

# **C.V. RAMAN POLYTECHNIC BHUBANESWAR**



## **DEPARTMENT OF ELECTRONICS AND TELECOMMUNICATION ENGINEERING**

### **LAB MANUAL**

*Year & Semester: 2<sup>ND</sup> Year, IV Semester*

*Subject Code/Name: PR- 4, ANALOG ELECTRONICS & LINEAR IC*

## LIST OF THE EXPERIMENTS

SL.NO.	NAME OF THE EXPERIMENT
1	Determine the forward & reverse characteristics of PN Junction diode.
2	Determine the forward & reverse characteristics of Zener diode.
3	Construct Bridge Rectifier using different filter circuit and to determine ripple factor & analyze wave form with filter & without filter.
4	To study the input and output characteristics of common emitter connection.
5	To study the input and output characteristics of common base connection.
6	Construct & test the transistor regulator using Zener diode.
7	To study two stage RC coupled amplifier, find gain & draw the frequency response curve.
8	Determine drain & Transfer characteristics of JFET.
9	Determine drain & Transfer characteristics of MOSFET.
10	Construct & test timer circuit using IC 555 timer.
11	Construct and observe the waveform of Clipper circuits.
12	Construct and observe the waveform of clamper circuits.
13	Construct and test voltage power supply using 78XX and 79XX.
14	Construct & test voltage power supply using LM723.
15	Study of Operational Amplifier 741 & draw its pin diagram.
16	To construct and study inverting amplifier using OPAMP.
17	To construct and study inverting amplifier using OPAMP.
18	To construct and study the differentiator using OPAMP.
19	To construct and study the integrator using OPAMP.

## EXPERIMENT NO.: -1(a)

### AIM OF THE EXPERIMENT:

Determine the forward & reverse characteristics of PN Junction diode.

### EQUIPMENT REQUIRED:

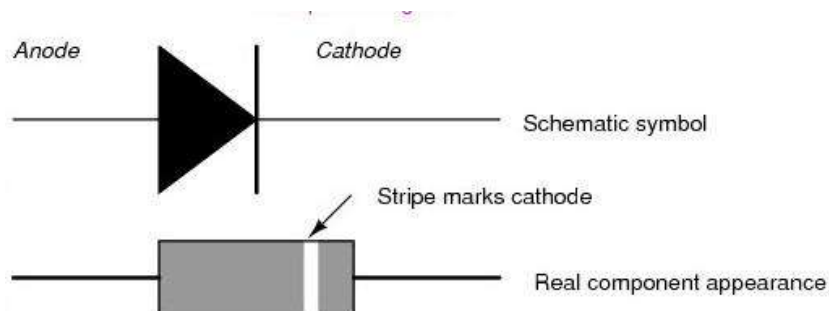
Hardware- Personal Computer

Software- Multisim Software

### THEORY:

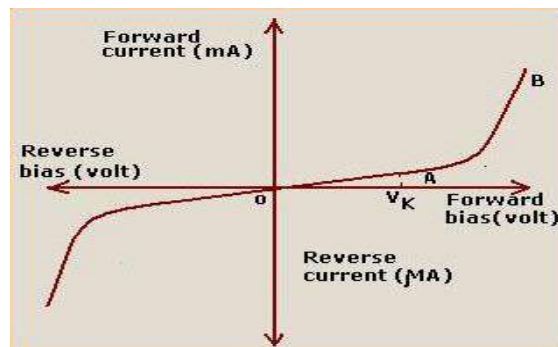
There are three possible biasing conditions and two operating regions for the typical PN-Junction Diode, they are zero bias, forward bias and reverse bias.

When no voltage is applied across the PN junction diode then the electrons will diffuse to P-side and holes will diffuse to N-side through the junction and they combine. Therefore, the acceptor atom close to the P-type and donor atom near to the N-side is left unutilized. An electronic field is generated by these charge carriers. This opposes further diffusion of charge carriers. Thus, no movement of the region is known as the depletion region or space charge.



If we apply forward bias to the PN-junction diode that means the negative terminal is connected to the N-type material and the positive terminal is connected to the P-type material across the diode which has the effect of decreasing the width of the PN junction diode.

If we apply a reverse bias to the PN-junction diode, that means the positive terminal is connected to the N-type material and the negative terminal is connected to the P-type material across the diode which has the effect of increasing the width of the PN junction diode and no charge can flow across the junction

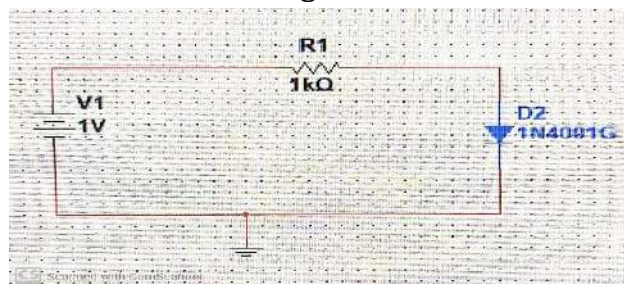


**PROCEDURE:**

1. Start MULTISIM. A blank circuit window will appear on the screen along with a component tool bar.
2. Using component tool bar, place all the components on the circuit window and wire the circuit.

**For Forward Characteristics-**

3. Connect the circuit as shown in circuit diagram.



4. Components Required-
  - a) Diode- 1N4001G
  - b) Ground
  - c) Resistor- 1KΩ
  - d) V1- 1V

5. Go to analyses and simulation-→DC sweep.
6. In the DC sweep window set the following values:

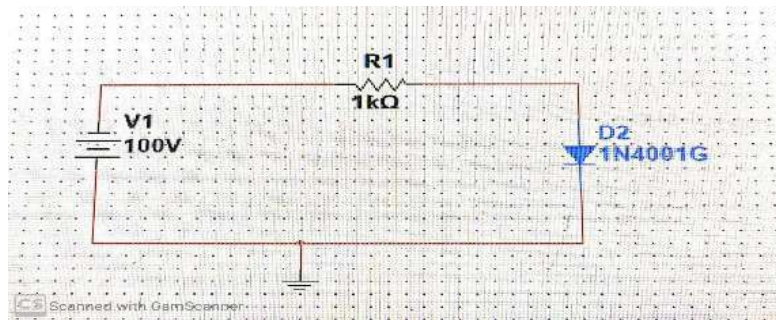
Source1

- Source-  $V_1$
- Start Value- 0
- Stop Value- 1
- Increment- 0.1

7. Then save it.
8. Go to analyses and simulation-→ DC sweep →output and select the value I (D1 [ID]) and click on add option.
9. Then run the simulation and observe the forward characteristics.

**For Reverse Characteristics-**

10. Connect the circuit as shown in circuit diagram.



11. Components Required-

- e) Diode- 1N4001G
- f) Ground
- g) Resistor- 1KΩ
- h) V1- 100V

12. Go to analyses and simulation → DC sweep.

13. In the DC sweep window set the following values:

Source1

Source-  $V_1$

Start Value- 0

Stop Value- 60

Increment- 10

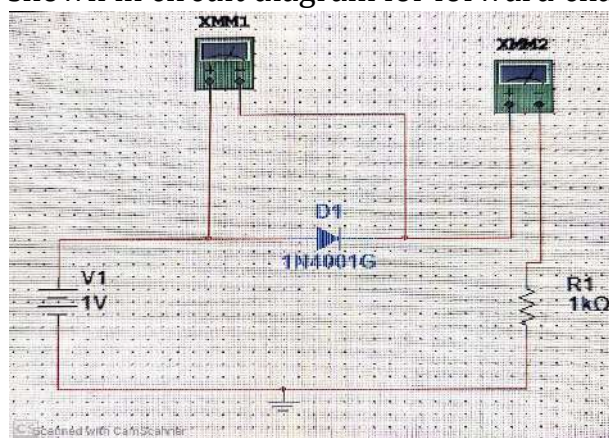
14. Then save it.

15. Go to analyses and simulation → DC sweep → output and select the value I (D1 [ID]) and click on add option.

16. Then run the simulation and observe the reverse characteristics.

**For Manual Plot of graph:**

1. Connect the circuit as shown in circuit diagram for forward characteristics.



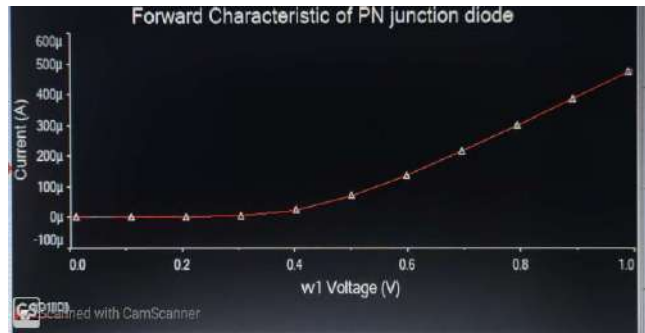
2. Use two multi meter to measure  $I_d$  and  $V_d$ .

**OBSERVATION-**

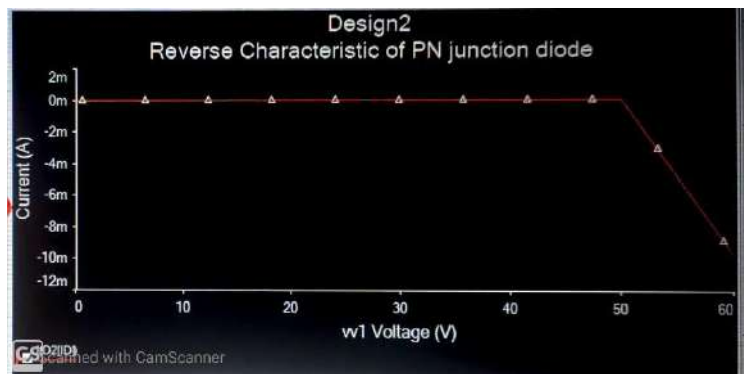
SL.NO.	Applied Voltage	Voltage drop across the ( $V_d$ in V)	Current flowing through the diode ( $I_d$ in mA)


**CHARACTERISTICS:**

Forward Characteristics:



Reverse Characteristic:



**CONCLUSION:**

## EXPERIMENT NO.-1(b)

### AIM OF THE EXPERIMENT:

Determine the forward & reverse characteristics of Zener diode.

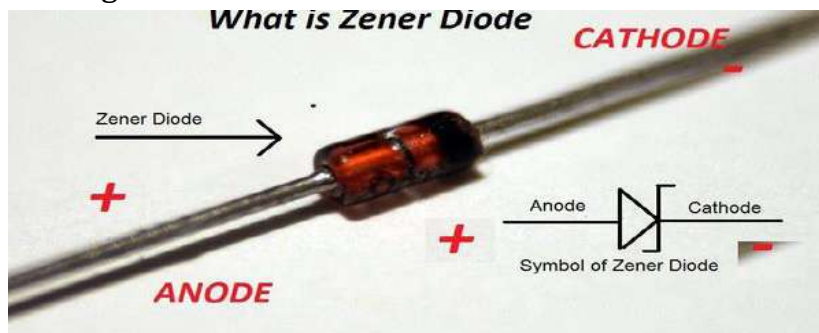
### EQUIPMENT REQUIRED:

Hardware- Personal Computer

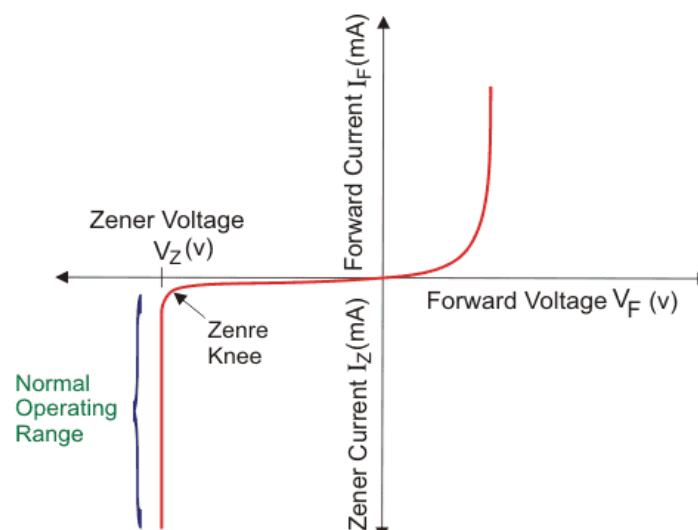
Software- Multisim Software

### THEORY: -

A Zener diode is one of the specially designed diodes that predominately work in reverse biased conditions. They are more heavily doped than ordinary diodes, due to which they have narrow depletion region. While regular diodes get damaged when the voltage across them exceeds the reverse breakdown voltage, Zener diodes work exclusively in this region.



The depletion region in Zener diode goes back to its normal state when the reverse voltage gets removed. This particular property of Zener diodes makes it useful as a voltage regulator. When we apply a reverse voltage to a Zener diode, a negligible amount of current flows through the circuit. When a voltage higher than Zener breakdown voltage is applied, Zener breakdown occurs.



Zener breakdown is a phenomenon where a significant amount of current flows through the diode with a negligible drop in voltage. When we increase the reverse

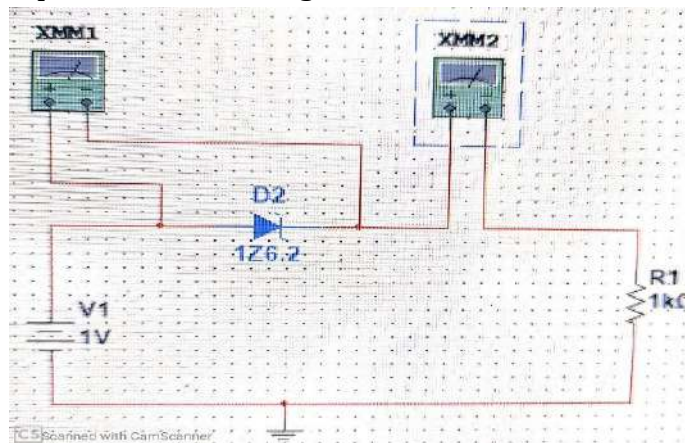
voltage further, the voltage across the diode remains at the same value of Zener breakdown voltage whereas the current through it keeps on rising as seen in the graph above. Here in the graph  $V_z$  refers to the Zener breakdown voltage. Zener breakdown voltage typically can range from 1.2 V to 200 V depending on its application.

**PROCEDURE:**

1. Start MULTISIM. A blank circuit window will appear on the screen along with a component tool bar.
2. Using component tool bar, place all the components on the circuit window and wire the circuit.
3. Components required:
  - a) Zener diode- 1Z6.2
  - b) Resistor – 1K $\Omega$
  - c) DC power supply- 12V
  - d) Ground
  - e) Two multimeter
4. Multimeter XMM1 connected across diode. So it will display voltage.
5. Multimeter XMM2 connected in series with diode, so it will measure current.

**For Forward Characteristic:**

6. Connect the circuit as per the circuit diagram.

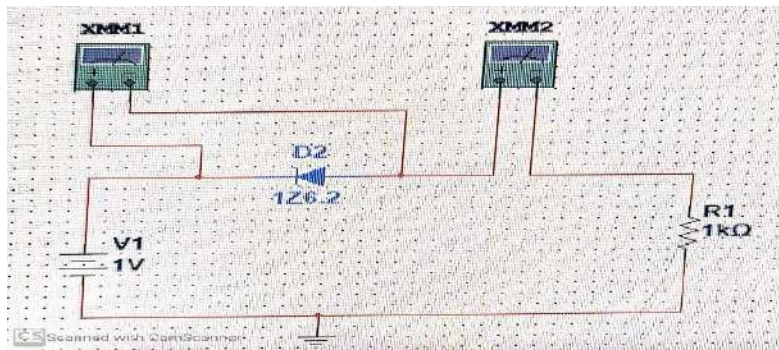


7. Apply different values of input voltage, measure the Voltage and current value showing in the multimeters and note down the reading in tabular form.
8. Plot the graph.

**For reverse Characteristic:**

9. Connect the circuit as per the circuit diagram.





10. Apply different values of input voltage, measure the Voltage and current value showing in the multi meters and note down the reading in tabular form.

11. Plot the graph.

**OBSERVATION:**

SL.NO.	Applied Voltage	Voltage drop across Zener diode	Current flowing through zener diode

**GRAPH:**

**CONCLUSION:**

## EXPERIMENT NO.: -2

### AIM OF THE EXPERIMENT:

Construct Bridge Rectifier using different filter circuit and to determine ripple factor & analyze wave form with filter & without filter.

### EQUIPMENT REQUIRED:

- 1) Analog board of AB09.
- 2) AC power supplies of 0-9Vrms from external source or Sciencetech 2612 Analog Lab.
- 3) Oscilloscope.
- 4) 2 mm. patch cords.

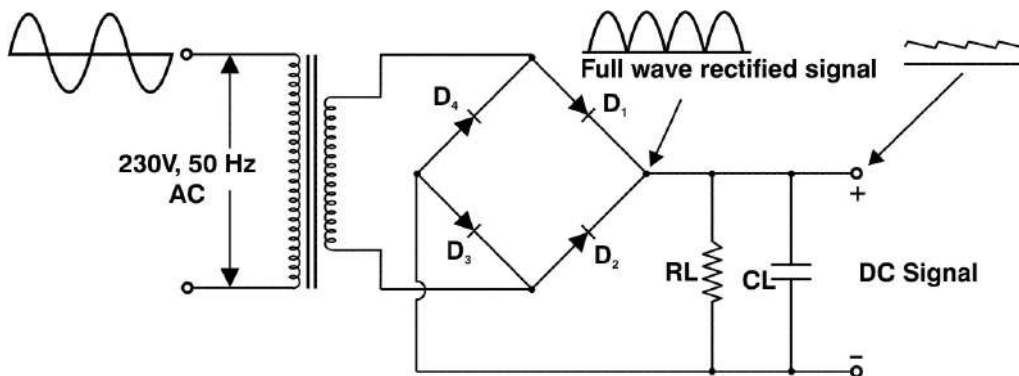
### THEORY:

In a full wave rectifier current flows through the load in the same direction for both half cycles of input ac voltages. Full wave bridge rectifier employs 4 diodes.

✓ Full wave rectifiers are classified into:-

- i. full wave center tap rectifier
- ii. Full wave bridge rectifier

### Full wave bridge rectifier: -



This rectifier employs 4 diodes i.e. D<sub>1</sub>, D<sub>2</sub>, D<sub>3</sub> & D<sub>4</sub>. During the +ve half cycle of the input ac voltage, end A of secondary winding becomes +ve & end B becomes -ve. This makes diode D<sub>1</sub> & D<sub>3</sub> forward biased while diode D<sub>2</sub> & D<sub>4</sub> becomes reverse biased. Thus, only diode D<sub>1</sub> & D<sub>3</sub> conducts. The conventional current flow is shown by dotted arrows.

During the -ve half cycle of input voltage, end A becomes -ve & end B becomes +ve. This makes diode D<sub>2</sub> & D<sub>4</sub> forward biased while diode D<sub>1</sub> & D<sub>3</sub> reverse biased. Thus, only diode D<sub>2</sub> & D<sub>4</sub> conducts. The conventional current flow is shown by solid arrows.

### Advantages:-

- ✓ PIV is one half that of center tap circuit.
- ✓ Output is twice that of center tap circuit.

✓ Need for center tapped transformer is eliminated.

DISADVANTAGES:-

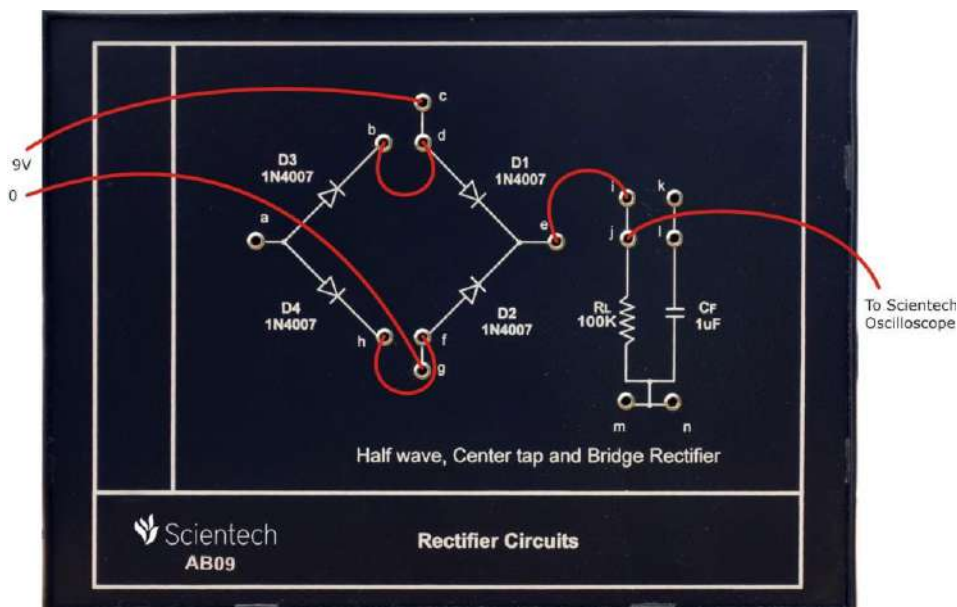
✓ Requires 4 diodes which increase the cost.

**PROCEDURE: -**

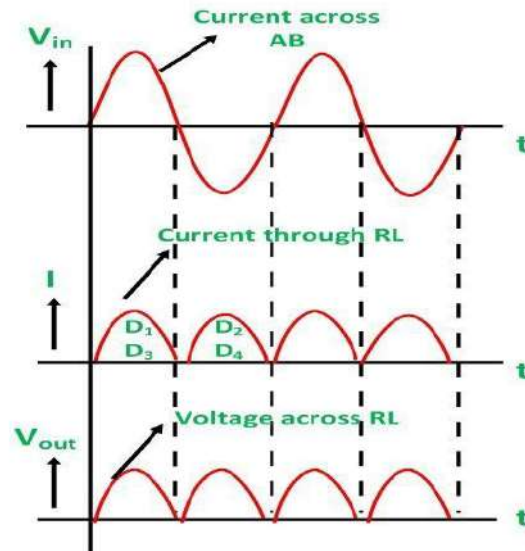
- 1) Connect 0-9Vrms from **Sciencetech 2612 Analog Lab** or from **Analog Digital Lab Power Supply** or from any external source (transformer o/p) to sockets g and c of **AB09** Board respectively using 2mm patch cords.
- 2) Connect sockets b with d and h with f using 2mm patch cords to complete the bridge rectifier circuit and also connect a patch cord between socket a and m/n.
- 3) Connect 2mm patch cord between sockets e and j. This will connect load resistance  $R_L$  across the output of Bridge rectifier (keep AC/DC push button switch of Oscilloscope in AC position)
- 4) Output DC voltage can be measured by pressing the AC/DC push button switch of the Oscilloscope or connecting a digital multimeter across sockets j and n i.e. across the load resistor and carry out following calculations.
- 5) Now connect the 2mm patch cord between i and l socket. This will connect the filter capacitor across the o/p of full wave bridge rectifier.
- 6) Observe the filtered output on Oscilloscope.

### CIRCUIT DIAGRAM:

Circuit used to study the full wave bridge Rectifier is shown in the below figure.



## INPUT & OUTPUT WAVEFORMS OF VOLTAGE & CURRENT: -



## IMPORTANT PARAMETERS OF FULL WAVE BRIDGE RECTIFIER: -

### EFFICIENCY-

Let  $v = V_m \sin \theta$  be the ac voltage to be rectified.

$r_f$  = diode resistance

$R_L$  = load resistance

$$i = \frac{V_m \sin \theta}{r_f + R_L}$$

### **D.C current:-**

$$\begin{aligned}
 I_{dc} = I_{avg} &= 2 * \frac{1}{2\pi} \int_0^{\pi} i * d\theta \\
 &= \frac{1}{\pi} \int_0^{\pi} \frac{V_m \sin \theta}{r_f + R_L} * d\theta \\
 &= \frac{V_m}{\pi(r_f + R_L)} \int_0^{\pi} \sin \theta * d\theta \\
 I_{dc} &= \frac{V_m}{\pi(r_f + R_L)} [-\cos \theta]_0^{\pi} \\
 &= \frac{V_m}{\pi(r_f + R_L)} [-\cos \pi + \cos 0] \\
 &= \frac{V_m}{\pi(r_f + R_L)} [-(-1) + 1] \\
 &= \frac{V_m}{\pi(r_f + R_L)} * 2
 \end{aligned}$$

$$= \frac{2I_m}{\pi}$$

**AC current:-**

$$\begin{aligned} I_{ac}^2 &= \frac{1}{\pi} \int_0^\pi (I_m \sin\theta)^2 * d\theta \\ &= \frac{1}{\pi} \int_0^\pi I_m^2 * \sin^2\theta * d\theta \\ &= \frac{I_m^2}{\pi} \int_0^\pi \sin^2\theta * d\theta \\ &= \frac{I_m^2}{2\pi} \int_0^\pi (1 - \cos 2\theta) * d\theta \\ &= \frac{I_m^2}{2\pi} [\int_0^\pi 1 * d\theta - \int_0^\pi (\cos 2\theta) * d\theta] \\ &= \frac{I_m^2}{2\pi} [[\theta]_0^\pi - [\frac{\sin 2\theta}{2}]_0^\pi] \\ &= \frac{I_m^2}{2\pi} [\pi - 0 - \{\frac{\sin 2\pi}{2} - \frac{\sin 2*0}{2}\}] \\ &= \frac{I_m^2}{2\pi} [\pi - \{\pi - \frac{1}{2} [0 - 0]\}] \\ &= \frac{I_m^2}{2\pi} [\pi - 0] \end{aligned}$$

$$I_{rms}^2 = \frac{I_m^2}{2\pi} * \pi$$

$$I_{rms} = \sqrt{\frac{I_m^2}{2}}$$

$$I_{rms} = \frac{I_m}{\sqrt{2}}$$

$$P_{ac} = I_{rms}^2 (r_f + R_L)$$

$$= \left(\frac{I_m}{\sqrt{2}}\right)^2 (r_f + R_L)$$

$$= \left(\frac{I_m^2}{2}\right) (r_f + R_L)$$

$$\text{Rectifier efficiency } (\eta) = \frac{P_{dc}}{P_{ac}}$$

$$= \frac{\left(\frac{2I_m}{\pi}\right)^2 * R_L}{\left(\frac{I_m^2}{2}\right) * 2 * (r_f + R_L)}$$

$$= \frac{4I_m^2 R_L}{\pi^2} * \frac{2}{I_m^2 (r_f + R_L)}$$

$$= \frac{8R_L}{\pi^2 * (r_f + R_L)}$$

$$= \frac{8R_L}{\pi^2 * R_L \left( \frac{r_f}{R_L} + 1 \right)}$$

$$= \frac{8}{\pi^2 * \left( \frac{r_f}{R_L} + 1 \right)}$$

As  $\frac{r_f}{R_L}$  is a very small value, so it can be neglected from the denominator.

Thus, we get,  $\eta = \frac{8}{\pi^2}$

$\eta$  (in % age) =  $\frac{8}{\pi^2} * 100$

= 81.2%

As efficiency is double than that of half wave rectifier, so full wave rectifier is twice more effective.

RIPPLE FACTOR:-

$$\text{Ripple factor} = \sqrt{\left( \frac{I_{rms}}{I_{dc}} \right)^2 - 1}$$

$$= \sqrt{\left( \frac{I_m/\sqrt{2}}{2I_m/\pi} \right)^2 - 1} = 0.48$$

FORM FACTOR:-

✓ It is the ratio of the rms value to the average value.

$$\text{Form factor} = \frac{\text{RMS value}}{\text{Average value}} = \frac{I_m/\sqrt{2}}{2I_m/\pi} = 1.11$$

PEAK INVERSE VOLTAGE:-

✓ It is the maximum reverse voltage that a diode can withstand without destroying the junction.

$$\text{PIV} = 2V_m$$

**CONCLUSION: -**

## EXPERIMENT NO.-3(a)

### **AIM OF THE EXPERIMENT: -**

To study the input and output characteristics of common emitter connection.

### **EQUIPMENTS REQUIRED: -**

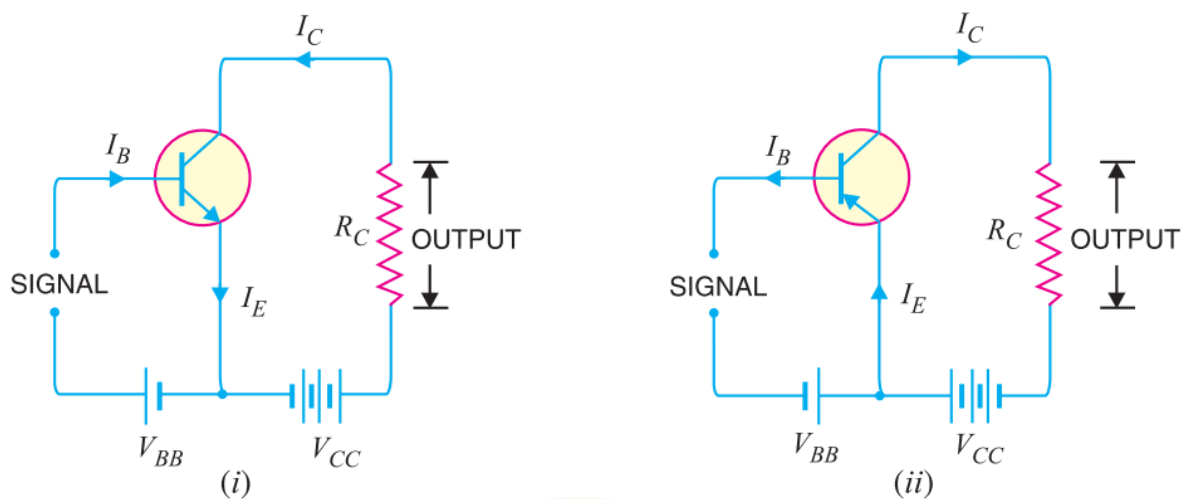
- a) Transistor characteristics trainer kit
- b) Patch cords

### **THEORY: -**

A transistor can be connected in a circuit in the following three ways:

- a) common base connection
- b) common emitter connection
- c) common collector connection

### **COMMON EMITTER CONNECTION:-**



- ✓ In the above circuit arrangement, input is applied between base & emitter & output is obtained from collector & emitter.
- ✓ Here, emitter is common to both input & output circuits, hence named as CE connection.

### CURRENT AMPLIFICATION FACTOR ( $\beta$ )

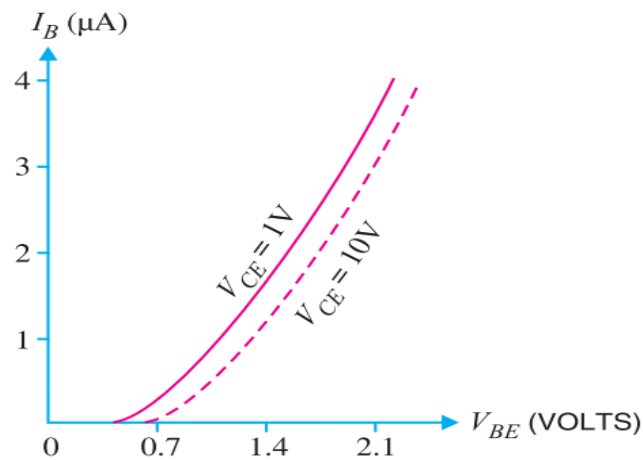
- ✓ It is the ratio of the change in  $I_C$  to change in  $I_B$ .

$$\beta = \Delta I_C / \Delta I_B$$

### INPUT CHARACTERISTICS

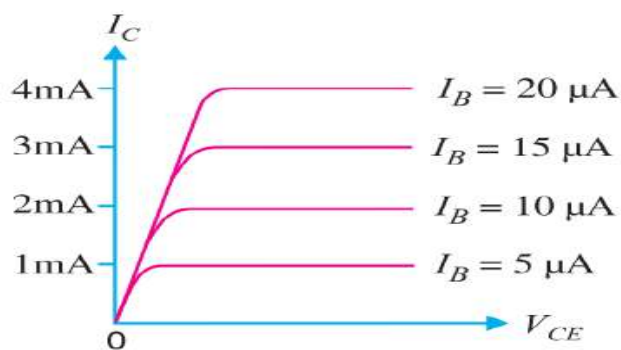
- ✓ It is the curve between  $I_B$  and  $V_{BE}$ .
- ✓ Keeping  $V_{CE}$  constant, when  $V_{BE}$  is increased,  $I_B$  increases less rapidly. This means that it has high input resistance than that of CB circuit.

- ✓ Input Resistance ( $r_i$ ) is the ratio of the change in  $V_{BE}$  to change in  $I_B$  at constant  $V_{CE}$ .  
Input resistance ( $r_i$ ) =  $\Delta V_{BE} / \Delta I_B$  at constant  $V_{CE}$ .



### OUTPUT CHARACTERISTICS

- ✓ It is the curve drawn between  $I_C$  and  $V_{CE}$  at constant  $I_B$ .
- ✓ By keeping  $I_B$  constant when  $V_{CE}$  is increased,  $I_C$  also increases slowly up to knee voltage.
- ✓ When  $V_{CE}$  is increased beyond knee voltage, the collector current becomes almost constant.
- ✓ Output resistance ( $r_o$ ) is the ratio of the change in  $V_{CE}$  to change in  $I_C$ .  
Output resistance ( $r_o$ ) =  $\Delta V_{CE} / \Delta I_C$  at constant  $I_B$ .



### **PROCEDURE FOR INPUT CHARACTERISTICS: -**

1. Adjust collector to emitter voltage  $V_{CE}$  (using  $VR_2$ ) at some suitable value (say at -2 v) and keep it constant.
2. Adjust base to emitter voltage  $V_{BE}$  (using  $VR_1$ ) so that base current shows value 20  $\mu A$ .
3. Note down base to emitter voltage  $V_{BE}$ .
4. Increase  $V_{BE}$  in small steps and note the corresponding base current  $I_B$ .
5. Repeat step number 1, 2, 3 and 4 for other values of  $V_{CE}$  (say at -4v, -6v, -8v).



6. Plot a graph by taking base voltage  $V_{BE}$  along X axis and base current along Y axis as shown in the figure 1.
7. Draw tangent  $V_{BE}$ - $I_B$  curve and determine its slope.

**PROCEDURE FOR OUPUT CHARATERISTICS: -**

1. Set collector voltage  $V_{CE} = 0.5$  v.
2. Adjust the base current  $I_B$  to  $50 \mu A$  using  $VR_1$ .
3. Note down the corresponding collector current  $I_C$ .
4. Gradually increase the collector voltage in small steps (i.e., say -2v, -2.5v, -3.0v ...-8v).
5. Note the corresponding collector current  $I_C$  keeping the base current  $I_C$  keeping the base current  $I_B$  constant.
6. Repeat step number 6 and 7 for other values of base current  $I_B$  (say  $75 \mu A$ ,  $100 \mu A$  etc).
7. Plot a graph by taking collector voltage  $V_{CE}$  along X axis and collector current  $I_C$  along Y axis.
8. Draw a tangent  $V_{CE}$ -  $I_C$  curve and determine its slope.

**OBSERVATIONS FOR INPUT CHACTERISTICS: -**

SERIAL NUMBER	Collector base voltage( $V_{CE}$ in volts)	Base current in ( $I_B$ ) $\mu A$	Collector emitter voltage( $V_{BE}$ in volts)
1	-2V		
2			
3			
4			
5			

SERIAL NUMBER	Collector base voltage( $V_{CE}$ in volts)	Base current in ( $I_B$ ) $\mu A$	Collector emitter voltage( $V_{BE}$ in volts)
1	-4V		
2			
3			
4			
5			

SERIAL NUMBER	Collector base voltage( $V_{CE}$ in volts)	Base current in ( $I_B$ ) $\mu A$	Collector emitter voltage( $V_{BE}$ in volts)

1	-6V		
2			
3			
4			
5			

**OBSERVATIONS FOR OUPUT CHARATERISTICS: -**

Serial number	Base current ( $I_B$ ) in $\mu A$	Collector current in ( $I_C$ ) $\mu A$	Collector emitter voltage( $V_{CE}$ in volts)
1			
2			
3			
4			
5			

Serial number	Base current ( $I_B$ ) in $\mu A$	Collector current in ( $I_C$ ) $\mu A$	Collector emitter voltage( $V_{CE}$ in volts)
1			
2			
3			
4			
5			

Serial number	Base current ( $I_B$ ) in $\mu A$	Collector current in ( $I_C$ ) $\mu A$	Collector emitter voltage( $V_{CE}$ in volts)
1			
2			
3			
4			
5			

**GRAPH:**

**CONCLUSION: -**

## EXPERIMENT NO.-3(b)

### **AIM OF THE EXPERIMENT: -**

To study the input and output characteristics of common base connection.

### **EQUIPMENTS REQUIRED: -**

- c) Transistor characteristics trainer kit
- d) Patch cords

### **THEORY: -**

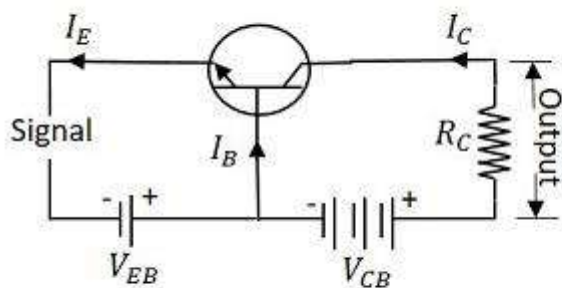
A transistor can be connected in a circuit in the following three ways:

- d) common base connection
- e) common emitter connection
- f) common collector connection

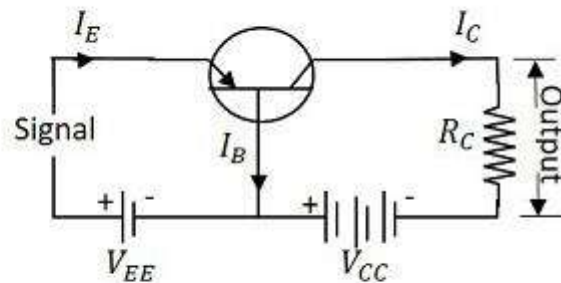
### **COMMON BASE CONNECTION**

- ✓ In this circuit arrangement, input is applied between emitter and base & output is obtained from collector base.
- ✓ In CB connection, base of the transistor is common to input & output circuit.

#### **Common Base Connection**



Using NPN transistor



Using PNP transistor

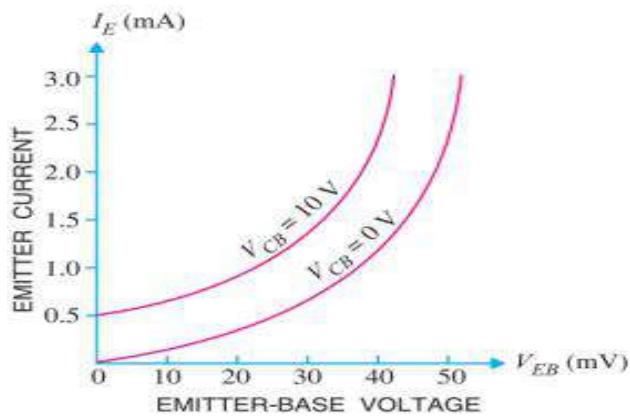
### CURRENT AMPLIFICATION FACTOR ( $\alpha$ )

- ✓ It is the ratio of the change in collector current to change in emitter current at constant collector base voltage.

$$\alpha = \frac{\Delta I_C}{\Delta I_E}$$

- ✓  $\alpha$  is less than unity.
- ✓ It can be increased by decreasing the base current.
- ✓ Base current can be decreased by doping the base lightly & making it thin.

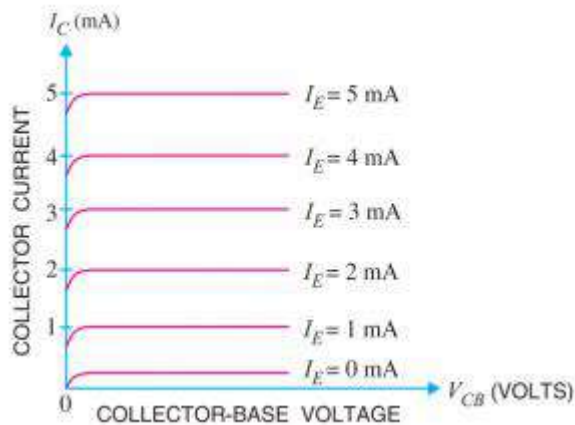
### INPUT CHARACTERISTICS



- ✓ It is the curve between emitter current ( $I_E$ ) & base emitter voltage ( $V_{BE}$ ) at constant  $V_{CB}$ .
- ✓ By keeping  $V_{CB}$  constant at a particular value, when there is small increase in  $V_{BE}$ , the  $I_E$  increases rapidly. This means that the input resistance is very small.
- ✓ Input resistance is the ratio of change in  $V_{BE}$  to the change in  $I_E$  at constant  $V_{CB}$ .

$$\text{Input resistance } (r_i) = \frac{\Delta V_{BE}}{\Delta I_E} \text{ at constant } V_{CB}.$$

### OUTPUT CHARACTERISTICS



- ✓ It is the curve between  $I_C$  &  $V_{CB}$  At constant  $I_E$ .
- ✓ By keeping  $I_E$  constant at a particular value, when there is an increase between  $V_{CB}$ , there is a small size in collector, current.
- ✓ But when voltage  $V_{CB}$  is increased above 1-2 volts, the collector current becomes constant.
- ✓ This means that the output resistance is very high.
- ✓ Output resistance is the ratio of change in  $V_{CB}$  Is change in  $I_C$  at constant  $I_E$ .

$$\text{Output resistance } (r_o) = \frac{\Delta V_{CB}}{\Delta I_C} \text{ at constant } I_E$$

### **PROCEDURE OF INPUT CHARACTERISTICS: -**

1. Adjust collector to base voltage  $V_{CB}$  (using VR2) at some suitable value (say at -2v) and keep it constant.
2. By adjusting input supply set the emitter current to a small but measurable value say 5mA, note down the corresponding emitter to base voltage  $V_{EB}$ . Increase  $V_{EB}$  in small steps and note down the corresponding emitter current  $I_E$ .
3. Repeat the step no. 2 & 3 for other values of collector voltages (say -6v, -8v etc).
4. Plot the graph by taking emitter base voltage  $V_{EB}$  along X axis and emitter current  $I_E$  along Y-axis.
5. Draw a tangent to  $V_{EB} - I_E$  curve.

**PROCEDURE OF OUTPUT CHARACTERISTICS:**

1. Adjust the emitter current  $I_E$  to a suitable value (say 10mA).
2. Set collector voltage  $V_{CB}$  to 0.5v and note the corresponding collector current  $I_C$ .
3. Gradually increase the collector voltage in small steps (i.e. make it -2v, -2.5v, -3.0v...-10v etc).
4. Note down corresponding values of collector current  $I_C$  keeping the emitter current  $I_E$  constant.
5. Repeat steps 1 & 2 for other value of emitter current  $I_E$  (say 15mA, 20mA etc).
6. Plot graphs by taking collector voltage  $V_{CB}$  along X-axis & collector current  $I_C$  along Y-axis.
7. Draw a tangent on a  $V_{CB} - I_C$  curve.

**OBSERVATIONS INPUT CHARACTERISTICS: -**

SERIAL NUMBER	Collector base voltage( $V_{CB}$ in volts)	Emitter current in ( $I_E$ ) $\mu$ A	Base emitter voltage( $V_{BE}$ in volts)
1	-2V		
2			
3			
4			
5			

SERIAL NUMBER	Collector base voltage( $V_{CB}$ in volts)	Emitter current in ( $I_E$ ) $\mu$ A	Base emitter voltage( $V_{BE}$ in volts)
1	-4V		
2			
3			
4			
5			

SERIAL NUMBER	Collector base voltage( $V_{CB}$ in volts)	Base current in ( $I_E$ ) $\mu A$	Base emitter voltage( $V_{BE}$ in volts)
1	-6V		
2			
3			
4			
5			

**OBSERVATIONS OUTPUT CHARACTERISTICS: -**

Serial number	Emitter current ( $I_E$ ) in $\mu A$	Collector current in ( $I_C$ ) $\mu A$	Collector base voltage( $V_{CB}$ in volts)
1			
2			
3			
4			
5			

Serial number	Emitter current ( $I_E$ ) in $\mu A$	Collector current in ( $I_C$ ) $\mu A$	Collector base voltage( $V_{CB}$ in volts)
1			
2			
3			
4			
5			

Serial number	Emitter current ( $I_E$ ) in $\mu A$	Collector current in ( $I_C$ ) $\mu A$	Collector base voltage( $V_{CB}$ in volts)
1			
2			
3			
4			
5			

**GRAPH:**

**CONCLUSION: -**

## EXPERIMENT NO:- 4

### AIM OF THE EXPERIMENT-

Construct & test the transistor regulator using Zener diode.

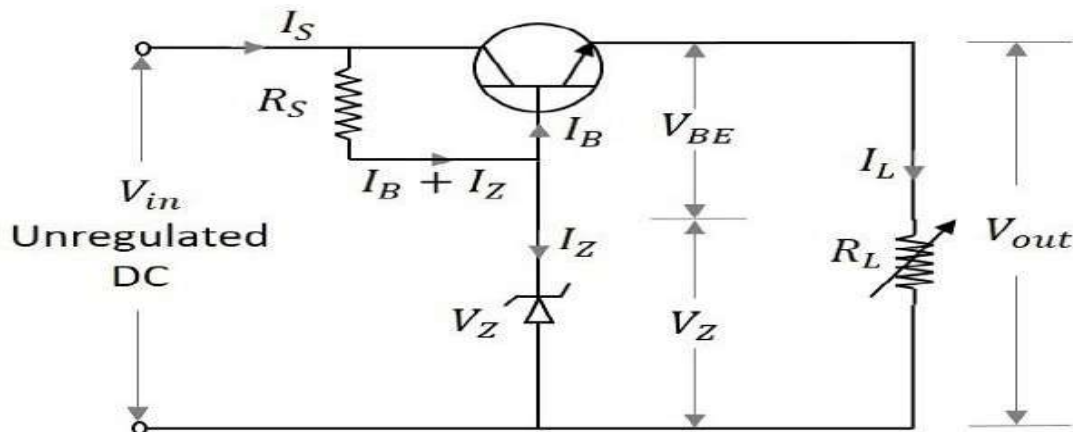
### EQUIPMENT REQUIRED:

Hardware- Computer

Software- Multisim software

### THEORY:

This regulator has a transistor in series to the Zener regulator and both in parallel to the load. The transistor works as a variable resistor regulating its collector emitter voltage in order to maintain the output voltage constant. The figure below shows the transistor series voltage regulator.



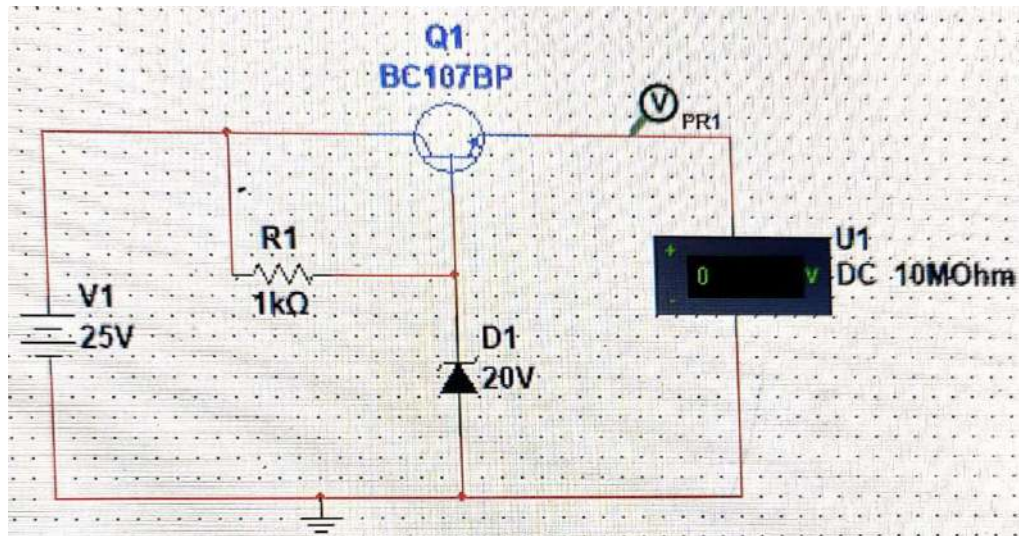
With the input operating conditions, the current through the base of the transistor changes. This effects the voltage across the base emitter junction of the transistor  $V_{BE}$ . The output voltage is maintained by the Zener voltage  $V_Z$  which is constant. As both of them are maintained equal, any change in the input supply is indicated by the change in emitter base voltage  $V_{BE}$ .

Hence the output voltage  $V_o$  can be understood as

$$V_o = V_Z + V_{BE}$$

### PROCEDURE:

1. Start MULTISIM. A blank circuit window will appear on the screen along with a component tool bar.
2. Using component tool bar, place all the components on the circuit window and wire the circuit.
3. Connect the circuit as shown in circuit diagram.



4. Components Required-

- A) Transistor-BC107BP
- B) Ground
- C) Resistor- 1K $\Omega$
- D) V1- 25V
- E) Zener diode- 20V
- F) Voltmeter -V

5. Go to analyses and simulation- $\rightarrow$ DC sweep.

6. In the DC sweep window set the following values:

Source1

Source-  $V_1$

Start Value- 0

Stop Value- 40

Increment- 0.5

7. Then save it.

8. Go to analyses and simulation- $\rightarrow$  DC sweep  $\rightarrow$ output and select the value V(PR1) and click on add option.

9. Then run the simulation and observe the graph.

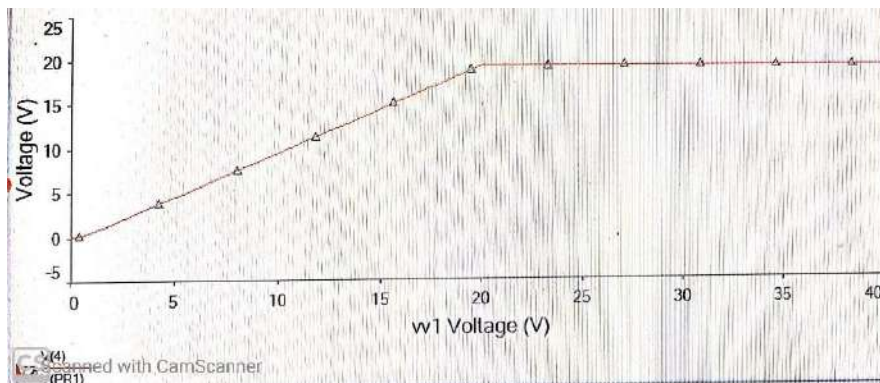
10.Note down the output voltage for different input voltage for observation.

**OBSERVATION-**

SL.NO.	$V_1$ (in V)	O/P Voltage



## GRAPH-



## CONCLUSION:

## EXPERIMENT NO.: -5

### AIM OF THE EXPERIMENT:

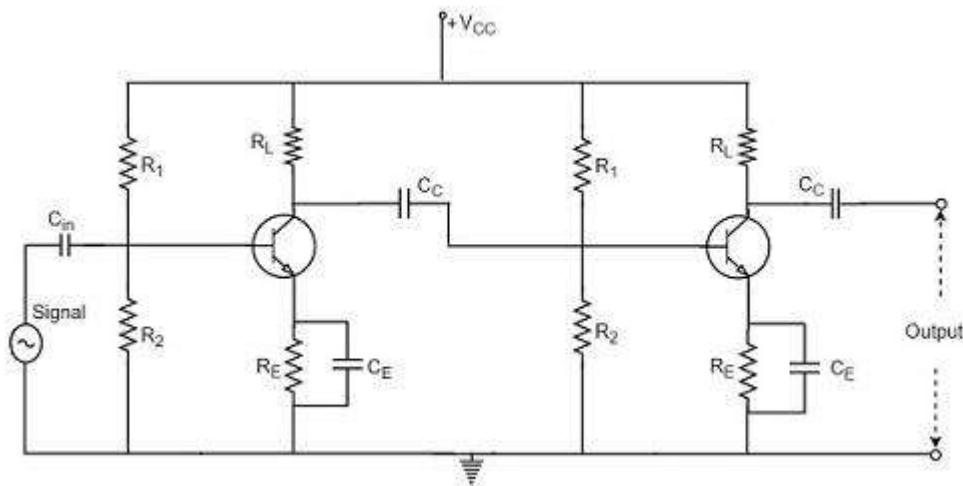
To study two stage RC coupled amplifier, find gain & draw the frequency response curve.

### EQUIPMENT REQUIRED:

- 1) RC coupled amplifier trainer kit
- 2) Patch cords
- 3) CRO
- 4) Function generator

### THEORY: -

- ✓ When two amplifiers are connected in such a way that the output signal of the first serves as the input signal to the second, the amplifiers are said to be connected in cascade.
- ✓ Amplifiers are connected in cascade to extend the gains possible with single stage amplifiers.
- ✓ R.C. Coupling is the most widely used method because it is cheap and provides excellent audio facility over a wide range of frequency.
- ✓ It is usually employed for voltage amplification.
- ✓ The fig. Shows two stage R.C. Coupled amplifier.



- ✓ The signal developed across the collector resistor  $R_L$  of the first stage is coupled to base of second stage through the coupled capacitor  $C_2$ .
- ✓ This capacitor blocks the DC component of first stage from reaching the base of the second stage.
- ✓ In this way DC biasing of the next stage is not interfered, for this reason capacitor  $C_2$  is also called a blocking capacitor.
- ✓ As the coupling from one stage to next is achieved by a coupling capacitor followed by a shunt resistor, therefore the amplifiers are called resistance – capacitance coupled amplifiers.
- ✓ Resistances  $R_1, R_2, R_{E1}$  form the biasing and stabilization network.

- ✓ The emitter bypass capacitors offers low reactance path to the signal.
- ✓ Without it, the voltage gain of each stage would be lost.
- ✓ The total gain is less than the product of the gains of individual stages.
- ✓ Its reason is that when the second stage follows the first stage, the effective load resistance of the first stage is reduced due to loading effect of the next stage.
- ✓ However, the gain of the third stage which has no loading effect of subsequent stage remains unchanged. The overall gain will be the product of the gains of three stages.

### **Advantages-**

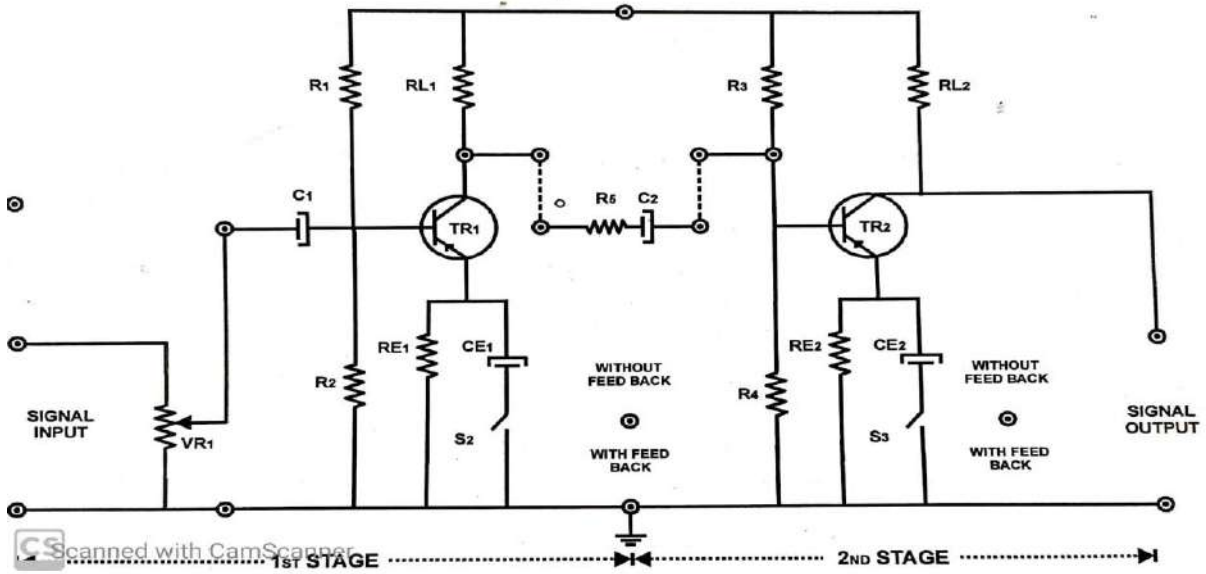
1. It requires no expensive or bulky components and no adjustments. Hence, it is small and expensive.
2. It has excellent frequency response. The gain is constant over the audio frequency range.
3. Its overall amplification is higher than that of the other couplings.
4. It has minimum possible non -linear distortion because it does not use any coils or transformer which might pick up undesirable signals. Hence, there are no magnetic fields to interfere with the signal.

### **PROCEDURE: -**

1. Connect audio frequency signal generator across input terminals. Set it at sine wave signal of 100Hz, 10mv – 20mv peak to peak amplitude.
2. Connect CRO across output of first stage amplifier at red socket above transistor (TR1).
3. Switch ON the instrument as well as CRO.
4. Connect the Red to Red & blue to blue dotted sockets through patch cord.
5. Connect signal input to input sockets and CRO probe to output sockets.
6. Keep both switches (S2 & S3) towards without feedback position and take observation as mentioned in previous cases.
7. Repeat the same procedure for switches (S2&S3) towards without feed-back position.
8. Observe the output signal on CRO. Adjust output signal with the help of potentiometer (VR<sub>1</sub>) provided on the front panel. Note down all the observation as shown in sample observation table. calculate the gain as per formula :  

$$\text{Voltage Gain } (A_{v3}) = \frac{\text{output voltage (P-P)}}{\text{Input Voltage (P-P)}}$$
9. Plot a graph between frequency vs. output gain by taking along X-axis & output gain along Y-axis.

**CIRCUIT DIAGRAM:**



**OBSERVATIONS: -**

**Sample observation table**

Serial no.	frequency	Amplitude	gain
1.			
2.			
3.			
4.			
5.			
6.			

**GRAPH:**

**CONCLUSION: -**

## **EXPERIMENT No.-6(a)**

### **AIM OF THE EXPERIMENT:**

Determine drain & Transfer characteristics of JFET.

### **EQUIPMENT REQUIRED:**

Hardware- Personal Computer

Software- Multisim Software

### **THEORY:**

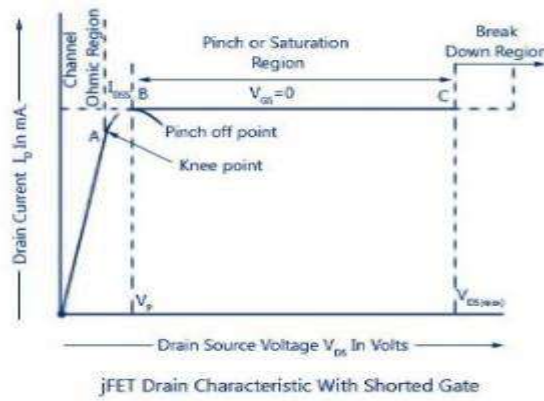
#### **Characteristics of JFETS**

There are two types of characteristics.

1. Output or drain characteristics and
2. Transfer characteristic.

#### **1) Output or Drain Characteristic.**

- The curve drawn between drain current  $I_p$  and drain-source voltage  $V_{DS}$  with gate-to source voltage  $V_{GS}$  as the parameter is called the drain or output characteristic. This characteristic is analogous to collector characteristic of a BJT:

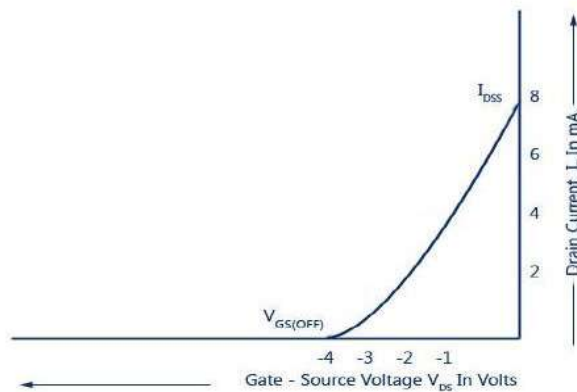


## 2) Transfer Characteristics of JFET

- The transfer characteristic for a JFET can be determined experimentally, keeping drain-source voltage,  $V_{DS}$  constant and determining drain current,  $I_D$  for various values of gate-source voltage,  $V_{GS}$ . The curve is plotted between gate-source voltage,  $V_{GS}$  and drain current,  $I_D$ , as illustrated in fig. It is similar to the Trans conductance characteristics of a vacuum tube or a transistor. It is observed that

(i) Drain current decreases with the increase in negative gate-source bias (ii) Drain current,  $I_D = I_{DSS}$  when  $V_{GS} = 0$  (iii) Drain current,  $I_D = 0$  when  $V_{GS} = V_D$

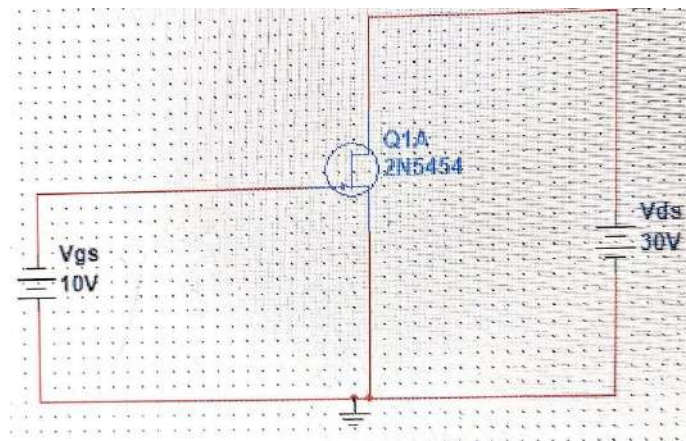
The transfer characteristic can also be derived from the drain characteristic by noting values of drain current,  $I_D$  corresponding to various values of gate-source voltage,  $V_{GS}$  for a constant drain-source voltage and plotting them.



Transfer Characteristics of JFET

### PROCEDURE -

1. Start MULTISIM. A blank circuit window will appear on the screen along with a component tool bar.
2. Using component tool bar, place all the components on the circuit window and wire the circuit.
3. Connect the circuit as shown in circuit diagram.



4. Components Required-

- i) JFET (2N5454)
- j)  $V_{gs}$  (V1)- 10V
- k)  $V_{ds}$  (V2)- 30V

**For Drain Characteristic:**

5. Go to analyses and simulation-→DC sweep.
6. In the DC sweep window set the following values:  
Source1  
 Source-  $V_{ds}$   
 Start Value- 0  
 Stop Value- 10  
 Increment- 2
7. Tick out the “use source 2” and set the following values:  
Source1  
 Source-  $V_{gs}$   
 Start Value- 0  
 Stop Value- 5  
 Increment- 1
8. Then save it.
9. Go to analyses and simulation-→ DC sweep →output and select the value  $I(JQ1\_A[ID])$  and click on add option.
10. Then run the simulation and observe the Drain characteristic..

**For Transfer Characteristic:**

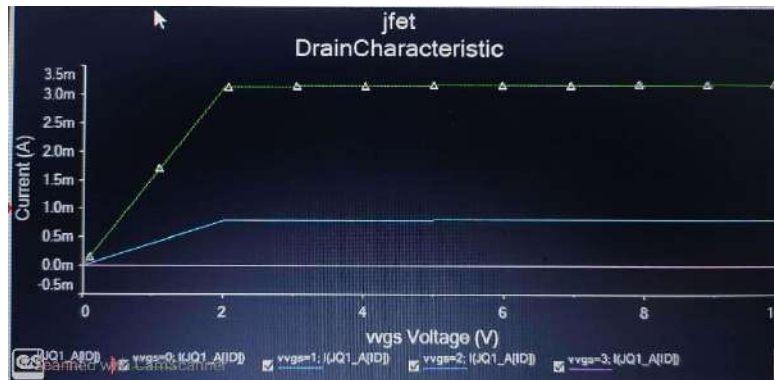
11. Go to analyses and simulation-→DC sweep.
12. In the DC sweep window set the following values:  
Source1  
 Source-  $V_{gs}$   
 Start Value- 0  
 Stop Value- 5  
 Increment- 2
13. Then save it.

14. Go to analyses and simulation → DC sweep → output and select the value I(JQ1\_A[ID]) and click on add option.

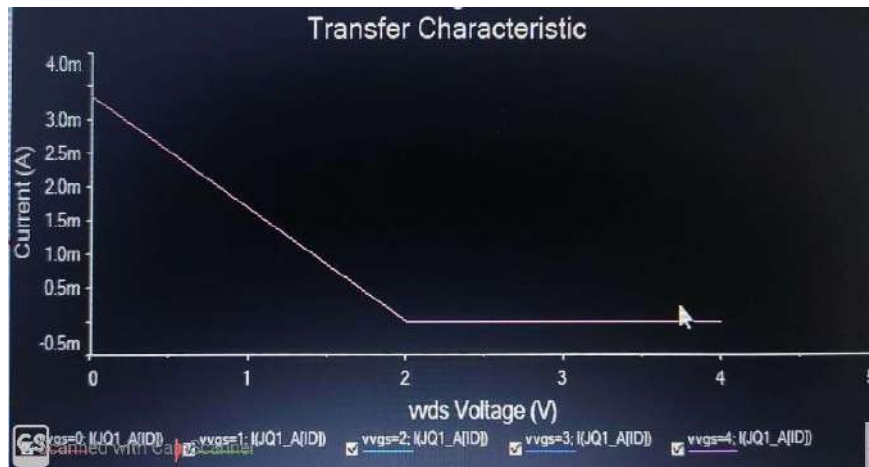
15. Then run the simulation and observe the transfer characteristic.

### CHARACTERISTICS:

Drain Characteristic-



Transfer Characteristic-



CONCLUSION:-



## **EXPERIMENT-6(b)**

### **AIM OF THE EXPERIMENT:**

Determine drain & Transfer characteristics of MOSFET.

### **EQUIPMENT REQUIRED:**

Hardware- Personal Computer

Software- Multisim Software

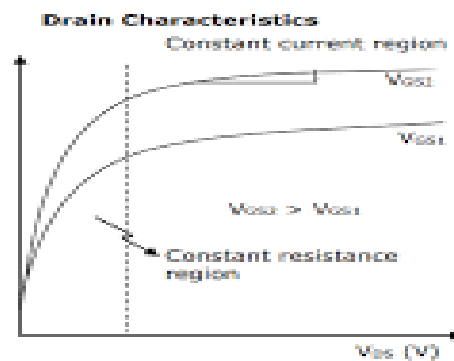
### **THEORY:**

There are two types of characteristics

1. Drain Characteristics
2. Transfer Characteristics

## Drain Characteristics

- In Drain Characteristics, the output current is plotted with respect to the Drain to source voltage  $V_{DS}$ . We make  $V_{GS}$  (Gate to source voltage constant). It helps us in understanding three regions of operation.
- On the X-axis we plot Drain to Source voltage while on Y- Axis we plot  $I_d$  (Drain current).

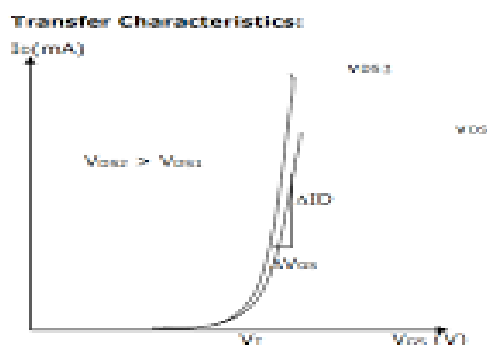


- We plot the current values for different values of  $V_{GS}$ .
- As we can see the current remains constant after some drain voltage. Hence, minimum drain to source voltage is needed for Mosfet to work.
- Hence, as we increase  $V_{GS}$  the channel width increases and it results in more drain current  $I_D$

## Transfer Characteristics

- Transfer characteristics is the graph of output current to input voltage.
- Hence, we plot current  $I_d$  (output) with respect to input voltage  $V_{GS}$ .

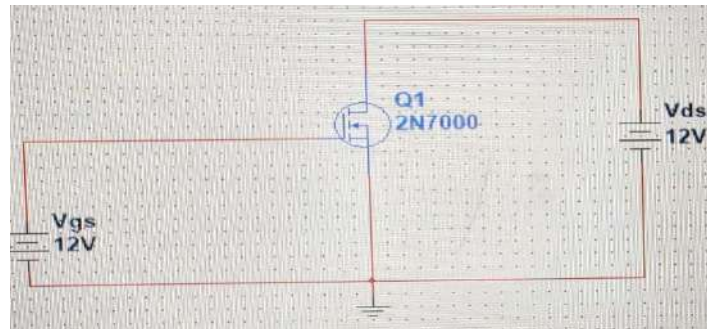
Ideal Graph:



- The above figure shows transfer characteristics. It is also known as Trans conductance curve.
- Initially, when there is no  $V_{GS}$  (gate to source voltage) very less current flows. It is in micro amps.
- When  $V_{GS}$  is positive, the drain current increases slowly.
- After that there is a rapid increase in drain current corresponding to increase in  $V_{GS}$ .

## PROCEDURE -

1. Start MULTISIM. A blank circuit window will appear on the screen along with a component tool bar.
2. Using component tool bar, place all the components on the circuit window and wire the circuit.
3. Connect the circuit as shown in circuit diagram.



4. Components Required-  
MOSFET (2N7000)  
Vgs (V1)- 12V  
Vds (V2)- 12V

### For Drain Characteristic:

5. Go to analyses and simulation-→DC sweep.
6. In the DC sweep window set the following values:  
Source1  
Source- Vds  
Start Value- 0  
Stop Value- 5  
Increment- 0.5
7. Tick out the “use source 2” and set the following values:  
Source1  
Source- Vgs  
Start Value- 0  
Stop Value- 5  
Increment- .5
8. Then save it.
9. Go to analyses and simulation-→ DC sweep →output
10. In output window click on add expression and select “-” and I (Vds). Then run the simulation and observe the Drain characteristic.

### For Transfer Characteristic:

11. Go to analyses and simulation-→DC sweep.
12. In the DC sweep window set the following values:  
Source1  
Source- Vgs

Start Value- 0  
Stop Value- 5  
Increment- 0.5

13. Then save it.

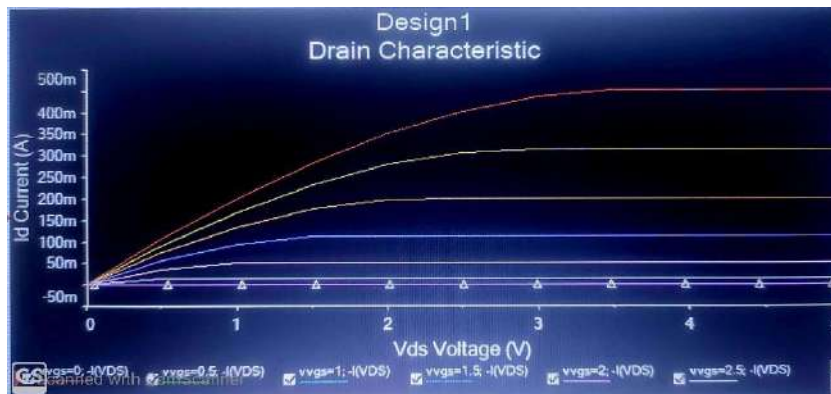
14. Go to analyses and simulation → DC sweep → output

15. In output window click on add expression and select “-” and I (Vds). Then run the simulation and observe the Drain characteristic.

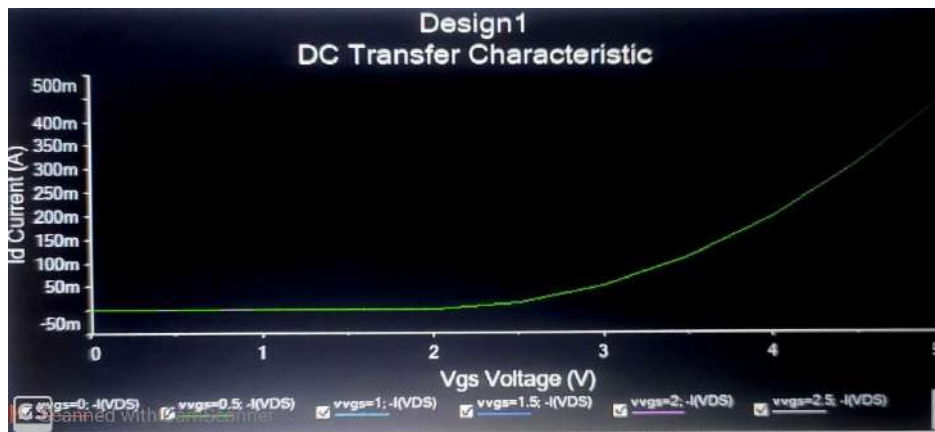
16. Then run the simulation and observe the transfer characteristic.

## CHARACTERISTICS:

Drain Characteristic-



Transfer Characteristics-



CONCLUSION:

## **EXPERIMENT-7**

### **AIM OF THE EXPERIMENT:**

Construct & test timer circuit using IC 555 timer.

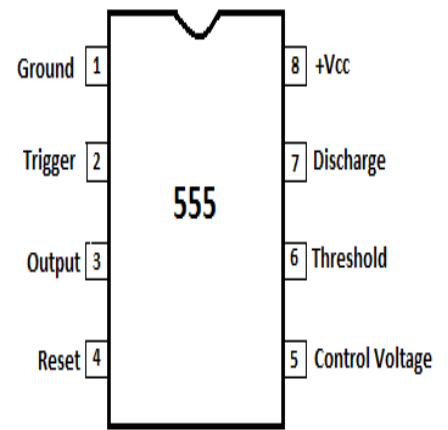
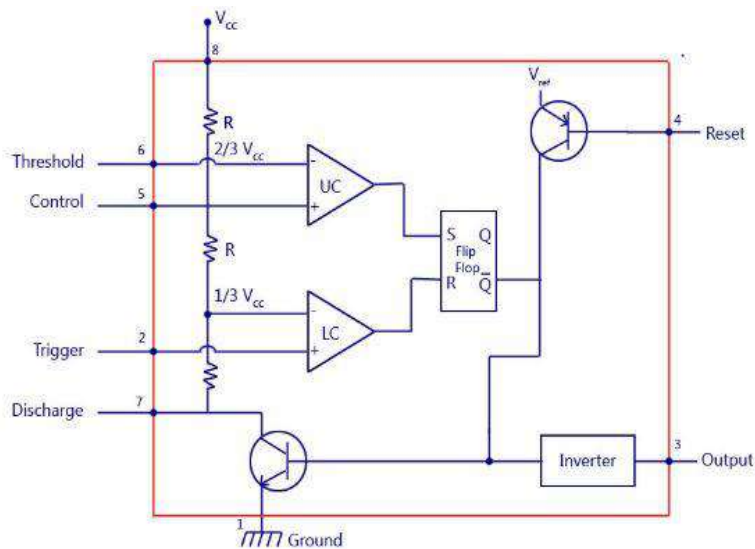
### **EQUIPMENT REQUIRED:**

Hardware- Personal Computer

Software- Multisim Software

### **THEORY:**

The standard 555 timer package includes 25 transistors, 2 diodes and 15 resistors on a silicon chip installed in an 8-pin mini dual-in-line package (DIP-8). Variants consist of combining multiple chips on one board.

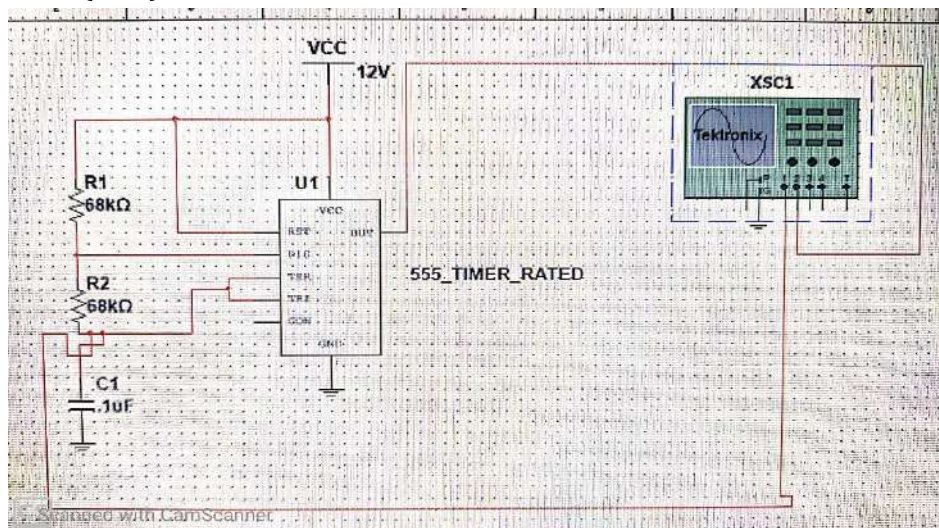


### Pin diagram and description

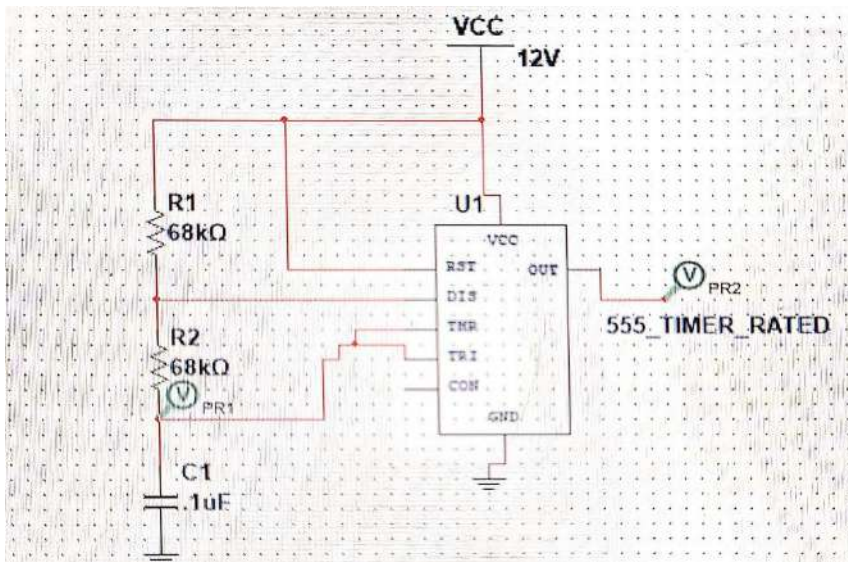
Pin	Name	Purpose
1	GND	Ground reference voltage, low level (0 V)
2	TRIG	The OUT pin goes high and a timing interval starts when this input falls below 1/2 of CTRL voltage (which is typically 1/3 Vcc, CTRL being 2/3 Vcc by default if CTRL is left open). In other words, OUT is high as long as the trigger low. Output of the timer totally depends upon the amplitude of the external trigger voltage applied to this pin.
3	OUT	This output is driven to approximately 1.7 V below +Vcc, or to GND.
4	RESET	A timing interval may be reset by driving this input to GND, but the timing does not begin again until RESET rises above approximately 0.7 volts. Overrides TRIG which overrides threshold.
5	CTRL	Provides "control" access to the internal voltage divider (by default, 2/3 Vcc).
6	THR	The timing (OUT high) interval ends when the voltage at threshold is greater than that at CTRL (2/3 Vcc if CTRL is open).
7	DIS	Open collector output which may discharge a capacitor between intervals. In phase with output.
8	Vcc	Positive supply voltage, which is usually between 3 and 15 V depending on the variation.

## PROCEDURE:

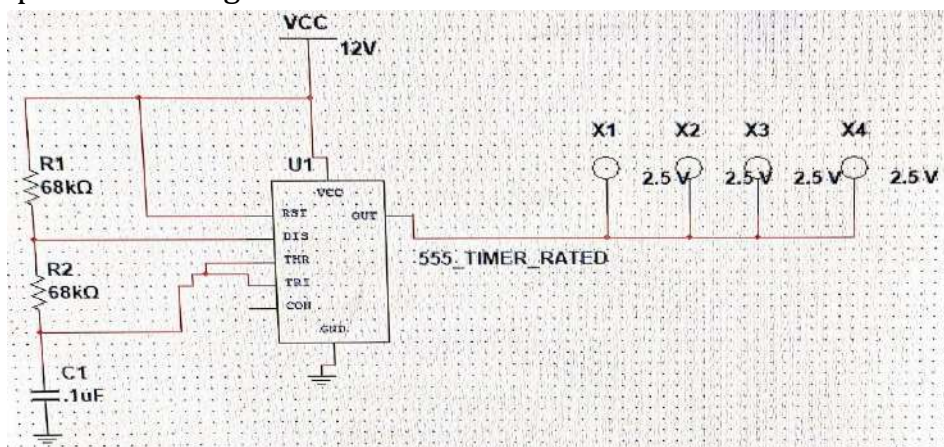
1. Start Multisim. A blank circuit window will appear on the screen on the screen along with a component tool bar.
2. Using component tool bar, place all the components on the circuit window and wire the circuit.
3. Connect the circuit as shown in the circuit diagram.
4. Components required:
  - ii. Vcc- 12V
  - iii. 555 timer rated
  - iv. Ground
  - v. Registers (R1 and R2)- 68K $\Omega$
  - vi. Capacitor- 0.1 $\mu$ F
  - vii. CRO (XSC)



5. Then select analyses and simulation-→DC sweep
6. In the DC sweep window set the following values:
  - Source-1
  - Source- Vcc
  - Start value- 0
  - Stop Value- 1
  - Increment- 0.5
7. Then Run the simulation and click on the CRO and observe the waveform.
8. Go to place option present in top most portion in that go to probe--→ voltage. Then connect the voltage as per below circuit diagram.



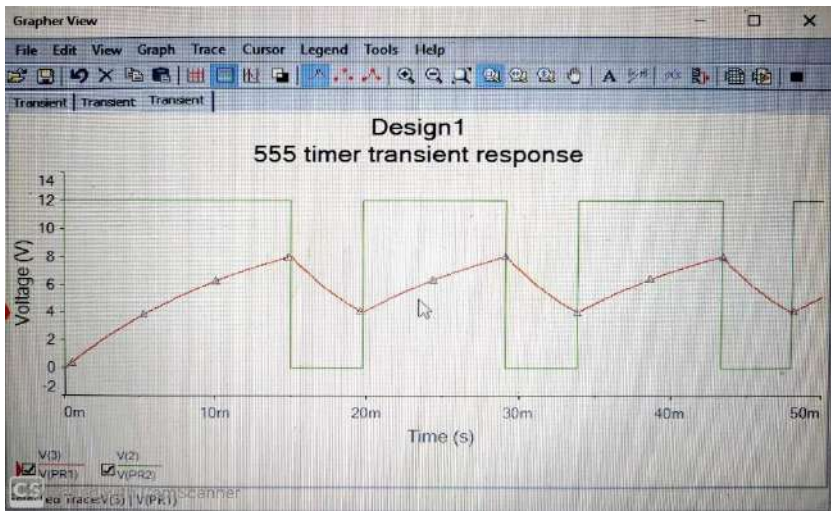
9. Go to Analyses & Simulation → transient. In transient window set the values :  
 Initial Conditions- User defined  
 Start time- 0  
 End time- 0.05
10. Tick out the maximum time step and initial time step.
11. Click on RUN option and observe waveform.
12. Then add component, Probe-Dig- Blue, Green, Orange, Red (any different color probes) as per circuit diagram.



13. At last select interactive simulation in analyses and simulation window and run the circuit to see the output.

**WAVEFORM-**





**CONCLUSION-**

## EXPERIMENT:- 8(a)

### AIM OF THE EXPERIMENT:

Construct and observe the waveform of Clipper circuits.

### COMPONENTS AND EQUIPMENT REQUIRED:

1. CRO (Dual Channel 0 to 20 MHz)
2. Signal Generator ( 1Hz to 1 MHz)
3. Diode (1N4007)
4. Resistor (2.2 K $\Omega$ )
5. D.C Power Supply (0 - 30 V (dual))
6. Connecting wires
7. Bread board

### Theory:

The circuit which the waveform is shaped by removing a portion of the applied wave is known as clipping circuit. Clipper finds extensive use in radar, digital and other electronic systems. Although several clipping circuits have been developed to change the wave shape. These clippers can remove signal voltages above or below a specific level. The important diode clippers are:

- i. Positive clipper
- ii. Negative clipper

#### Positive Clipper:

A positive clipper is that which removes the positive half cycles of the input voltage. In a positive clipper circuit, the output voltage has all the positive half cycles removed or clipped off. During the positive half cycle of the input voltage, the diode is forward biased and conducts heavily. Therefore the voltage across the diode and hence across the load  $R_L$  is zero. Hence output voltage during the half cycles is zero. During the negative half cycle of the input voltage, the diode is reverse biased and behaves as an open.

#### Negative Clipper:

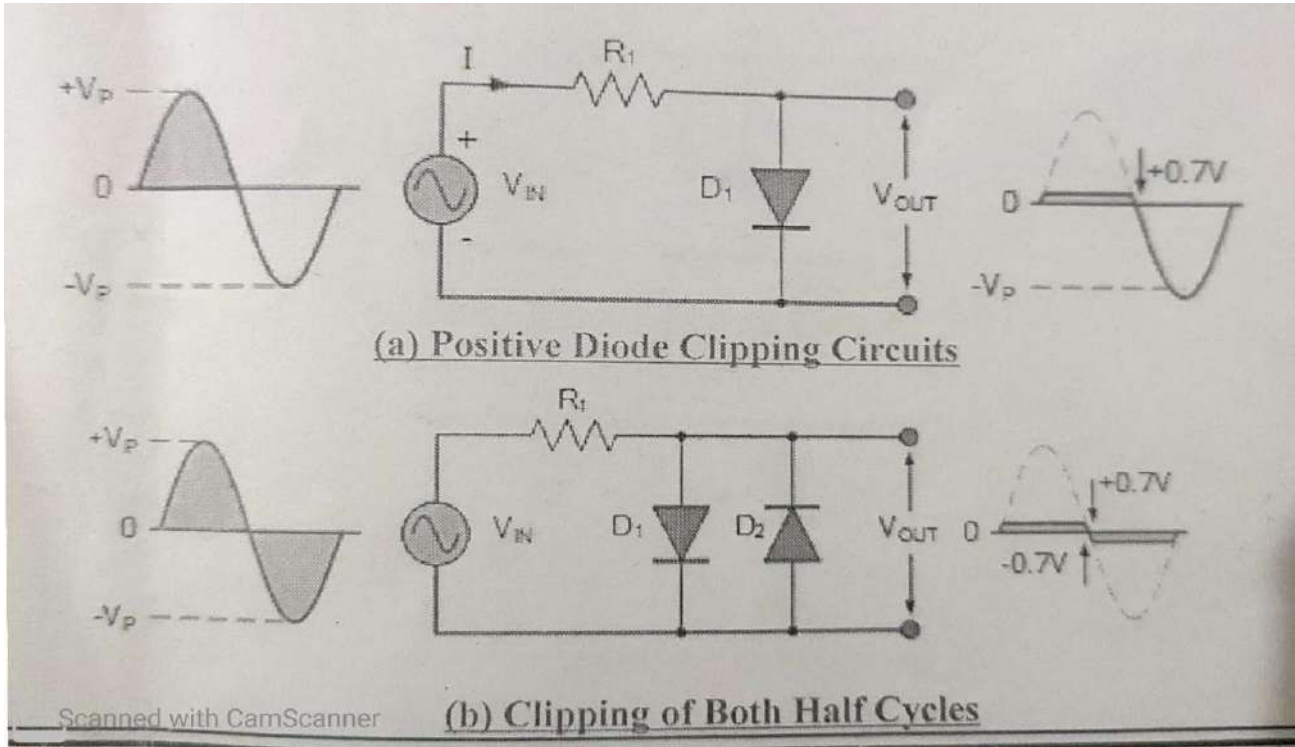
A negative clipper is that which removes the negative half cycles of the input voltage. In a negative clipper the output voltage has all the negative half cycles removed or clipped off. During the negative half cycle of the input voltage, the diode is forward biased and conducts heavily. Therefore the voltage across the diode and hence across the load  $R_L$  is zero. Hence output voltage during negative half cycles is zero. During positive half cycle of the input voltage, the diode is reverse biased and behaves as an open.

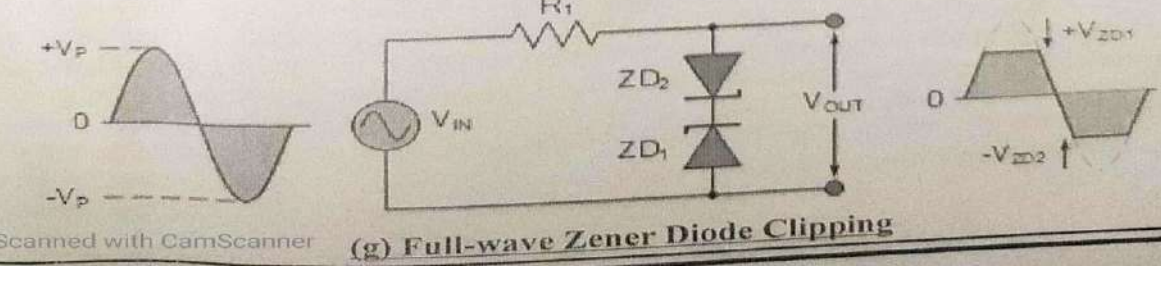
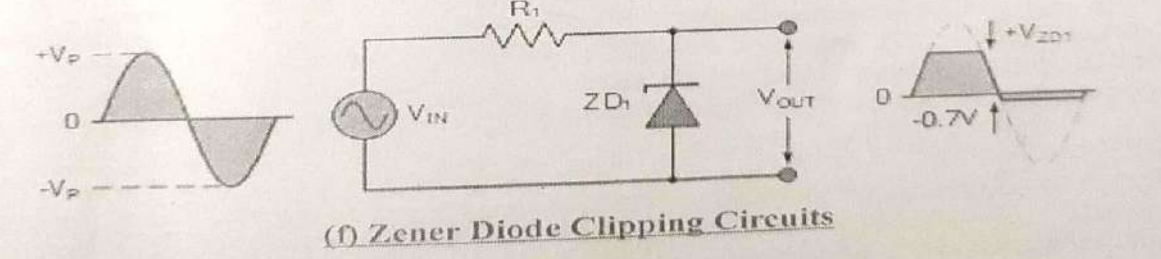
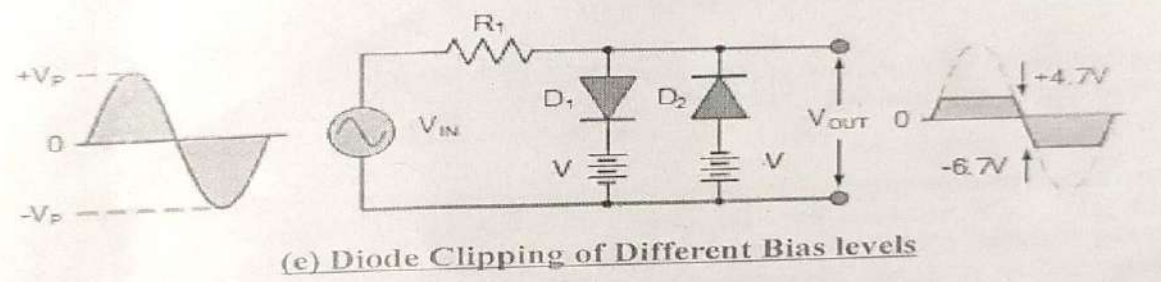
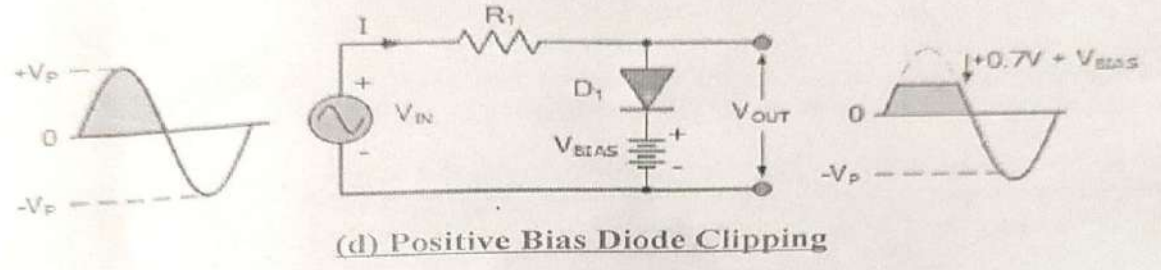
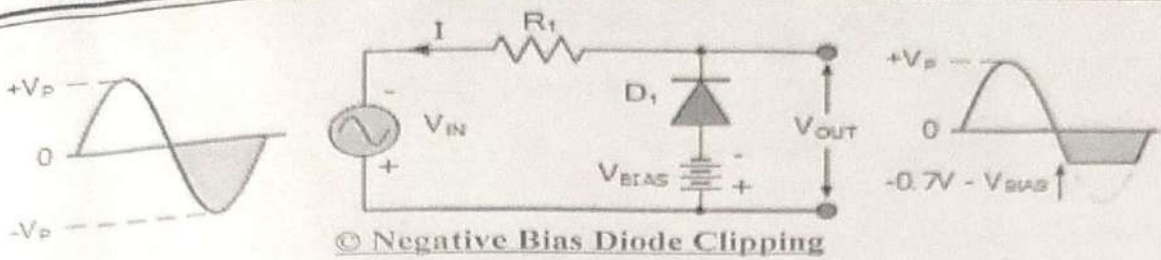
### Procedure:

1. Connect the circuit as shown in the figures given below.
2. Apply input signal of 20VP-P, 1 kHz from function generator at input terminals. Also connect CRO at output.
3. Complete the clipper circuit through patch cords.

4. Check the output wave form at CRO.
5. In positive clipper positive part will be clip and clipping can be shifted according to variation of voltage from power supply as shown in figure.
6. For negative clipper diode connection will be reverse as shown in figure. Now we will observe negative portion of input signal has been clipped and it can be shifted according variation of voltage from power supply.

**CIRCUIT DIAGRAMS:**





Scanned with CamScanner

**CONCLUSION:**

## **EXPERIMENT:- 8(B)**

### **AIM OF THE EXPERIMENT:**

Construct and observe the waveform of clamper circuits.

### **EQUIPMENTS REQUIRED:**

1. Clamper trainer Kit
2. Function generator
3. CRO
4. Patch cords

### **THEORY:**

The process where sinusoidal signals are going to be altered by transmitting through a non-linear network is called non-linear wave shaping. Non-linear elements (like diodes) in combination with resistors and capacitors can function as clamping circuit.

Clamping circuits add a DC level to an AC signal. A clamper is also referred to as DC restorer or DC re-inserter. The Clampers clamp the given waveform either above or below the reference level, which are known as positive or negative clampers respectively.

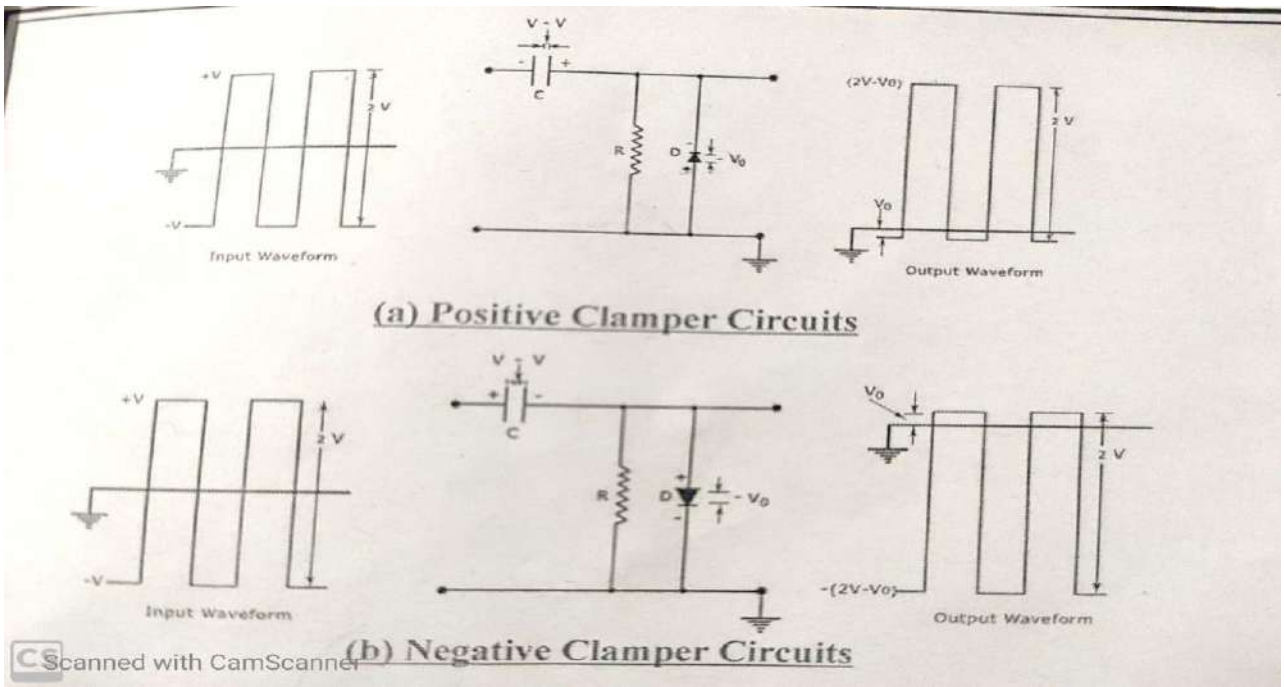
Clamping circuits are classified as two types.

- i. Negative Clampers
- ii. Positive Clampers

### **PROCEDURE:**

1. Connect the circuit as shown in the figure.
2. Apply input signal of 5V P-P, 1KHz from function generator at input terminals. Also connect CRO at output.
3. Set CRO at DC level.
4. Connect 5V DC regulated power supply across battery terminals (battery sign marked).
5. Switch ON the instrument using ON-OFF toggle switch provided on the front panel.
6. Check the output wave shape at CRO. We will observe that the DC level of the sine wave is shifted upward i.e. in positive side. For further shifting of DC level, we will have to increase the value of DC supply.
7. For negative clamping, reverse the polarity of diode, capacitor and supply connected as shown in the figure.
8. Trace the input and output wave form

### **CIRCUIT DIAGRAMS:**



**CONCLUSION-**

## EXPERIMENT NO.-9

### AIM OF THE EXPERIMENT:

Construct and test voltage power supply using 78XX and 79XX.

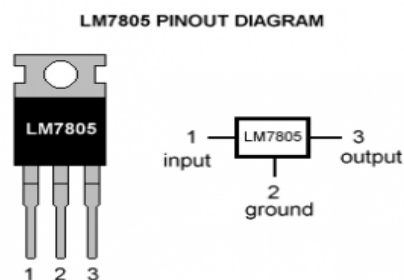
### EQUIPMENT AND COMPONENT REQUIRED:

1. Bread board
2. ICs 7805, 7809, 7912 ICs - 1No. each
3. Regulated power supply
4. DRB / potentiometer 10K $\Omega$  - 1No.
5. Capacitors 1000 $\mu$ F, 22  $\mu$ F - 1No. each
6. Voltmeter - 0-20V
7. Connecting wires

### THEORY:

#### 78XX:

Voltage sources in a circuit may have fluctuations resulting in not providing fixed voltage outputs. A voltage regulator IC maintains the output voltage at a constant value. 7805 IC, a member of 78xx series of fixed linear voltage regulators used to maintain such fluctuations, is a popular voltage regulator integrated circuit (IC). The xx in 78xx indicates the output voltage it provides. 7805 IC provides +5 volts regulated power supply with provisions to add a heat sink.



#### 7805 IC Rating

- Input voltage range 7V- 35V
- Current rating  $I_c = 1A$
- Output voltage range  $V_{Max}=5.2V, V_{Min}=4.8V$

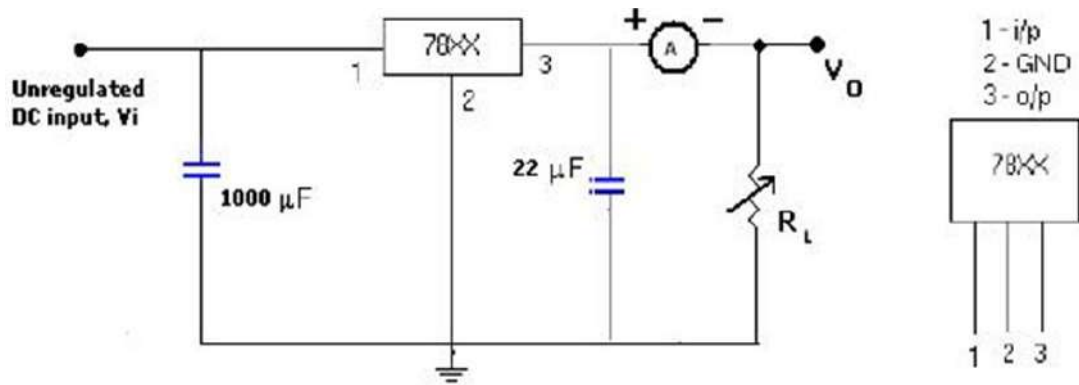


Figure.1 Fixed Positive Voltage regulator

### 79XX:

IC 79xx is a three pin negative voltage controller IC. It is a small integrated circuit used in a circuit to supply a constant negative input voltage. The number 79 indicates that it is a negative voltage regulator and xx indicates the output voltage of the IC. 'xx' can be replaced by the controlled output voltage provided by the regulator, for example, if it is 7905, then the output voltage of the IC is -5 V. Similarly if it is 7912, then output voltage of the IC is -12 volts and so on. The name of the IC may vary based on the manufacturer as LM79xx, L79xx and MC79xx etc.

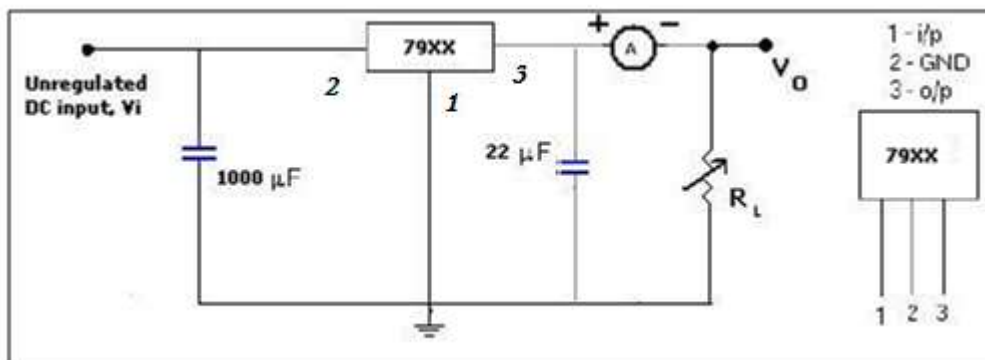


Figure.2 Fixed Negative Voltage Regulator

### PROCEDURE:

#### For fixed positive voltage regulator (7805):

1. Connect the circuit as per diagram figure 1.
2. Apply the unregulated voltage to the IC 7805 and note down the regulator output voltage. Vary input voltage from 7V to 20V and record the output voltages.
3. Calculate the line regulation of the regulator using the formula.
4. Line Regulation =  $\Delta V_o / \Delta V_i$ .

#### For fixed negative voltage regulator (7912):

1. Connect the circuit diagram as shown in figure.2.



2. Apply the unregulated voltage to the IC 7912 and note down the regulator output voltage.
3. Vary input voltage from 7V to 20V and record the output voltages.
4. Calculate the line regulation of the regulator using the formula.
5. Line Regulation =  $\Delta V_o / \Delta V_i$ .

**OBSERVATIONS:**

**1). For +Ve Voltage Regulator 7805**

**Line Regulation: ( $R_L$  is constant)**

SL.No.	Unregulated DC Input, $V_i$ in Volts	Regulated DC Output, $V_o$ in Volts

**2). For -Ve Voltage Regulator 7912**

**Line Regulation: ( $R_L$  is constant)**

SL.NO	Unregulated DC Input, $V_i$ in Volts	Regulated DC Output, $V_o$ in Volts

**CONCLUSION:**

## EXPERIMENT: 10

### **AIM OF THE EXPERIMENT:**

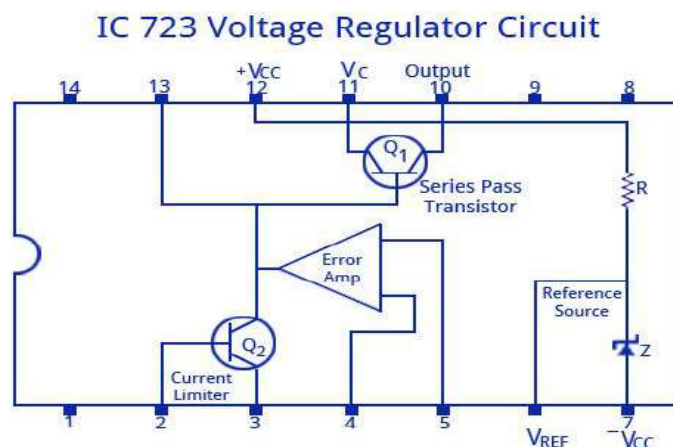
Construct & test voltage power supply using LM723.

### **EQUIPMENT AND COMPONENTS REQUIRED:**

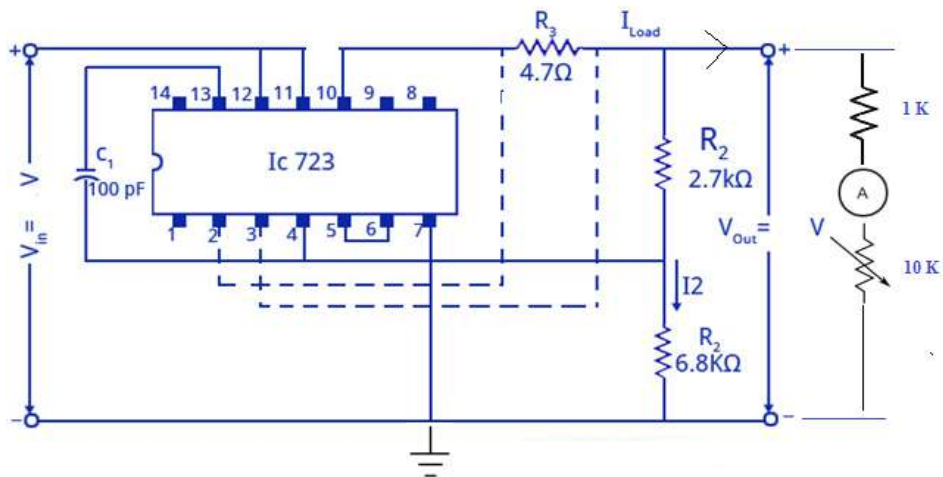
- Bread board
- IC LM723 - 1No.
- Resistors (1K $\Omega$ , 2.7K $\Omega$ , 4.7K $\Omega$ , 6.8K $\Omega$ ) - 1No. each
- Regulated power supply
- DRB / Potentiometer 10K - 1No.
- Capacitors 100pF - 1No.
- Connecting wires
- Ammeter 0-20 mA - 1No.
- Voltmeter 0-20V - 1No.

### **THEORY:**

It consists of a voltage reference source (Pin 6), an error amplifier with its inverting input on pin 4 and non-inverting input on pin 5, a series pass transistor (pins 10 and 11), and a current limiting transistor on pins 2 and 3. The device can be set to work as both positive and negative voltage regulators with an output voltage ranging from 2 V to 37 V, and output current levels upto 150 m A. The maximum supply voltage is 40 V, and the line and load regulations are each specified as 0.01%.



### **POSITIVE VOLTAGE REGULATOR USING IC 723**



### a. To get output voltage > 7V

The output voltage can be set to any desired positive voltage between (7-37) volts. 7 volts is the reference starting voltage. All these variations are brought with the change of values in resistors R1 and R2 with the help of a potentiometer. A Darlington connection is made by the transistor to Q1 to handle large load current. The broken lines in the image indicate the internal connections for current limiting. Even fold back current limiting is possible in this IC. A regulator output voltage less than the 7 V reference level can be obtained by using a voltage divider across the reference source. The potentially divided reference voltage is then connected to terminal 5.

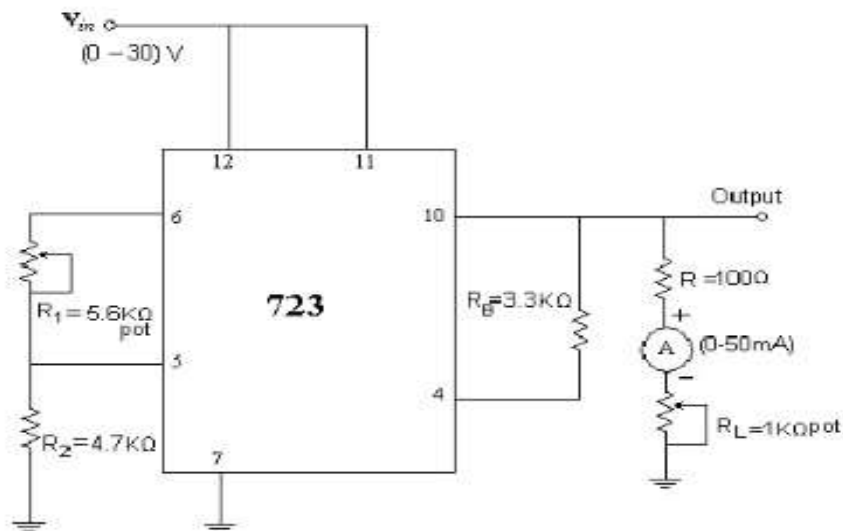


Fig1: Voltage Regulator

### b. To get output voltage < 7V

## PROCEDURE:

### I. LINE REGULATION

1. Connections are made as per the circuit diagram.
2. RPS is connected as  $V_i$ .
3. A fixed load of 1K is kept at the output.
4. Input  $V_i$  is varied from 15V to 25V in steps of 2V and Output voltage is measured.

**Observations:**

14. Line regulation =  $(\Delta V_{out} / \Delta V_{in}) / 100\%$

15.  $V_{nl} =$

Line Voltage (V)	Output Voltage (V)

**CONCLUSION:**

## **EXPERIMENT NO.-11**

### **AIM OF THE EXPERIMENT-**

Study of Operational Amplifier 741 & draw its pin diagram.

**EQUIPMENTS REQUIRED: Not required**

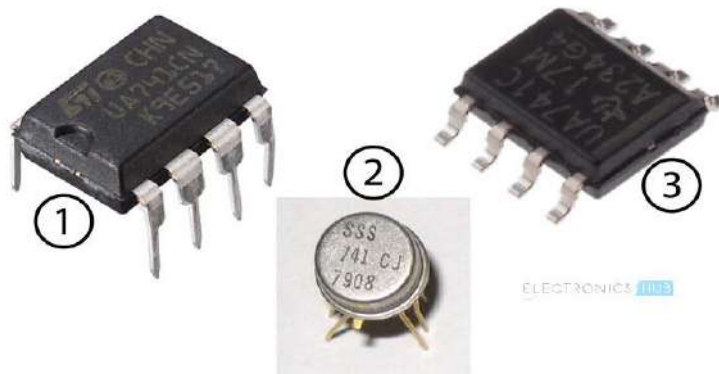
### **THEORY:**

#### **IC 741 Op Amp (Operational Amplifier)**

The 741 Op Amp IC is a monolithic integrated circuit, comprising of a general purpose Operational Amplifier. It was first manufactured by Fairchild semiconductors in the year 1963. The number 741 indicates that this operational amplifier IC has 7 functional pins, 4 pins capable of taking input and 1 output pin.

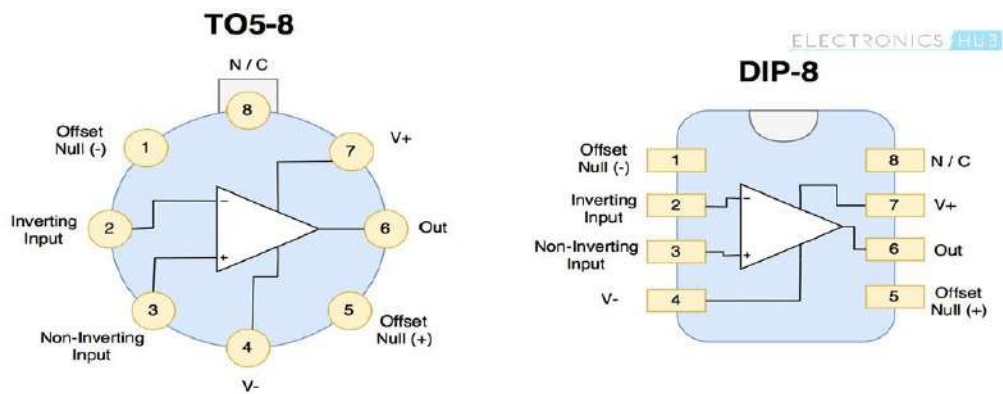
IC 741 Op Amp can provide high voltage gain and can be operated over a wide range of voltages, which makes it the best choice for use in integrators, summing amplifiers and general feedback applications. It also features short circuit protection and internal frequency compensation circuits built in it. This Op-amp IC comes in the following form factors:

- 8 Pin DIP Package
- TO5-8 Metal can package
- 8 Pin SOIC



#### **Pinout of IC 741 Op Amp and their Functions**

The below figure illustrates the pin configurations and internal block diagram of IC 741 in 8 pin DIP and TO5-8 metal can package.



## Pin-Outs of 741 IC

Now let's take a look at the functions of different pins of 741 IC:

- **Pin4 & Pin7 (Power Supply):** Pin7 is the positive voltage supply terminal and Pin4 is the negative voltage supply terminal. The 741 IC draws in power for its operation from these pins. The voltage between these two pins can be anywhere between 5V and 18V.
- **Pin6 (Output):** This is the output pin of IC 741. The voltage at this pin depends on the signals at the input pins and the feedback mechanism used. If the output is said to be high, it means that voltage at the output is equal to positive supply voltage. Similarly, if the output is said to be low, it means that voltage at the output is equal to negative supply voltage.
- **Pin2 & Pin3 (Input):** These are input pins for the IC. Pin2 is the inverting input and Pin3 is the non-inverting input. If the voltage at Pin2 is greater than the voltage at Pin3, i.e., the voltage at inverting input is higher, the output signal stays low. Similarly, if the voltage at Pin3 is greater than the voltage at Pin2, i.e., the voltage at non-inverting input is high, the output goes high.
- **Pin1 & Pin5 (Offset Null):** Because of high gain provided by 741 Op-Amp, even slight differences in voltages at the inverting and non-inverting inputs, caused due to irregularities in manufacturing process or external disturbances, can influence the output. To nullify this effect, an offset voltage can be applied at pin1 and pin5, and is usually done using a potentiometer.
- **Pin8 (N/C):** This pin is not connected to any circuit inside 741 IC. It's just a dummy lead used to fill the void space in standard 8 pin packages.

## Specifications

The following are the basic specifications of IC 741:

Power Supply: Requires a Minimum voltage of 5V and can withstand upto 18V

Input Impedance: About 2 mega ohms

Output impedance: About 75 ohms

Voltage Gain: 200,000 for low frequencies

Maximum Output Current: 20mA

Recommended Output Load: Greater than 2 Kohms

Input Offset: Ranges between 2mV and 6mV

Slew Rate: 0.5V/microsecond (It is the rate at which an Op-Amp can detect voltage changes)

The high input impedance and very small output impedance makes IC 741 a near ideal voltage amplifier.

**CONCLUSION:**

## EXPERIMENT NO.- 12

### AIM OF THE EXPERIMENT:

To construct and study inverting and non-inverting amplifier using OPAMP.

### EQUIPMENT REQUIRED:

5. Operational amplifier characteristics trainer
6. CRO
7. Digital Multimeter
8. Set of patching wires
9. Function Generator

### THEORY:

#### Inverting amplifier:

The op-amp is connected as an inverting amplifier using the circuit diagram given below.

$R_A$  is called the input element and  $R_B$  is called the feedback element. For this circuit, both elements are resistors. The input is applied to the inverting input via  $R_A$  and the non-inverting input is grounded.  $R_B$  allows a fraction of the output voltage ( $V_0$ ) to be fed back to the inverting input. In terms of  $R_A$  and  $R_B$  the output voltage is

$$V_0 = -\frac{R_B}{R_A} V_i$$

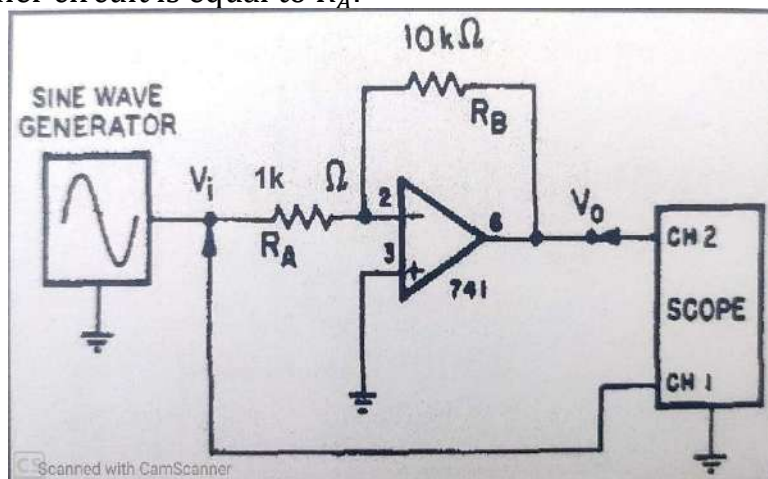
The voltage gain or the ratio of the output voltage to the input voltage is

$$\text{Voltage gain} = \frac{V_0}{V_i} = -\frac{R_B}{R_A}$$

And depends only in the ratio of the feedback resistance  $R_B$  to the input resistance  $R_A$ . Consequently the voltage gain can either be less than 1, equal to 1 or greater than 1.

#### Circuit Diagram of Inverting Amplifier:

Typically  $R_A$  is at least  $1K\Omega$  and  $R_B$  is  $10K\Omega$ , since the input impedance of an inverting amplifier circuit is equal to  $R_A$ .





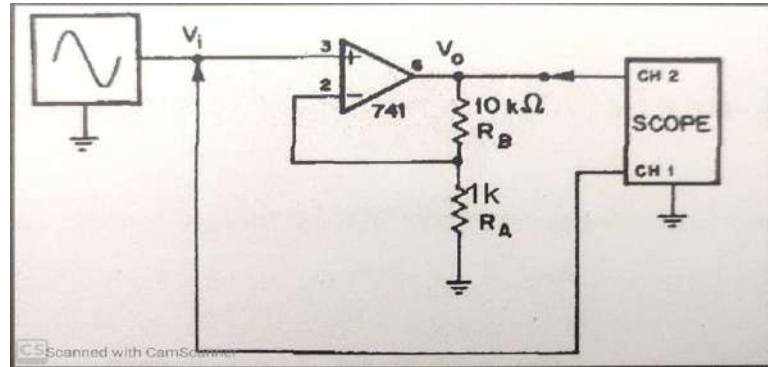
### Non-inverting amplifier:

The op-amp is connected as a non-inverting amplifier using the circuit given below. The input signal is applied directly to the non-inverting input, while the input resistor is grounded. In terms of  $R_A$  and  $R_B$ , the voltage gain is,

$$\text{Voltage gain} = \frac{V_o}{V_i} = 1 + \frac{R_B}{R_A}$$

The voltage gain will always be greater than 1.

### Circuit Diagram:



### DESIGN:

#### Inverting amplifier:

Gain of an inverting amplifier  $A_v = V_o/V_i = -R_B/R_A$

The required gain = -10,

That is  $A_v = -R_B/R_A = -10$

Let  $R_A = 1\text{K}\Omega$ ,

Then  $R_B = 10\text{K}\Omega$

#### Non-inverting amplifier:

Gain of an inverting amplifier

$$A_v = V_o/V_i = 1 + R_B/R_A$$

Let the required gain be 11,

$$\text{Therefore } A_v = 1 + R_B/R_A = 11$$

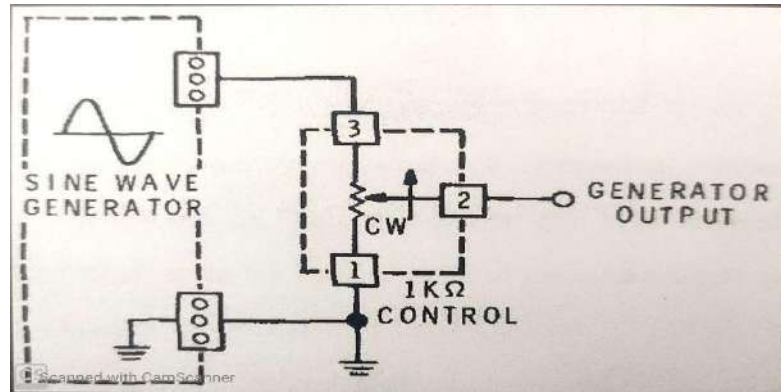
$$R_B/R_A = 10$$

Take  $R_A = 1\text{K}\Omega$ , Then  $R_B = 10\text{K}\Omega$

### PROCEDURE:

#### Inverting amplifier:

1. Patch the circuit as shown in the above figure and refer wiring diagram for inverting amplifier. The sine wave generator must have variable output amplitude. You can vary the generator's output level by adjusting the  $1\text{K}\Omega$  control.



2. Switch on the trainer and observe the two traces on the CRO screen. (Since we are concerned with both the input and output signals, the input signal will be in channel 1 and the output signal will be in channel 2.)
3. Now adjust the amplitude level of your function generator so that the peak to peak input voltage ( $V_i$ ) is 0.2V.
4. Adjust the frequency of the function generator so that there are 2 complete cycles on the screen.
5. Observe the waveform and trace it. (The output signal is opposite or inverted compared to the input signal.)
6. Measure output peak to peak voltage = \_\_\_\_\_ V. using the voltage gain as the ratio of the output voltage to the input voltage, calculate the voltage gain = \_\_\_\_\_.
7. Compare Voltage gain.

$$\frac{V_0}{V_i} = -\frac{R_B}{R_A}$$

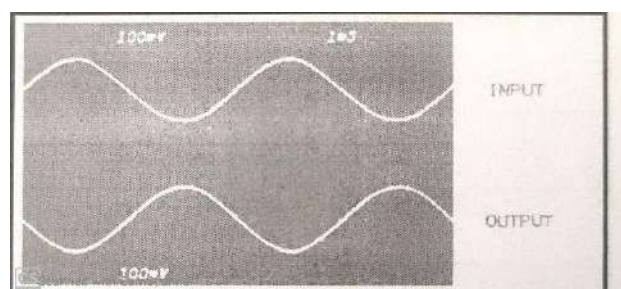
#### Non-inverting amplifier:

1. Patch the circuit as shown in the above figure and refer wiring diagram for non-inverting amplifier.
2. Switch on the trainer. If the peak to peak input level is not 200mV, adjust the control to this level.
3. Observe the waveform of both channels. (the only difference between the two signals is that the output is larger than the input signal.)
4. Measure output peak-to-peak voltage = \_\_\_\_\_ V.
5. Calculate the voltage gain.
6. Compare the voltage gain.

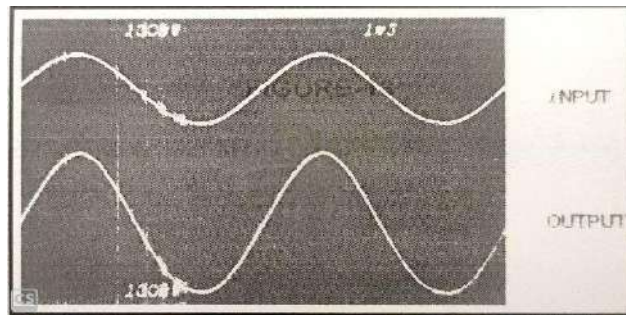
$$\frac{V_0}{V_i} = 1 + \frac{R_B}{R_A}$$

#### Waveform:

#### Inverting Amplifier:



**Non-Inverting Amplifier:**



**CONCLUSION:**

## EXPERIMENT NO.-13

### **Aim of the experiment:**

To construct and study the differentiator and integrator using OPAMP.

### **EQUIPMENTS/COMPONENTS REQUIRED:**

1. Operational amplifier characteristics trainer
2. CRO
3. Digital Multimeter
4. Set of patching wires
5. Function Generator

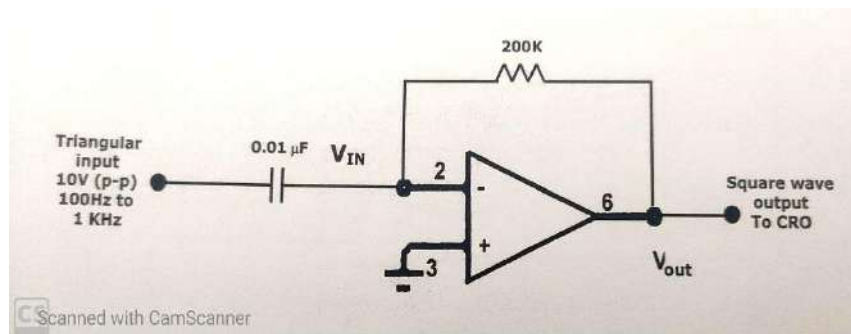
### **THEORY:**

#### **Differentiator:**

It is an op-amp circuit which performs the mathematical operation of differentiation. That is the output waveform is the derivative or differential of the input voltage.

That is  $V_o = -R_A C (V_i)/dt$ . The differentiator circuit is constructed from basic inverting amplifier by replacing the input resistance  $R_A$  with capacitor  $C$ . This circuit also works as high pass filter.

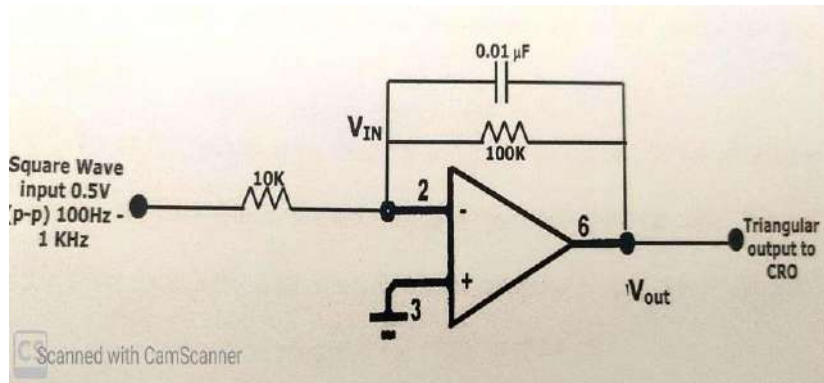
#### **Circuit diagram:**



#### **Integrator:**

It is a closed loop op-amp circuit which performs the mathematical operation of integration. That is the output waveform is the integral of the input voltage and is given by  $V_o = (-1/R_f C) \int V_i dt$ . The integrator circuit is constructed from basic inverting amplifier by replacing the feedback resistance  $R_f$  with capacitor  $C$ . This circuit also works as low pass filter.

#### **Circuit Diagram:**



## PROCEDURE:

### Differentiator:

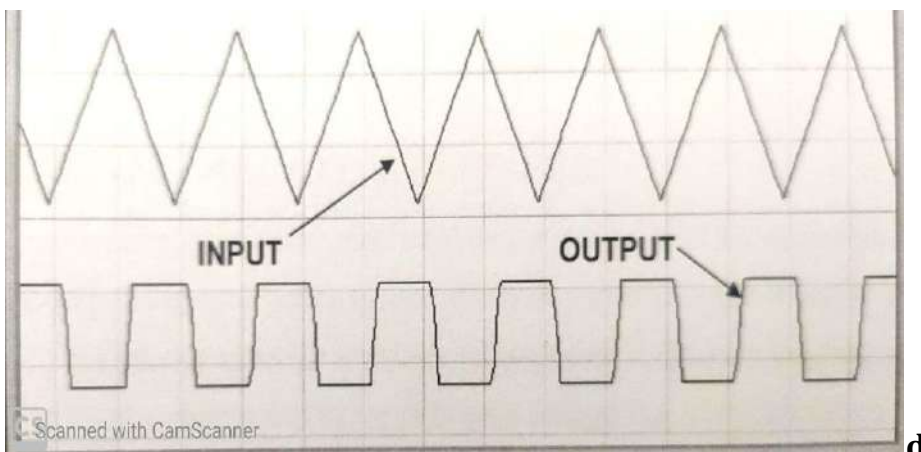
1. Patch the circuit as shown in the figure-1.
2. Switch ON the trainer and check the power supply to be +15V and -15V.
3. Connect 500Hz triangular wave input from function generator to the input of differentiator and set the amplitude to be +10V(p-p).
4. You will observe that the output is a square wave on CRO.
5. Vary the input frequency and observe the output.

### Integrator:

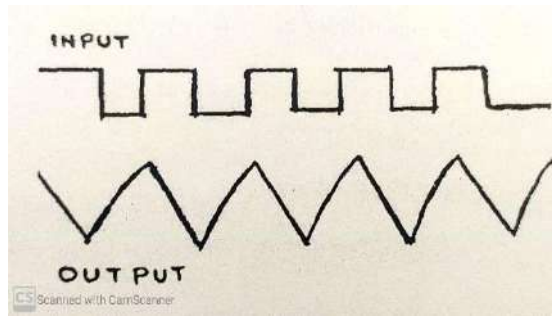
1. Patch the circuit as shown in figure-2.
2. Switch ON the trainer and check the power supply to be +15V and -15V.
3. Set the function generator for a square wave output of 1KHz.
4. Adjust the square wave's peak to peak amplitude 0.5V to 5V.
5. Observe the output waveform.(The output signal should resemble a triangular wave. Since the op-amp is used in the inverting configuration, the triangular waveform which is the integrated square wave is inverted.)
6. Vary the input frequency and observe the output.

## WAVEFORM:c

### Differentiator:



### Integrator:



**CONCLUSION:**