Fundamentals of Electronics

Laboratory Manual

Course code: EECE103L   Semester: First Sem.   Year: First Yr.

BENNETT UNIVERSITY

Department of Electronics and Communication Engineering

School of Engineering and Applied Sciences

	Name	Date of Revision	Revision no.
Prepared by	Dr Arjun Kumar, Dr. Surjana Kagita, Prof. S. Chandra		

Table of Contents

Safety Rules and Laboratory Safety Information

1. Enter in the lab in time. your lab timings will be observed strictly.
2. Strictly follow the written and verbal instructions given by the teacher / Lab Instructor. If you do not understand the instructions, the handouts and the procedures, ask the instructor or teacher.
3. Never work alone! You should be accompanied by your laboratory partner and / or the instructors / teaching assistants all the time.
4. It is mandatory to come to lab in a formal dress and wear your ID cards.
5. Do not wear loose-fitting clothing or jewellery in the lab. Rings and necklaces are usual excellent conductors of electricity.
6. Mobile phones should be switched off in the lab. Keep bags in the bag rack.
7. Keep the labs clean always, no food and drinks allowed inside the lab.
8. Intentional misconduct will lead to expulsion from the lab.
9. Do not handle any equipment without reading the safety instructions. Read the handout and procedures in the Lab Manual before starting the experiments.
10. Do your wiring, setup, and a careful circuit checkout before applying power. Do not make circuit changes or perform any wiring when power is on.
11. Avoid contact with energized electrical circuits.
12. **NEVER** try to experiment with the power from the wall plug.
13. Immediately report dangerous or exceptional conditions to the Lab instructor / teacher: Equipment that is not working as expected, wires or connectors are broken, the equipment that smells or “smokes”. If you are not sure what the problem is or what's going on, switch off the Emergency shutdown.
14. Never use damaged instruments, wires or connectors. Hand over these parts to the Lab instructor/Teacher.
15. After completion of Experiment, return the bread board, trainer kits, wires, CRO probes and other components to lab staff. Do not take any item from the lab without permission.
16. Observation book and lab record should be carried to each lab. Readings of current lab experiment are to be entered in Observation book and previous lab experiment should be written in Lab record book. Both the books should be corrected by the faculty in each lab.
17. Handling of Semiconductor Components: Sensitive electronic circuits and electronic components must be handled with great care. The inappropriate handling of electronic component can damage or destroy the devices. The devices can be destroyed by driving to high currents through the device, by overheating the device, by mixing up the polarity, or by electrostatic discharge (ESD). Therefore, always handle the electronic devices as indicated by the handout, the specifications in the data sheet or other documentation.
18. Special Precautions during soldering practice
    a. Hold the soldering iron away from your body. Don't point the iron towards you.
    b. Don't use a spread solder on the board as it may cause short circuit.

c. Do not overheat the components as excess heat may damage the components/board.
d. In case of burn or injury seek first aid available in the lab or at the college dispensary

Experiment -1

Aim: - Study of various measurement equipment listed below:

a) To find the amplitude and frequency using CRO
b) Study of Multimeter
c) Bread Board
d) Signal Generator
e) DC voltage Source

Components and Equipment's Required: Cathode Rays Oscilloscope (CRO), Signal generator, resistances, capacitors, multimeter, bread board, CRO probes, testing probes and DC voltage source

Theory: The block diagram shown in the Fig.1, explain how an oscilloscope works.

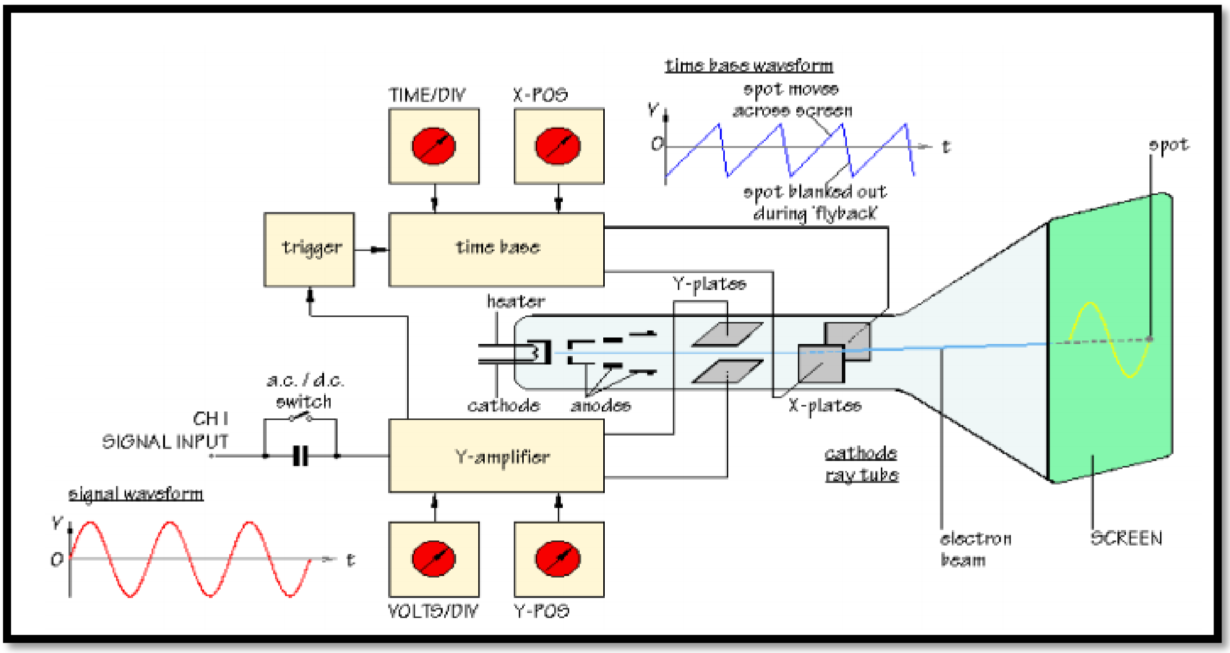


Fig.1: Cathode Rays Oscilloscope

Like a television screen, the screen of an oscilloscope consists of a Cathode Ray Tube (CRT) which is also called the heart of CRO. Although the size and shape are different, the operating principle is the same. Inside the tube is a vacuum. The electron beam emitted by the heated cathode at the rear end of the tube is accelerated and focused by one or more anodes, and strikes the front of the tube, producing a bright spot on the phosphorescent screen.

The electron beam is bent, or deflected, by voltages applied to two sets of plates fixed in the tube. The horizontal deflection plates or **X-plates** produce side to side movement. As you can see, they are linked to a system block called the time base. This produces a saw tooth waveform. During the rising phase of the saw tooth, the spot is driven at a uniform rate from left to right across the front of the screen. During the falling phase, the electron beam returns rapidly from right to left, but the spot is 'blanked out' so that nothing appears on the screen. In this way, the time base generates the **X-axis** of the **V/t graph**. The slope of the rising phase varies with the frequency of the saw tooth and can be adjusted, using the **TIME/DIV** control, to change the scale of the **X-axis**. Dividing the oscilloscope screen into squares allows the horizontal scale to be expressed in seconds, milliseconds or microseconds per division (s/DIV, ms/DIV, μs/DIV). The signal to be displayed is connected to the input. The AC/DC switch is usually kept in the DC position (switch closed) so that there is a direct connection to the Y-amplifier. In the AC position (switch open) a capacitor is placed in the signal path. The capacitor blocks DC signals but allows AC signals to pass. The Y-amplifier is linked in turn to a pair of Y-plates so that it provides the Y-axis of the Y-amplifier is linked in turn to a pair of Y-plates so that it provides the Y-axis of the V/t graph.

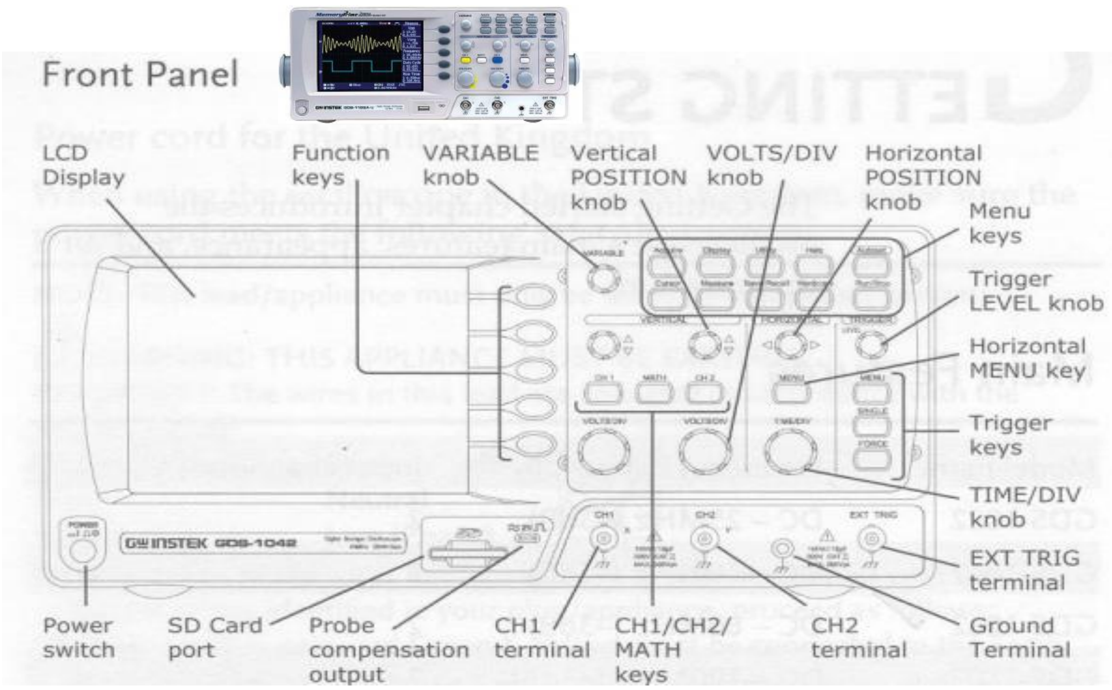


Fig. 2: Front view of Digital Oscilloscope

The overall gain of the Y-amplifier can be adjusted, using the VOLTS/DIV control, so that the resulting display is neither too small nor too large, but fits the screen and can be seen clearly. The vertical scale is usually given in V/DIV or mV/DIV. The trigger circuit is used to delay the time base waveform so that the same section of the input signal is displayed on the screen each time the spot moves across. The effect of this is to give a stable picture on the oscilloscope screen, making it easier to measure and interpret the signal Changing the scales of the X-axis and Y-axis allows many different signals to be displayed. Sometimes, it is also useful to be able to change the positions of the axes. This is possible using the X-POS and Y-POS controls. For example, with no signal applied, the normal trace is a straight line across the centre of the screen. Adjusting Y-POS allows the zero level on the Y-axis to be changed, moving the whole trace up or down on the screen to give an effective display of signals like pulse waveforms which do not alternate between positive and negative

Measurement of Amplitude and Frequency:

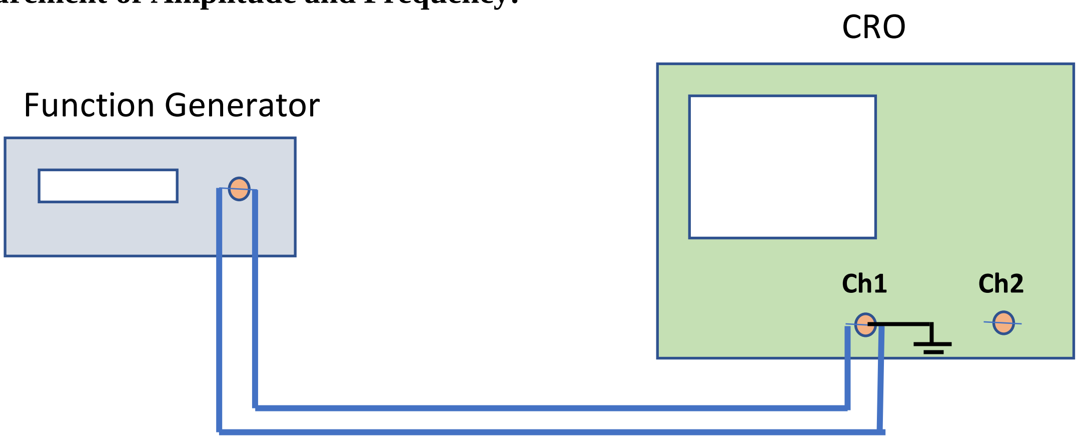


Fig. 3: Measurement of Amplitude and Frequency

A) Measurement of Amplitude:

Procedure:

1. Make the connections as per the diagram shown above.

2. Put the CRO on a single channel mode and bring the CRO into operation by adjusting the trace of the beam to a normal brightness and into a thin line.

3. Now apply the sinusoidal wave of different amplitudes by using the LEVEL and COARSE buttons of the function generator.

4. Note on the vertical scale the peak to peak amplitude (Vpp).

Observation Table:

S.no.	No. of Vertical Divisions (X)	Voltage/Division (Y)	V(p-p) = X*Y	Vm = V(p-p)/2

B) Measurement of Frequency:

Procedure:

1. Make the connections as per the diagram shown above.

2. Put the CRO on a single channel mode and bring the CRO into operation by adjusting the trace of the beam to a normal brightness and into a thin line.

3. Now apply the sinusoidal wave of different frequencies by using the LEVEL and COARSE buttons of the function generator.

4. Note down the horizontal scale period (T) in second by observing difference between the two successive peaks of the waveform.

Observation Table:

S.no.	No. of Horizontal Divisions (X)	Time/Division (Y)	T = X*Y	F = 1/T

b) Multimeter:

A **multimeter** is an instrument that allows us to make multiple electrical measurements using the same tool. We can use a multimeter as:

- A **voltmeter** to measure voltage
- An **ammeter** to measure current
- An **ohmmeter** to measure resistance

There are two basic types of multimeters: **digital multimeters** and **analog multimeters**. Digital multimeters are superior to analog multimeters because of their better accuracy in measurements, sensitivity to very small changes in input voltages, and clear and easy-to-read displays.

However, unlike analog multimeters, digital multimeters need a power supply, such as batteries. Also, because they digitize the analog signals, multimeters can add noise, and it sometimes becomes difficult to isolate the signal from the noise. In addition, digital multimeters are not the best when it comes to testing semiconductor electronic parts.

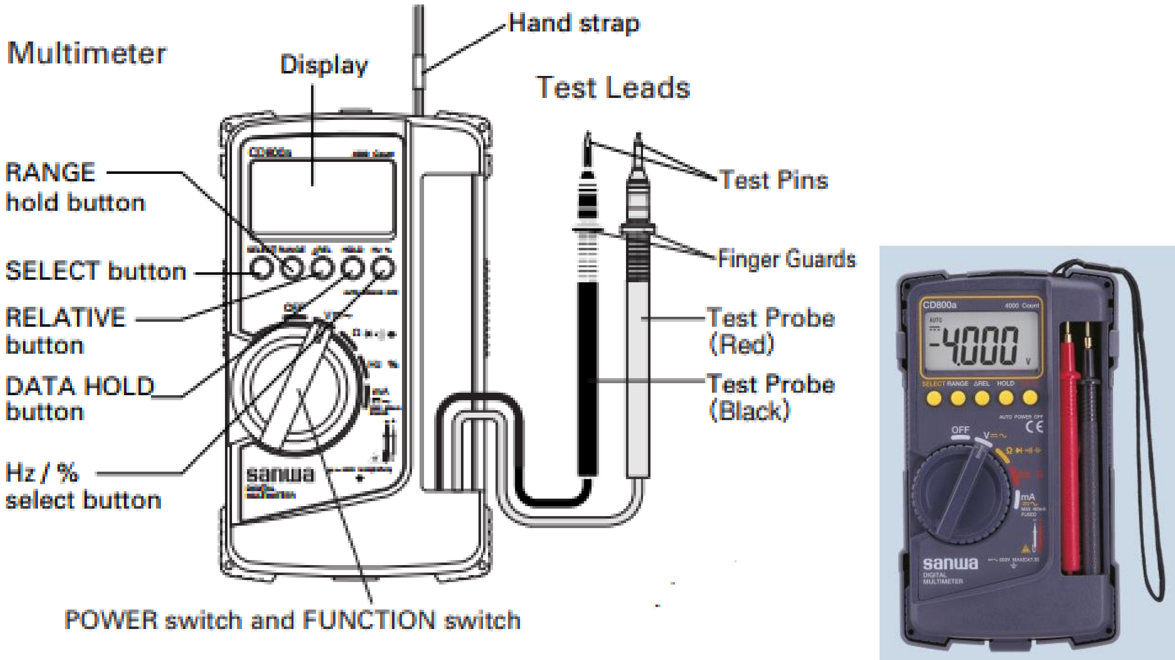


Fig. 4: Digital Multimeter with front panel

Start-up Measurement Setup:

(1) Check the lead and fuse by continuity check as shown in fig. below

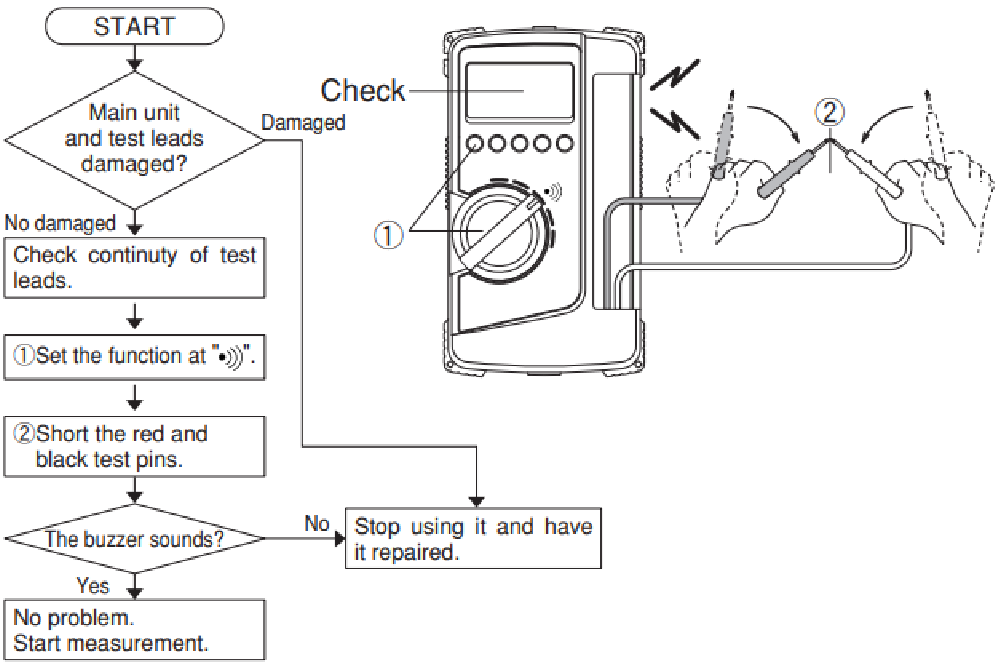


Fig. 5: Continuity checking of wire or fuse

(2) Voltage Measurement:

1. Set the FUNCTION switch at "V" and select either DC or AC with the SELECT button.
2. Apply the red and black test pins to the circuit to measure.
3. For measurement of DCV, apply the black test pin to the negative potential side of the circuit to measure and the red test pin to the positive potential side.
4. For measurement of ACV, apply the red and black test pins to the circuit to measure.
5. The reading of Voltage is shown on the display.
6. After measurement, release the red and black test pins from the object measured.

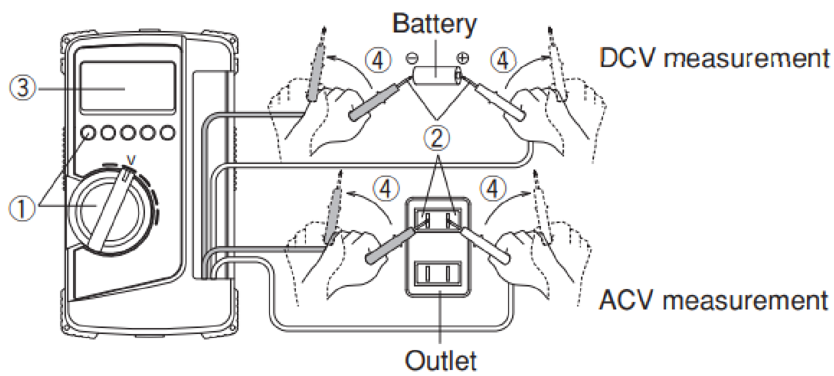


Fig. 6: Testing of AC/DC voltage

(3) Current Measurement:

1. Never apply voltage to the input terminals.
2. Be sure to make a series connection via load.
3. Do not apply an input exceeding the maximum rated current to the input terminals.
4. Before starting measurement, turn OFF the power switch of the circuit to separate the measuring part, and then connect the test leads firmly.
5. Set the function switch at “mA” and select either DC or AC with the SELECT button.
6. In the circuit to measure and apply the red and black test pins in series with load.
7. For measurement of DCA, apply the black test pin to the negative potential side of the circuit to measure and the red test pin to the positive potential side in series with load.
8. For measurement of ACV, apply the red and black test pins to the circuit to measure in series with load.
9. Read the value on the display.

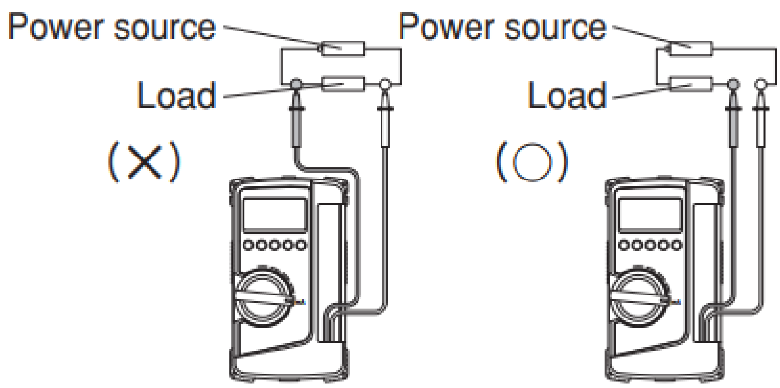


Fig. 7 : Current Measurement setup

(4) Resistance Measurement:

1. Set the FUNTION switch at Ω/▶|/•)))/⊣⊢ and select Ω with the SELECT button.
2. Apply the red and black test pins to an object to measure.
3. The reading is shown in the display.
4. After measurement, release the red and black test pins from the object measured.
5. Note : If measurement is likely to be influenced by noise, shield the object to measure with negative potential (COM). If a finger touches a test pin during measurement, measurement will be influenced by the resistance in the human body, and that results in measurement error. Open Circuit Voltage:

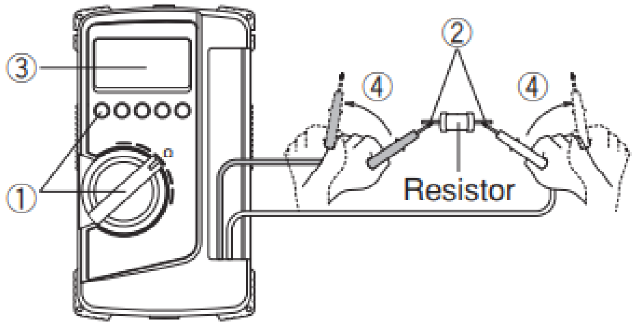


Fig. 8 : Resistance Measurement setup

(5) Capacitance Measurement:

1. Set the FUNTION switch at Ω/ ⊣▶⊢/ •))) / ⊣⊢ .
2. Select by pressing the SELECT button ⊣⊢ .
3. Press the REL button for zero setting (00.00 nF).
4. Apply the red and black test pins to a conductor to measure.
5. Read the value on the display.
6. After measurement, release the red and black test pins from the object measured.
   - ● Manual range is not available in capacitance measurement.
   - ● Readings are unstable because of stray capacitance in test leads or noise

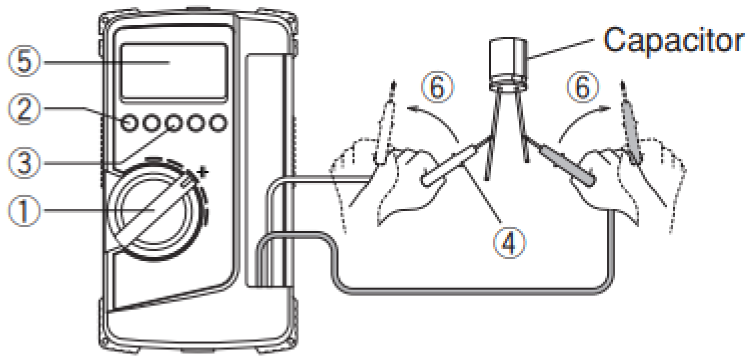


Fig. 9: Capacitor testing

(6) Diode Testing:

Applications The quality of diodes is tested.

How to use:

(1) Set the FUNTION switch at Ω/ ⊣▶⊢/ •))) / ⊣⊢
(2) Select by pressing the SELECT button.
(3) Apply the black test pins to the cathode of the diode and the red test pin to the anode.

(4) Make sure that the display shows a diode forward voltage drop.
(5) Replace the red and black test pins, make sure that the display is “OL” reading.
(6) After measurement, release the red and black test pins from the object measured.
(7) The input terminals open voltage is about 1.5 V

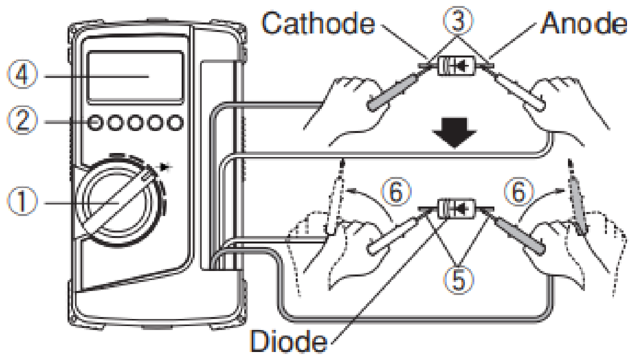


Fig. 10: Diode testing

c) Bread-Board

In order to temporarily construct a circuit without damaging the components used to build it, we must have some sort of a platform that will both hold the components in place and provide the needed electrical connections. In the early days of electronics, most experimenters were amateur radio operators. They constructed their radio circuits on wooden breadboards. Although more sophisticated techniques and devices have been developed to make the assembly and testing of electronic circuits easier, the concept of the breadboard still remains in assembling components on a temporary platform.

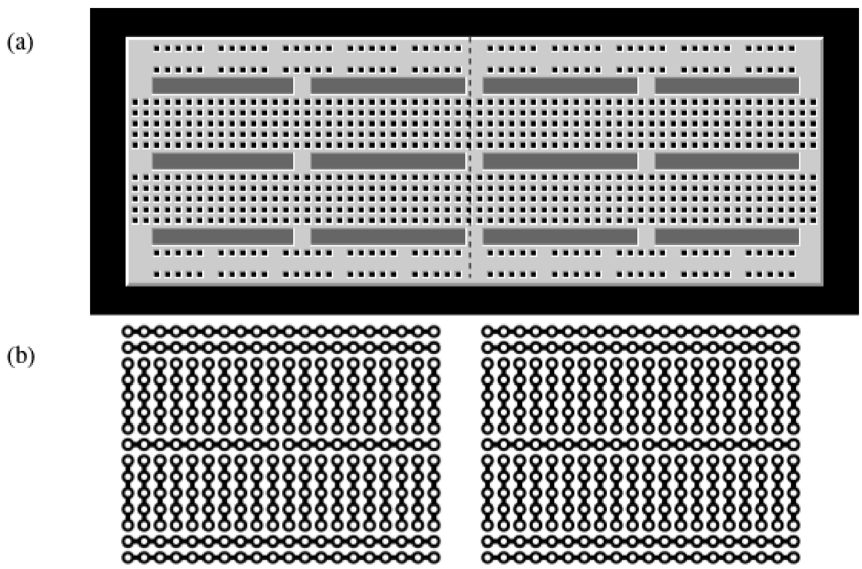


Fig. 11: Bread-Board internal connection structure

A real breadboard is shown in Fig. 11(a) and the connection details on its rear side are shown in Fig. 11(b). The five holes in each individual column on either side of the central groove are electrically connected to each other, but remain insulated from all other sets of holes. In addition to the main columns of holes, however, you'll note four sets or groups of holes along the top and bottom. Each of these consists of five separate sets of five holes each, for a total of 25 holes. These groups of 25 holes are all connected on either side of the dotted line indicated on Fig.1(a) and needs an external connection if one wishes the entire row to be connected. This makes them ideal for distributing power to multiple ICs or other circuits.

These breadboard sockets are sturdy and rugged, and can take quite a bit of handling. However, there are a few rules you need to observe, to extend the useful life of the electrical contacts and to avoid damage to components.

These rules are:

- Always make sure power is disconnected when constructing or modifying your experimental circuit. It is possible to damage components or incur an electrical shock if you leave power connected when making changes.
- Never use larger wire as jumpers. #24 wire (used for normal telephone wiring) is an excellent choice for this application.
- Observe the same limitation with respect to the size of component leads
- Whenever possible, use ¼ watt resistors in your circuits. ½ watt resistors may be used when necessary; resistors of higher power ratings should never be inserted directly into a breadboard socket.
- Never force component leads into contact holes on the b Doing so can damage the contact and make it useless.
- Do not insert stranded wire or soldered wire into the breadboard socket. If you must have stranded wire (as with an inductor or transformer lead), solder (or use a wire nut to connect) the stranded wire to a short length of solid hookup wire, and insert only the solid wire into the breadboard.

d) Signal Generator:

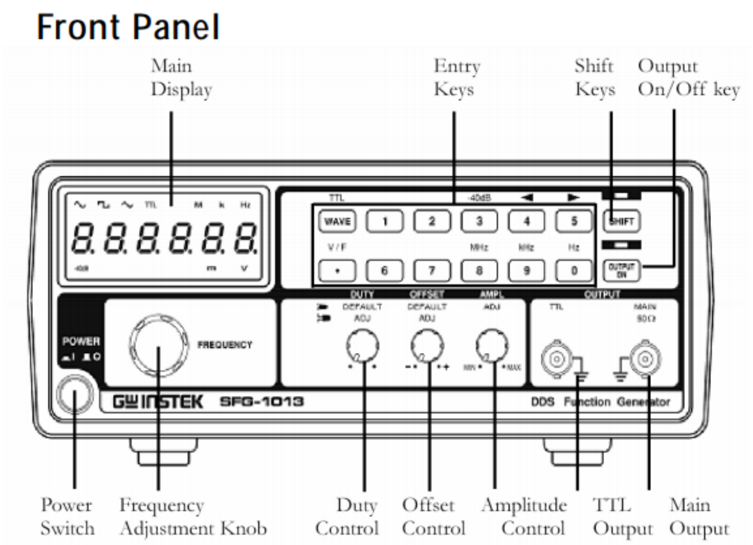


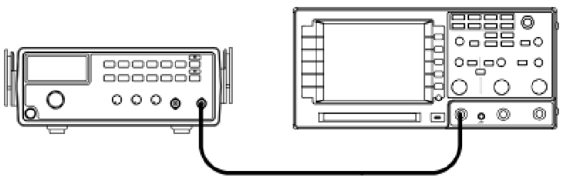
Fig. 12: Front Panel of Function Generator

Waveform key	WAVE	Selects the waveform: sine, square, and triangle. For details, see page20.
TTL activation	SHIFT → WAVE (TTL)	Activates TTL output. For details, see page25.
Numerical keys	1 ~ 0	Specifies frequency.
Frequency unit selection	SHIFT → 8 (MHz) (9 (kHz), 0 (Hz))	Specifies the frequency unit: MHz, kHz, or Hz.
Cursor selection	SHIFT → 4 (◄) or 5 (►)	Moves the cursor (frequency editing point) left or right. For details, see page21.

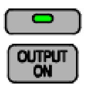
−40dB attenuation (SFG-1013 only)	SHIFT → 3 (−40dB)	Attenuates amplitude by −40dB. For details, see page22. Key operation is for SFG-1013 only.
Frequency / Voltage display selection (SFG-1013 only)	• (V/F)	Switches the display between frequency and voltage. For details, see page22. For SFG-1013 only.
Shift key	SHIFT	Selects the 2nd function associated to the entry keys. The LED lights when Shift is activated.
Output On/Off key	OUTPUT ON	Turns the output On/Off. The LED lights when the output is On.

Functionality check

1. Connect SFG main output to measurement device such as oscilloscope.



2. Press the output key. The output is activated and the LED turns On.



3. Observe the output waveform: 1kHz, sine wave.

Fig. 13: Functionality Testing

e) DC Source or Power Supply

The power supply has an electrically floating output. This permits easy series or parallel connection with other power supply units, to increase supply voltage or current respectively. Basically, the power supply is constituted by rectifier, filters and regulator circuits for constant voltage supply. Here in the figure given below shows the test setup for supply the DC supply.

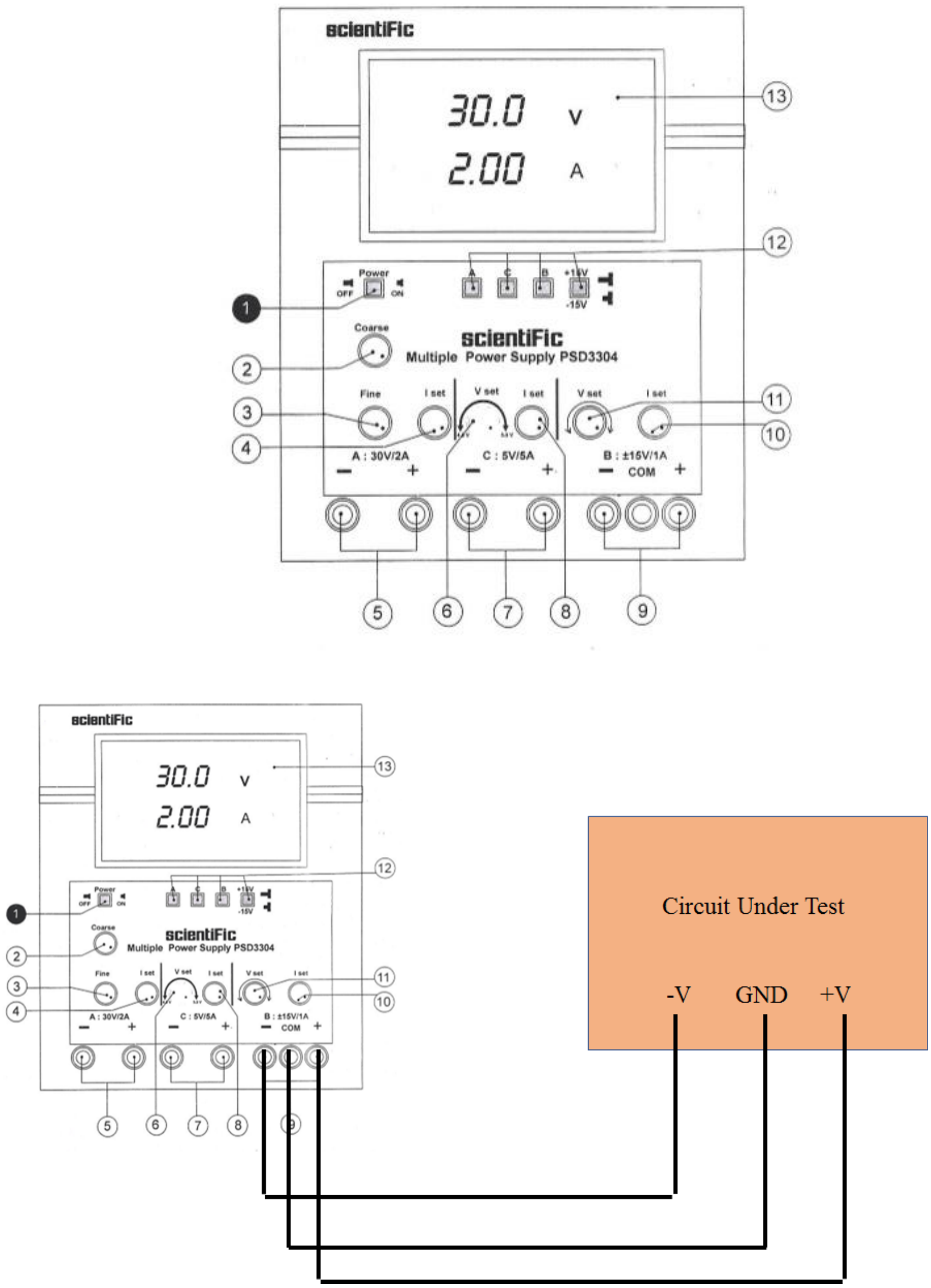


Fig. 14: Front Panel of DC Power Supply

Experiment -2

AIM: Verification of Ohm's law, Kirchhoff's current law and voltage law

APPARATUS REQUIRED: Resistances, Bread-board, Power Supply, Multi meter, connecting wires etc.

THEORY: Ohm's law, Kirchoff's Voltage Law and Kirchoff's Current Law are essential in the analysis of linear circuitry. Kirchoff's laws deal with the voltage and current in the circuit. Ohm's law relates voltage, current and resistance to one another. These three laws apply to resistive circuits where the only elements are voltage and/or current sources and resistors. Using the three laws any resistance of, current through or voltage across a resistor can be found if any two are already known.

1. **Ohm's Law:** Ohm's law is used to relate voltage to current and resistance. It states that voltage is directly proportional to current and resistance. This is stated mathematically as

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

where V is the voltage across an element of the circuit in volts, I is the current passing through the element in amps and R is the resistance of the element in ohms. Given any two of these quantities Ohm's law can be used to solve for the third.

2. **Kirchhoff's Voltage Law:** Kirchhoff's Voltage Law (KVL) states that the sum of all voltages in a closed loop must be zero.

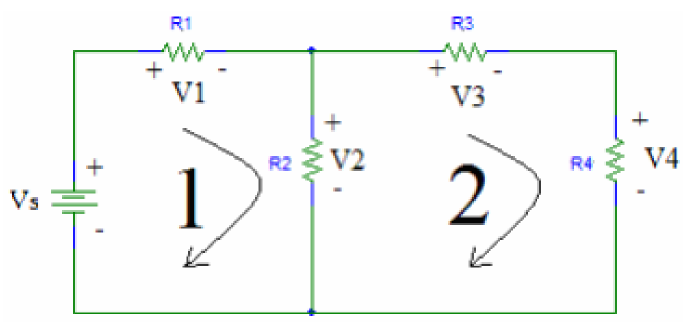


Fig. 2.1: An Example of KVL

A closed loop is a path in a circuit that doesn't contain any other closed loops. Loops 1 and 2 in Figure 1 are examples of closed loops. The perimeter of the circuit is also a closed loop, but since it includes loops 1 and 2 it would be repetitive to include a KVL equation for it. If loop 1 is followed clockwise the KVL equation is

$$V1 + V2 - Vs = 0 \tag{2}$$

This equation holds true only if the passive sign convention is satisfied. In the case of KVL the passive sign convention states that when a positive node is encountered while following a loop the voltage across the element is positive. If a negative node is encountered the corresponding element voltage is negative. In order to simplify the KVL equations, the polarities should be assigned to satisfy the passive sign convention whenever possible.

3. **Kirchoff's Current Law:** Kirchoff's Current Law (KCL) deals with the currents flowing into and out of a given node. KCL states that the sum of all currents at a node must equal zero. This is illustrated in Fig. 2.2.

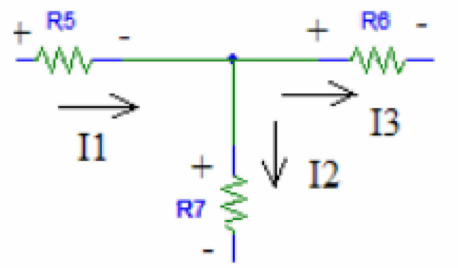


Fig. 2.2: An Example of KCL

The equation obtained by KCL for the node shown in Fig. 2 is

$$I1 - I2 - I3 = 0 \tag{3}$$

In the case of KCL the passive sign convention deals with the direction of currents with respect to the node. Currents entering the node must have opposite signs as those exiting the node. The passive sign convention with respect to KVL can also be applied to KCL. On many schematics, the polarities of resistors are already assigned, so the directions of the currents should be assigned such that the current is entering the positive terminal. This will simplify later calculations.

4. Circuit Analysis:

The circuit analyzed in this laboratory is shown in Fig.2.3.

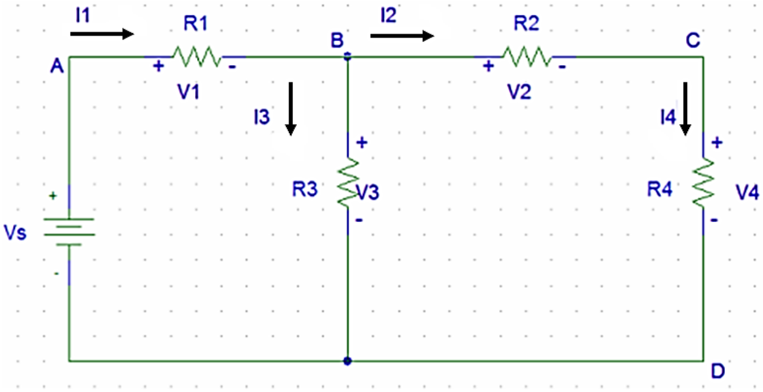


Fig. 2.3: Circuit to be analyzed

The currents, voltages and polarities were labelled as shown on the given schematic. The current directions and voltage polarities have all been assigned such that the passive sign convention has been satisfied wherever possible.

5. Observations:

KVL and KCL circuits are analyzed and the values of different parameters are mentioned in observation table.

Observation Table 1

S.No.	**Resistances**		**Voltage**		**Current**	
1	Calculated	Observed	Calculated	Observed	Calculated	Observed
2						
3						
4						
5						

Results and Conclusions: Ohm's law, KVL and KCL are three of the most basic techniques for the analysis of linear circuits. The purpose of this lab was to prove these three laws valid. A circuit was provided with four unknown voltages and three unknown currents. Then the circuit was built on a breadboard. The voltage and current values were measured and placed into KVL and KCL equations to determine whether they turned out as predicted. These measured values were then used in the Ohm's law equation to find resistance. The calculated and measured values were then compared to the expected results from the theories. All three's law has been verified successfully.

Experiment-3

AIM: Study and verifications of Thevenin's and Norton's Theorem.

APPARATUS REQUIRED: Bread-board, Multi-meter, Resistances, Voltage /Current Source, Connecting wires, etc.

THEORY: (a) Thevenin's Theorem: It states that "*Any linear circuit containing several voltages and resistances can be replaced by just one single voltage in series with a single resistance connected across the load*". In other words, it is possible to simplify any electrical circuit, no matter how complex, to an equivalent two-terminal circuit with just a single constant voltage source in series with a resistance (or impedance) connected to a load as shown below.

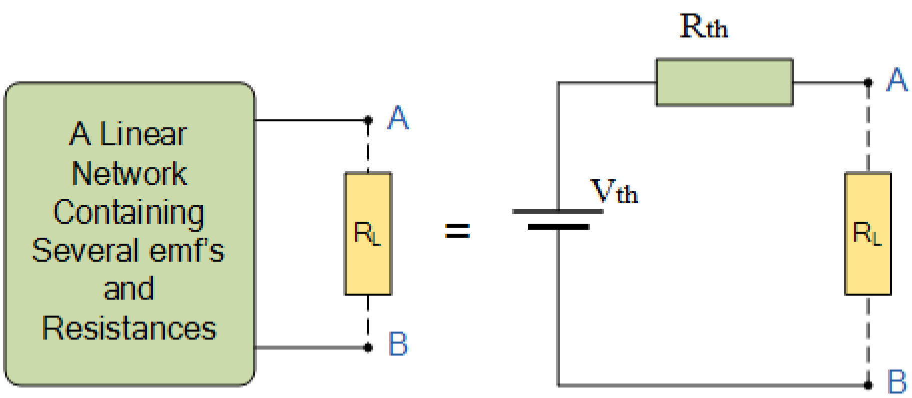


Fig. 3.1: Thevenin's Equivalent Circuit

Simple steps to analyse electronic circuit given in Fig. 3.2, through Thevenin's theorem

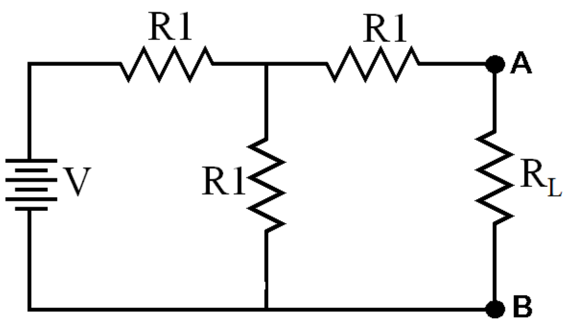
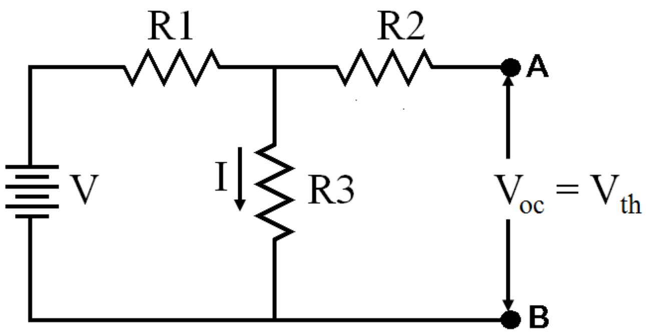


Fig. 3.2: Electronic circuit to be analyzed through Thevenin's Theorem

1. Open the load resistor
2. Calculate/measure the open circuit voltage. This is the Thevenin's voltage (Vth)

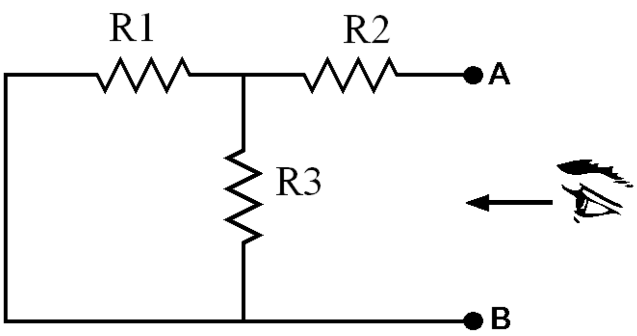


$$Voc = I\ R3$$

$$I = V/(R1 + R3)$$

$$Voc\ or\ Vth = V * R3/(R1 + R3)$$

3. Open current sources and short the voltage sources.
4. Calculate / measure the open circuit resistance. This is the Thevenin's resistance (Rth)



$$Rth = R2 + \frac{R1 * R3}{R1 + R2}$$

5. Now, Redraw the circuit with measured open circuit voltage (Vth) in step (2) as voltage source and measured the open circuit resistance (Rth) in step (4) as a series resistance and connect the load resistor which had been removed in step (1). This is the equivalent Thevenin circuit of that linear electric network or complex circuit which had to be simplified and analyzed by Thevenin's Theorem.

OBSERVATION TABLE:

S.NO.	**Thevenin's Voltage**		**Thevenin's Resistance**	
	Theoretical calculation	Practical or Measured	Theoretical calculation	Practical or Measured

(b) Norton's Theorem: It states that "*Any linear circuit containing several energy sources and resistances can be replaced by a single Constant Current generator in parallel with a Single Resistor*". As far as the load resistance, R_L is concerned this single resistance, Rs is the value of the resistance looking back into the network with all the current sources open circuited and Is is the short circuit current at the output terminals as shown below.

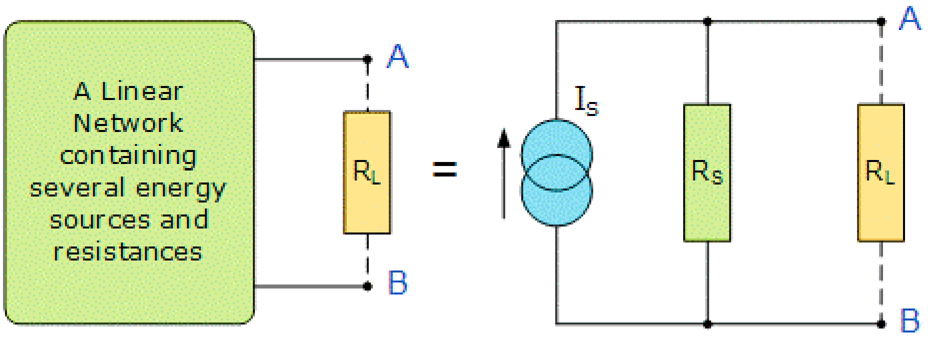
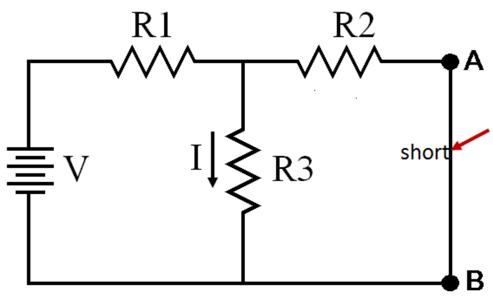


Fig. 3.3: Norton's Equivalent Circuit

Simple steps to analyse electronic circuit given in Figure 2, through Norton's theorem

1. Short the load resistor



2. Calculate/Measure the short circuit current. This is the Norton current (I_N)

   We have shorted the AB terminals to determine the Norton current, I_N. The R3 and R2 are then in parallel and this parallel combination of R3 and R2 are then in series with R1.

   So the Total Resistance of the circuit to the Source is:-

   R1 + (R3 || R2) ….. (|| = in parallel with).

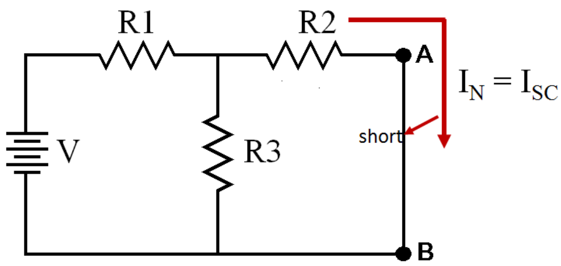
   R_T = R1 + [(R3 x R2) / (R3+ R2)].

   I_T = V / R_T

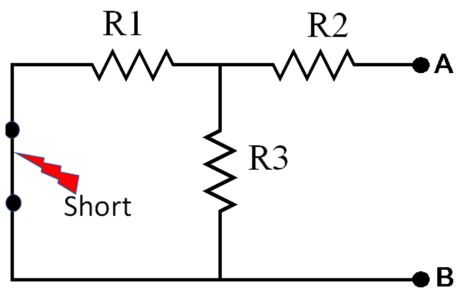
   Now we must find I_{SC} = I_N… Apply Current Divider Rule

   I_{SC} = I_N = I_T x [(R3 / (R2+R3)]

   I_{SC}= I_N



3. Open the current sources, short the voltage sources and open the load resister

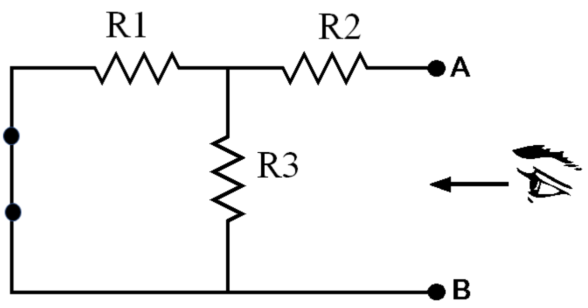


4. Calculate/Measure the open circuit resistance. This is the Norton resistance (R_N).

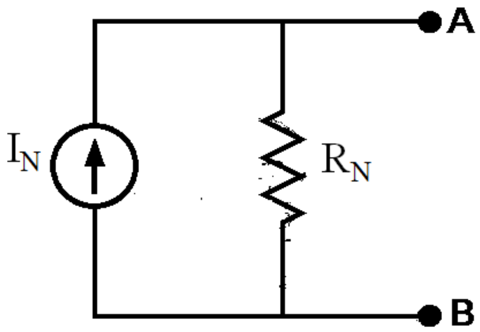
   We have reduced the DC source to zero is equivalent to replace it with a short in step (3), as shown in above figure. We can see that R2 resistor is in series with a parallel combination of R3 resistor and R1 resistor. i.e.:

   R2 + (R3 || R1) ….. (|| = in parallel with)

   R_N = R2 + [(R3 x R1) / (R3+ R1)]



5. Now, Redraw the circuit with measured short circuit current (I_N) in step (2) as current source and measured open circuit resistance (R_N) in step (4) as parallel resistance and connect the load resistor which had been removed in step (3). This is the equivalent Norton circuit of that linear electric network.



6. Now find the load current through and load voltage across the load resistor by using the current divider rule.

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

OBSERVATION TABLE:

	V (Volts)	IN (mA)	RN (Ω)	IL (mA)
Theoretical values				
Practical values				

CONCLUSIONS: Thevenin's theorem and Norton's theorem have been verified successfully.

Experiment No: 4

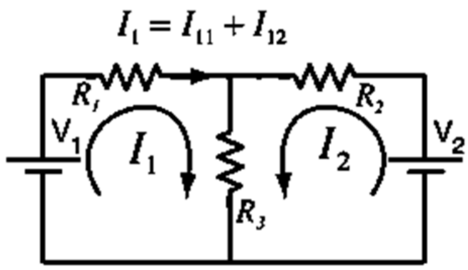
OBJECTIVE: Verification of Superposition theorem

APPARATUS REQUIRED: Power Supply, resistances, breadboard, multi-meter, connecting wires etc.

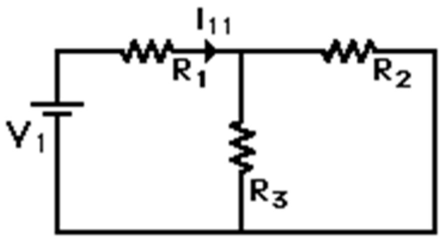
THEORY:

This theorem states that in a linear bilateral network containing several sources the overall response at any point in the network equal the sum of the responses of each individual sources considered separately with all other sources made inoperative i.e., replaced by resistances equal to their internal resistances.

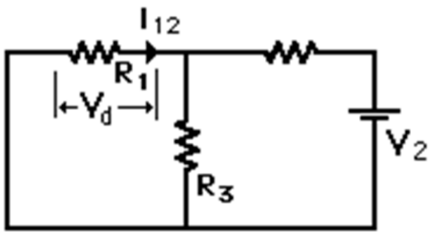
CIRCUIT DIAGRAM:



(a)



(b)



(c)

Fig. 4: (a) Circuit with two voltage sources with current direction, (b) considering voltage source V1, (c) Considering voltage source V2

$$I_{11} = \frac{V_1}{R_1 + R_2 \| R_3} \tag{1}$$

$$I_{12} = \frac{-V_d}{R_1} \tag{2}$$

$$V_d = \frac{(R_1 \| R_3)V_2}{R_2 + R_1 \| R_3} \tag{3}$$

OBSERVATION TABLE:

S.no.	Current "I"(mA) Practically Ckt. diagram	Current "I_{11}"(mA) Practically Fig. 4(b)	Current "I_{12}"(mA) Practically Fig.4 (c)	$I=I_{11} + I_{12}$ With sign

PROCEDURE:

i) Connect the circuit on bread board as shown in Fig.4
ii) For verifying theorem, consider one source at a time if you have multiple sources
iii) Connect the multi-meter in series with branch.
iv) Consider source V1 and short the V2 terminal
v) Observe the reading of current "I_{11}"
vi) Consider source V2 and short the V1
vii) Observe the reading of I_{12}

PRECAUTION:

(i) Do not make interconnection on the board with mains switched ON.
(ii) As soon as mains is ON the reading in the meters must be zero. If the reading in the meters is not zero, check the meter.

RESULT: The net current I is the algebraic sum of I_{11} & I_{12} due to the individual voltage source taking one at a time $I=I_{11} + I_{12}$

Experiment No: 5

OBJECTIVE: Verification of maximum power transfer theorem (MPT).

APPARTAUS REQUIRED:

S. no.	Instrument	Type	Range/specification	Qty.
1	Supply	DC	0-12V	1
2	Experimental board			1
3	Voltmeter	MC	0-5V	1
4	Ammeter	MC	0-10mA	1
5	Resistance	CFR/MFR	25 Ω /50Ω/90 Ω/100 Ω/500 Ω	1 each
6	Connecting leads			As required

THEORY:

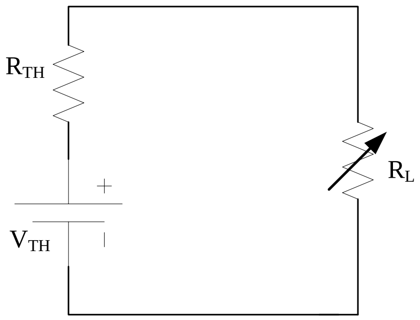
This theorem states that in a DC network, a resistive load will abstract maximum power from a network when the load resistance is equal to the resistance of the network as viewed from the output terminals, with all energy sources replaced by their internal resistance.

$P_o = P_{max}$ when $R_L = R_T = R_{TH}$

$P_{max} = V_{TH}^2 / 4R_L$

Power delivered to load resistance,

$P_o = I_L^2 \ R_L$



CIRCUIT DIAGRAM

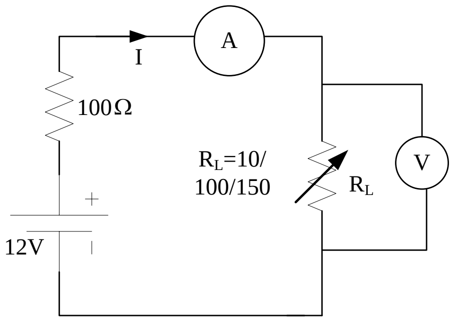


Fig.5

OBSERVATION TABLE

S. no.	R_L (Ω)	I(mA)	V(volts)	P=VI	R_I (Ω)	V_{th} (volts)	$P_{max} = V_{TH}^2 / 4R_L$ calculated
1	50(Ω)				100(Ω)	12 V	
2	80(Ω)						
3	100(Ω)						
4	120(Ω)						

PROCEDURE:

i. Connect the circuit According to the fig.1
ii. Select the different value of R_L (Ω) as 50,80,100,120 R_L (Ω)
iii. Note down the reading of voltmeter to find V(volts)
iv. Note down the reading of ammeter to find I(mA)

PRECAUTION:

i. Do not make interconnection on the board with mains switched ON.
ii. As soon as mains is ON. The reading in the meters must be zero. If the reading in the meters is not zero, check the meters.

RESULT:

From the observation table we can observe that power delivered is maximum when the load resistance is equal to internal resistances. And the maximum value of power is

P_{max} = ……….

Experiment No. 6

AIM: Study of Low Pass and High Pass Characteristics of RC Filter Circuit

a) Study the transfer function and phase shift of a low pass RC filter network.
b) Study the transfer function and phase shift of a high pass RC filter network.

APPARATUS REQUIRED: Bread-Board, Digital Multimeter, Digital Oscilloscope, Resistors, Capacitors, Connecting Wires, Testing probes, CRO probes, Function Generator, etc.

THEORY: Filter is an essential component for any communication system. Filter is nothing but a frequency selective circuit which reject the undesired frequency band and pass the desired frequency band. Frequency-selective or filter circuits pass only those input signals to the output that are in a desired range of frequencies (called pass band). The amplitude of signals outside this range of frequencies (called stop band) is reduced (ideally reduced to zero). The frequency between pass and stop bands is called the cut-off frequency (ω_c). Typically, in these circuits, the input and output currents are kept to a small value and as such, the current transfer function is not an important parameter. The main parameter is the voltage transfer function in the frequency domain, $Hv(j\omega) = Vo/Vi$. Subscript v of Hv is frequently dropped. As $H(j\omega)$ is complex number, it has both a magnitude and a phase, filters in general introduce a phase difference between input and output signals.

LOW AND HIGH PASS FILTER:

A low pass filter or LPF attenuates or rejects all high frequency signals and passes only low frequency signals below its characteristic frequency called as cut-off frequency, ωc. An ideal low-pass fillter's transfer function is shown in Fig. 6.1. A high pass filter or HPF, is the exact opposite of the LPF circuit. It attenuates or rejects all low frequency signals and passes only high frequency signals above ωc. In practical filters, pass and stop bands are not clearly defined, $|H(j\omega)|$ varies continuously from its maximum towards zero. The cut-off frequency is, therefore, defined as the frequency at which $|H(j\omega)|$ is reduced to $1/\sqrt{2}$ or 0.7 of its maximum value. This corresponds to signal power being reduced by 1/2 as $P \propto V^2$.

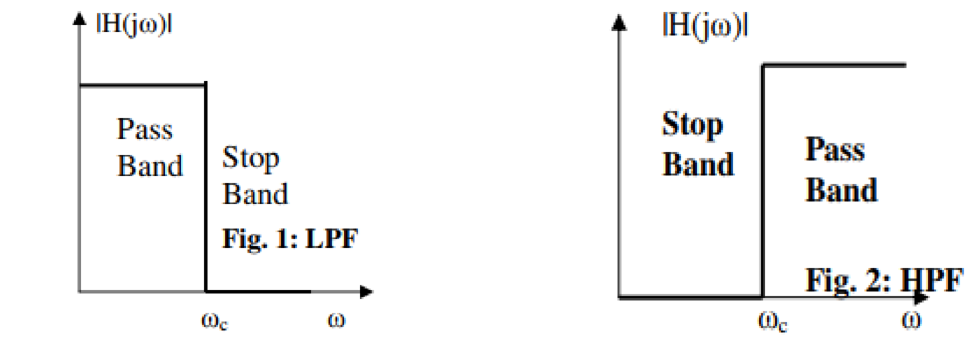


Fig. 6.1 : Transfer function of ideal low and high pass filter

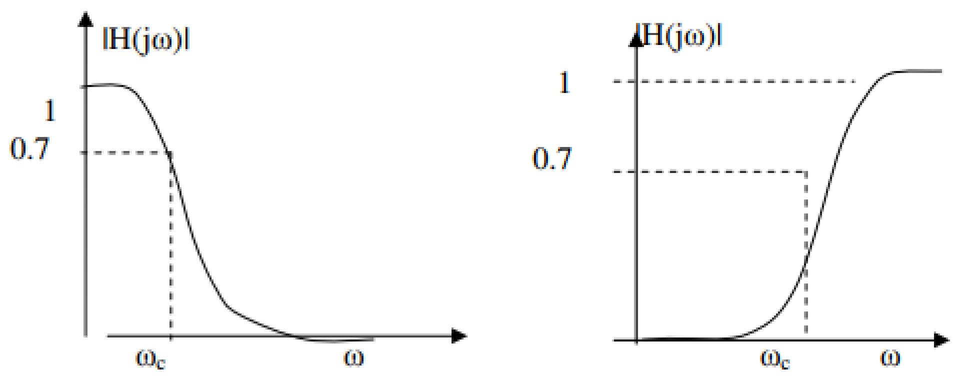


Fig. 6.2 : Transfer function of practical low and high pass filter

RC Filter: The simplest passive filter circuit can be made by connecting a single resistor and a single capacitor in series across an input signal, (Vin) with the output signal, (Vout) taken from the junction of these two components. Depending on which way around we connect the resistor and the capacitor with regards to the output signal determines the type of filter construction resulting in either a Low Pass or a High Pass Filter. As there are two passive components within this type of filter design the output signal has amplitude smaller than its corresponding input signal, therefore passive RC filters attenuate the signal and have a gain of less than one, (unity).

Low-Pass RC Filter: A series RC circuit as shown also acts as a low-pass filter. For no load resistance (output is open circuit, $R \to \infty$):

$$Vo = \frac{\frac{1}{j\omega C}}{R + \left(\frac{1}{j\omega C}\right)} Vi = \frac{1}{1 + j(\omega RC)} Vi$$

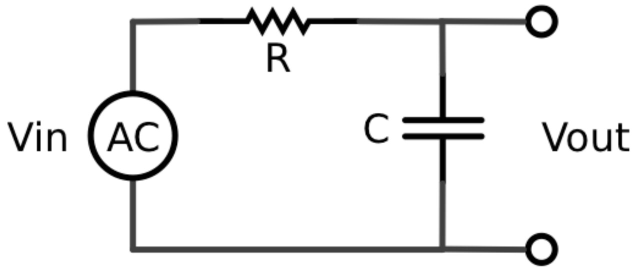


Fig. 6.3: Low-Pass RC Circuit

$$|H(j\omega)| = \frac{Vo}{Vi} = \frac{1}{\sqrt{1 + (\omega RC)^2}}$$

When ω→0, |H(jω)| is maximum and →1. For ω = ωc, |H(jωc)| = 1/√2. Thus

$$|H(j\omega_c)| = \frac{Vo}{Vi} = \frac{1}{\sqrt{1 + (\omega_c RC)^2}} = 1/\sqrt{2}$$

$$\omega_c \ = 1/RC,$$

$$H(j\omega) = \frac{1}{1 + \frac{j\omega}{\omega_c}}$$

$$|H(j\omega)| = \frac{1}{\sqrt{1 + \left(\frac{\omega}{\omega_c}\right)^2}}$$

$$Phase, \emptyset = \ -\tan^{-1}\left(\frac{\omega}{\omega_c}\right)$$

High-Pass RC Filter: A series RC circuit as shown in Fig.6.4 acts as a high-pass filter.

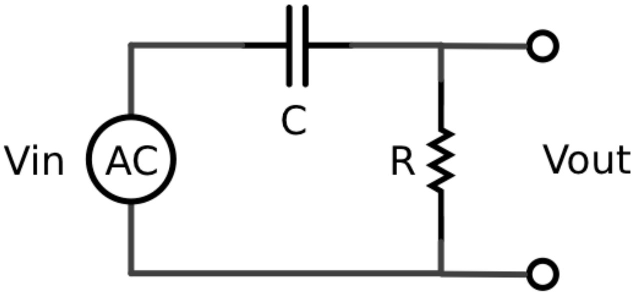


Fig. 6.4: High-Pass RC Circuit

For no load resistance (output open circuit), we have

$$V_o = \frac{R}{R + \left(\frac{1}{j\omega C}\right)} V_i$$

$$H(j\omega) = \frac{V_o}{V_i} = \frac{1}{1 - j(\frac{1}{\omega RC})}$$

$$|H(j\omega_c)| = \frac{Vo}{Vi} = \frac{1}{\sqrt{1 + (1/\omega_c RC)^2}} = 1/\sqrt{2}$$

$$\omega_c \;= 1/RC,$$

$$H(j\omega) = \frac{1}{1 + \frac{j\omega_c}{\omega}}$$

$$|H(j\omega)| = \frac{1}{\sqrt{1 + \left(\frac{\omega_c}{\omega}\right)^2}}$$

$$Phase, \emptyset = -\tan^{-1}\left(\frac{\omega_c}{\omega}\right)$$

PROCEDURE:

1. Begin lab by familiarizing yourself with the function generator and oscilloscope.

2. Read and measure the values of R and C.

3. Using the scope set the function generator to produce a 10 V(pp) sine wave. This signal will be used for the input. Do not change the amplitude of this signal during the experiment.

4. Set up the low/high pass RC filter on the breadboard as shown in the circuit diagram. Use the function generator to apply a 10 V(pp) sine wave signal to the input. Use the dual trace oscilloscope to look at both Vin and Vout. Be sure that the two oscilloscope probes have their grounds connected to the function generator ground.

5. For several frequencies between 20 Hz and 20 kHz (the audio frequency range) measure the peak-to-peak amplitude of Vout . Check often to see that V in remains roughly at the set value and that VOLTS/DIV dials are in their calibrated positions. Take enough data (at least up to 10 times the cut-off frequency, for low pass and down to 1/10 times cut-off frequency, for high pass filter) to make your analysis complete. If needed use the STOP button of oscilloscope at a desired frequency to acquire data.

6. From your measurements determine the ratio

$$|H(j\omega)| = \left|\frac{V_o}{V_i}\right| = \frac{V_o(pp)}{V_i(pp)}$$

and compute this ratio by using the formula

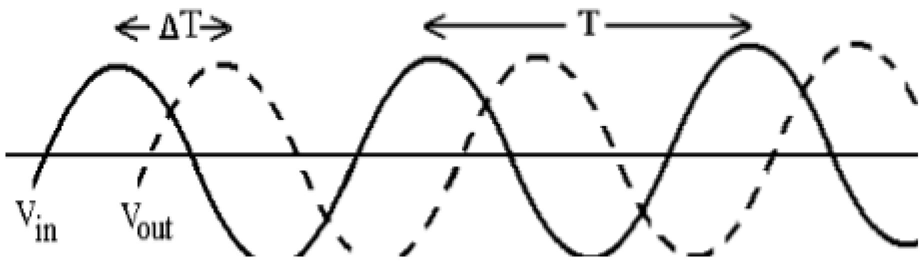
For low pass filter:

$$|H(j\omega)| = \frac{1}{\sqrt{1 + \left(\frac{f}{f_c}\right)^2}}$$

For high pass filter:

$$|H(j\omega)| = \frac{1}{\sqrt{1 + \left(\frac{f_c}{f}\right)^2}}$$

7. For each listed frequency, measure the phase shift angle φ with proper sign as shown in the diagram below.



8. The phase shift angle in degrees is

$$\emptyset = \left(\frac{\Delta T}{T}\right) X\ 360°$$

9. Compute the phase shift angle for each frequency for low/high pass filter.

OBSERVATIONS:

R = ______10KΩ_________,   C = ___0.01µf_________

(I) For Low Pass Filter: V_{in} (pp) = 10V

$$\boldsymbol{f_c = \frac{1}{2\pi RC} = 1.6KHz}$$

(a) Observation Table for $|\boldsymbol{H(j\omega)}|$

S.no.	Frequency (kHz) (ω =2πf)	$\frac{f}{f_c}$	Vo (PP) volts	$\lvert H(j\omega)\rvert = \frac{V_o(pp)}{V_i(pp)}$	$\lvert H(j\omega)\rvert = \frac{1}{\sqrt{1+\left(\frac{f}{f_c}\right)^2}}$
1	0.5	0.3125	9.5	0.95	
2	1	0.625	8.56	0.856	
3	2		6.5	0.65	
4	3		5.0	0.50	
5	4		3.9	0.39	
6	5		3.3	0.33	
7	6		2.72	0.272	

8	7		2.48	0.248	
9	8		2.08	0.208	
10	9		1.92	0.192	
11	10		1.76	0.176	
12	11		1.68	0.168	
13	12		1.52	0.152	

(a) Observation Table for $|\boldsymbol{H(j\omega)}|$

S.no.	Frequency (kHz) (ω =2πf)	$\frac{\omega}{\omega_c}$	ΔT (ms)	T (ms)	$\emptyset = \left(\frac{\Delta T}{T}\right) X\, 360°$ (deg)	$Phase, \emptyset = -\tan^{-1}\left(\frac{\omega}{\omega_c}\right)$
1						
2						
3						
4						
5						
6						

II. For High Pass Filter: $\mathbf{V_{in}}$ (pp) =

$$\boldsymbol{\omega_c = \frac{1}{RC} =}$$

(b) Observation Table for $|\boldsymbol{H(j\omega)}|$

S.no.	Frequency (kHz) (ω =2πf)	$\frac{\omega_c}{\omega}$	Vo (PP) volts	$\lvert H(j\omega)\rvert = \frac{V_o(pp)}{V_i(pp)}$	$\lvert H(j\omega)\rvert_{dB}$	$\lvert H(j\omega)\rvert = \frac{1}{\sqrt{1+\left(\frac{\omega_c}{\omega}\right)^2}}$
1						
2						
3						

4						
5						
6						

(b) Observation Table for $|\boldsymbol{H(j\omega)}|$

S.no.	Frequency (kHz) (ω =2πf)	$\frac{\omega_c}{\omega}$	ΔT (ms)	T (ms)	$\emptyset = \left(\frac{\Delta T}{T}\right) X\ 360°$ (deg)	$Phase, \emptyset = -\tan^{-1}\left(\frac{\omega_c}{\omega}\right)$
1						
2						
3						
4						
5						
6						

Graphs: Trace and study bode plots of |H(jω)|dB and φ versus f(×2π) in a semi-log format for low/high pass RC filter. Determine the cut-off frequency from graph. Also, estimate the frequency roll-off for each filter

RESULTS:

Experiment No. 7

AIM: Study and measurement of current-voltage (I-V) characteristics of a PN junction Diode (Ge and Si)

APPARATUS REQUIRED: Variable DC power supply (0-30 V), Breadboard, Multimeters (two Nos.), pn junction diode, resistors of different values, connecting wires and crocodile clips. Resistors of different values: 100 Ω, 1 kΩ, 10 kΩ, 100 kΩ and 1MΩ

THEORY: A pn junction diode is active electronics component made of semiconductor material such as silicon (Si). The basic feature of the diode is that it conducts significantly when forward biased, beyond a certain critical voltage called "***diode cut-in voltage***". Forward biasing refers to the p-side connected to positive and n-side connected to negative terminal of battery. For pn junction diodes made in silicon, the value of cut-in voltage is about 0.7 V. For diodes made in germanium (Ge) semiconductor material, the corresponding value is of the order of 0.3 V. In the reverse bias conditions (n side connected to positive terminal and p-side to negative terminal of battery), negligible current flows through the diode till the voltage reaches the breakdown value. The diode breakdown voltage may vary from a few volts to few hundreds of volts and is controlled by various design, process and material parameters of the diode. The precise measurement and understanding of diode I-V characteristics is important for using these components in a variety of electronic circuits.

SETUP FOR I-V MEASUREMENT: The circuit used for measuring the I-V characteristics of a pn junction diode under forward biasing is schematically illustrated in Fig. 7.1. For measurement under reverse biasing, the polarity of the diode is reversed. In the measurement setup presented here, the current through the diode is determined by measuring the voltage across the resistor R using a multimeter as shown in Fig. 7.1. The value of the resistor is chosen such that the voltage across it falls in the measurement range of the multimeter used. With this arrangement, current as low as 10 nA could be measured accurately. One can also use an ammeter for current measurement, if the instrument with required range and accuracy is available in the Lab.

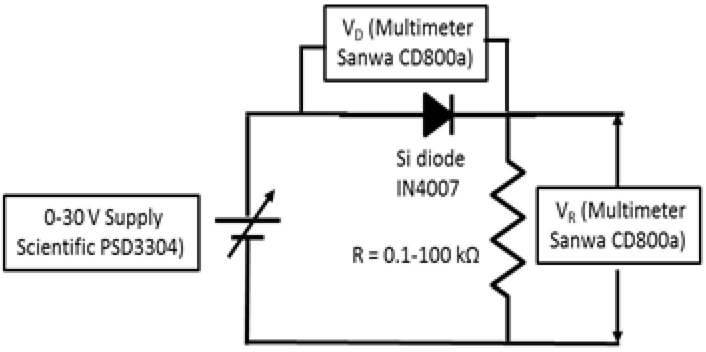
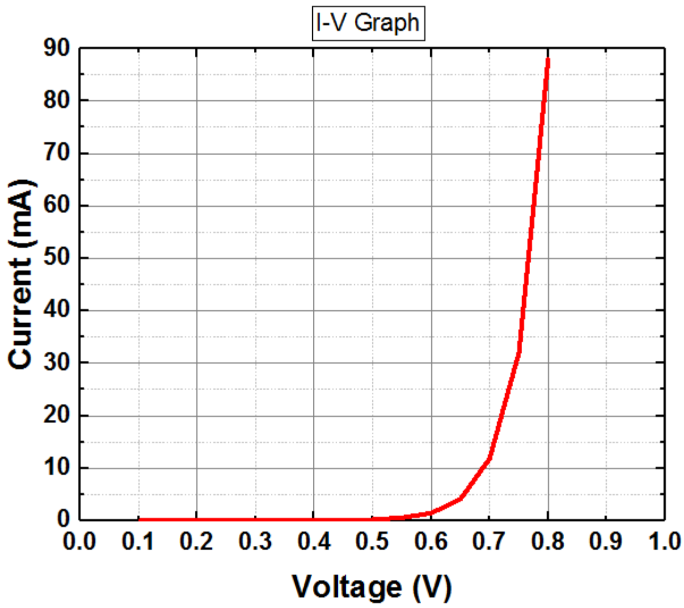


Fig.7.1: Measurement Setup for I-V Characteristics of PN Junction Diode

OBSERVATION TABLE: The I-V measurement results under **forward biasing** are presented in the Table below.

S. No.	V_D (mV)	V_R	R	$I = V_R / R$
1	103.7	1.1 mV	100 kΩ	11 nA
2	150	3.5 mV	100 kΩ	35 nA
3	200	11.6 mV	100 kΩ	116 nA
4	250	38.4 mV	100 kΩ	384 nA
5	300	128.3 mV	100 kΩ	1.2 µA
6	350	0.435 V	100 kΩ	4.35 µA
7	400	1.555 V	100 kΩ	15.5 µA
8	450	5.65 V	100 kΩ	56.5 µA
9	500	17.7 V	100 kΩ	177 µA
10	550	5.3 V	10 kΩ	0.53 mA
11	600	14.18 V	10 kΩ	1.40 mA
12	650	4.08 V	1 kΩ	4.08 mA
13	700	11.78 V	1 kΩ	11.78 mA
14	750	3.21 V	100 Ω	32 mA
15	800	8.79 V	100 Ω	87.9 mA



The I-V measurement results under **reverse biasing** are presented in the Table below.

S. No.	V_D, V	V_R	R	$I = V_R / R$
1	1	116.5 mV	1 MΩ	0.11 μA
2	5	0.516 V	1 MΩ	0.516 μA
3	7	0.720 V	1 MΩ	0.72 μA
4	10	1.02 V	1 MΩ	1.02 μA
5	12	1.212 V	1 MΩ	1.21 μA
6	15	1.50 V	1 MΩ	1.5 μA
7	17	1.70 V	1 MΩ	1.7 μA
8	20	1.90 V	1 MΩ	1.9 μA
9	22	2.20 V	1 MΩ	2.2 μA
10	25	2.49 V	1 MΩ	2.49 μA
11	27	2.67 V	1 MΩ	2.6 μA
12	30	2.86 V	1 MΩ	2.86 μA

Note: From the measurements, it is apparent that the breakdown voltage of the diode is more than 30 V.

RESULTS AND DISCUSSIONS: The data on I-V measurement is plotted on a graph sheet to comprehend the diode behaviour in forward and reverse bias conditions. The accuracy of measurement lies in selecting the proper value of the resistor R connected in series with diode for current measurement. Its value should be chosen such that the voltage across it falls in the measurement range of the multimeter. With proper selection of the resistor value, low current values in the range of 10 nA can also be measured using inexpensive multimeter which are readily available in the Lab.

Experiment No. 8

HALF WAVE RECTIFIER

Aim

a) To build a half –wave rectifier circuit with given component values on a bread board. (with out and with capacitor)

b) To observe the out-put waveforms on the oscilloscope.

c) To calculate the peak value, rms value and ripple factor of the output waveforms

d) To plot the input and output waveforms on a graph sheet to scale.

Equipment required

Cathode Ray Oscilloscope (CRO), Signal generator, Multimeter, Bread board, Probes and connecting wires

Theory

Half-wave rectifier is primarily used to convert a given ac signal into a DC signal. Power transmitted from power stations is an AC signal (220V/50 Hz) whereas most of the home appliances or other electrical equipment we use in our day-to-day life require DC power. The method by which we convert a given AC signal into a DC signal is called 'rectification'. In this experiment, one of such basic circuits used for rectification is built. The output current

Circuit Diagram

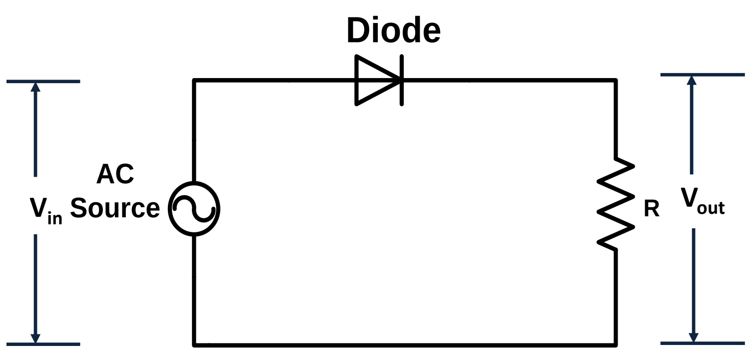


Fig (a)

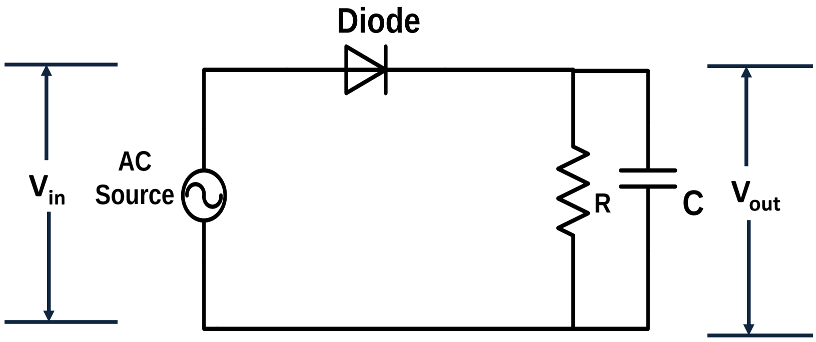


Fig (b)

List of components

1. Diode- Model number 1N4007
2. Resistor (R)-10KΩ
3. Capacitor (C)- 5µF

Operation

During the positive half of the input sinusoidal signal the diode is forward biased and is conducting. The diode has a very low resistance in the forward bias state. So the output taken across the resistor 'R' in fig (a) almost follows the input after a drop of around 0. Volts across the diode (cut in voltage of the diod) Its value is V_{in} - $V_{cutin..}$ During the negative half if the input signal the diose is reverse biased and has a very high resistance. This prevents any current from flowing through the resistor 'R'. Hence the output voltage is zero in this cycle.

When a capacitor is connected across the resistor as in Fig. (b) , during the positive half cycle the circuit is conducting and the capacitor is quickly charged to the peak value of the input signal as the time constant during the positive cycle ($\tau = \frac{RR_f}{R+R_f}C$) .very low. Here '$R_f$ ' is the forward resistance of the diode. Voltage measured across the resistor is equal to input voltage. During the negative half cycle diode is reverse biased and the circuit is broken across the diode due to this reverse bias operation. The capacitor then slowly discharges its charge into the resistor 'R'. Time constant ($\tau = RC$) is relatively high for this case.

Experiment Procedure

1. Connect the circuit shown in Fig (a) (without capacitor) on the bread board.
2. Apply a sinusoidal signal of a given frequency *f* across V_{in} and observe the waveforms across V_{out} in Fig (a)
3. Plot the output waveforms with respect to the input wave form to scale on a graph sheet.
4. Now connect the circuit given in Fig (b) (with capacitor) and repeat steps 2 and 3.
5. Make a table of the following observations.

Without capacitor

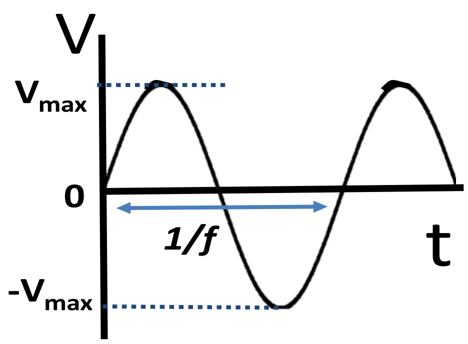
1. AC Input voltage (rms) Vrms= ____________
2. DC output voltage V_{DC} = ____________
3. DC current: I_{DC} =_______________
4. AC output voltage (Ripple voltage) V_r: ___________
5. Ripple factor: (V_r/V_{DC}) = _______________

With capacitor

1. AC Input voltage (rms) Vrms= ____________
2. DC output voltage V_{DC} = ____________
3. DC current: I_{DC} =_______________
4. AC output voltage (Ripple voltage) V_r: ___________
5. Ripple factor: (V_r/V_{DC}) = _______________

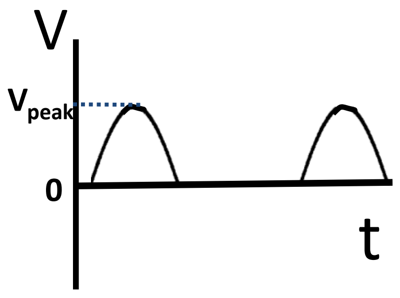
Expected waveforms

Input Waveform



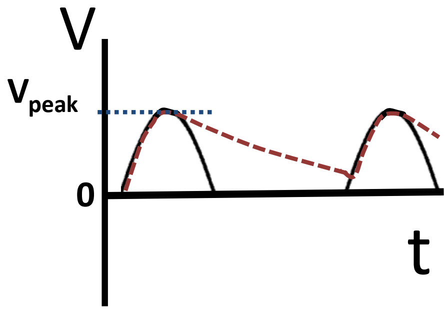
Output Waveform

For the circuit in Fig (a) [Without Capacitor]



For the circuit in Fig (b) [With Capacitor]

Output waveform



Observed Waveforms:

Without capacitor:

With capacitor:

Conclusion:

Experiment No. 9

FULL WAVE RECTIFIER

Aim

a) To build a full –wave rectifier circuit with given component values on a bread board. (with out and with capacitor)

b) To observe the out-put waveforms on the oscilloscope.

c) To calculate the peak value, rms value and ripple factor of the output waveforms

d) To plot the input and output waveforms on a graph sheet to scale.

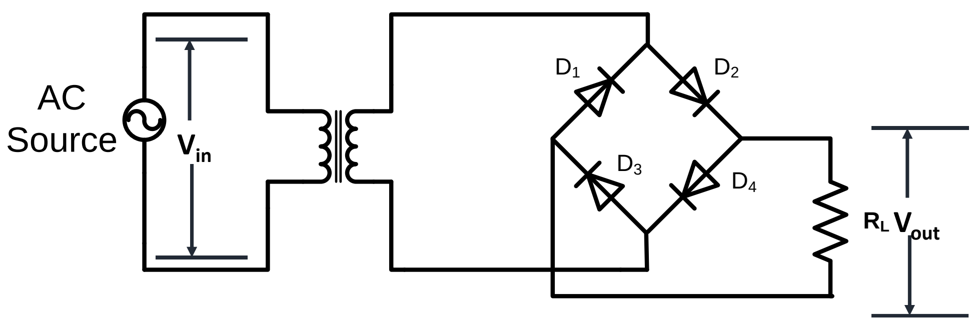
Equipment required

Cathode Ray Oscilloscope (CRO), Signal generator, Multimeter, Bread board, Probes and connecting wires

Theory

Full-wave rectifier is also used to convert a given AC signal into a DC signal as in half- wave rectifier. The main difference between the operation of a half -wave rectifier and full-wave rectifier is that the full-wave rectifier converts input AC signal into half sinusoidal pulse in both positive and negative cycles of the input sinusoidal wave signal unlike the half wave rectifier whose output is zero during the negative cycle. Hence full wave rectifier is efficient than a half wave rectifier.

Circuit Diagram



Circuit Operation

During the positive half cycle diodes D2 and D3 are turned ON whereas diodes D1 and D4 are in off state. Current flows through diodes D2, D3 into the output resistor ‘R’. Output voltage is $V_{out} = V_{in} – 2V_{cutin}$ .Here V_{cutin} is the diode cut in voltage. During the negative half cycle of the input , dioses D1 and D4 are conducting while diodes D2 and D3 are OFF. Current flows through D1 and D4 into the resistor. The direction of current flow in the resistor is same for both the cases. Hence $V_{out} = V_{in} – 2V_{cutin}$ for negative half cycle also.

List of components

1. Diode- Model number 1N4007 – 4 No's
2. Resistor (R)-10KΩ

Experiment Procedure

1. Connect the circuit shown in Fig (a) (without capacitor) on the bread board.
2. Apply a sinusoidal signal of a given frequency f across V_{in} and observe the waveforms across V_{out} in Fig (a)
3. Plot the output waveforms with respect to the input wave form to scale on a graph sheet.
4. Now connect a capacitor of value C μF across and repeat step 3.
5. Make a table of the following observations.

Without capacitor

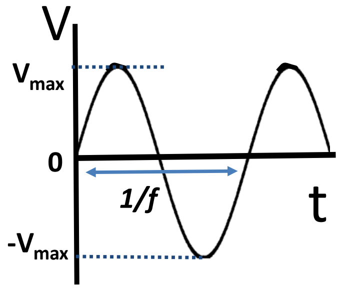
1. AC Input voltage (rms) Vrms= ____________
2. DC output voltage V_{DC} = ____________
3. DC current: I_{DC} =_______________
4. AC output voltage (Ripple voltage) V_r: ___________
5. Ripple factor: (Vr/V_{DC}) = _______________

With capacitor

1. AC Input voltage (rms) Vrms= ____________
2. DC output voltage V_{DC} = ____________
3. DC current: I_{DC} =_______________
4. AC output voltage (Ripple voltage) V_r: ___________
5. Ripple factor: (Vr/V_{DC}) = _______________

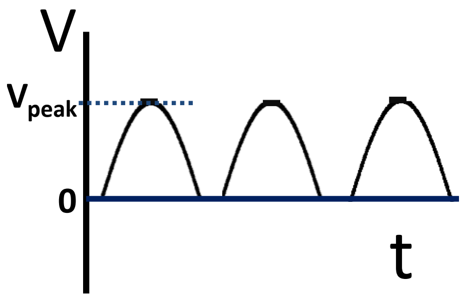
Expected waveforms

Input Waveform

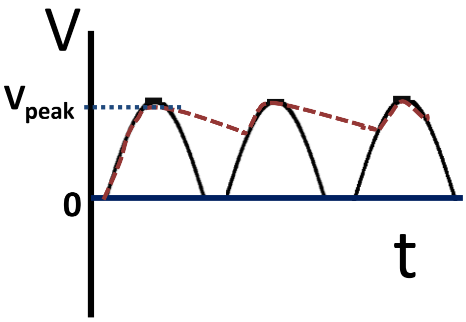


Output Waveform

For the circuit in Fig (a) [Without Capacitor]



For the circuit in Fig (b) [With Capacitor]



For the circuit in Fig (b) [With Capacitor]

Observed Waveforms:

Without capacitor:

With capacitor:

Conclusion:

Experiment No. 10

CLIPPER

Aim

a) To be able to design a clipper circuit for a desired clipping value.
b) To construct the circuit on the bread board
c) To observe out-put waveforms on the oscilloscope.
d) To calculate the peak values of waveforms for positive and negative cycles
e) To plot input and output waveforms on a graph sheet to scale.

Equipment required

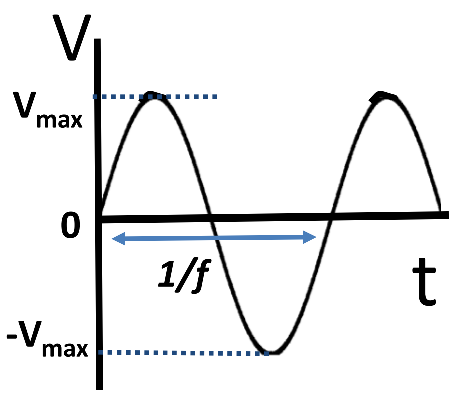
Cathode Ray Oscilloscope (CRO), Signal generator, Multimeter, DC power supply, Bread board, Probes and connecting wires

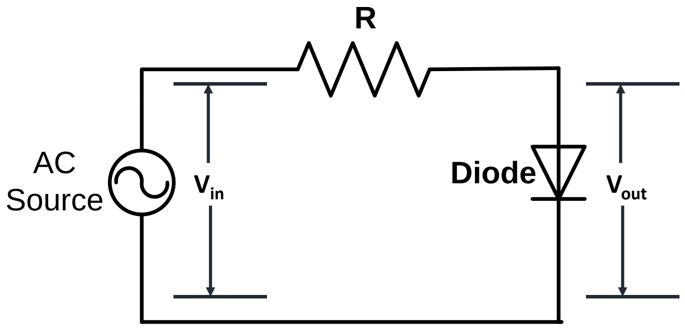
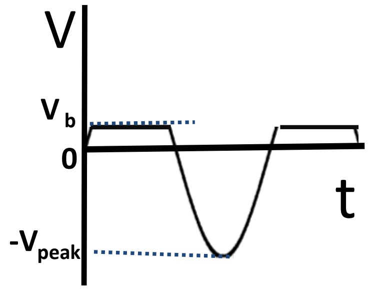
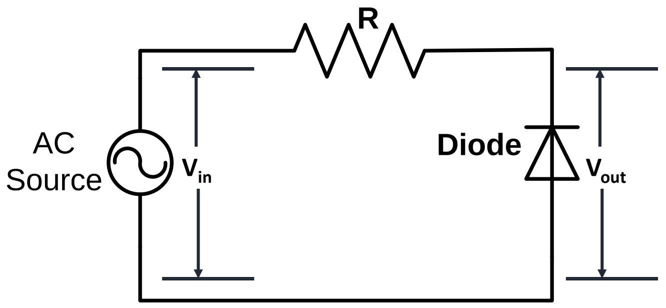
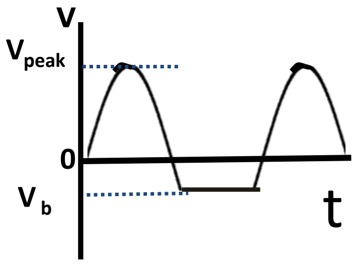
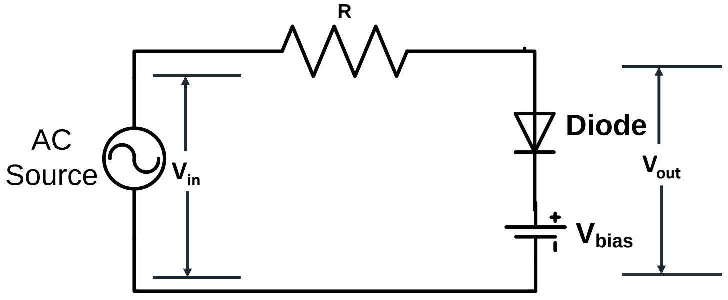
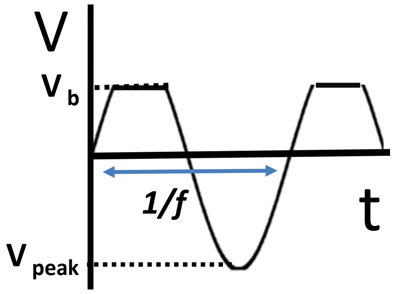
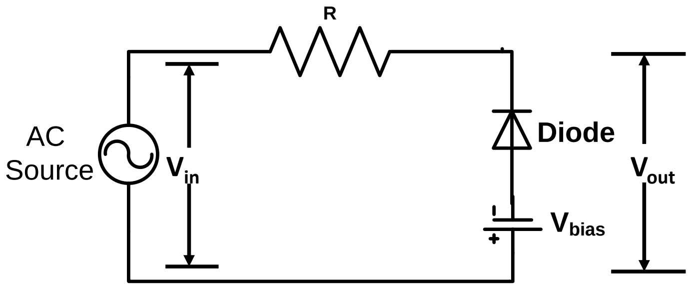
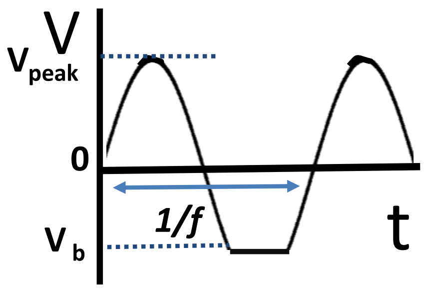
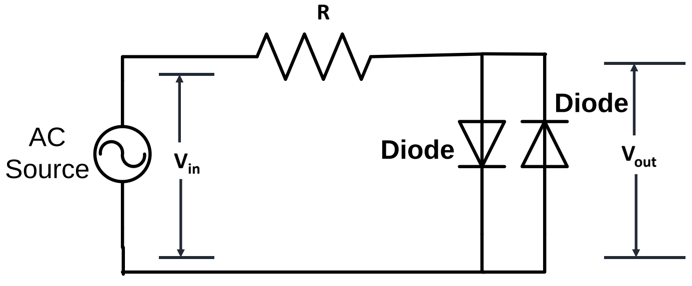
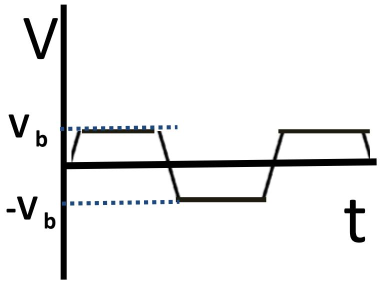
Theory

Circuit with which waveforms can be shaped by removing a portion of the signal is known as clipper circuit. There are used as limiters / slicers to chip the waveform above/ below a certain level.

Circuit Diagrams and corresponding waveforms

Input signal waveform



<table>
<tr><th>Circuit Diagram</th><th>Expected wave form</th></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>
</td><td>
</td></tr>
</table>

Components Required:

1) Diode (1N4007) -1 No.
2) Resistor -1 No.

Operation

When the diode is in OFF state the input wave form follows the output waveform as we have a open circuit condition across the output. When it is in the ON state the output waveform is the sum of the DC voltage drop across the diode and dc power supply.

Experiment Procedure

1. Connect the circuit one after the other as shown in Table 1
2. Connect a DC supply in series with the diode where ever required in the circuit. Initially set the dc value to 0 and vary it slowly to the desired value.
3. Apply a sinusoidal signal of a given frequency f across V_{in} and observe the waveforms across V_{out} in Fig (a)
4. Measure the voltage levels of all the waveform for positive and negative cycles.
5. Plot the output waveforms with respect to the input wave form to scale on a graph sheet.

Conclusion:

Experiment No. 11

CLAMPER

Aim

a) To design a clamper circuit for various clamping values

b) To observe the out-put waveforms on the oscilloscope.

d) To plot the input and output waveforms on a graph sheet to scale.

Equipment required

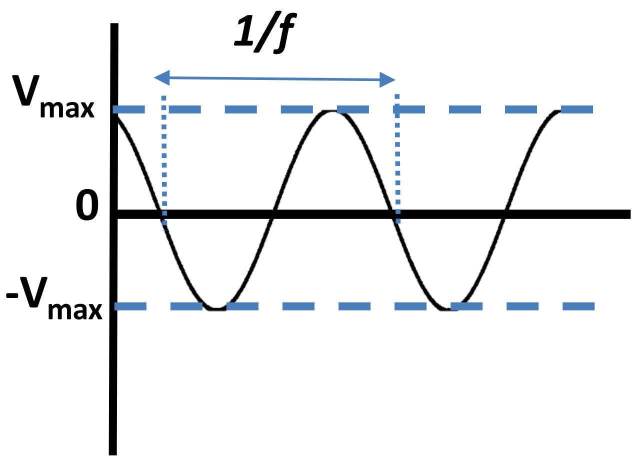
Cathode Ray Oscilloscope (CRO), Signal generator, Multimeter, DC power supply, Bread board, Probes and connecting wires

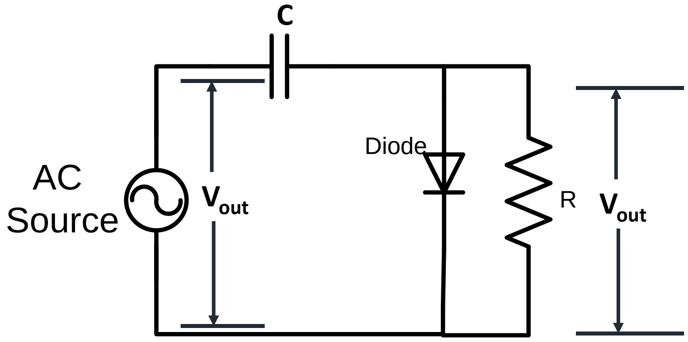
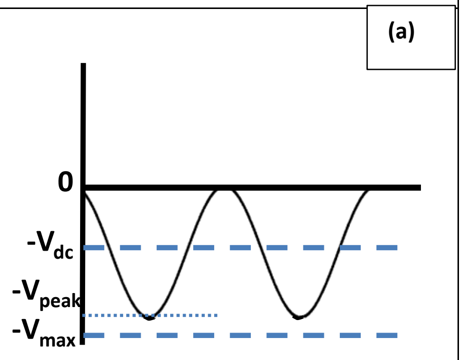
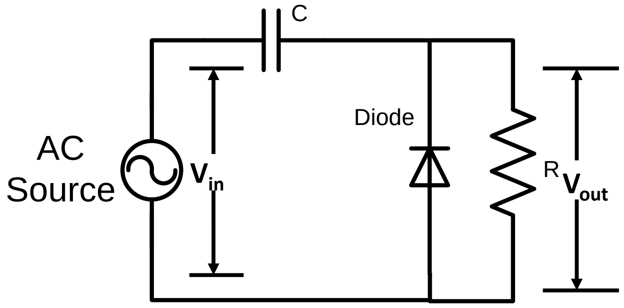
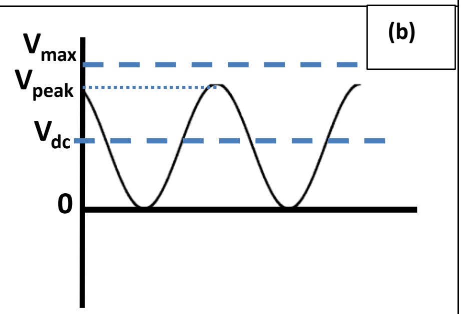
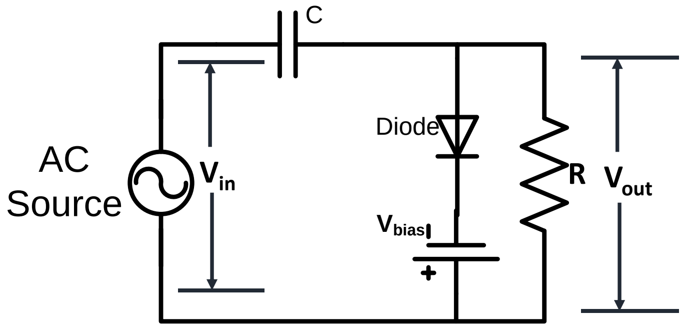
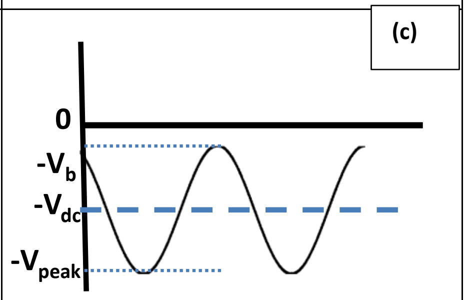
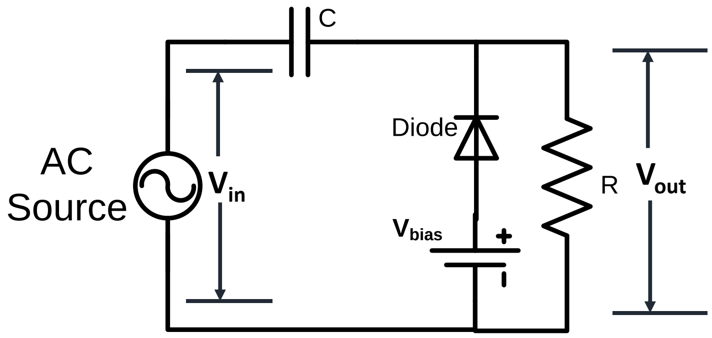
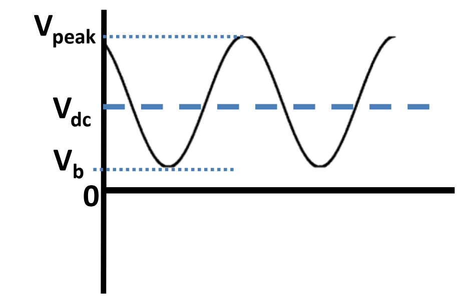
Theory

Clamping circuits are used to change the dc levels of an alternating waveform without changing the shape of the waveform. This is one more application of a diode circuits.

Circuit Diagram and corresponding output waveform

Input signal waveform



Circuit Diagram	Expected waveform
	
	
	
	

Operation

For the circuit in fig (a) , when an alternating signal say $V_m \sin\omega t$,during the first positive cycle of the input waveform the diode is in conducting state and due to a very low resistance of the diode in this state , the capacitor is charged to the peak value of the input. Voltage across the capacitor is $-V_m$ During the negative cycle the diode if OFF and the voltage across the resistor is now $-V_m-V_m \sin\omega t$. Note that the time constant in this state is very large and the capacitor

voltage is almost constant. During the next positive cycle the voltage across the resistor is $-V_m+V_m\sin\omega t$. The diode remains OFF for rest of the time.
When an additional bias voltage V_{bias} is applied in series with the diode as shown in fig (c) capacitor gets charges to a value of $-V_m-V_{bias}$. Rest of the operation is same as in Fig (a).

List of components

1. Diode- Model number 1N4007
2. Resistor (R)-10KΩ
3. Capacitor (C)- 5µF

Experiment Procedure

1. Connect the circuit shown in Fig (a) on the bread board.
2. Apply a sinusoidal signal of a given frequency f across V_{in} and observe the waveforms across V_{out} in Fig (a)
3. Plot the output waveforms with respect to the input wave form to scale on a graph sheet.
4. Repeat steps 1,2,3 for circuits in Fig (a) to Fig (d).

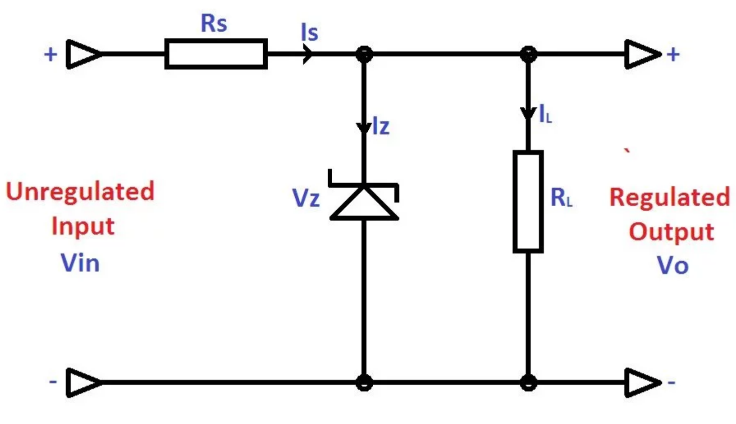
Conclusion:

Experiment No. 12

AIM: To study Zener diode as a voltage regulator

APPRATUS REQUIRED: Zener diode, multimeter, bread board, power supply, connecting wires.

THEORY: Zener diode is a P-N junction diode specially designed to operate in the reverse biased mode. It is acting as normal diode while forward biasing. It has a particular voltage known as break down voltage, at which the diode break downs while reverse biased. In the case of normal diodes, the diode damages at the break down voltage. But Zener diode is specially designed to operate in the reverse breakdown region. The basic principle of Zener diode is the Zener breakdown. When a diode is heavily doped, it's depletion region will be narrow. When a high reverse voltage is applied across the junction, there will be very strong electric field at the junction. And the electron hole pair generation takes place. Thus, heavy current flows. This is known as Zener break down. So, a Zener diode, in a forward biased condition acts as a normal diode. In reverse biased mode, after the break down of junction current through diode increases sharply. But the voltage across it remains constant. This principle is used in voltage regulator using Zener diodes The figure shows the zener voltage regulator, it consists of a current limiting resistor RS connected in series with the input voltage Vs and zener diode is connected in parallel with the load RL in reverse biased condition. The output voltage is always selected with a breakdown voltage Vz of the diode.



The input source current, $I_S = I_Z + I_L$ (1)

The drop across the series resistance, $R_S = Vin - Vz$ (2)

And current flowing through it, $I_S = (Vin - Vz)/ R_S$ (3)

From eqn (1) & (2), we get, $(Vin - Vz)/ R_S = I_Z + I_L$ (4)

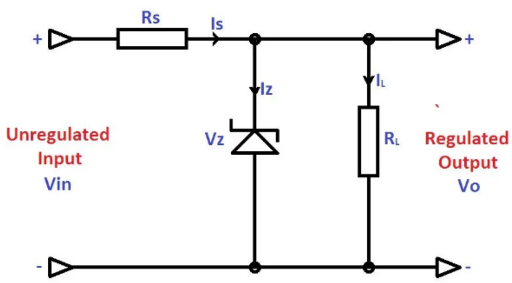
Regulation with a varying input voltage (line regulation): It is defined as the change in regulated voltage with respect to variation in line voltage. It is denoted by 'LR'. In this, input voltage varies but load resistance remains constant hence, the load current remains constant. As the input voltage increases, form equation (3) Is also varies accordingly. Therefore, zener current Iz will increase. The extra voltage is dropped across the Rs. Since, increased Iz will still have a constant Vz and Vz is equal to Vout. The output voltage will remain constant. If there is decrease in Vin, Iz decreases as load current remains constant and voltage drop across Rs is reduced. But even though Iz may change, Vz remains constant hence, output voltage remains constant.

Regulation with the varying load (load regulation): It is defined as change in load voltage with respect to variations in load current. To calculate this regulation, input voltage is constant and output voltage varies due to change in the load resistance value. Consider output voltage is increased due to increasing in the load current. The left side of the equation (4) is constant as input voltage Vin, IS and Rs is constant. Then as load current changes, the zener current Iz will also change but in opposite way such that the sum of Iz and IL will remain constant. Thus, the load current increases, the zener current decreases and sum remain constant. Form reverse bias characteristics even Iz changes, Vz remains same hence, and output voltage remains fairly constant.

PROCEDURE:-

A) Line Regulation:

1. Make the connections as shown in figure below.



2. Keep load resistance fixed value; vary DC input voltage from 5V to 15V.

Observation Table for Line Regulation:

S.no.	Vin (volts)	V_L (in volts)

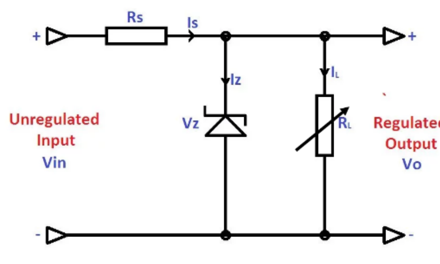
3. Plot the graph between V_L vs Vin

3. Note down output voltage as a load voltage with high line voltage 'V_{HL}' and as a load voltage with low line voltage 'V_{LL}'.

4. Using formula, % Line Regulation = $(V_{HL}-V_{LL})/ V_{NOM}$ x100, where V_{NOM} = the nominal load voltage under the typical operating conditions. For ex. VNOM = 9.5 ± 4.5 V)

Load Regulation:

1. For finding load regulation, make connections as shown in figure below.



2. Keep input voltage constant say 10V, vary load resistance value.

Observation Table for Load Regulation:

S.no.	R_L (in ohm)	V_L (in volts)

3. Plot the graph between V_L vs R_L

3. Note down no load voltage 'V_{NL}' for maximum load resistance value and full load voltage 'V_{FL}' for minimum load resistance value.

4. Calculate load regulation using, % load regulation = $(V_{NL}-V_{FL})/ V_{FL}$ x100

CONCLUSIONS: