (left turn) the frequency
Main output	Outputs sine, square, and triangle waveform. BNC, 50Ω output impedance
TTL output	Outputs TTL output waveform, BNC terminal. For TTL mode
Amplitude Control	Sets the sine/square/triangle waveform amplitude. Turn left (decrease) or right (increase)
DC offset Control	When pulled out, sets the DC offset level for sine/square/triangle waveform. Turn left (decrease) or right (increase). The range is −5V ~ +5V, in 50Ω load
Duty Cycle Control	When pulled out, sets the square / TTL wave Duty Cycle
Power Switch	Turn the main power ON/OFF

EXPERIMENT 5

Aim: **To Verify Kirchoff's Voltage Law and Verification of the Kirchoff's Current Law.**

Acitvity-1: **To Verify Kirchoff's Voltage Law.**

Components Required: Resistor,Digital Multi-meter,DC Power Supply (0-32V),Bread board, Connecting Wires.

Theory: In any closed loop of an electric network, the algebraic sum of all EMF and the entire voltage drop is zero i.e. in the closed loop of any electrical circuit the algebraic sum of total voltage source present in the circuit and the voltage drop across various resistances is zero.It can be given by the equation

$$\sum \boldsymbol{EMF} + \sum \boldsymbol{IR} = \mathbf{0}$$

This theorem is applicable to linear circuits only.

Circuit diagram:

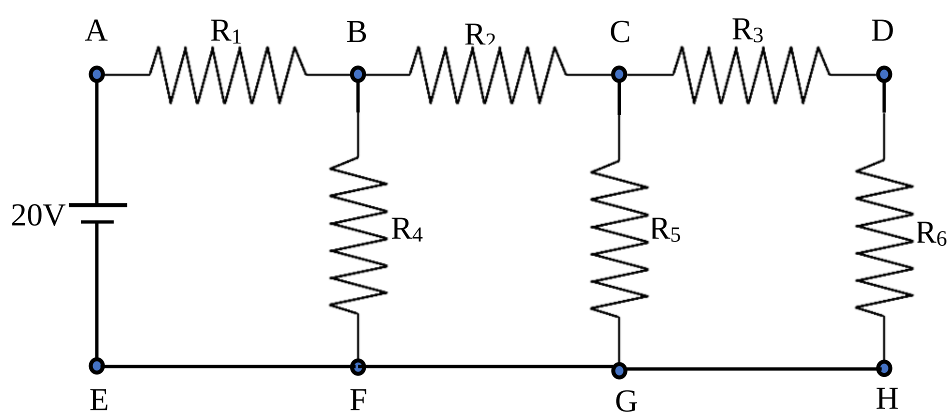


Figure5.1: Kirchoff's Voltage Law

Procedure:

1. Take six resistances and determine their value using colour code and verify using Multi-meter.
2. Connect the circuit on the bread board as shown in the figure 5.1
3. Power supply of 20 volts should be applied across the circuit.
4. Name the nodes and select one loop, say ABEF.
5. Calculate the voltage across the resistances in this loop with the help of multi-meter.
6. Find the algebraic sum of the voltages in this loop.

7. Repeat for other loops in the circuit.
8. If the algebraic sum of EMFs in the loops equals zero, the theory is verified.

Precautions:

1. Care should be taken that low value resistances are not connected across the circuit.
2. Ammeter should be connected in series and voltmeter should be connected in parallel.
3. Take care to use the proper polarity when measuring voltage and current.

Observations: Resistance Values

R_1 =	R_2 =	R_3 =
R_4=	R_5 =	R_6 =

Loop ABFE: Voltage across R_1 = Voltage across R_4 = Voltage across EA = Algebraic sum =	**Loop ADHE:** Voltage across R_1 = Voltage across R_2 = Voltage across R_3 = Voltage across R_6 = Voltage across EA = Algebraic sum =
Loop BCGF: Voltage across R_2 = Voltage across R_4 = Voltage across R_5 = Algebraic sum =	**Loop CDHG:** Voltage across R_3 = Voltage across R_5 = Voltage across R_6 = Algebraic sum =
Loop BDHF: Voltage across R_2 = Voltage across R_3 = Voltage across R_4 = Voltage across R_6 = Algebraic sum =	**Loop ACGE:** Voltage across R_1 = Voltage across R_2 = Voltage across R_5 = Voltage across EA = Algebraic sum =

Results& Discussion:

Acitvity-2: To Verify Kirchoff's Current Law.

Components Required: Resistors, Digital Multi-meter, Regulated DC Power Supply(0-32V), Bread Board, Connecting Wires.

Theory: In any electrical network, the algebraic sum of currents meeting at any node or junction is zero i.e. the total value of electric current entering the node and the total current leaving the node is equal.

It is given by the equation

$$\sum I = 0$$

This theorem is restricted to linear elements only.

Circuit diagram:

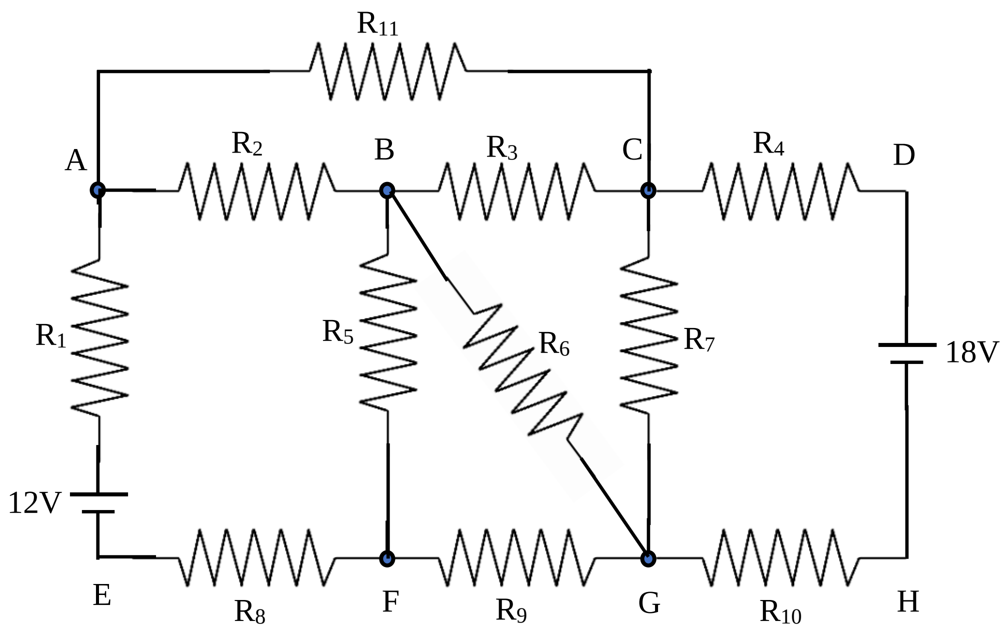


Figure 5.2: Kirchoff's Current Law

Procedure:

1. Take eleven resistances and determine their value using colour code and verify using Multi-meter.
2. Connect the circuit on the bread board as shown in the figure 5.2.
3. Name the nodes and select one node, say A.
4. Measure the current flowing through the branches connected at this node with the help of multi-meter.

5. Find the algebraic sum of the currents at this node.
6. Repeat for other nodes in the circuit.

Observations: Resistance Values

R_1 =	**R_2 =**	**R_3 =**
R_4=	**R_5 =**	**R_6 =**
R_7 =	**R_8 =**	**R_9 =**
R_{10}=	**R_{11}=**	

Node A: Current through R_1 = Current through R_2 = Current through R_{11} = Algebraic sum =	**Node B:** Current through R_2 = Current through R_3 = Current through R_5 = Current through R_6 = Algebraic sum =
Node C: Current through R_3 = Current through R_4 = Current through R_{11} = Current through R_7 = Algebraic sum =	**Node F:** Current through R_5 = Current through R_8 = Current through R_9 = Algebraic sum =
Node G: Current through R_6 = Current through R_7 = Current through R_9 = Current through R_{10} = Algebraic sum =	

Precautions:

1. Care should be taken that low value resistances are not connected across the circuit.
2. Ammeter should be connected in series and voltmeter should be connected in parallel.
3. Take care to use the proper polarity when measuring voltage and current.

Results& Discussion:

EXPERIMENT 6

Aim: **To verify superposition theorem**.

Components Required: Resistor, Digital Multi-meter, DC Power Supply (0-32V), Bread Board.

Theory: According to superposition theorem for an electrical circuit, the total current flowing in part of electrical circuit is equal to algebraic sum of currents produced by each source acting separately. To measure the current due to an individual source, all other sources are removed in the following manner -

1. All other voltage sources are replaced by short circuit.
2. All other current sources are replaced by open circuit.

i.e. voltage across the resistance R_3 due to the presence of two sources V_1 and V_2 is equal to the algebraic sum of voltage present due to voltage source $V_1$and V_2 when present alone.

Circuit diagram:

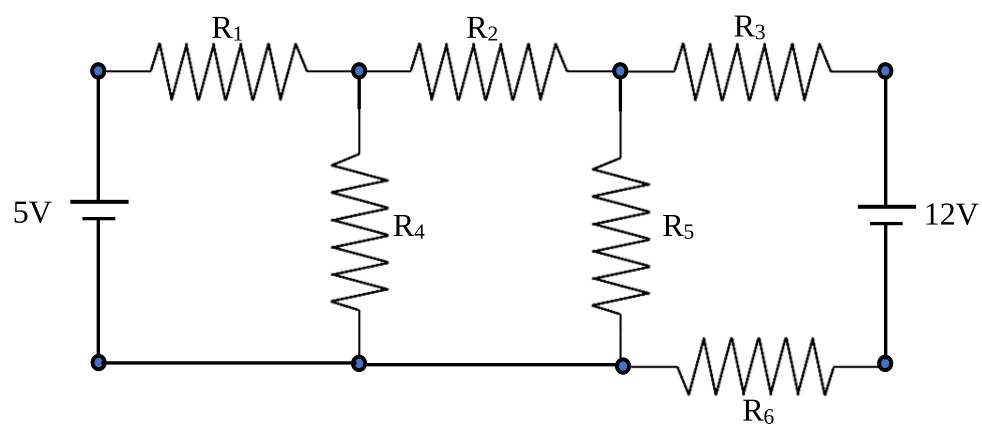


Figure 6.1: Superposition Theorem

Procedure:

1. Connect the components on the bread current as shown in the figure 6.1.
2. Connect the two-power supply V_1 & V_2 as shown in the circuit and apply voltage of value 5V & 12V respectively through the supply.
3. Measure the voltage across the resistances.
4. Now replace the voltage source V_1 with a short and measure the voltage across the resistances.
5. Now repeat the above procedure using V_2.

6. If the sum of voltages appearing across the resistance is equal to the voltage appearing across it when both the sources are in the circuit, the theorem stands verified.

Precautions:

1. Care should be taken that low value resistances are not connected across the circuit.
2. Ammeter should be connected in series and voltmeter should Be connected in parallel.
3. Take care to use the proper polarity when measuring voltage and current.
4. The positive side of V_1 and V_2 must have at least one resistance**.**

Observations: Resistance Values:

R_1=	**R_2=**	**R_3=**
R_4=	**R_5=**	**R_6=**

Measured Potentials:

Active *Vs*	V_{R1}	V_{R2}	V_{R3}	V_{R4}	V_{R5}	V_{R6}
V_1 Only						
V_2Only						
Both						
%Error						

Precautions:

1. Care should be taken that low value resistances are not connected across the circuit.
2. Ammeter should be connected in series and voltmeter should be connected in parallel.
3. Take care to use the proper polarity when measuring voltage and current.

Results& Discussion:

EXPERIMENT 7

Aim: To verify Norton's theorem.

Components required: Resistors, Digital Multi-Meter, DC Power Supply (0-32V), Current Source, Bread Board, Connecting Wires.

Theory: Norton theorem states that any combination of voltage sources, current sources and resistors with two terminals is electrically equivalent to a single current source I and a single series resistor R i.e. consider a circuit with a large number of components like resistances, inductances, capacitances, voltage source &current sources. All the components can be replaced by a current source and resistor.

Circuit diagram:

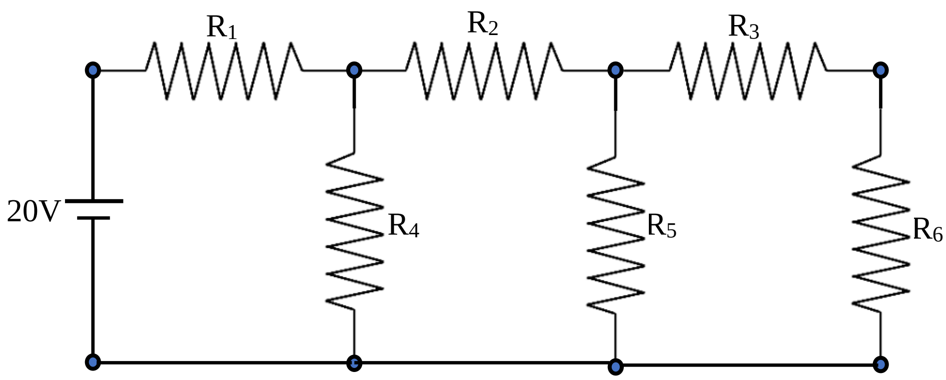


Figure7.1: Norton's theorem

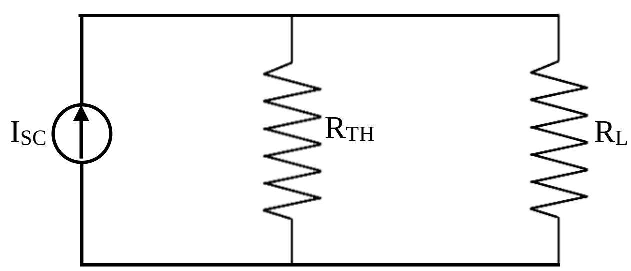


Figure7.2: Norton's Equivalent Circuit

Procedure:

1. Connect the circuit on the bread board as shown in the figure 7.1.
2. Replace the voltage sources in the circuit with their internal resistances.
3. Remove load resistance and measure equivalent resistance R_{TH} looking into the circuit from these terminals.

4. Remove load resistance and measure short circuit current, I_{SC} flowing through these terminals
5. Replace load resistance in the circuit and measure the voltage V_{L1} across R_L due to the presence of the voltage source V_{S1}.
6. Mount the Norton's equivalent circuit using current I_{SC} and resistance R_{TH} in parallel across the load resistance as shown in Fig .7.2.
7. Measure the voltage drop V_{L2} across the load resistance in this circuit. If both these V_{L1} & V_{L2} are equal, the theorem stands verified.

Observations: Resistance Values

R_1=	**R_2=**	**R_3=**
R_4=	**R_5=**	**R_6=**
R_{Th}=	**I_{sc}=**	

Observation Table:

Value Determination	**V_{L1} in original circuit**	**V_{L2} in Norton's Equivalent circuit**
Calculated		
Measured		

Precautions:

1. Care should be taken that low value resistances are not connected across the circuit.
2. Ammeter should be connected in series and voltmeter should be connected in parallel.
3. Take care to use the proper polarity when measuring voltage and current.

Results& Discussion:

EXPERIMENT 8

Aim: To verify Thevenin's Theorem and Maximum Power Transfer Theorem.

Activity-1: To verify Thevenin's Theorem.

Components required: Resistors, Digital Multi-Meter, DC Power Supply (0-32V), Bread Board, Connecting Wires.

Theory: Thevenin's theorem states that any combination of voltage sources, current sources and resistor with two terminals is electrically equivalent to a single voltage source V and a single series resistor R i.e. consider a circuit with a large number of components like resistances, inductances, capacitances, voltage source &current sources. All the components can be replaced by a voltage sources and resistor.

Circuit diagram:

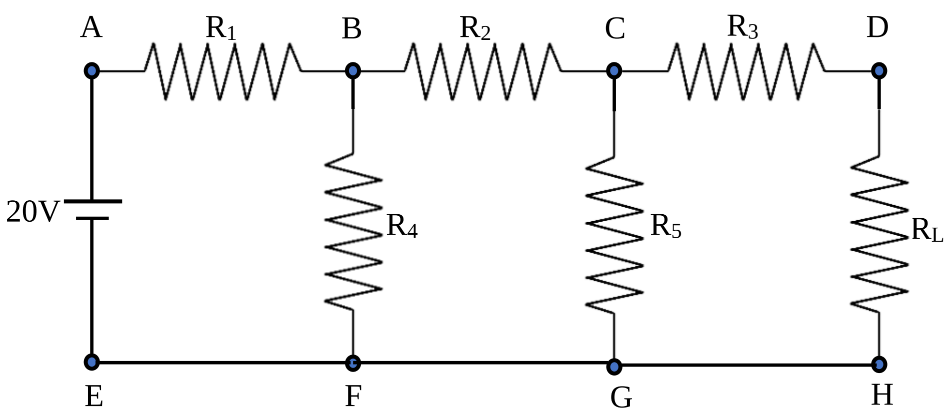


Figure8.1: Thevenin's Theorem

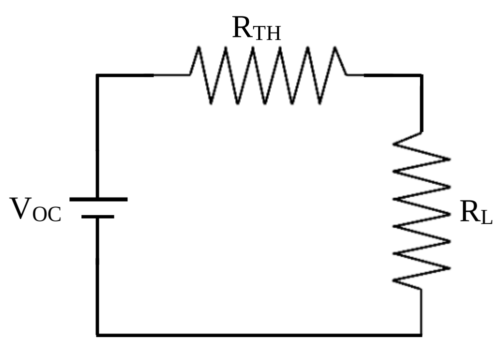


Figure 8.2: Thevenin's Equivalent Circuit

Procedure:

1. Connect the circuit on the bread board as shown in the figure 8.1.
2. Replace the voltage sources in the circuit with their internal resistances.

3. Remove load resistance and measure equivalent resistance R_{TH} looking into the circuit from these terminals.
4. Remove load resistance and measure open circuit voltage, V_{OC} at these terminals
5. Replace load resistance in the circuit and now measure the voltage V_{L1} across R_L due to the presence of the voltage source V_1.
6. Mount the Thevenin's equivalent circuit using voltage V_{OC} and resistance R_{TH} in series across the load resistance as shown in Fig 8.2
7. Measure the voltage drop V_{L2} across the load resistance in this circuit.
8. If both these V_{L1} & V_{L2} are equal, the theorem stands verified.

Observations: Resistance Values

R_1=	**R_2 =**	**R_3 =**
R_4=	**R_5=**	**R_6 =**
R_{TH} =	**V_{OC} =**	

Observation Table:

Value Determination	V_{L1} in original circuit	V_{L2} in Thevenin's Equivalent circuit
Calculated		
Measured		

Activity-2: To verify maximum power transfer theorem.

Components required: Resistors, Digital Multi-meter, DC Power Supply (0-32V), Bread board, Connecting Wires.

Theory: The maximum power (transfer) theorem states that, to obtain maximum external power from a source with a finite internal resistance, the resistance of the load must be made the same as that of the source. The theorem applies to maximum power. Again, if the resistance of the load is made larger than the resistance of the source, then efficiency is higher, since most of the power is generated in the load. But the overall power is lower since the total circuit resistance goes up.

Circuit diagram:

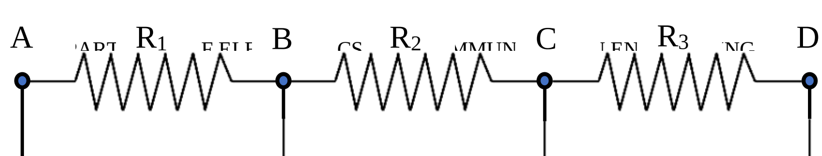


Figure8.3: Maximum Power Transfer Theorem

Procedure:

1. Connect the circuit as shown by Figure 8.3 on the bread board.
2. Replace the voltage sources in the circuit with their internal resistances.
3. Remove load resistance and measure equivalent resistance R_{TH} looking into the circuit from these terminals.
4. Replace the load resistance and the voltage source in the circuit.
5. Measure the voltage drop V_L across the load resistance in this circuit.
6. Find the power consumed by R_L.
7. Now select different values of R_L and measure the voltage drop V_L and the power consumed by R_L.
8. Determine the value at which maximum power is consumed by R_L.
9. If this value of R_L is equal to R_{TH}, the theorem is verified.

Observation Table: R_{TH}=______________

S. No.	Value of R_L	V_L across R_L	Power $P=(V_L)^2/R_L$
1			
2			
3			

Precautions:

1. Care should be taken that low value resistances are not connected across the circuit.
2. Ammeter should be connected in series and voltmeter should be connected in parallel.
3. Take care to use the proper polarity when measuring voltage and current.

Results& Discussion:

EXPERIMENT 9

Aim: To study the transient response of series RC circuits using different values of R and C.

Apparatus Required: DSO, Multi-meter, Breadboard, Function Generator, Connecting Wires.

Component Required: Resistors 2.2kΩ, 100kΩ, Capacitors 0.1µF, 0.01µF

Theory: A capacitor has the ability to store an electrical charge and energy. The voltage across the capacitor is related to the charge by the equation *V=Q/C* for steady state values, or expressed as an instantaneous value *dv=dq/C* we will study the transient response of the *RC* circuit, which is the response to a sudden change in voltage.

In this experiment, we apply a pulse waveform to the *RC* circuit to analyze the transient response of the circuit. The pulse-width relative to a circuit's time constant determines how it is affected by an *RC* circuit.

Time Constant (τ): A measure of time required for certain changes in voltages and currents in *RC* and *RL* circuits. Generally, when the elapsed time exceeds five-time constants (5τ) after switching has occurred, the currents and voltages have reached their final value, which is also called steady-state response.

The time constant of an *RC* circuit is the product of equivalent capacitance and the Thevenin's resistance as viewed from the terminals of the equivalent capacitor.

$$\tau = RC \tag{1}$$

A Pulse is a voltage or current that changes from one level to the other and back again. If a waveform's high time equals its low time, it is called a square wave. The length of each cycle of a pulse train is termed its period (*T*).

The pulse width (t_p) of an ideal square wave is equal to half the time period. The relation between pulse width and frequency is then given by,

$$f = \frac{1}{2t_p} \tag{2}$$

A series *RC* circuit is shown in Figure1. From Kirchoff's laws, it can be shown that the charging voltage $V_C(t)$ across the capacitor is given by:

$$V_c(t) = V\left(1 - e^{-t/RC}\right), \quad t \geq 0 \tag{3}$$

Where, V is the applied source voltage to the circuit for $t \geq 0$. $\tau = RC$ is the time constant. The response curve, showing capacitor charging for Series RC circuit to a step input with time axis normalized by τ is shown in Figure 9.1.

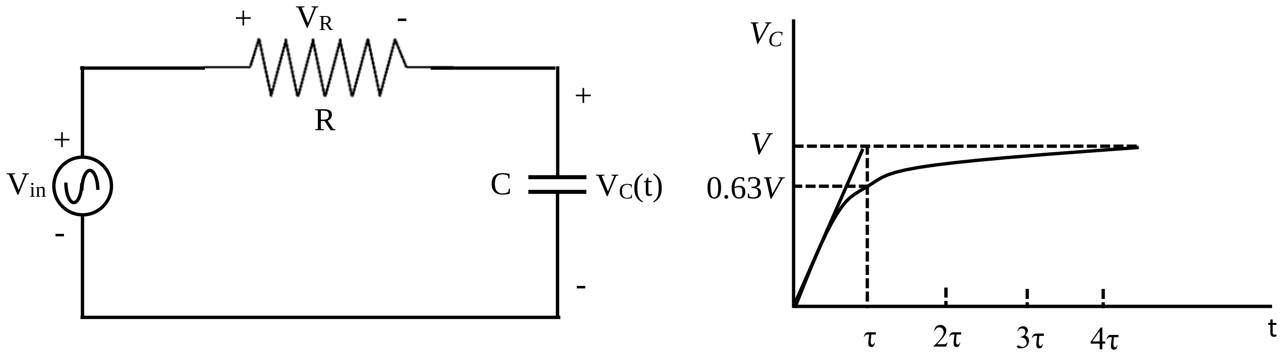
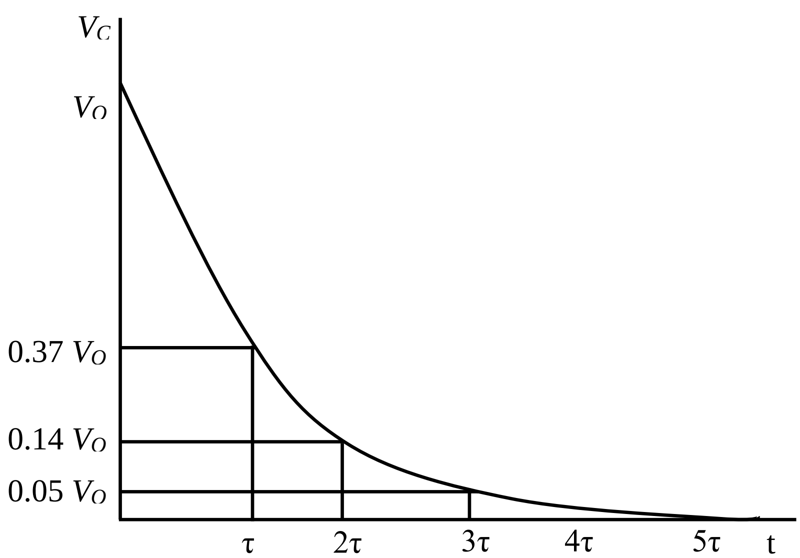


Figure 9.1: RC Circuit

The discharge voltage for the capacitor is given by:

$$V_C(t) = V_o e^{-t/RC}, \qquad t \geq 0 \tag{4}$$

Where V_o is the initial voltage stored in capacitor at $t = 0$, and $\tau = RC$ is time constant. The response curve is a decaying exponential as shown in Figure 9.2.



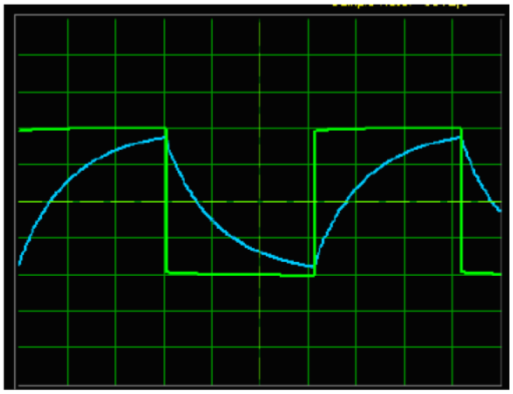
t	$\boldsymbol{e^{-1/\tau}}$
τ	**0.37**
2τ	**0.14**
3τ	**0.05**
4τ	**0.02**
5τ	**0.01**

Figure 9.2: RC response curve

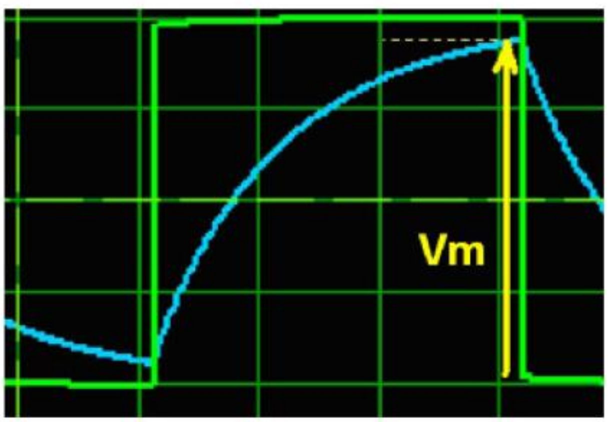
Procedure:

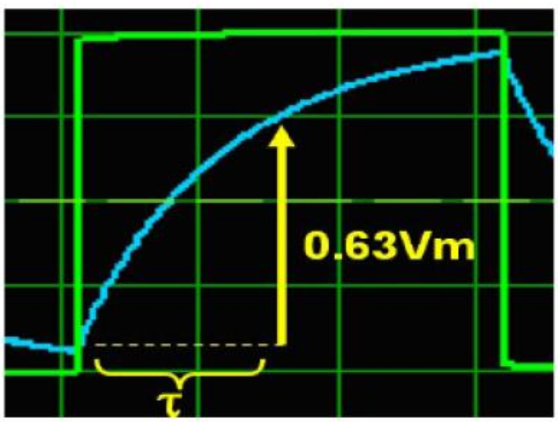
1. Set up the circuit shown in Figure9.1 with the component values R=2.2 kΩ and C=0.1μF.
2. Set the Function Generator to generate a 4Vp-p square wave and apply as input voltage to the circuit.
3. Observe the input square wave on channel 1 and output, across the capacitor, on channel 2 of the CRO. Set the volt/div same for both the channels, as shown in Figure.
4. Observe the response of the circuit for the following three cases and record the results.

a). $t_p >> 5\tau$: Set the frequency of the function generator output such that the capacitor has enough time to fully charge and discharge during each cycle of the square wave. So, let $t_p = 15\tau$ and accordingly set the function generator frequency using equation (2). The value you have found should be approximately 150 Hz. Determine the time constant from the waveforms obtained on the CRO. (At $t = \tau$, $V_c(t) = 0.63V$ from equation (3)).



b). $t_p = 5\tau$: Set the frequency such that $t_p = 5\tau$ (this should be 450 Hz). Since the pulse width is exactly 5τ, the capacitor should just be able to fully charge and discharge during each pulse cycle. From the figure determine τ.





c).t_p<< 5τ : In this case the capacitor does not have time to charge significantly before it is switched to discharge, and vice versa. Let $t_p = 0.5\tau$ in this case and set the frequency accordingly.

5. Repeat the procedure using R = 100 kΩ and C = 0.01 μF and record the measurements.

Precaution:

1. Care should be taken that low value resistances are not connected across the circuit.
2. Capacitors should be connected in same polarity.
3. Voltmeter should be connected in parallel.
4. Take care to use the proper polarity when measuring voltage.

Results& Discussion:

EXPERIMENT 10

Aim: **Determination of frequency response of current in RLC circuit with sinusoidal ac input.**

Apparatus Required: DSO, Multi-meter, Breadboard, Function Generator.

Component Required: Resistors - 2.2kΩ, 100kΩ, Capacitors - 0.1μF, 0.01μF, Inductor.

Theory: The voltage through an *RLC* series circuit will be measured as a function of frequency for a fixed applied voltage. The frequency for which the *rms* voltage attains a maximum value is the resonance frequency. The expected resonance frequency is given by equation

$$f_0 = \frac{1}{2\pi} \cdot \frac{1}{\sqrt{LC}}$$

Circuit diagram:

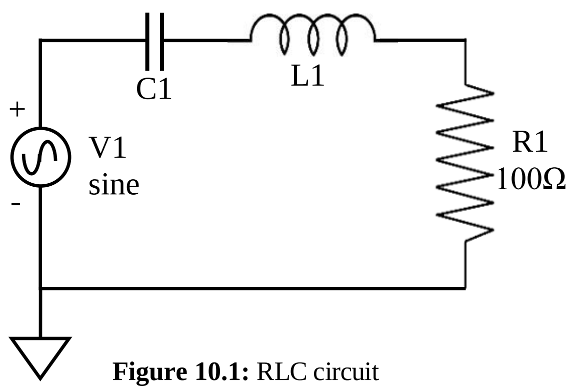


Figure 10.1: RLC circuit

Procedure:

1. Before you connect the circuit to the function generator set the frequency to 60 Hz. Then, using the voltmeter set the generator's output to 5 volts (rms).
2. Using the proto-board and wire leads connect the resistor, capacitor, and inductor along with the output of the function generator to construct the circuit shown in Figure 1. Here we are measuring the peak to peak voltage across the resistor using the oscilloscope. The three components are connected in series with the function generator acting as the power supply. Connect the black leads together at the end of the resistor as noted in Figure 1.
3. Record the values of R, L, and C for this circuit in the space provided in the data section.
4. Use equation 1 to compute the expected resonance frequency and record your result in data table 1.
5. Change the function generator frequency to 50Hz and record the peak to peak voltage from the oscilloscope in data table 2. Then, adjust the output frequency to 100 Hz and record the

voltage. Adjust the output frequency to 200 Hz and record the voltage. Continue adjusting the output frequency to each value below the expected resonance frequency computed in step 4. Record the voltage for each of these values.

6. Determine an experimental value for resonance frequency by finding the frequency that produces the largest voltage on the oscilloscope. Record this frequency and voltage.
7. Record the voltage for frequency values that are above the resonance frequency determined in step 6.
8. Turn all equipment off and disconnect the circuit.

Analysis:

1. Use Excel to produce a plot of frequency vs. voltage. Set the frequency axis to a logarithmic scale. To do this right click on the axis; chose 'format axis' and check the box for 'logarithmic scale'.
2. Draw a smooth curve through all the data points. This curve should be similar to figure 1.
3. Use the graph to determine the resonance frequency.
4. Compare the experimental resonance frequency to the expected value obtained from equation 1.

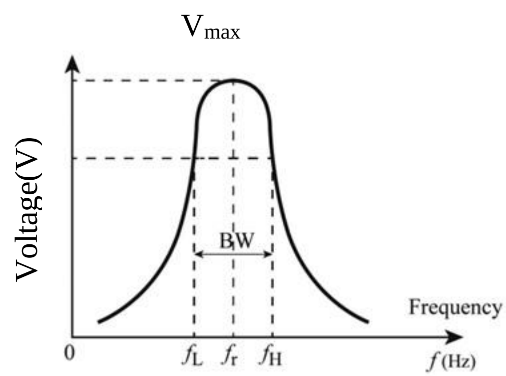


Figure 10.2: Frequency Response Curve

Table1

Observations:

Calculated Resonant Frequency	
Experimental Resonant Frequency	
%Difference	

Circuit Parameters:

R=

L=

C=

Table2

S No	Frequency	Peak to Peak Voltage(V_o)
1	50Hz	
2	100Hz	
3	400Hz	
4	800Hz	
5	1KHz	
6	5 KHz	
7	10 KHz	
8	50 KHz	
9	100 KHz	
10	200 KHz	
11	300 KHz	
12	400 KHz	
13	500 KHz	
14	600 KHz	
15	700 KHz	
16	800 KHz	
17	900 KHz	
18	1 MHz	
Resonant Frequency		

EXPERIMENT 11

Aim: To determine the turns ratio and polarities of transformer windings.

Apparatus Required: Single Phase Transformer

Wattmeter - 1	:	Ammeter - 1 MI
(0-5/10A)	:	(0-10A)
(0-150/300V)	:	
Voltmeters - 2 MI		
(0-150V)		
(0-300V)		

Theory: Transformer is a device, which has two windings -Primary and Secondary, and with the help of which an ac voltage can be step up or step down. It consists of a Magnetic circuit linked with two windings When any one winding is connected to the supply, an alternating flux will be set up in the core and this flux linking with the secondary induces an *emf* in it. Depending upon the number of turns in Primary and Secondary, the voltage can be step up or step down.

Voltage Transformation Ratio k =

$$\frac{Secondary\ Voltage}{Primary\ Voltage} = \frac{E_2}{E_1} = \frac{V_2}{V_1} = \frac{N_2}{N_1}$$

Procedure:

(i) Ratio Test

1. Connect the Voltmeter as shown in the figure.
2. Note the rating of the winding and then apply 220V across it.
3. Note the Voltmeter reading at 100% Tap and then at 50% Tap.
4. Calculate the ratio for the two readings.

(ii) Polarity Test

1. Connect as shown in Figure (a)

2. A Voltage of 220V is now applied across the Primary. If the Voltmeter reading is equal to E_1 - E_2, then the Secondary terminal connected to C_1 is positive and is marked c_1, the other terminal connected to C_2 through voltmeter is negative and is marked c_2. Polarity is subtractive.
3. If the Voltmeter reading is equal to E_1 + E_2 as shown in fig. (b), then the terminals connected to C_1 and C_2 are negative and positive and are marked c_1 and c_2. Polarity is additive.

(iii) Load Test

1. Connect various instruments as shown in the circuit diagram for load test.
2. Note down the KVA rating and voltage rating of the Transformer and hence calculate the maximum current carrying capacity of the Primary and Secondary.
3. Apply rated voltage to the Primary and put Load on the Secondary.
4. Take readings for various Loads and calculate efficiency at various Loads.
5. Take readings only upto the maximum current rating of the windings.
6. Also calculate Voltage Regulation of the Transformer.

Observations:

(i) Ratio Test

At 100% Tap	:	V_1	=	Volts
		V_2	=	Volts
At 50% Tap	:	V_1	=	Volts
		V_2	=	Volts

(ii) Polarity Test

V_1	=	Volts
V_2	=	Volts
$V_1 + V_2$	=	Volts
$V_1 - V_2$	=	Volts

(iii) Load Test

S. No.	Input	Output		Efficiency VI/W			SL.
		V	W	I	V	V x I	

Calculations: Calculate efficiency at various Loads and Plot efficiency curve.

Output Power = VI watts

Input Power = W watts

So % efficiency (η) = $\frac{VI}{W} \times 100$

Voltage Regulation of (VR) = $\frac{V\ at\ no\ load - V\ at\ full\ load}{V\ at\ no\ load} \times 100\%$

Results& Discussion:

EXPERIMENT 12

Aim: To obtain the equivalent circuit parameters from OC and SC tests, and to estimate efficiency & regulation at various loads.

Apparatus Required: Single Phase Transformer,

Wattmeter - 2	:	Ammeter - 2 MI
(0-2.5A) : (0-5A)	:	(0-1/2.5A), (0-5A)
(0-300V): (0-150V)		
Voltmeters - 2 MI	:	Single phase
(0-300V), (0-150V)	:	Variac

Theory: The open circuit test and the short circuit test determine core loss and copper loss of the transformer. The shunt branch parameters can be calculated by open circuit Test and other series parameters can be calculated by Short Circuit Test.

Procedure:

(i) Open Circuit Test

(1) Connect as shown in the circuit diagram for open circuit test.
(2) The high voltage winding is kept open circuited and full rated voltage of the low voltage winding is applied across the low voltage winding through variac.
(3) Note down the Ammeter, Voltmeter and Wattmeter reading. The Wattmeter reading directly gives coreless.

(ii) Short Circuit Test

(1) Connect various instruments as shown in the circuit diagram for short circuit test.
(2) In short circuit test usually the low voltage side is short circuited and instruments are placed on the high voltage side.
(3) Apply some voltage through variac and vary it such that rated current flows in the winding.
(4) Note down the Ammeter, Voltmeter and Wattmeter reading. Wattmeter reading here directly gives copper loss.

Observations:

OC Test

V_0	=	Voltmeter reading	=	Volts
I_0	=	Ammeter reading	=	Amps
		Wattmeter reading =		
		Multiplying Factor of Wattmeter =		
W_0	=	Actual Power Loss	=	Watts

SC Test

V	=	Voltmeter reading	=	Volts
I	=	Ammeter reading	=	Amps
		Wattmeter reading	=	
		Multiplying Factor of Wattmeter	=	
W	=	Actual Power Loss	=	Watts

Calculations:

From OC Test

$$No\ load\ Power\ factor\ cos\theta = \frac{W_0}{V_0 I_0}$$

$$I_C = I_0 cos\theta_0$$

$$I_M = I_0 sin\theta_0$$

$$R_C = \frac{V_0}{I_C} and\ X\mu = \frac{V_0}{I\mu}$$

From SC Test

$$Z_{eH} = \frac{V}{I}$$

$$r_{eH} = \frac{W}{I}\ and\ X_{eH} = \sqrt{Z_{eH}^2 - r_{eH}^2}$$

Determine the equivalent circuit parameters refer to h_v and l_v sides.

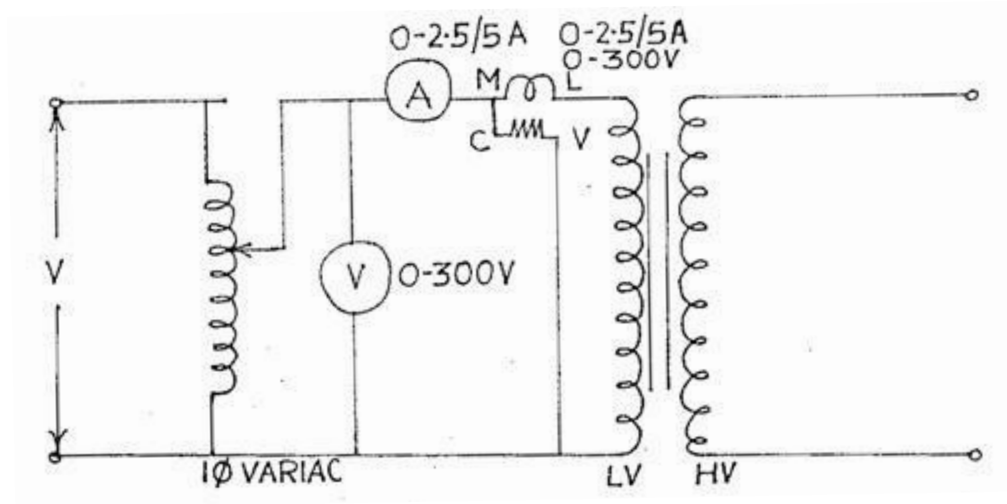


Figure 12.1: Open Circuit Test

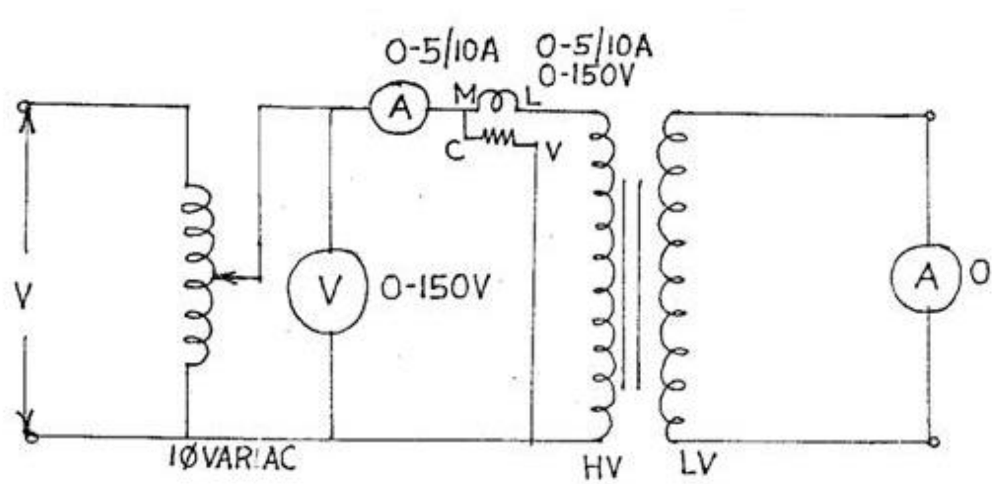


Figure 12.1: Short Circuit Test

Results& Discussion: