

# **ELECTRONIC CIRCUIT ANALYSIS-1**

## **LABORATORY MANUAL (ECE - 217)**

### **II/IV ECE SEM - I**



By  
Mrs. B.Deepa  
Mr. V.V.K.Raju

Dr. V. Rajya Lakshmi  
Professor & HOD, ECE

**DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING**

**ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES (A)**  
(Affiliated to AU, Approved by AICTE & Accredited by NBA) Sangivalasa-  
531 162, Visakhapatnam District, Phone: 08933-225083/84/87



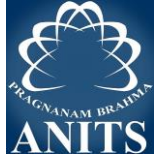
**DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING**  
**Anil Neerukonda Institute of Technology & Sciences (Autonomous)**  
**Sangivalasa-531 162, Bheemunipatnam Mandal, Visakhapatnam District**

**Vision of the Institute**

ANITS envisions to emerge as a world-class technical institution whose products represent a good blend of technological excellence and the best of human values.

**Mission of the Institute**

To train young men and women into competent and confident engineers with excellent communication skills, to face the challenges of future technology changes, by imparting holistic technical education using the best of infrastructure, outstanding technical and teaching expertise and an exemplary work culture, besides molding them into good citizens



**DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING**  
**Anil Neerukonda Institute of Technology & Sciences (Autonomous)**  
**Sangivalasa-531 162, Bheemunipatnam Mandal, Visakhapatnam District**

**Vision of the Department**

To become a centre of excellence in Education, research and produce high quality engineers in the field of Electronics and Communication Engineering to face the challenges of future technological changes.

**Mission of the Department**

To achieve vision department will

Transform students into valuable resources for industry and society by imparting contemporary technical education.

Develop interpersonal skills and leadership qualities among students by creating an ambience of academic integrity to participate in various professional activities

Create a suitable academic environment to promote research attitude among students.



**DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING**  
**Anil Neerukonda Institute of Technology & Sciences (Autonomous)**  
**Sangivalasa-531 162, Bheemunipatnam Mandal, Visakhapatnam District**

**Program Educational Objectives (PEOs):**

**PEO1 :** Graduates excel in their career in the domains of Electronics, Communication and Information Technology.

**PEO2 :** Graduates will practice professional ethics and excel in professional career through interpersonal skills and leadership qualities.

**PEO3 :** Graduates demonstrate passion for competence in higher education, research and participate in various professional activities.

**Program Outcomes (POs):**

Engineering Graduates will be able to:

1. **Engineering knowledge:** Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
2. **Problem analysis:** Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
3. **Design/development of solutions:** Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
4. **Conduct investigations of complex problems:** Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
5. **Modern tool usage:** Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.

6. **The engineer and society:** Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
7. **Environment and sustainability:** Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
8. **Ethics:** Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
9. **Individual and team work:** Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
10. **Communication:** Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
11. **Project management and finance:** Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.
12. **Life-long learning:** Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

**Program Specific Outcomes (PSOs):**

- PSO1 :** Implement Signal & Image Processing techniques using modern tools.
- PSO2 :** Design and analyze Communication systems using emerging techniques.
- PSO3 :** Solve real time problems with expertise in Embedded Systems.



### Electronic Circuits and Analysis-I Laboratory (ECE 217)

<b>ECE 217</b>	<b>Credits:2</b>
Instruction: 3 Practicals / Week	Sessional Marks:50
End Exam: 3 Hours	End Exam Marks:50

**Course Outcomes:**

**At the end of the course, students will be able to**

ECA-I Lab	CO1	Measure the important parameters of a PN diode from the V-I characteristics.
	CO2	Analyze different rectifier circuits and voltage regulation circuits used in regulated Power supplies.
	CO3	Design amplifier circuits using BJTs in different configurations and determine fL and fH from the frequency response characteristics.
	CO4	Analyze and Design linear wave shaping circuits and non linear waveshaping circuits.
	CO5	Analyze and Design different multivibrator circuits.

**CO-PO-PSO MAPPING:**

CO	PO1	PO3	PO4	PO8	PO9	PO10	PSO3
CO1	1		3	1	1	1	1
CO2	2		3	1	1	1	2
CO3	2	2	3	1	1	1	2
CO4	2	2	3	1	1	1	1
CO5	2	2	3	1	1	1	2

**Correlation levels    1: Slight (Low) 2: Moderate (Medium)    3:Substantial (High)**



**ELECTRONIC CIRCUIT ANALYSIS-I LABORATORY**  
**LIST OF EXPERIMENTS**

**CYCLE-I:DESIGN AND SIMULATION USING MULTISIM SOFTWARE**

<b>Sl.No</b>	<b>NAME OF THE EXPERIMENT</b>	<b>Page Number</b>
1.	Linear Wave Shaping	13
2.	Non-Linear Wave Shaping – Clippers & Clampers	17
3.	Halfwave rectifier	21
4.	Fullwave rectifier	24
5.	Bridge rectifier.	26
6.	Common Emitter amplifier.	29
7.	Common Collector amplifier	34
8.	Astable Multivibrator	36
9.	Monostable Multivibrator	38
10.	Schmitt trigger circuit	40
11.	Two Stage RC-Coupled Amplifier	42

**CYCLE-II: (Hardware experiments)**

<b>Sl.No</b>	<b>NAME OF THE EXPERIMENT</b>	<b>Page Number</b>
1.	Volt-Ampere Characteristics of PN junction diode and LED	44
2.	V-I Characteristics and regulation characteristics of Zener diode.	53
3.	Ripple factor calculation of Half wave rectifier and Full wave rectifier	59
4.	Diode Clippers	68
5.	Clamping Circuits	72
6.	Transistor Switch	78
7.	Linear wave shaping	80
8.	Common Emitter Configuration	84
9.	Common Base Configuration	89
10.	Drain and transfer characteristics of JFET	93
11.	Two Stage RC-Coupled Amplifier	99
12.	Bistable Multivibrator	103



**DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING**  
**Anil Neerukonda Institute of Technology & Sciences (Autonomous)**  
**Sangivalasa-531 162, Bheemunipatnam Mandal, Visakhapatnam District**

**ECA-1 LAB (ECE217) SCHEME OF EVALUATION**

Total marks for each student to evaluate in lab: 100 marks

Out of 100 marks:

- a. External exam Evaluation: 50 marks
- b. Internal Evaluation: 50 marks
  - Internal exam: 25 marks
  - Evaluation in Lab: 25 marks

<b>EXTERNAL EXAMS (50 Marks)</b>		
S.No.	Lab exam assessment	Marks allotted
1.	Write Up-Aim, apparatus, Circuit diagram	05
2.	Procedure	05
3.	Tabular columns , Theoretical calculations(if any),Formulas	05
4.	Model graphs & precautions	05
5.	Performance	10
6.	Results & graph	10
7.	Viva	10
<b>TOTAL MARKS</b>		<b>50</b>
<b>INTERNAL EVALUATION (50 Marks)</b>		
1.	Internal Exam at the end of semester(Marks division same as above)	25
2.	<b>Evaluation in lab</b>	
	Observation(write up for every experiment)	5
	Viva (viva in every lab)	5
	Record Aim & apparatus-1mark Circuit diagram and procedure-3 marks Theory-3 marks Calculations and tables-2 marks Result-1marks	10
	Attendance >90% - 5 marks >80% - 90% - 4 marks >75% - 80% - 3 marks >=66% - 75% -2 marks <66% - No Marks	5
<b>Total marks:</b>		<b>50</b>





**DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING**  
**Anil Neerukonda Institute of Technology & Sciences (Autonomous)**  
**Sangivalasa-531 162, Bheemunipatnam Mandal, Visakhapatnam District**

**RUBRICS**  
**(ECA-II LABORATORY)**

S.No	Competency	Performance Indicator
1.	Demonstrate an ability to conduct experiments consistent with their level of knowledge and understanding.	Laboratory preparation
		Stating clearly the aim of the experiment, its scope and importance for purpose of doing experiment.
		Ability to construct the circuit diagram on a bread board and use meters/ instruments to record the measured data according to the range selected.
2.	Demonstrate an ability to design experiments to get the desired output.	Finding the appropriate values of the components to meet the specifications.
		Analyze data for trends and correlations, stating possible errors and limitations in choosing the component values.
3.	Demonstrate an ability to analyze the data and reach valid conclusions.	Tabulate data (tabular form or in graphical form) from the results so as to facilitate analysis and explanations of the data, and draw conclusions.
		Ability to gather materials and writing in lab record

S.No	Performance Indicator	Excellent (A) 100%	Good(B) 80%	Need improvement (C) 60%	Fail (D) <40%
1.	Laboratory preparation & ability to construct the circuit diagram on a bread board and use meters/ instruments to record the measured data according to the range selected (Based on physical observation)(5M)	Read and understand the lab manual before coming to lab. Observations are completed with necessary theoretical calculations including the use of units and significant figures & Obtain the correct values of the components after calculations. Follow the given experimental procedures, to obtain the desired output.	Observations are completed with necessary theoretical Calculations but With-out proper understanding & Obtain the correct values for only few components after calculations. Follow the given experimental procedures, but obtained results with some errors.	Observations are incomplete & Obtain the incorrect values for components. Lacks the appropriate knowledge of the lab procedures. Has no idea what to do	No effort exhibited
2.	Stating clearly the aim of the experiment, its scope and importance for purpose of doing experiment & Oral Presentation (Based on viva)(5M)	Clearly describes the purpose of doing experiment and its scope. Responds confidently, and precisely in giving answers to questions correctly	Clearly describes the purpose of doing experiment. Responds in giving answers to questions but some answers are wrong.	Some idea of doing experiment but not very clear & responds in giving answers to questions but all answers are wrong.	No effort exhibited
3.	Presentation of record & Conclusions of the lab experiment performed. (Based on Lab record)(10M)	Well-organized, interesting, confident presentation of record & able to correlate the theoretical concepts with the concerned lab results with appropriate reasons.	Presentation of record acceptable	Presentation of record lacks clarity and organized	No effort exhibited



## About the Lab

In this lab students will be able to study and analyses the basic electronic devices like diodes, BJTs, JFETs, Half-wave & full-wave rectifiers. Rectifiers are introduced and their performances with different types of filters are observed and basic amplifiers. With this knowledge students will be able to do mini-projects with the help of diodes and transistors.





**DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING**  
**Anil Neerukonda Institute of Technology & Sciences (Autonomous)**  
**Sangivalasa-531 162, Bheemunipatnam Mandal, Visakhapatnam District**

**LIST OF MAJOR EQMENT IN EDC-I LABORATORY**

SL.NO	NAME OF THE EQUIPMENT	MAKE	QUANTITY
1.	20 MHz, 25MHz & 30MHz DUAL TRACE OSCILLOSCOPE	SCIENTECH/SCIENTIFIC/ CADDO/FALCON	16
2.	1 MHz FUNCTION GENERATOR WITH DIGITAL DISPLAY	SCIENTECH/SCIENTIFIC/ SYSTRONICS/ FUTURE TECH/ METRAVI/APLAB	18
3.	TRPS 0-30V, 2A DUAL CHANNEL	ITL HYD/FALCON	26
4.	TRPS 0-30V, 2A SINGLE CHANNEL	ITL HYD	10
5.	AC MICRO VOLTMETER	SYSTRONICS	10
6.	5KVA SERVO CONTROLLED STABILIZER	ITL	01
7.	BENCH TOP DIGITAL MULTIMETER	METRAVI/APLAB	23

**TOTAL EXPENDITURE OF THE LABORATORY (including consumables: Rs. 17,19,646/-**



**DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING**  
**Anil Neerukonda Institute of Technology & Sciences (Autonomous)**  
**Sangivalasa-531 162, Bheemunipatnam Mandal, Visakhapatnam District**

**During the lab class:**

**Do's**

1. Be punctual and regular to the laboratory.
2. Maintain Discipline all the time and obey the instructions.
3. Check the connections properly before turning ON the circuit.
4. Turn OFF the circuit immediately if you see any component heating.
5. Dismount all the components and wires before returning the kit.
6. Any failure / break-down of equipment must be reported to the faculty

**Don'ts**

1. Don't touch live electric wires.
2. Don't turn ON the circuit unless it is completed.
3. Avoid making loose connections.
4. Don't leave the lab without permission.
5. Do not handle any equipment without reading the instructions/Instruction manuals

# CYCLE-I: DESIGN AND SIMULATION USING MULTISIM SOFTWARE

## 1. LINEAR WAVE SHAPING

### Aim:

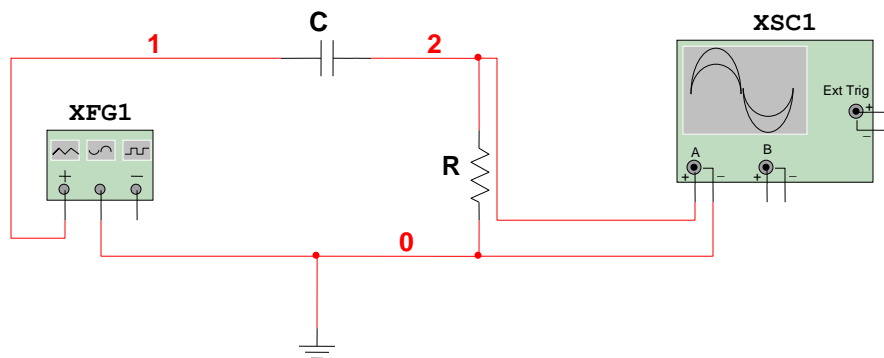
Design a RC LPF and HPF at various time constants and verify the responses for Square wave input (choose  $C = 0.1\mu\text{f}$ ,  $V_i = V = 4\text{ VP-P}$ ,  $f = 10\text{ K Hz}$ ) using Multisim

### SOFTWARE USED:

Multisim V10.

### Circuit diagram:

#### High Pass Filter:



### Design / Calculations:

a)  $RC = T$

$$\text{Given } T = 1/10\text{ KHz} = 0.1\text{ mSec}$$

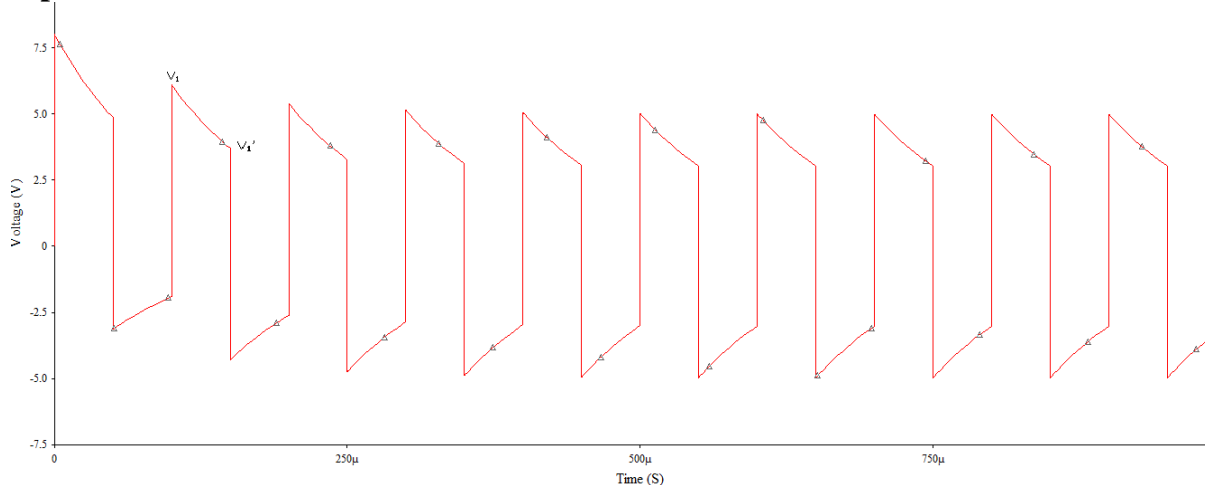
$$R = (0.1 \times 10^{-3}) / 0.1\mu\text{f} = 1\text{ K ohms}$$

$$V_1 = V / (1 + e^{-T/2RC}) = 2.49\text{v}$$

$$V_1' = V / (1 + e^{T/2RC}) = 1.51\text{ v}$$

$$\% \text{ tilt} = 2 (V_1 - V_1') / V = (2.49 - 1.51) / 2 = 49\%$$

### Output Waveform:



b)  $RC \gg T$

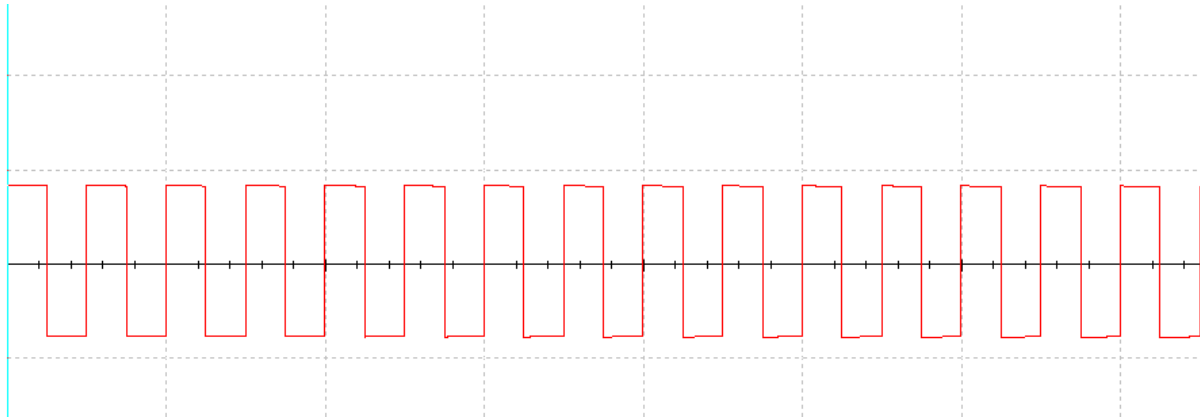
Choose  $RC = 100T = 1\text{ m Sec}$

$$R = (10^{-3}) / (0.1 \times 10^{-6})$$

$$= 100 \text{ k ohms}$$

The output waveform will be identical to input

**Output Waveform:**



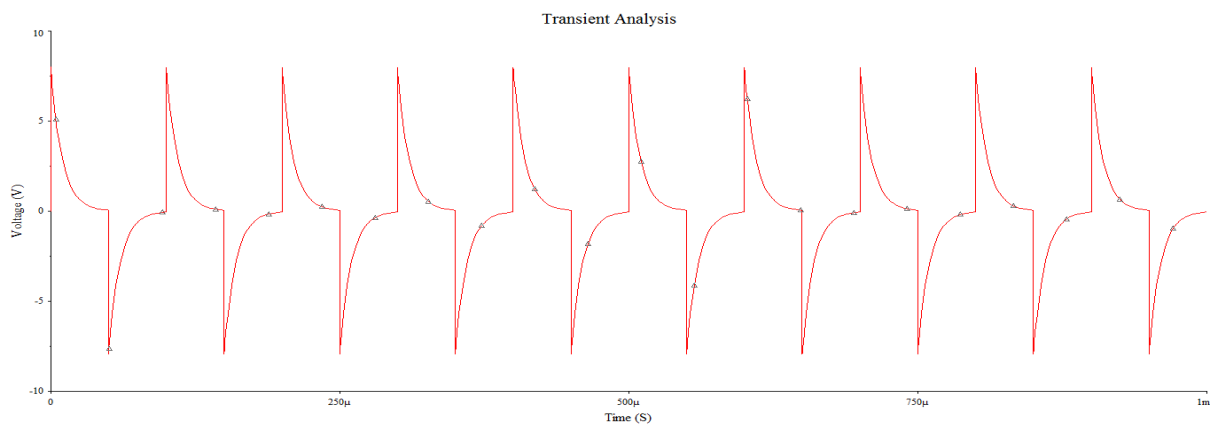
c)  $RC \ll T$

$$RC = 0.1 \text{ T}$$

$$R = (0.1 \times 10^{-4}) / (0.1 \times 10^{-6})$$

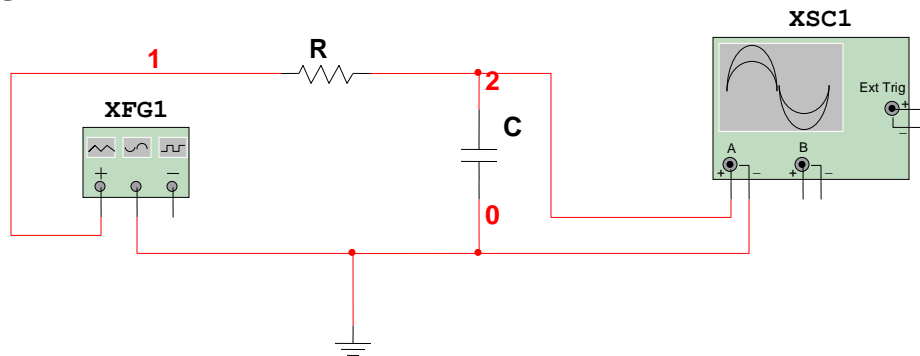
$$= 100 \text{ Ohms}$$

**Output Waveform:**



## Low Pass Filter:

### Circuit Diagram:



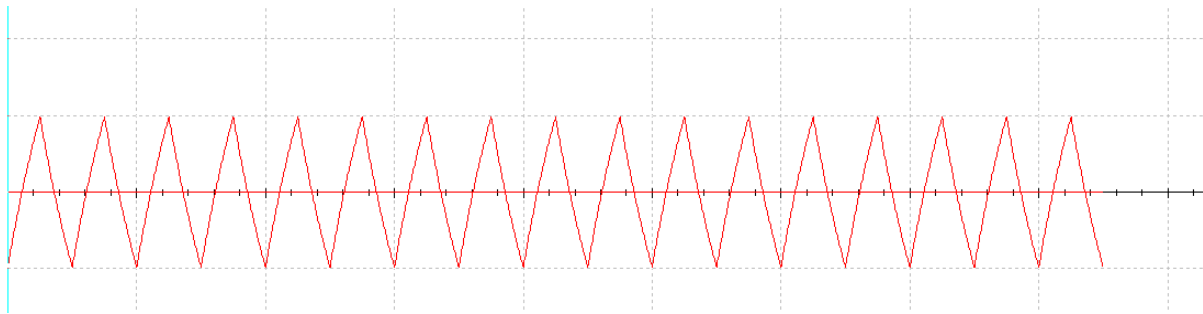
### Design / Calculations:

For  $f=1\text{KHz}$

(a)  $RC=T$

$$C=0.1\mu\text{f}, R= 10\text{K Ohms}$$

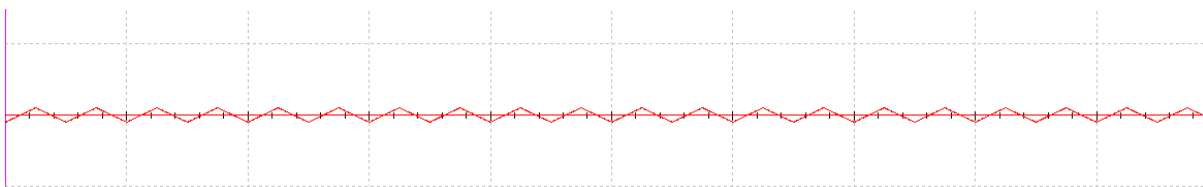
### Output Waveform:



b)  $RC \gg T$

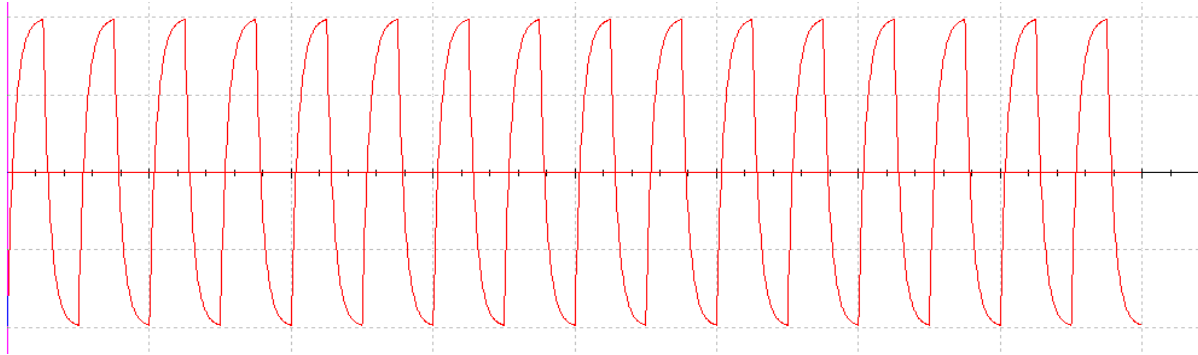
$$C=0.1\mu\text{f}, R= 100\text{K Ohms}$$

### Output Waveform:



c)  $RC \ll T$

$$C=0.1\mu\text{f}, R= 1\text{K Ohms}$$



**Procedure:**

**HPF:**

1. Open new schematic capture in Multisim.
2. Connect the circuit as shown in figure using Toolbar → components.
3. Now apply a square wave of frequency 1KHz and amplitude 8V(P-P) using function generator available in Multisim.
4. Now click on Simulate and then RUN the circuit.
5. Now observe the output waveform using oscilloscope.
6. For transient analysis....  
     Simulate → Analyses → Transient analysis
7. Enter the time duration and select output node .click on simulate.
8. Repeat same procedure for LPF .

**Result:**

LPF and HPF are designed at various time constants and the responses for square wave input is observed & hence plotted using Multisim software.

**Viva questions:**

1. What is linear wave shaping?
2. How low pass RC circuit works as an integrator?
3. How low pass RC circuit works as differentiator?
4. Define time constant?
5. Define tilt?
6. Explain the output wave forms of high pass and low pass circuit for different conditions.



## 2. NON-LINEAR WAVE SHAPING – CLIPPERS & CLAMPERS

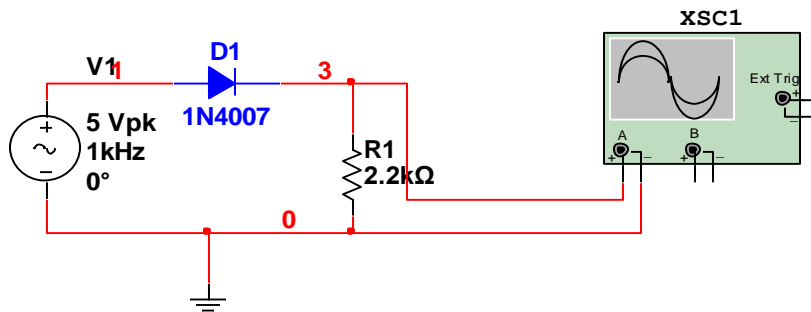
**Aim:** To study the clipping and clamping circuits using Multisim.

**SOFTWARE USED:**

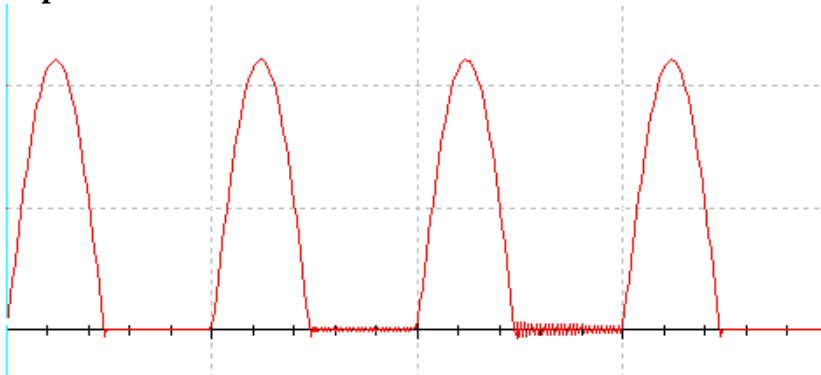
Multisim V10.

**Circuit diagram:**

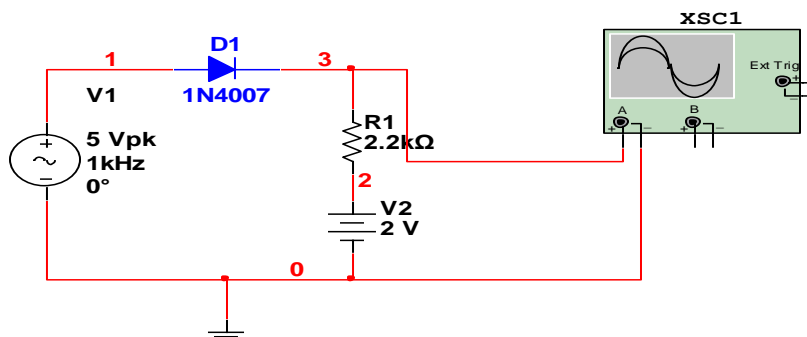
**Series Diode Clipper:-**



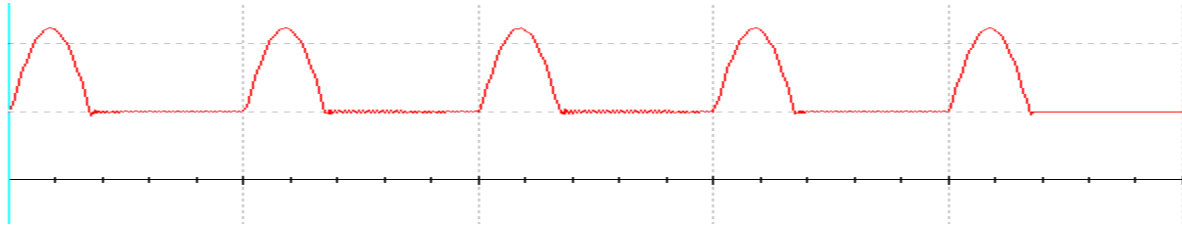
**Output Waveform:**



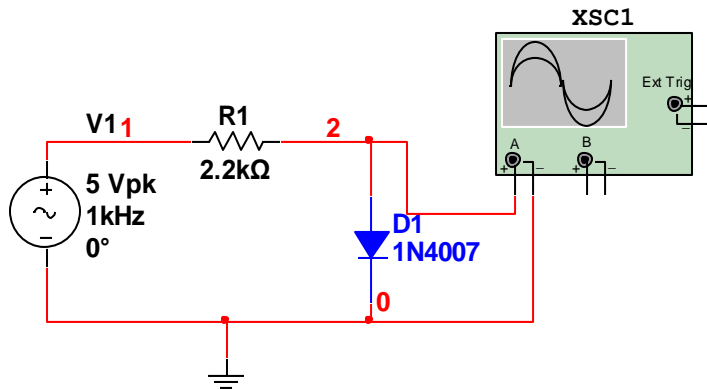
**Series Diode Clipper with Bias:**



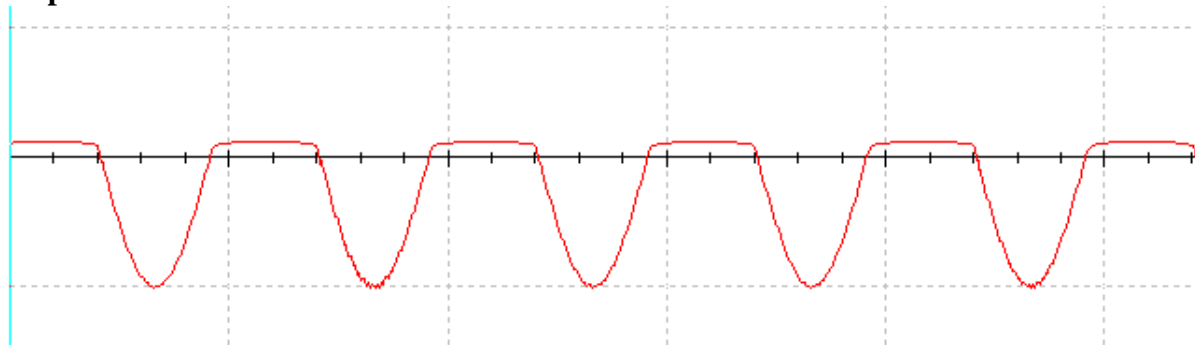
### Output Waveform:



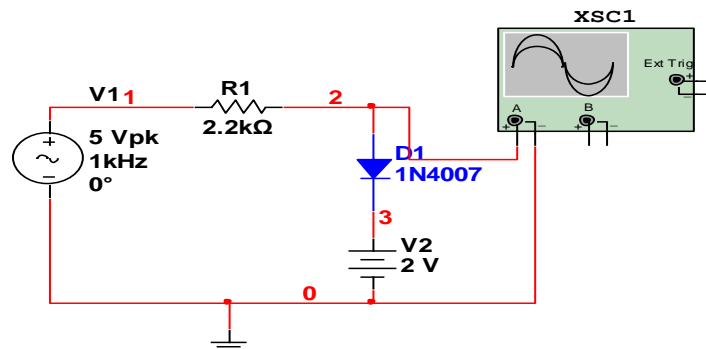
### Shunt Diode Clipper:



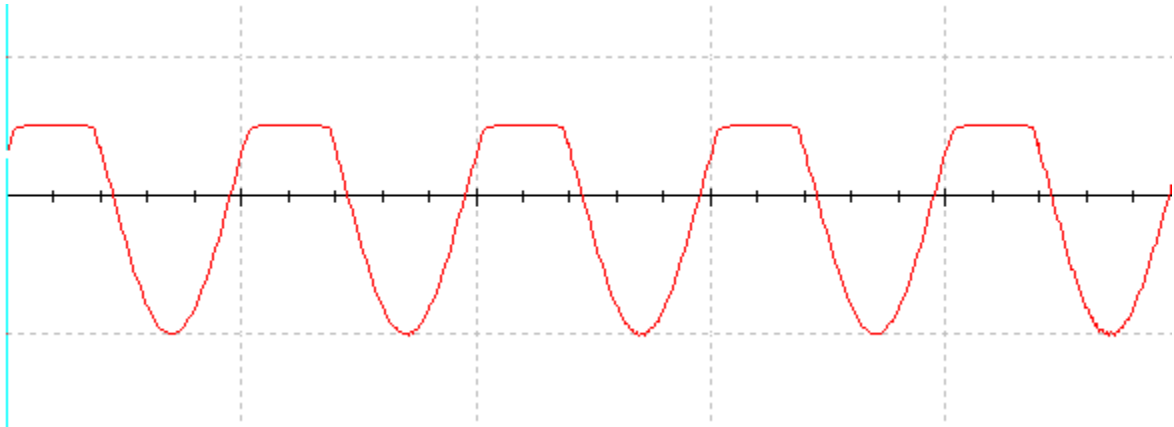
### Output Waveform:



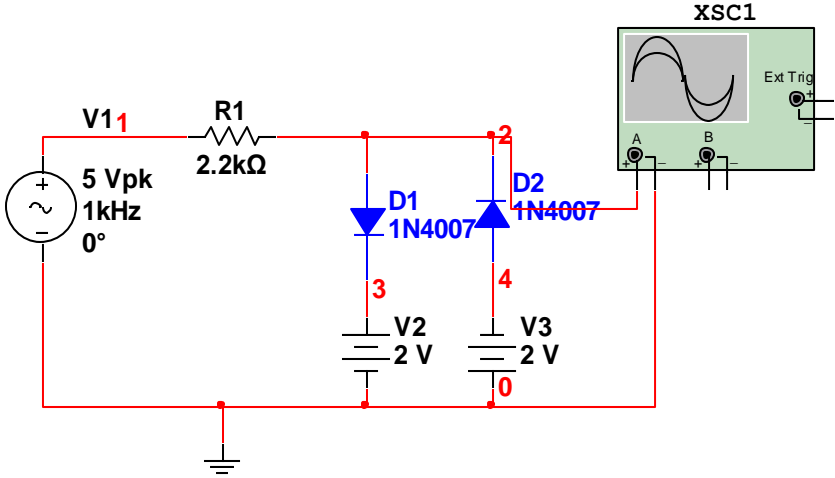
### Shunt Diode Clipper with Bias



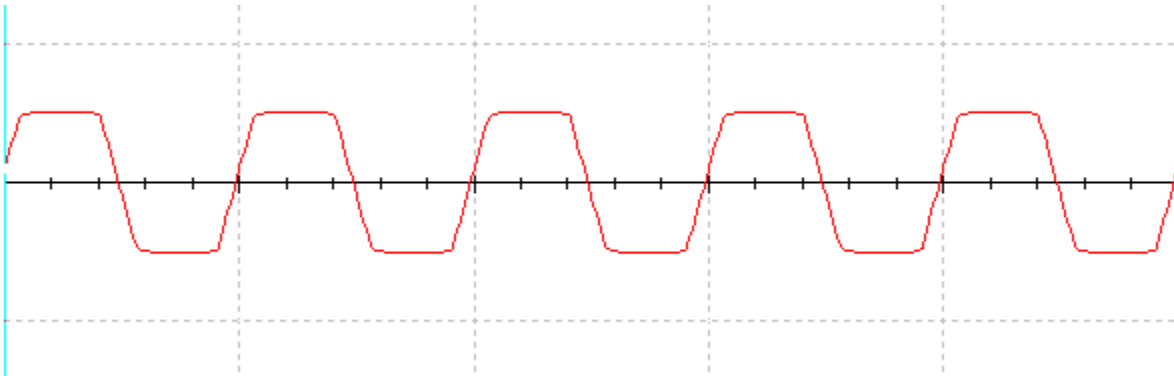
**Output Waveform:**



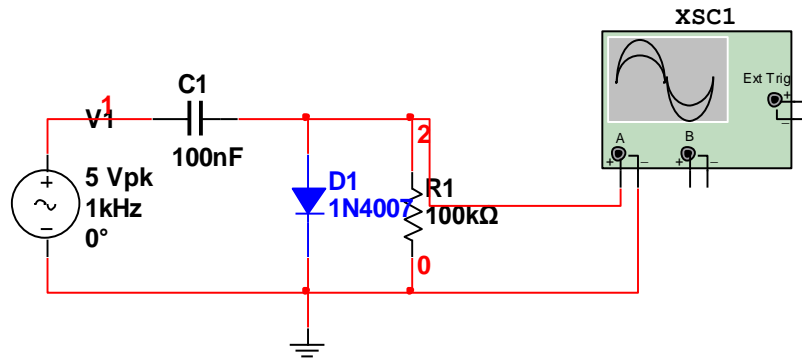
**Slicer:**



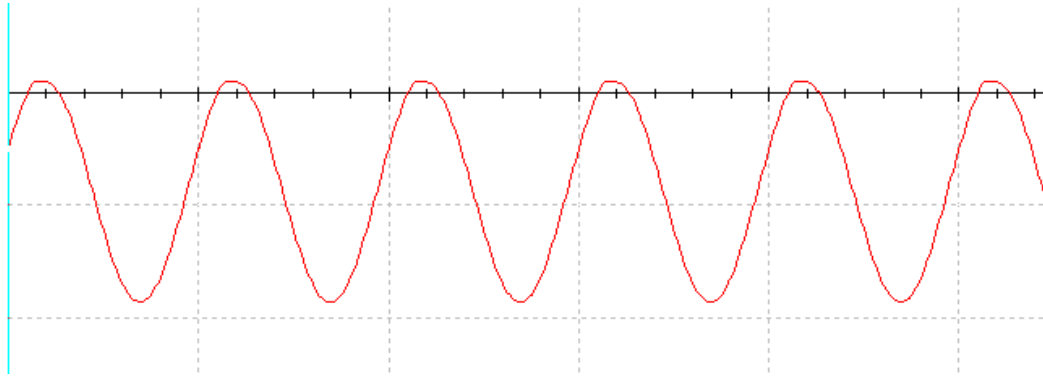
**Output Waveform:**



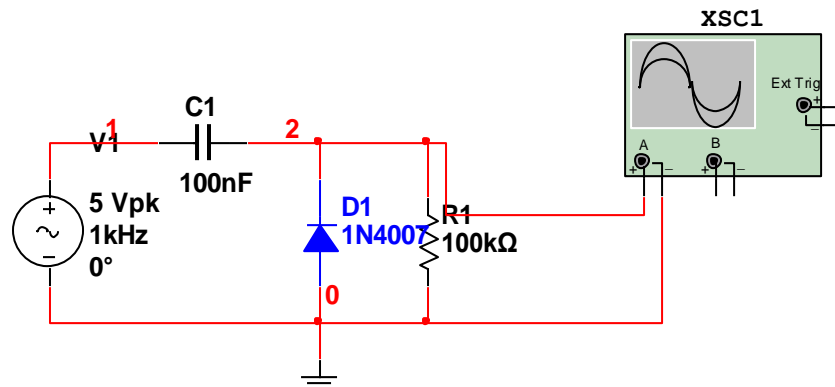
## Negative Clamper



## Output Waveform:



## Positive Clamper



### Output Waveform:



### Procedure:

1. Open new schematic capture in Multisim.
2. Connect the circuit as shown in figure using Toolbar  $\rightarrow$  components.
3. Now apply a sine wave of frequency 1KHz and amplitude 10V(P-P) using function generator available in Multisim.
4. Now click on Simulate and then RUN the circuit.
5. Now observe the output waveform using oscilloscope.

### Result:

Different types of clipping and clamping circuits are studied and observed the response for different combinations of  $V_R$  and diodes using Multisim..

### Viva questions

1. What is meant by non linear wave shaping?
2. What is clipper? What are the different types of clippers?
3. What are the different applications of clipper?
4. What is two level clipper
5. Explain the operation of positive and negative clipper?

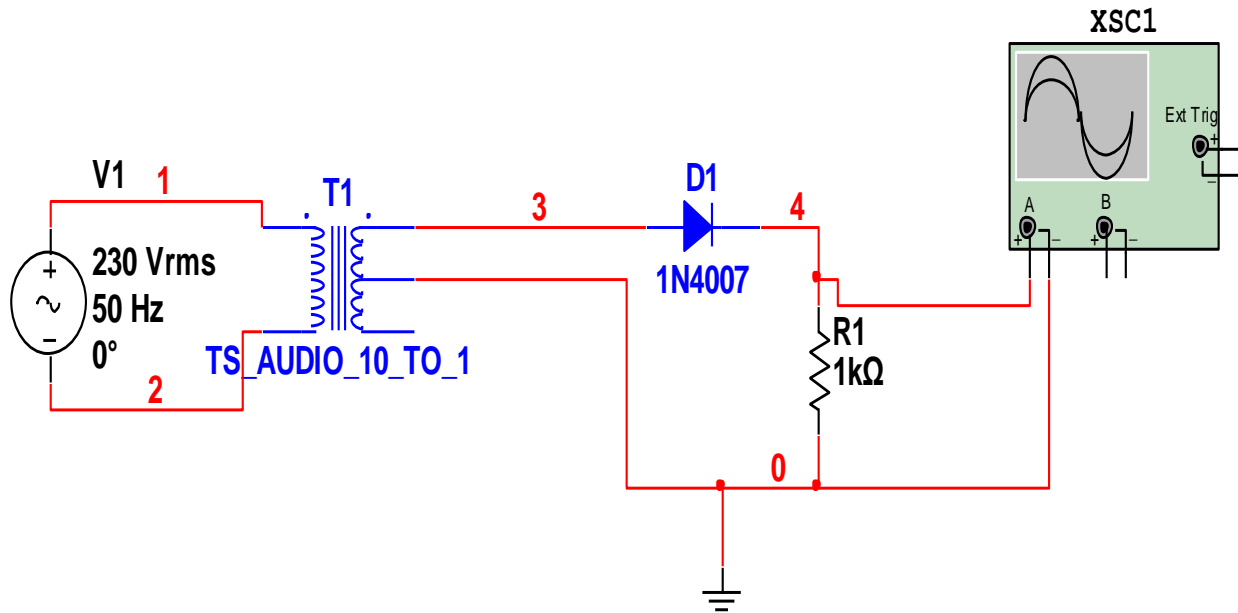
### 3. HALFWAVE RECTIFIER

**Aim:** To study the operation of half-wave rectifier and find its ripple factor using Multisim.

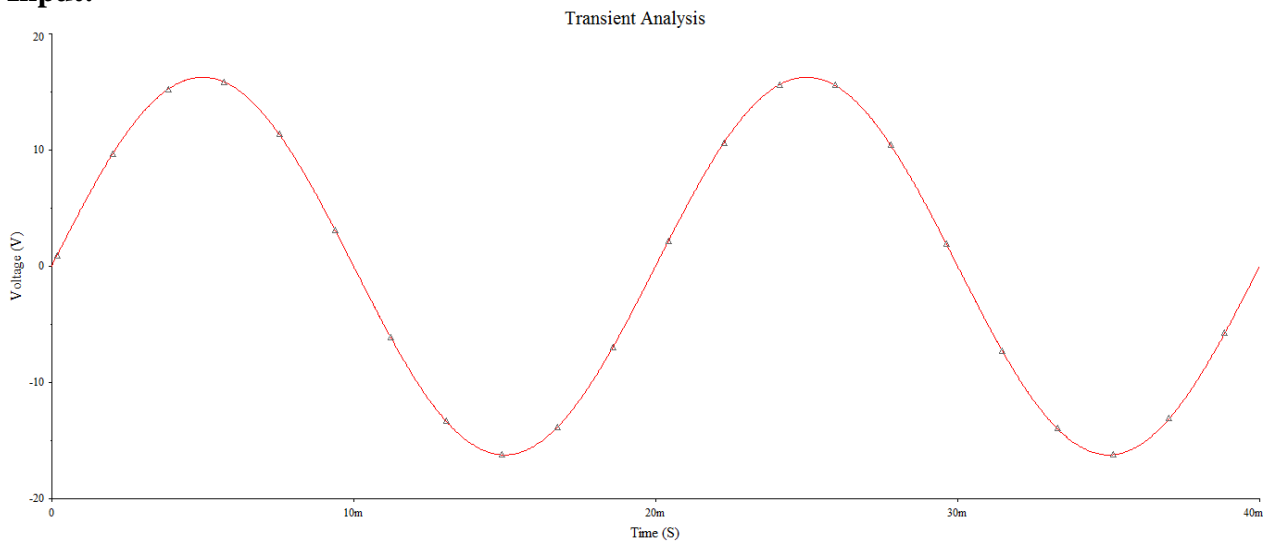
**Software:** Multisim

**Circuit Diagram:**

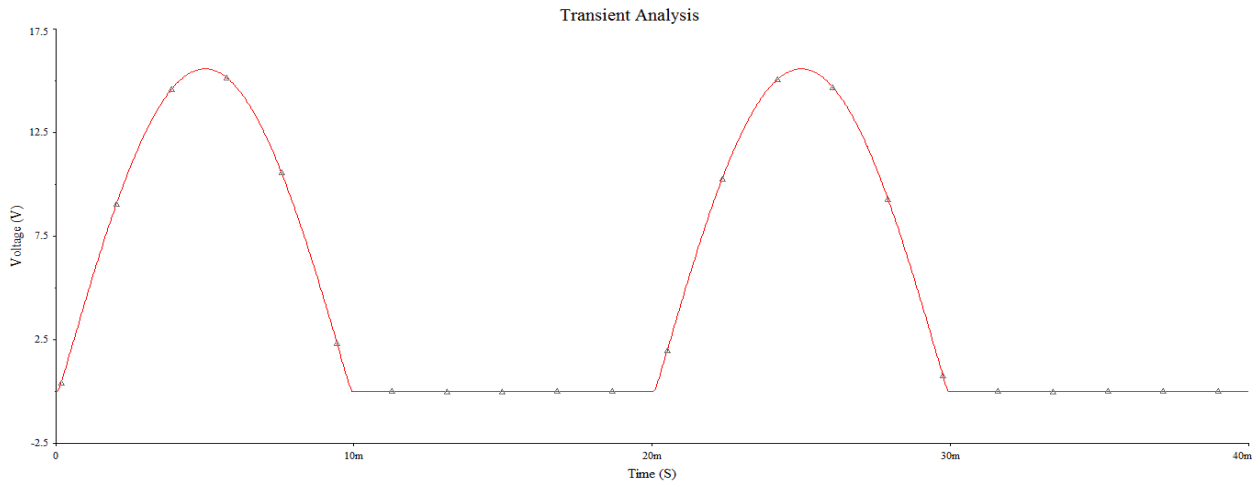
**Without filter:**



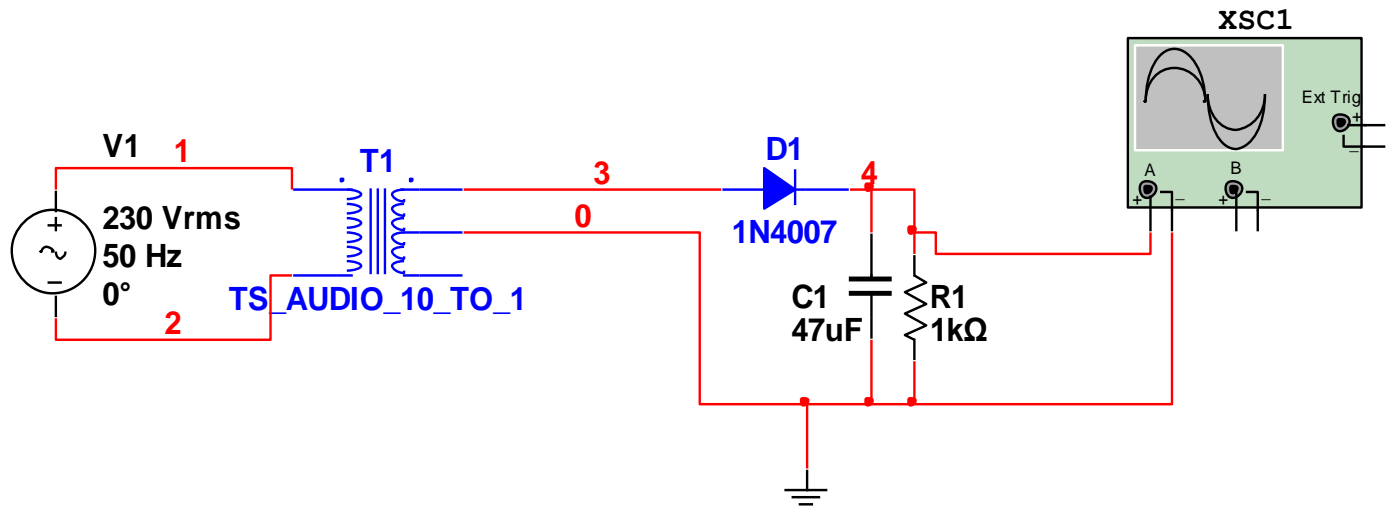
**Input:**



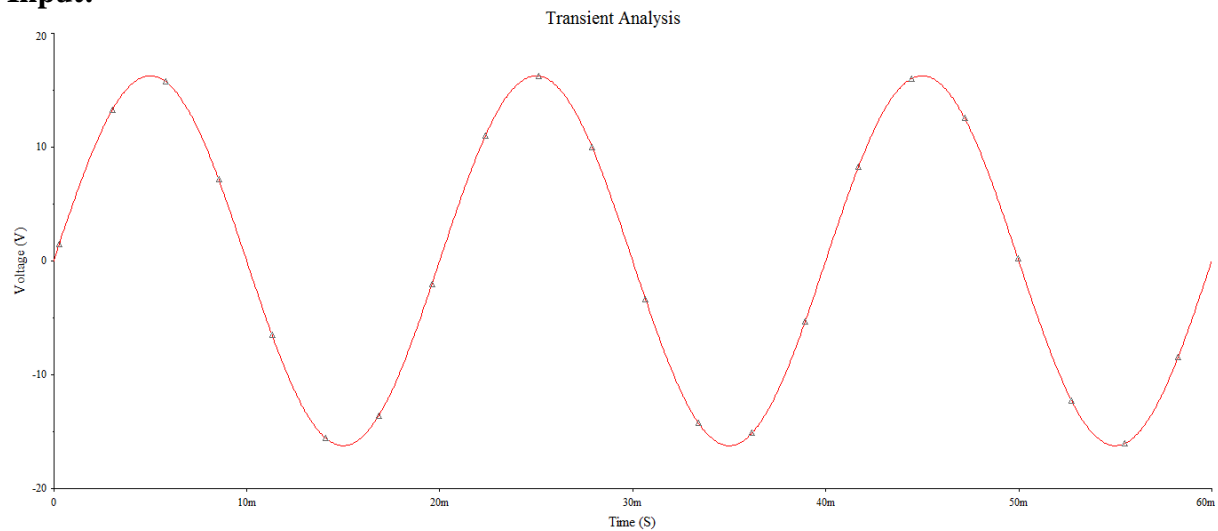
### Output:

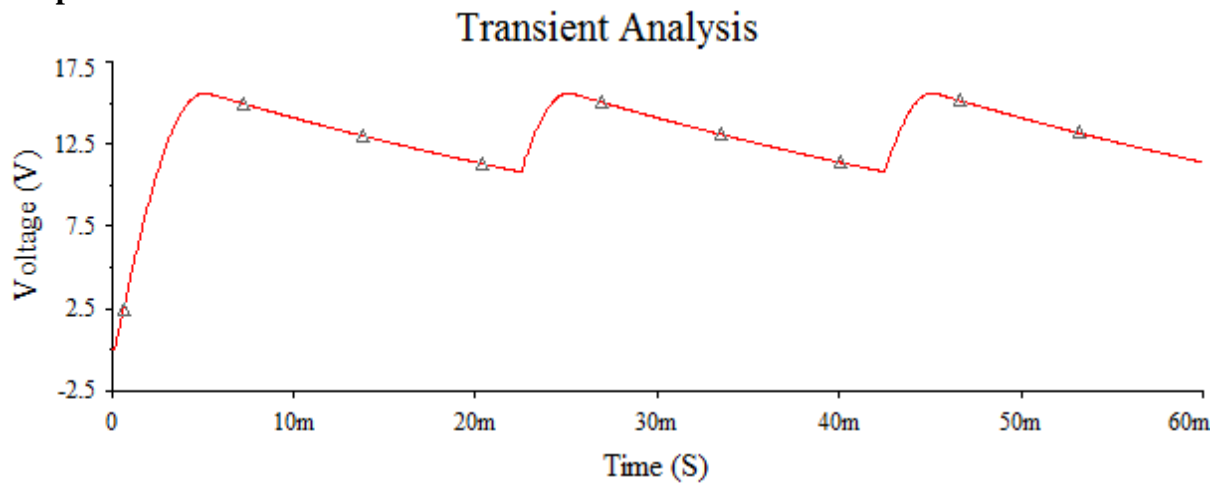


### Circuit Diagram with capacitor filter:



### Input:



**Output:****Procedure:**

1. Open new schematic capture in Multisim.
2. Connect the circuit as shown in figure using Toolbar → components.
3. Now apply a sine wave of frequency 50Hz ,230Vrms across primary coil of the transformer.
4. RUN the circuit and observe the output waveform using oscilloscope.

**Result:**

The operation of the half-wave rectifier is studied using Multisim and ripple factor is calculated.

**VIVA QUESTIONS:**

1. What is rectifier?
2. What is filter?
3. Define Ripple factor.
4. What is Peak inverse voltage?
5. How capacitor acts as filter.
6. Define regulation.
7. What are the applications of rectifiers?
8. Define transformer utilization factor.



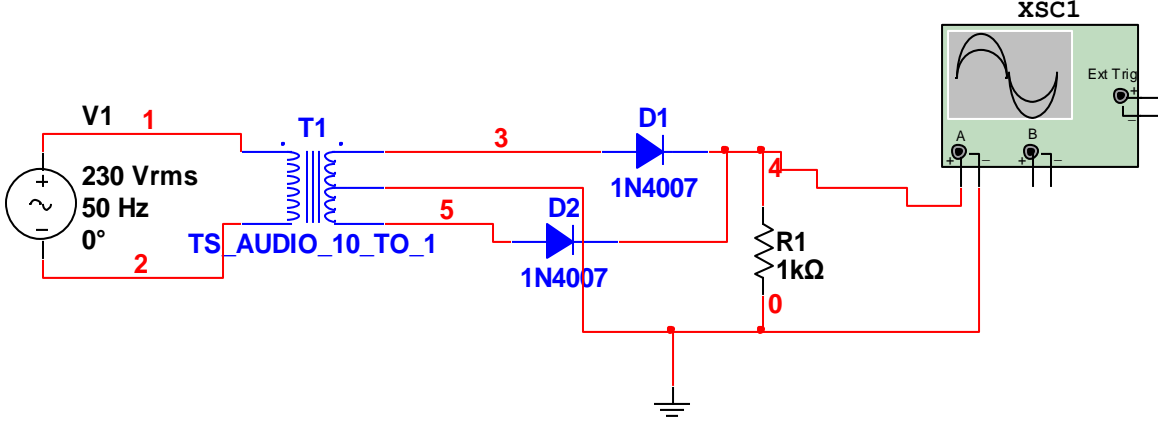
# 4. FULLWAVE RECTIFIER

**Aim:** To study the operation of full-wave rectifier and find its ripple factor using Multisim.

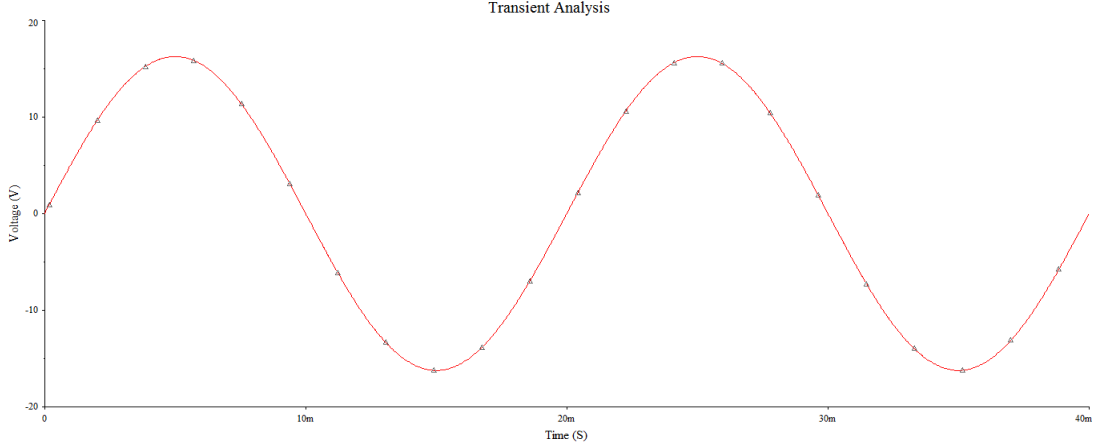
**Software:**  
Multisim

**Circuit Diagram:**

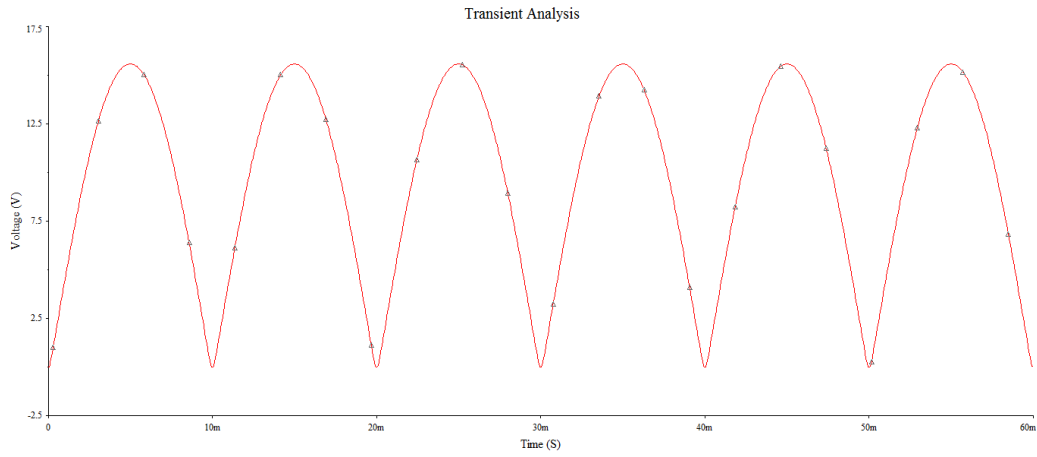
**Without filter:**



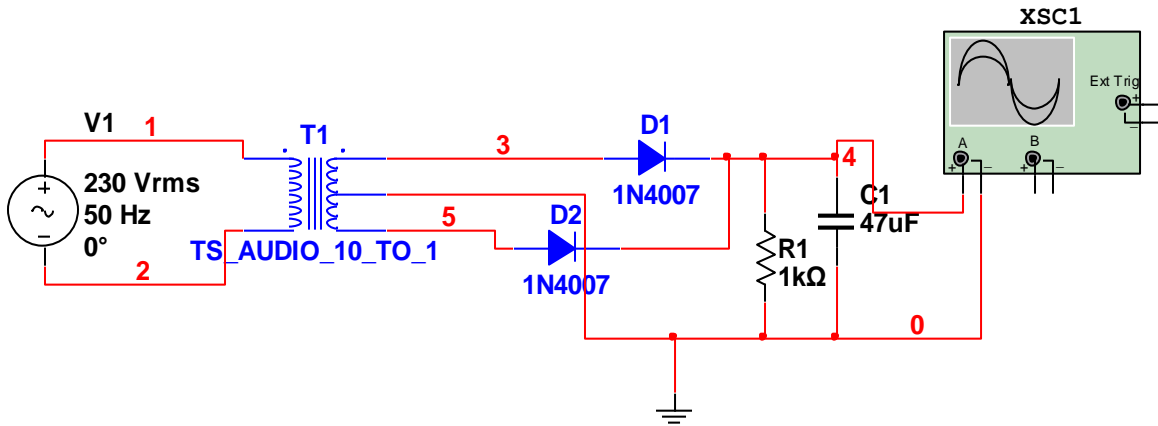
**Input:**



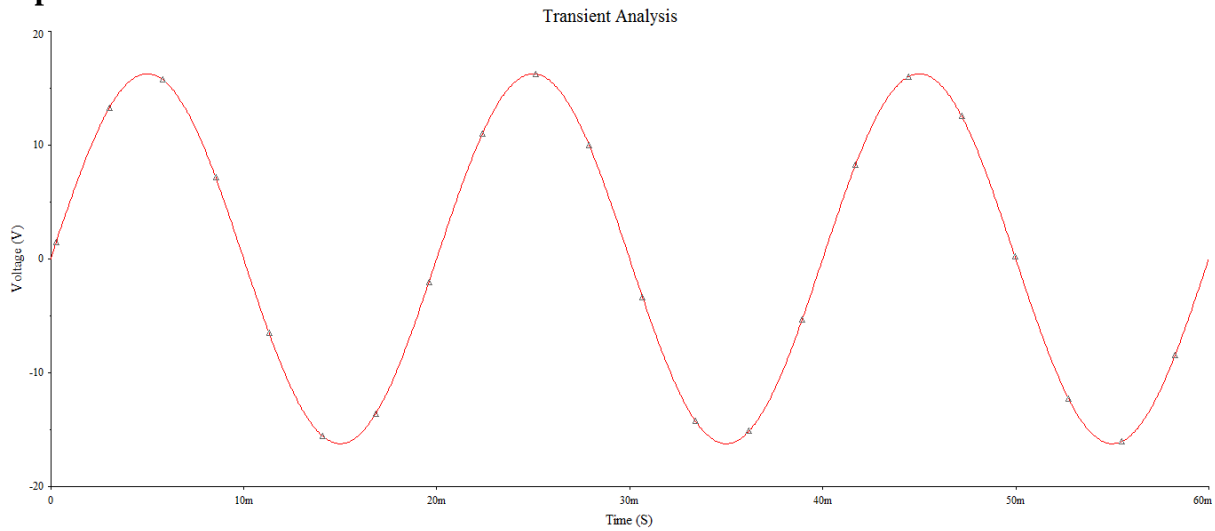
### Output:



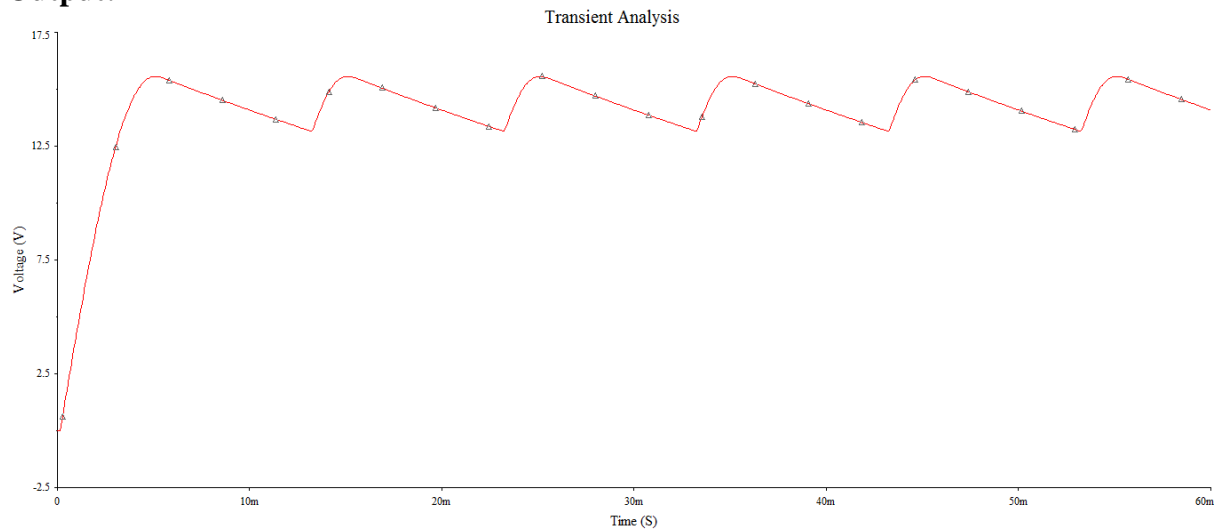
### With filter:



### Input:



## Output:



## Procedure:

1. Open new schematic capture in Multisim.
2. Connect the circuit as shown in figure using Toolbar  $\rightarrow$  components.
3. Now apply a sine wave of frequency 50Hz ,230Vrms across primary coil of the transformer.
4. RUN the circuit and observe the output waveform using oscilloscope.

## Result:

The operation of the full-wave rectifier is studied using Multisim and ripple factor is calculated.

## VIVA QUESTIONS:

1. What is rectifier?
2. What is filter?
3. Define Ripple factor.
4. What is Peak inverse voltage?
5. How capacitor acts as filter.
6. Define regulation.
7. What are the applications of rectifiers?
8. Define transformer utilization factor.

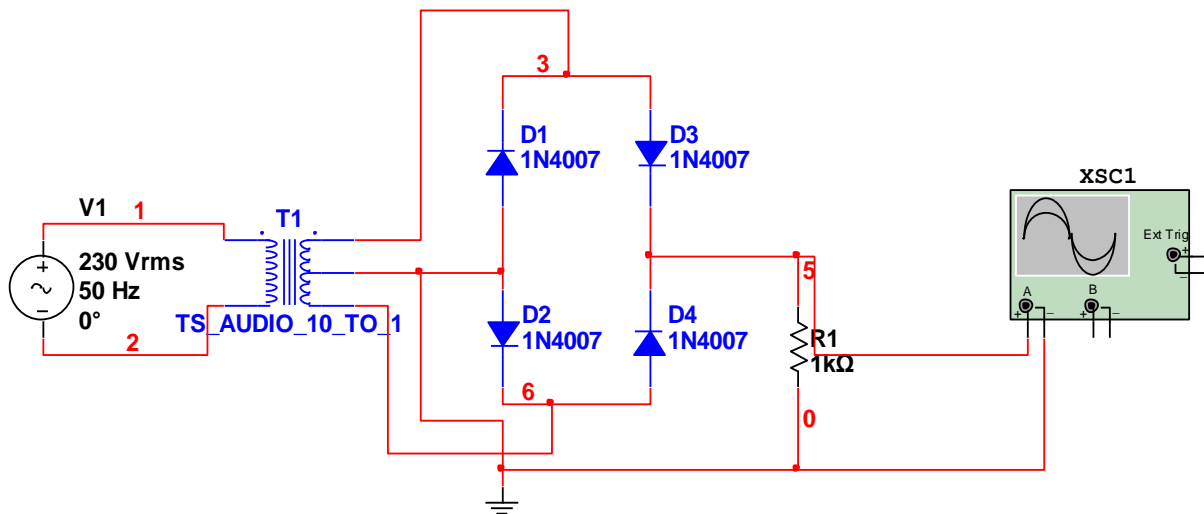
## 5. BRIDGE RECTIFIER

**Aim:** To study the operation of bridge rectifier and find its ripple factor using Multisim.

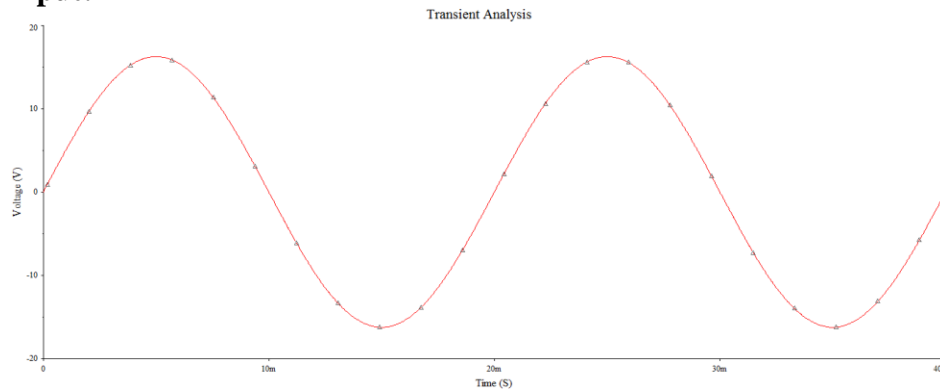
**Software:**  
Multisim

**Circuit Diagram:**

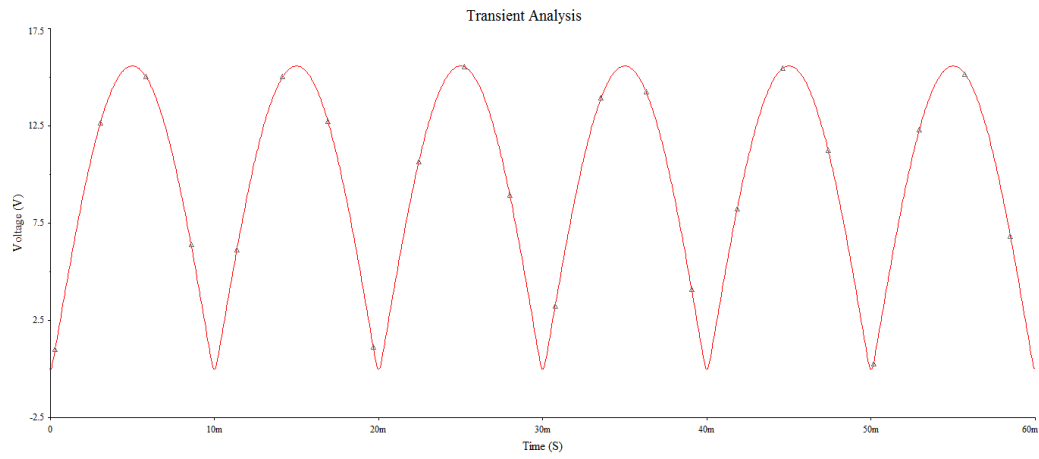
**Without filter:**



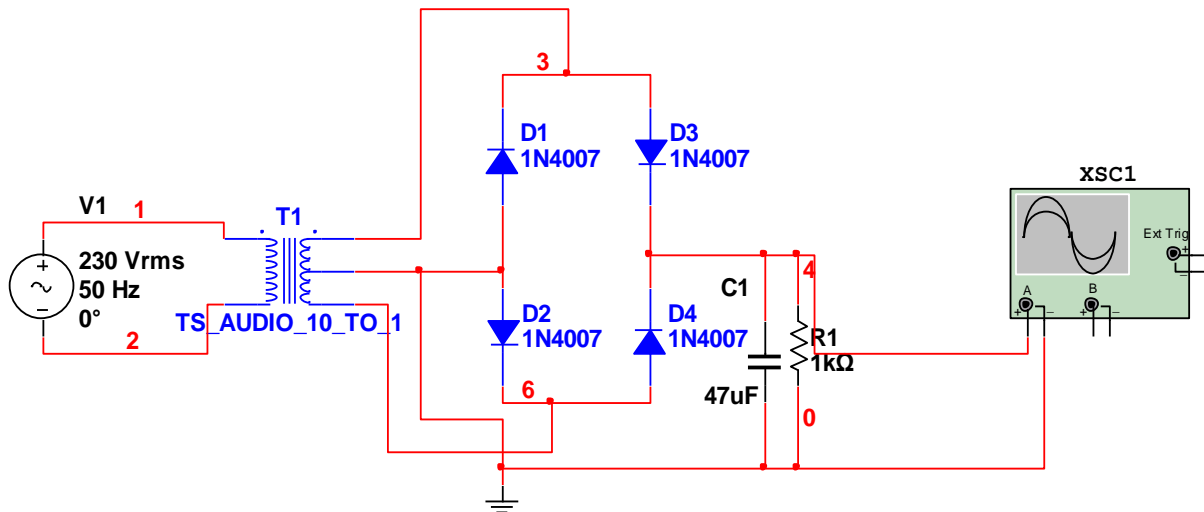
**Input:**



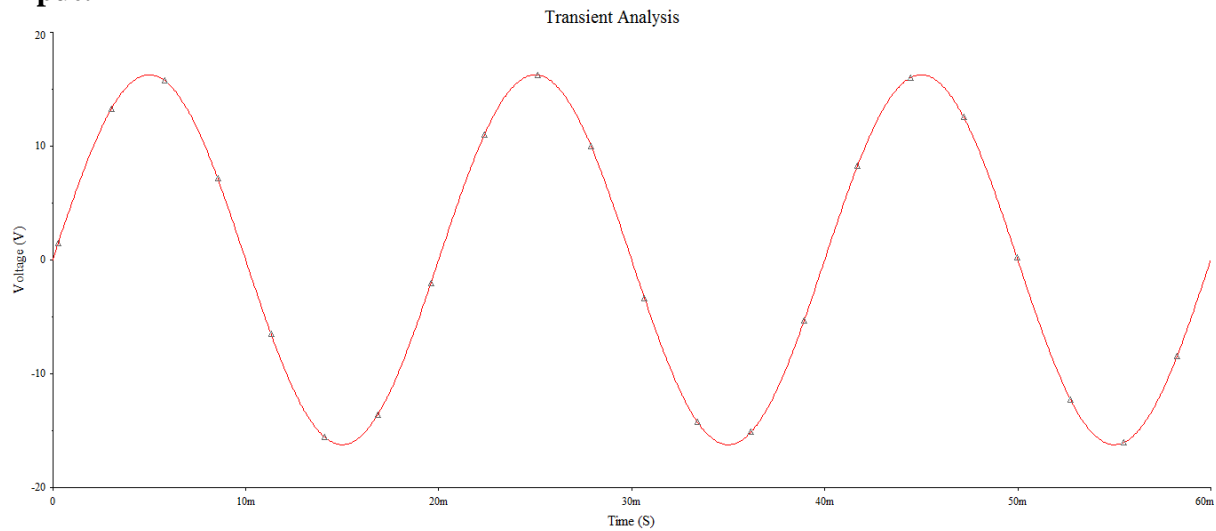
## Output:



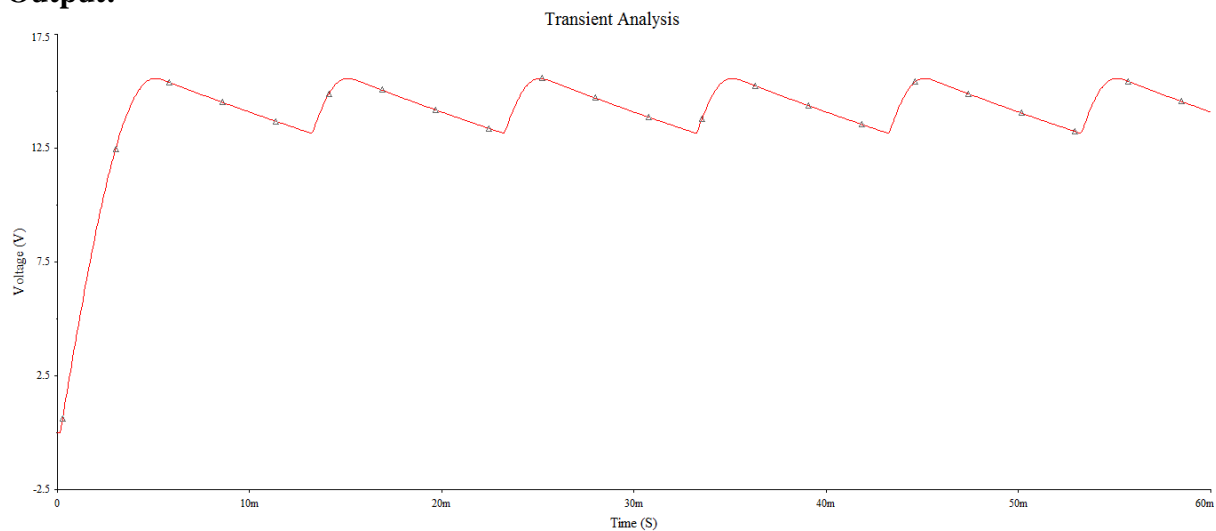
## With filter:



### Input:



### Output:



### Procedure:

1. Open new schematic capture in Multisim.
2. Connect the circuit as shown in figure using Toolbar → components.
3. Now apply a sine wave of frequency 50Hz, 230Vrms across primary coil of the transformer.
4. RUN the circuit and observe the output waveform using oscilloscope.

### Result:

The operation of the bridge rectifier is studied using Multisim and ripple factor is calculated.

### **VIVA QUESTIONS:**

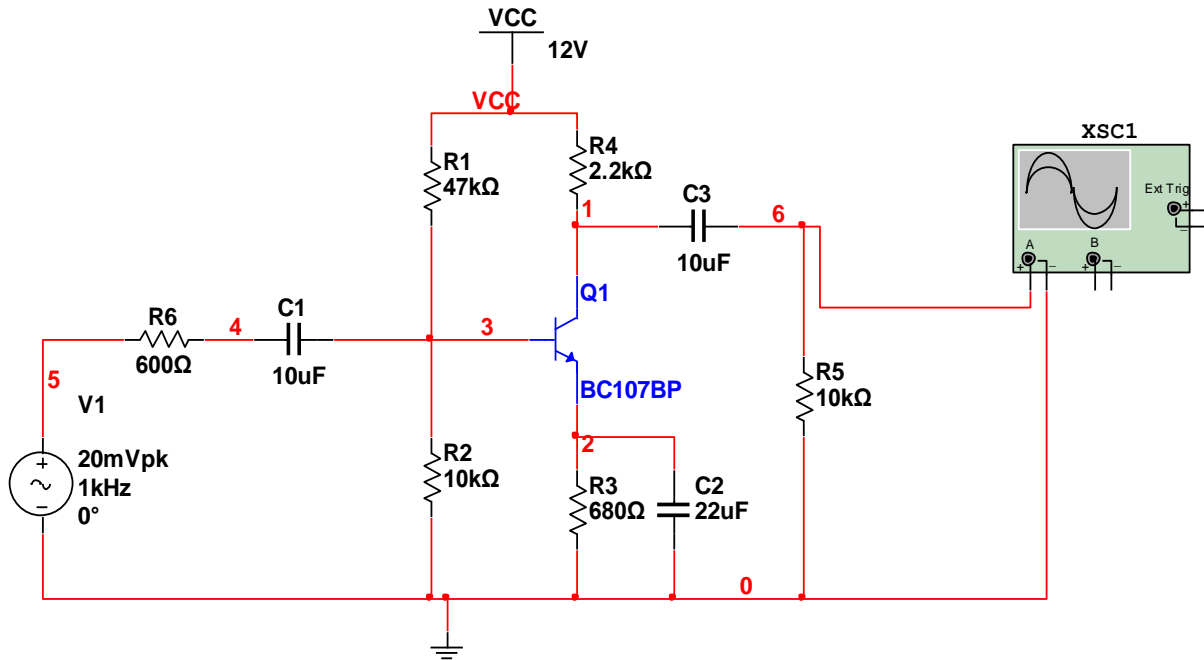
1. What is rectifier?
2. What is filter?
3. Define Ripple factor.
4. What is Peak inverse voltage?
5. How capacitor acts as filter.
6. Define regulation.
7. What are the applications of rectifiers?
8. Define transformer utilization factor.

## 6. COMMON EMITTER AMPLIFIER

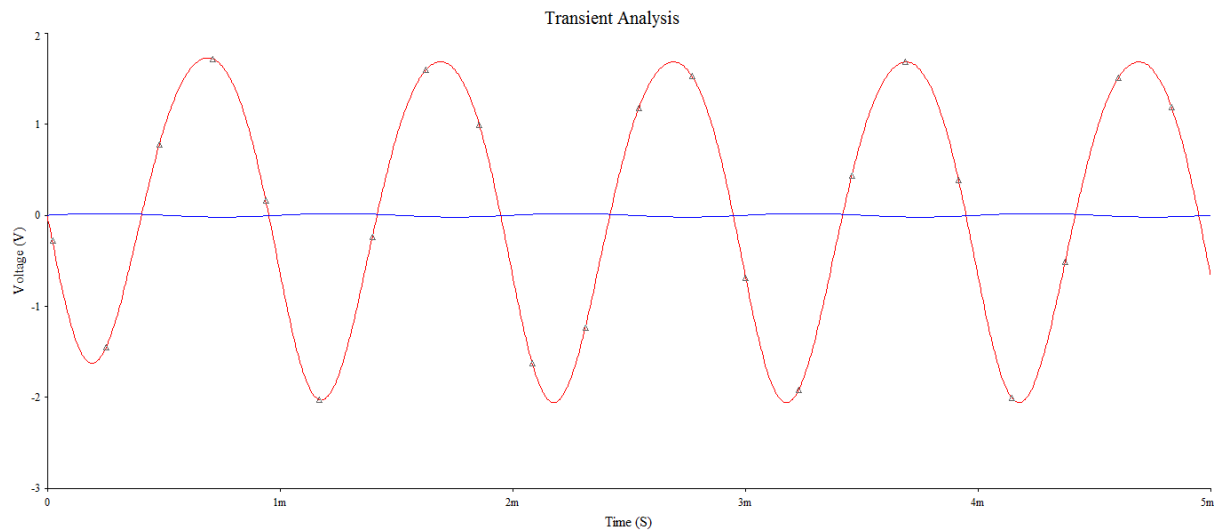
**AIM:** To design a common emitter amplifier for the given Specifications and simulate the same circuit using Multisim.

**SOFTWARE USED:**  
Multisim V10.

**Circuit diagram:**

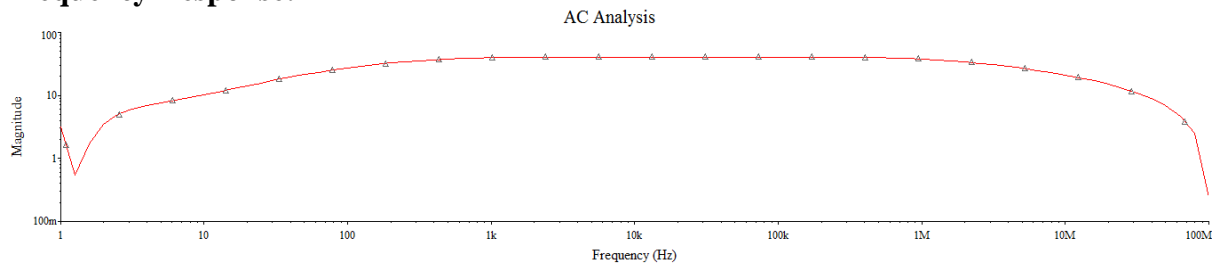


**Output waveform:**





## Frequency Response:



## Calculations:

1. Gain(dB ) =
2. Bandwidth =

## PROCEDURE:

1. Connect the circuit diagram as shown in Figure.
2. Apply a sine wave of frequency 1kHz, 20mv (p) amplitude as a input signal
3. Select simulate and then click on RUN.
4. Observe the amplified signal using oscilloscope.
5. For frequency response.....  
Simulate → Analyses → AC analysis
6. Enter start and stop frequencies
7. Write the expression for gain and click on simulate.
8. Now observe the magnitude and phase response of amplifier.
9. From magnitude response calculate gain and bandwidth.

## Result:

CE amplifier is designed and simulated using multisim.

## VIVA QUESTIONS:

1. Why CE configuration is most widely used?
2. Draw the equivalent Circuit of C.E
3. What is the Current Gain, voltage gain, i/p and o/p impedance in CE?.
4. Relation between 'α' and 'β' and γ
5. Give the condition to operate the given Transistor in active, saturation & Cut-off Regions
6. What is Emitter Efficiency?

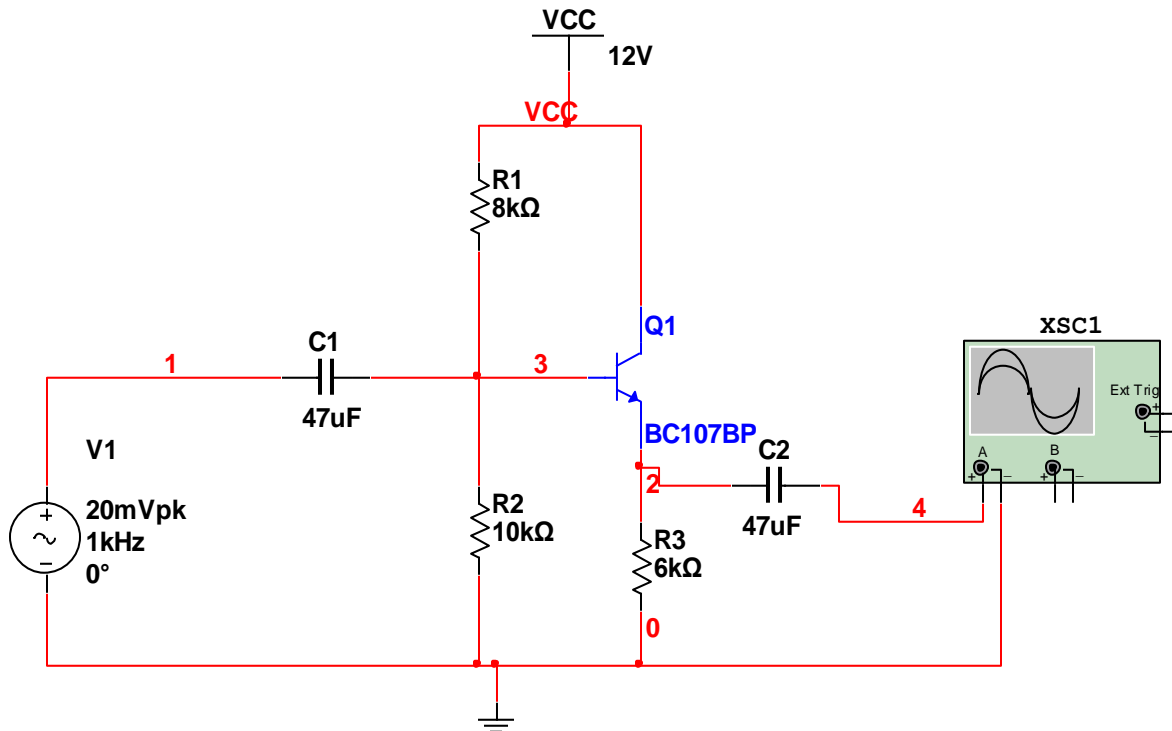
## 7. COMMON COLLECTOR AMPLIFIER

**AIM:** To design a common collector amplifier for the given Specifications and simulate the same circuit using Multisim.

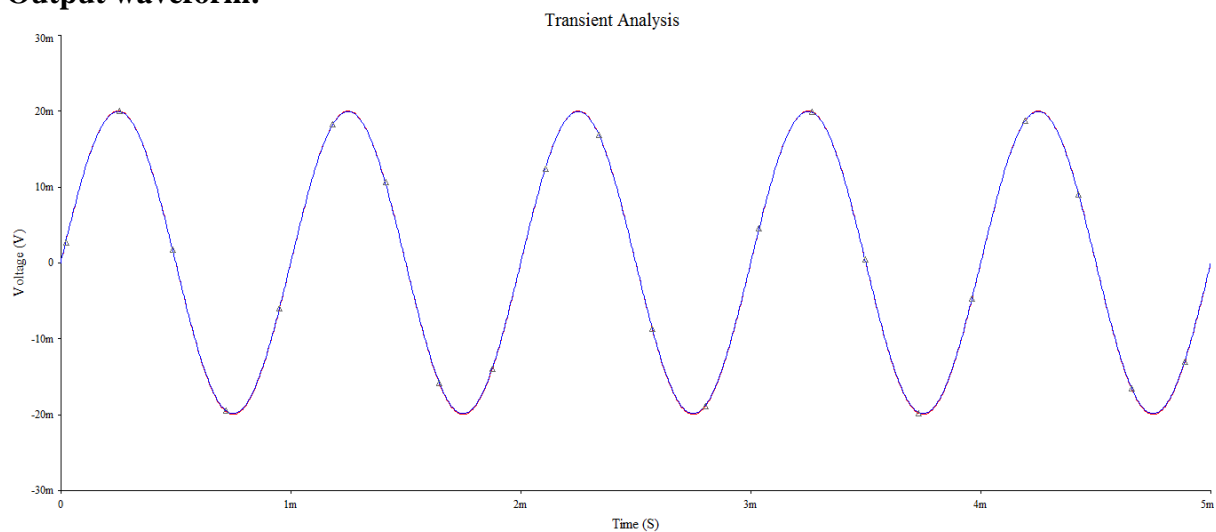
**SOFTWARE USED:**

Multisim V10.

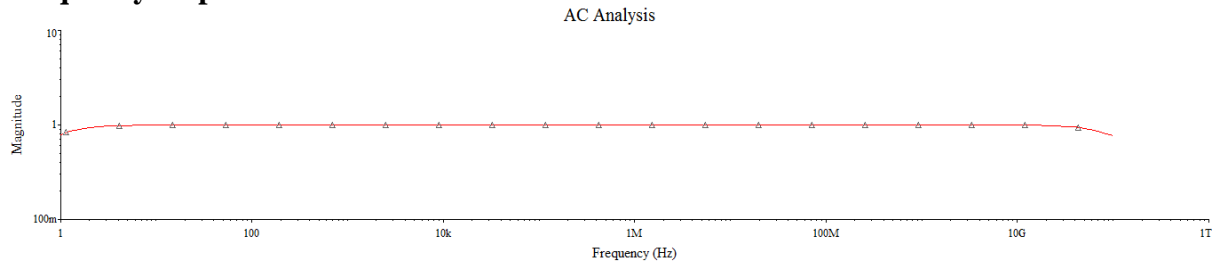
**Circuit diagram:**



**Output waveform:**



## Frequency response



### Calculations:

1. Gain(dB ) =
2. Bandwidth =

### PROCEDURE:

1. Connect the circuit diagram as shown in Figure.
2. Apply a sine wave of frequency 1kHz, 20mv (p) amplitude as a input signal
3. Select simulate and then click on RUN.
4. Observe the amplified signal using oscilloscope.
5. For frequency response.....  
Simulate → Analyses → AC analysis
6. Enter start and stop frequencies
7. Write the expression for gain and click on simulate.
8. Now observe the magnitude and phase response of amplifier.
9. From magnitude response calculate gain and bandwidth.

### Result:

CC amplifier is designed and simulated using multisim.

### Viva Questions:

1. Why CC amplifier is known as emitter follower?
2. Mention the applications of CC amplifier?
3. What are the differences between CE, CB and CC amplifier?
4. What are the basic characteristics of a CC amplifier?

## 8. ASTABLE MULTIVIBRATOR

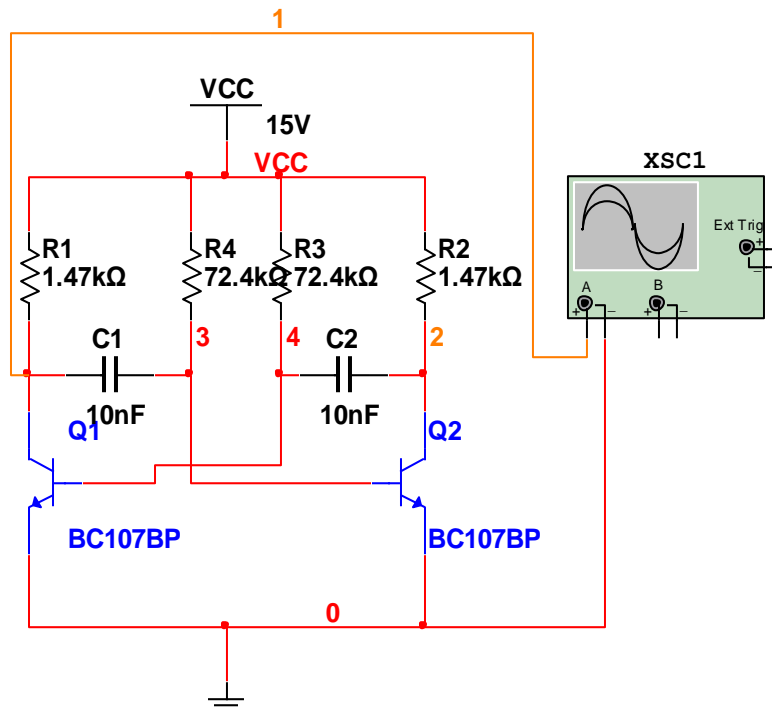
### Aim :-

To design an Astable Multivibrator to generate a Square wave of 1KHz frequency and simulate using Multisim software.

### SOFTWARE USED:

Multisim V10.

### Circuit diagram:



### Design:

The period T is given by

$$T = T_1 + T_2 = 0.69 (R_1 C_1 + R_2 C_2)$$

For symmetrical circuit with  $R_1 = R_2 = R$  &  $C_1 = C_2 = C$

$$T = 1.38 RC$$

$$10^{-3} = 1.38 \times 10^{-9} \times R$$

$$R = 724 K\Omega \text{ (When } c=1nf \text{);}$$

$$R = (10^{-3}) / 1.38 \times 10 \times 10^{-9}$$

$$= 72.4 K \Omega \text{ (where } c=10nf \text{)}$$

$$R = 7.24 K \Omega \text{ (where } c=100nf \text{)}$$

Let  $V_{cc} = 15V$ ,  $h_{fe} = 51$  (for BC 107)

$$V_{be \text{ sat}} = 0.7V, V_{ce \text{ sat}} = 0.3 V$$

Choose  $I_{c \text{ max}} = 10 \text{ mA}$

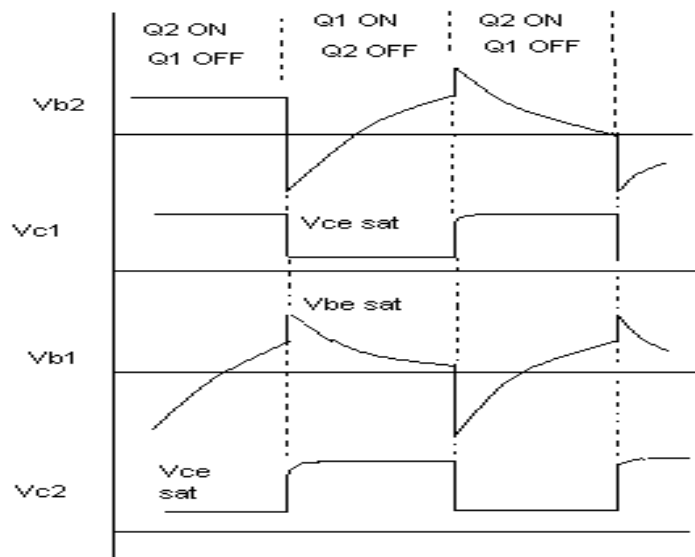
$$RC = (V_{CC} - V_{CESat}) / I_{Cmax}$$

$$= (15 - 0.3) / (10 \times 10^{-3}) = 1.47\text{K}\Omega$$

**Procedure:**

1. Connect the circuit as shown in figure using Multisim.
2. Apply the supply voltage  $V_{cc} = 15\text{V}$
3. Calculate the pulse width (T) of the Astable O/P with the selected values of R & C on the CRO. See that CRO is in DC mode.
4. Connect the CRO channel-1 to the collector and base of the Transistor Q1&Q2..
5. Measure the pulse width and verify with the theoretical value.
6. Obtain waveforms at different points like VB1, VB2, VC1 & VC2.

**Model waveforms:**



**Result :** An Astable Multivibrator is designed and simulated using Multisim software.

Viva Questions:

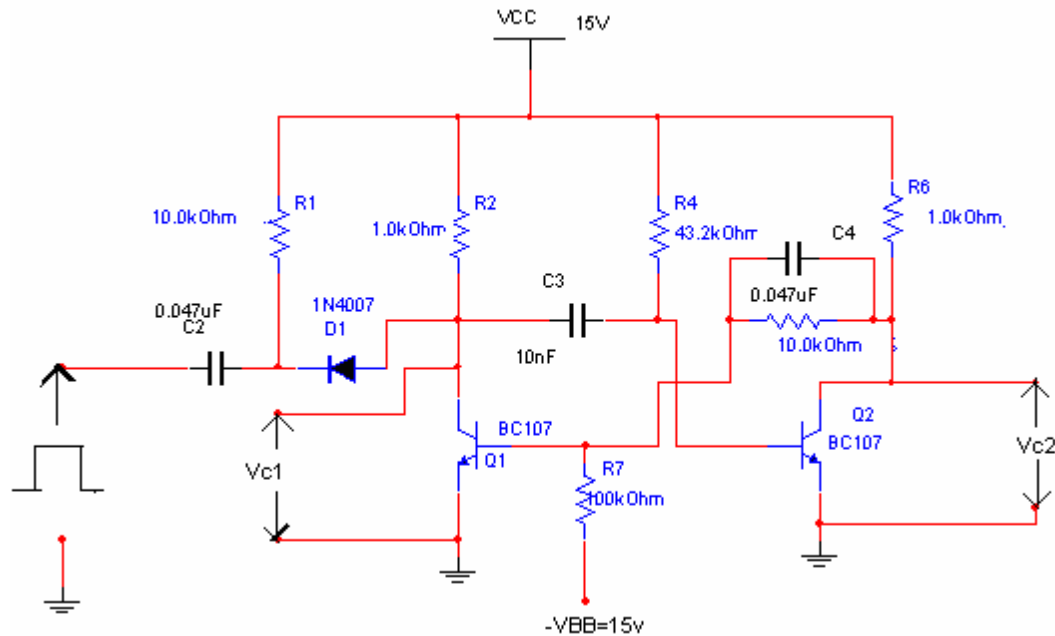
1. What is a multivibrator?
2. How many types of multivibrators exist?
3. What are the states?
4. What are the applications of Astable multi?
5. What is the significance of Astale multi?

## 10.MONOSTABLE MULTIVIBRATOR

**Aim :** To design a monostable multivibrator for the Pulse width of 0.03mSec and simulate using Multisim.

**SOFTWARE USED:**Multisim

**Circuit Diagram:-**



**Design:-**

To design a monostable multivibrator for the Pulse width of 0.03mSec.

Choose  $I_{Cmax} = 15mA$ ,  $V_{CC} = 15V$ ,  $V_{BB} = 15V$ ,  $R_1 = 10K\Omega$ .

$$T = RC \ln 2$$

$$T = 0.69 RC$$

Choose  $C = 10nf$

$$0.3 \times 10^{-3}Sec = 0.69 \times R \times 10 \times 10^{-9}$$

$$R = 43.47 K\Omega.$$

$$R_c = (V_{cc} - V_{ce sat}) / I_c \text{ max.}$$

$$R_c = (15 - 0.2) / 15mA = 1k\Omega$$

For more margin Given,

$$V_{B1} = -1.185$$

$$V_{B1} = -V_{BB} R_1 / (R_1 + R_2) + V_{ce sat} R_2 / (R_1 + R_2)$$

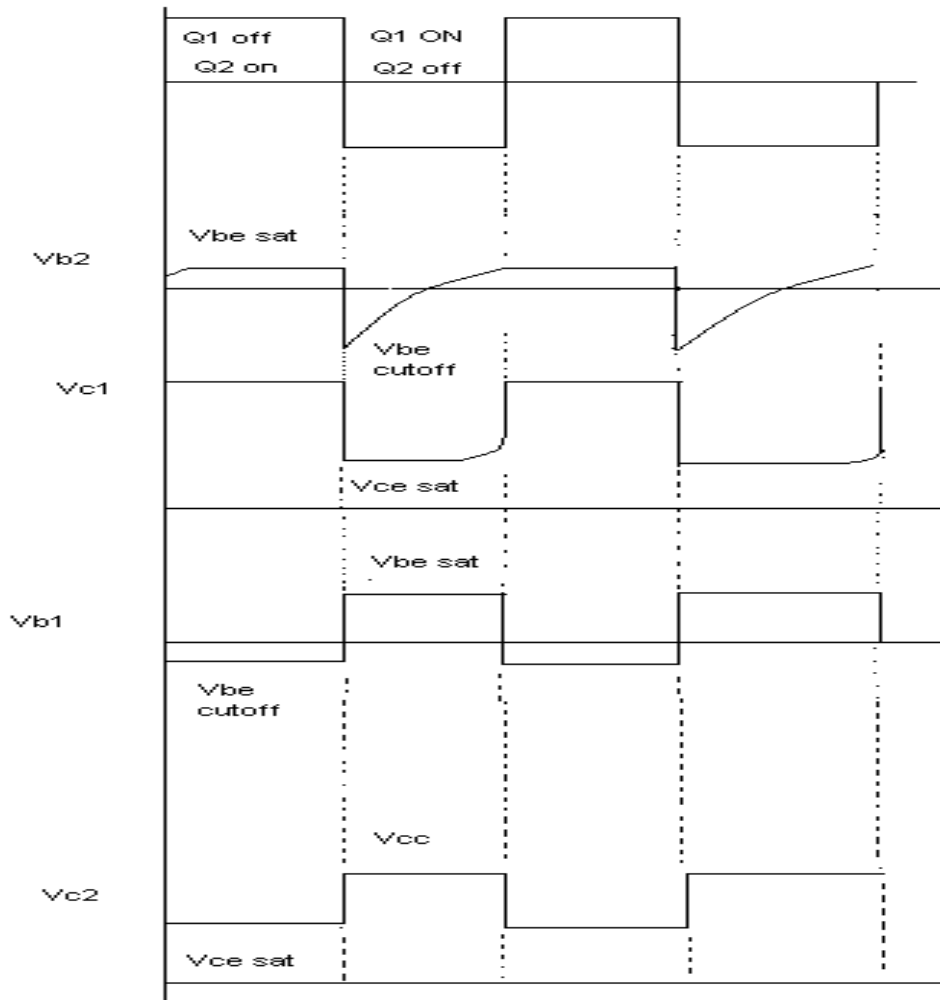
$$-1.18 = (-15 R_1 + 0.2 R_2) / (R_1 + R_2)$$

Given  $R_1 = 10 K\Omega$

We will  $R_2 = 100K\Omega$

**Procedure:**

1. Wire the circuit as shown in the circuit diagram.
2. Calculate the pulse width (T) of the Monostable O/P with the selected values of R & C on the CRO. See that CRO is in DC mode.
3. Select the triggering pulse such that the frequency is less than  $1/T$  Apply the triggering input to the circuit and to the CRO's channel 1 .
4. Connect the CRO channel-2 to the collector and base of the TransisterQ1&Q2..
5. Adjust the triggering pulse frequency to get stable pulse on the CRO and now measure the pulse width and verify with the theoretical value.
6. Obtain waveforms at different points like VB1, VB2, VC1 & VC2.
7. Repeat the experiment for different combinations of R & C (C = 1nf, 100nf). Calculate R for same value of T = 0.3 mSec.



**Result :**

A collector coupled Monostable Multivibrator is designed, the waveforms are observed using Multisim.

**Viva Questions:**

1. What is a multivibrator?
2. How many types of multivibrators exist?
3. What are the states?
4. What are the applications of Monostable multi?
5. What is the significance of Monostable multi?



## 10. SCHMITT TRIGGER

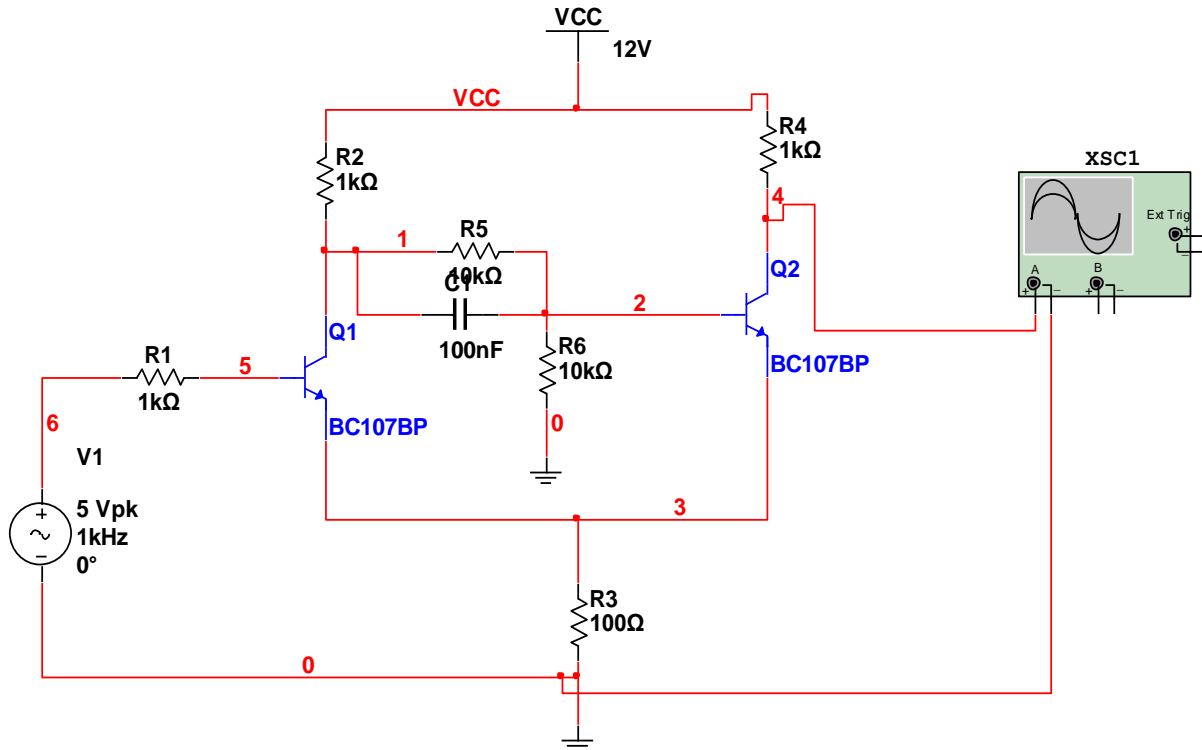
### Aim:-

To study the operation of Schmitt trigger circuit using Multisim and find the UTP and LTP voltages & compare with the theoretical values.

### SOFTWARE USED:

Multisim V10.

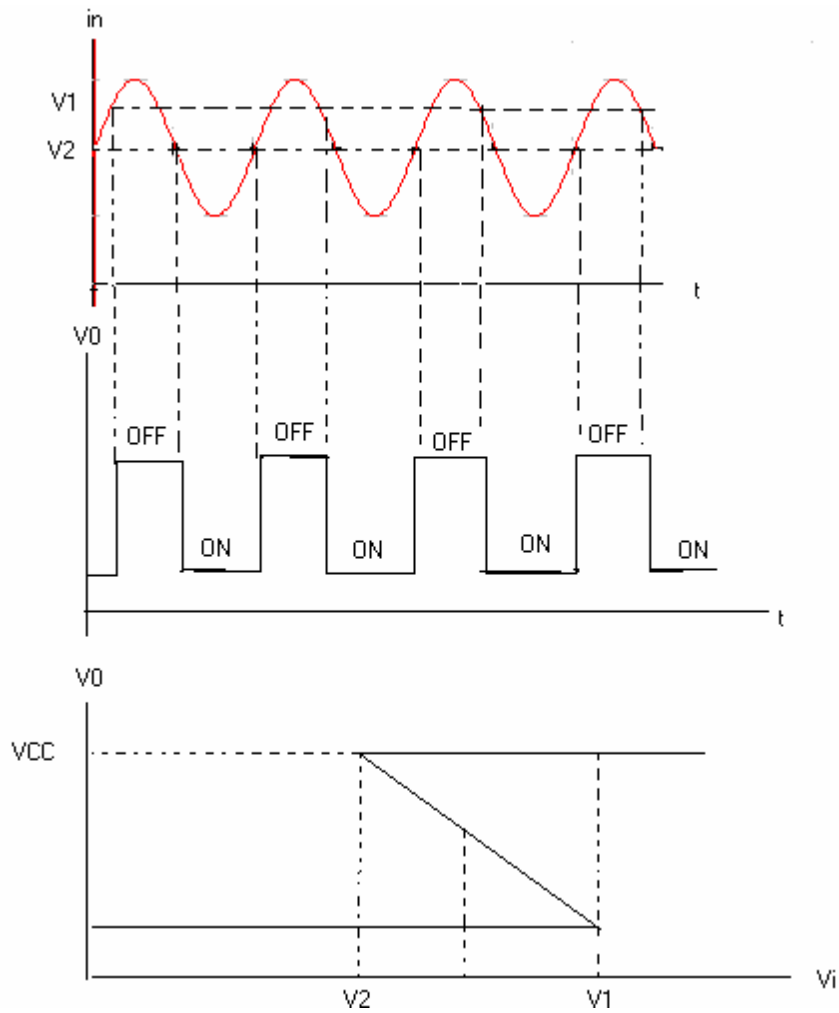
### Circuit diagram:



### Procedure:

1. Connect the circuit on the bread board as per circuit diagram
2. Keep the peak to peak input voltage 10V using function generator
3. Keep the Vcc voltage at 12V constant using regulated power supply
4. Observe the output waveform in the CRO
5. Plot the values and draw the graph
6. Calculate the upper triggering point & lower triggering point

### Model Waveform:



### Result:

The operation of Schmitt trigger is verified and the UTP & LTP voltages are measured using Multisim.

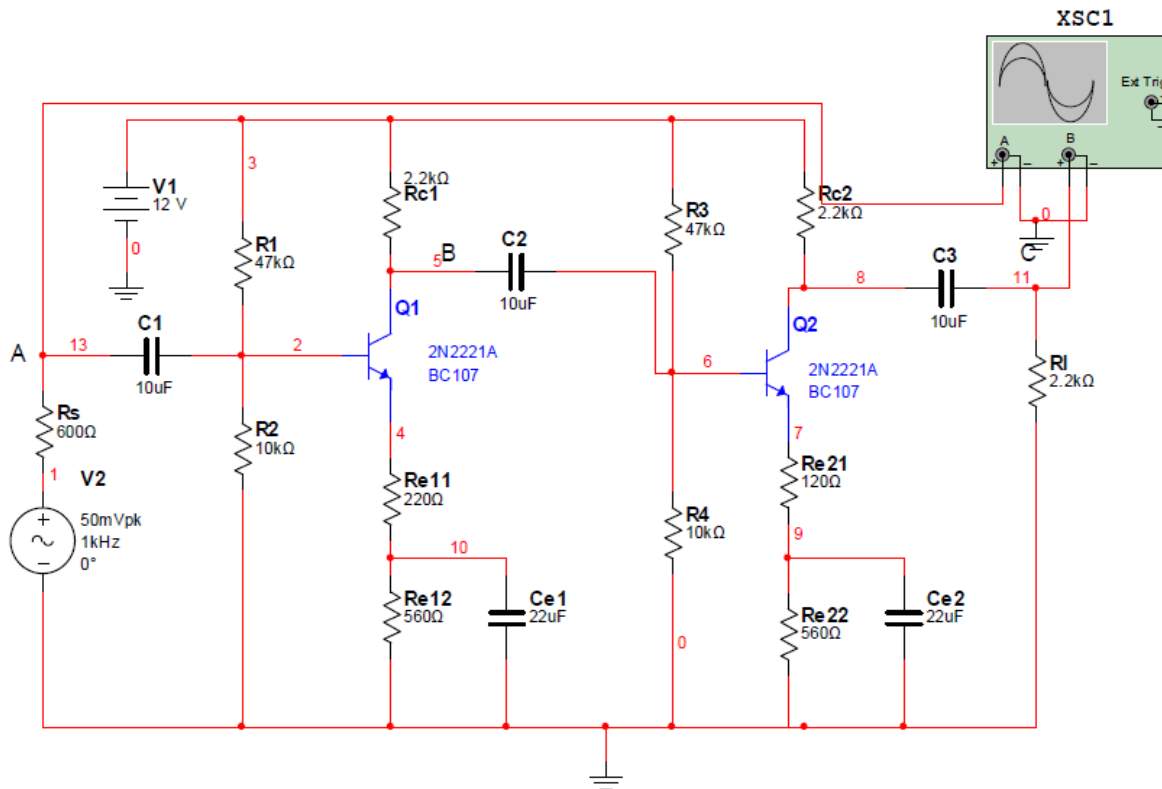
1. What is a multivibrator?
2. How many types of multivibrators exist?
3. What are the states?
4. What are the applications of Schmitt trigger?
5. What is the significance of Schmitt trigger??

## 11. TWO STAGE RC-COUPLED AMPLIFIER

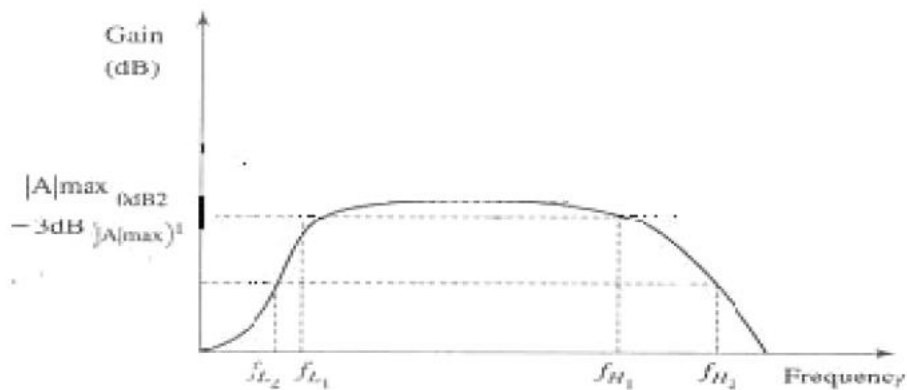
**AIM:** To design a two stage RC coupled amplifier for the given Specifications and simulate the same circuit using Multisim.

**SOFTWARE USED:** Multisim V10.

**Circuit diagram:**



**Frequency Response:**



**Calculations:**

1. Gain(dB ) =
2. Bandwidth =

**PROCEDURE:**

1. Connect the circuit diagram as shown in Figure.
2. Apply a sine wave of frequency 1kHz,20mv (p) amplitude as a input signal
3. Select simulate and then click on RUN.
4. Observe the amplified signal using oscilloscope.
5. For frequency response.....  
Simulate→Analyses→AC analysis
6. Enter start and stop frequencies
7. Write the expression for gain and click on simulate.
8. Now observe the magnitude and phase response of amplifier.
9. From magnitude response calculate gain and bandwidth.

**Result:**

Two stage RC coupled amplifier is designed and simulated using Mu

**Viva Questions:**

1. What is a Multistage amplifier?
2. What is the need of coupling device?
3. What are the applications of RC coupled amplifier?
4. What are the advantages and disadvantages of RC coupled multistage amplifier?
5. What is the effect of multistage on gain and bandwidth of the amplifier?

## CYCLE-II: (Hardware experiments)

### Exp-1: Volt-Ampere Characteristics of PN junction Diode and LED

#### Objective:

1. To plot Volt-Ampere Characteristics of Silicon P-N Junction Diode.
2. To find cut-in Voltage for Silicon P-N Junction diode.
3. To find static and dynamic resistances for P-N Junction diode.
4. To Plot the V-I Characteristics of LED and analyze the relation of current flowing through the diode and the intensity of glow.

#### Apparatus:

S.No	Apparatus	Type	Range	Quantity
01	PN Junction diode	1N4007		01
02	Resistance		470 $\Omega$ ,1K $\Omega$	01
03	Regulated Power supply		(0-30V)	01
04	Ammeter		(0-100mA),(0-100 $\mu$ A)	01
05	Voltmeter		(0-2V),(0-30V)	01
06	LED			01
07	Breadboard and Wires			

#### Theory:

The semi conductor diode is created by simply joining an n-type and a p-type material together nothing more just the joining of one material with a majority carrier of electrons to one with a majority carrier of holes.

The P-N junction supports uni-directional current flow. If +ve terminal of the input supply is connected to anode (P-side) and -ve terminal of the input supply is connected to cathode (N-side), then diode is said to be forward biased. In this condition the height of the potential barrier at the junction is lowered by an amount equal to given forward biasing voltage. Both the holes from p-side and electrons from n-side cross the junction simultaneously and constitute a forward

current( **injected minority current** – due to holes crossing the junction and entering N-side of the diode, due to electrons crossing the junction and entering P-side of the diode).

Assuming current flowing through the diode to be very large, the diode can be approximated as short-circuited switch. If –ve terminal of the input supply is connected to anode (p-side) and +ve terminal of the input supply is connected to cathode (n-side) then the diode is said to be reverse biased. In this condition an amount equal to reverse biasing voltage increases the height of the potential barrier at the junction. Both the holes on p-side and electrons on n-side tend to move away from the junction thereby increasing the depleted region. However the process cannot continue indefinitely, thus a small current called **reverse saturation current** continues to flow in the diode. This small current is due to thermally generated carriers. Assuming current flowing through the diode to be negligible, the diode can be approximated as an open circuited switch. The volt-ampere characteristics of a diode explained by following equation:

$$I = I_0 \left( e^{\frac{V}{\eta V_T}} - 1 \right)$$

I=current flowing in the diode

I<sub>0</sub>=reverse saturation current

V=voltage applied to the diode

$$V_T = \text{volt-equivalent of temperature} = \frac{KT}{q} = \frac{T}{11,600} = 26mV \text{ at room temp}$$

$$\eta = 1 \text{ (for Ge)}$$

$$\eta = 2 \text{ (for Si)}$$

It is observed that Ge diode has smaller cut-in-voltage when compared to Si diode. The reverse saturation current in Ge diode is larger in magnitude when compared to silicon diode.

### **LED-Theory:**

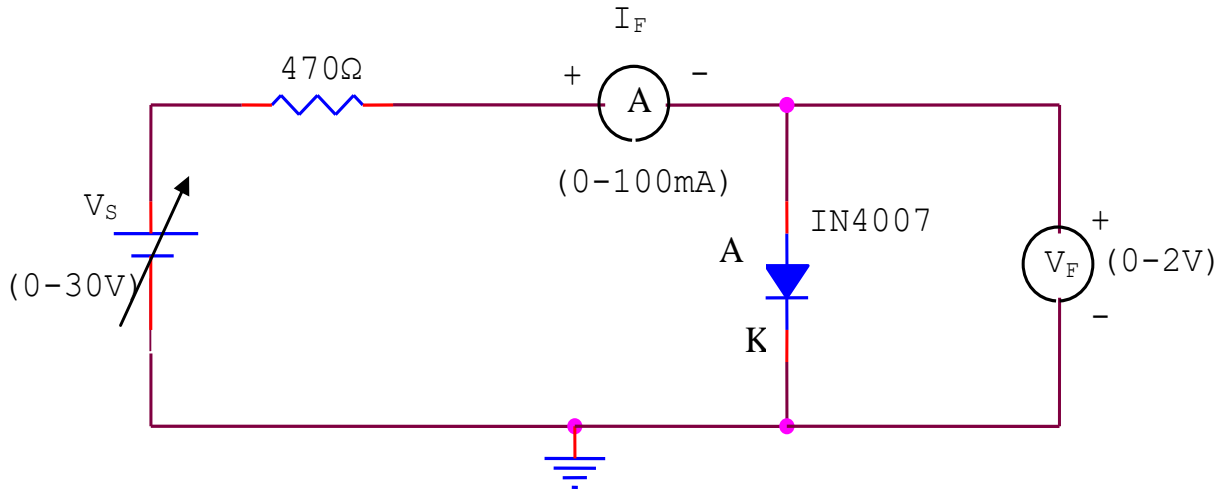
LEDs emit light when an electric current passes through them.

LED is connected in the circuit as shown in figure. LED operates only in forward biased condition. Under forward bias condition the anode is connected to the positive terminal and the cathode is connected to the negative terminal of the battery. It is like a normal pn junction diode except the basic semiconductor material is GaAs or InP which is responsible for the color of the

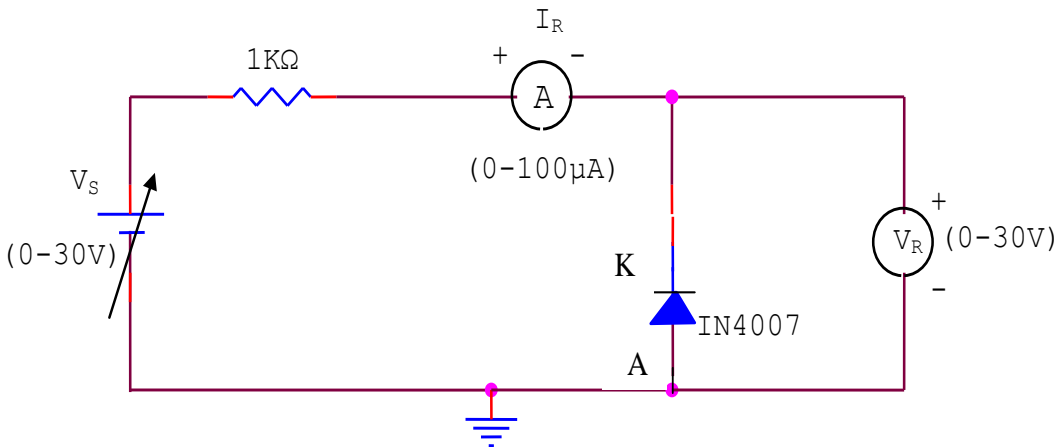
light. When it is forward biased the holes moves from p to n and electrons flow from n to p. In the junction the carriers recombine with each other and released the energy in the form of light. Thus LED emits light under forward biased condition. Under reverse biased condition, there is no recombination due to majority carriers, so there is no emission of light.

**Circuit Diagrams:**

**a) Forward Bias of PN Diode**



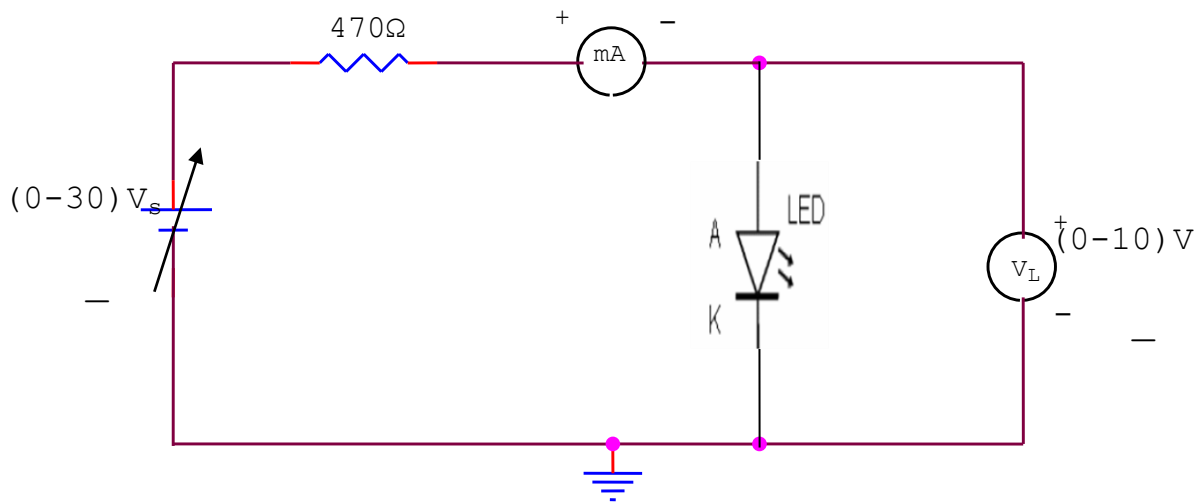
**b) Reverse Bias of PN diode:**



**c) Forward biased LED:**

(0-100 mA)

—



**Procedure:**

**a) Forward biased condition-PN diode**

1. Connect the PN Junction diode in forward bias i.e Anode is connected to positive of the power supply and cathode is connected to negative of the power supply .
2. Use a Regulated power supply of range (0-30)V and a series resistance of 470Ω
3. By varying the input voltage in steps of 0.1V, note down corresponding Ammeter readings.( $I_F$ ) and voltmeter reading.
4. Plot the graph between forward voltage ( $V_F$ ) and forward current ( $I_F$ ).

**b) Reverse biased condition-PN Diode**

1. Connect the PN Junction diode in Reverse bias i.e; anode is connected to negative of the power supply and cathode is connected to positive of the power supply.
2. Use a Regulated power supply of range (0-30)V and a series resistance of 1KΩ
3. By varying the input voltage vary voltage ( $V_R$ ) in steps of 1V and note down corresponding Ammeter readings.( $I_R$ )
4. Plot the graph between Reverse voltage ( $V_R$ ) and Reverse current ( $I_R$ ).

**c) V-I characteristics of LED**

1. Connections must be made as per the circuit diagram.
2. Varying the source voltage in steps of 0.1V note down the corresponding current and the voltage readings.
3. At the same time the glow intensity of the light emitting diode is also to be noted.
4. Plot the graph between voltage and current for forward bias



**Observation Table of PN diode:**

1. Table for observations of forward biased PN diode

S.No	$V_s$ (Volts)	$V_F$ (Volts)	$I_F$ (mA)

2. Table for observations of Reverse biased PN diode

S.No	$V_S$ (Volts)	$V_R$ (Volts)	$I_R$ ( $\mu A$ )

**Table for observations of Forward biased LED:**

S.No	$V_{S(V)}$	$V_{L(V)}$	$I_{f(mA)}$	Intensity of glow

**Model Graph- VI characteristics of PN Diode:**

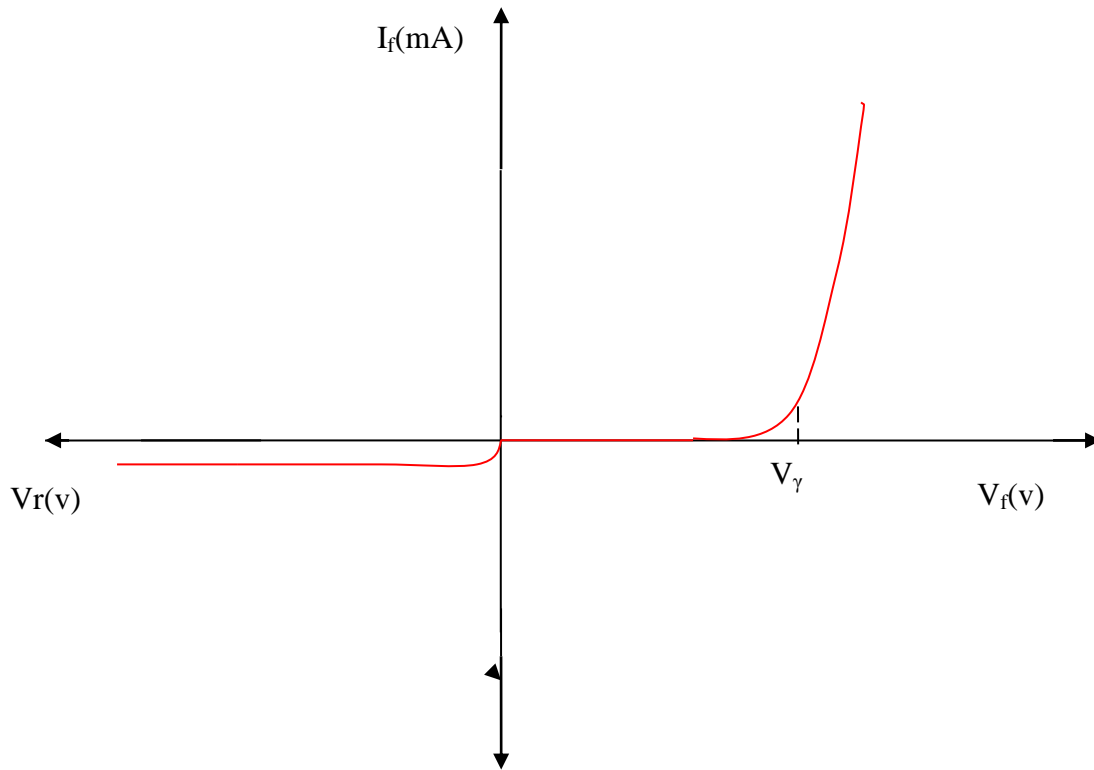


Fig: V-I Characteristics of PN diode

**Model Graph- Forward bias of LED:**

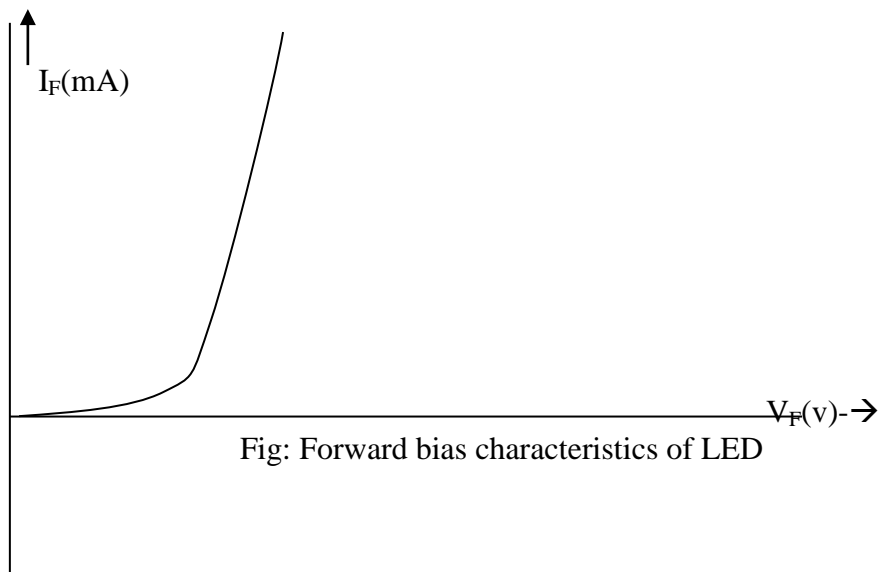


Fig: Forward bias characteristics of LED

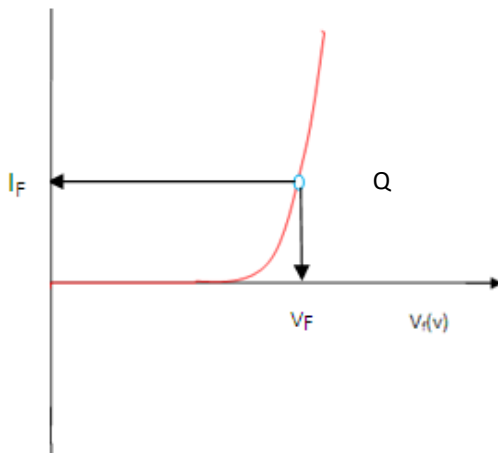
## Calculations- PN Diode:

**Static Resistance:** To find the forward static resistance locate a point on characteristic curve obtained from the forward bias characteristics which is called operating point Q and draw a line onto the X-axis and Y-axis to obtain  $V_F$  and  $I_F$ . Calculate static forward resistance using the formulae

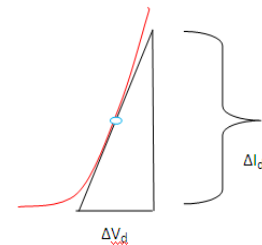
$$\text{Static forward Resistance } R_{DC} = \frac{V_F}{I_F} \Omega \text{ at Q-point.}$$

**Dynamic Resistance:** The dc resistance of a diode is independent of the shape of the characteristic in the region surrounding the point of interest. If a sinusoidal input is applied rather than a dc input, the varying input will move the instantaneous operating point up and down a region of the characteristics and thus defines a specific change in current and voltage. To find the ac or dynamic resistance draw a straight line drawn tangent to the curve through the Q-point as shown in the figure will define a particular change in voltage and current that can be used to determine the ac or dynamic resistance for this region of the diode characteristics.

$$\text{Dynamic Resistance } r_d = \frac{\Delta V_d}{\Delta I_d} \Omega \text{ at Q-point}$$



Forward characteristics



## Calculating LED resistor value

LED must have a resistor connected in series to limit the current through the LED; otherwise it will burn out almost instantly. The resistor value, R is given by:

$$R = \frac{(V_S - V_L)}{I}$$

$V_S$  = supply voltage

$V_L$  = LED voltage (usually 2V, but 4V for blue and white LEDs)

I = LED current (e.g. 20mA), this must be less than the maximum permitted

### For example:

If the supply voltage  $V_S = 10V$ , and you have a red LED ( $V_L = 2V$ ), requiring a current  $I = 20mA$

$R = (10V - 2V) / 0.02A = 400 \Omega$ , so choose  $470\Omega$  (the nearest standard value which is greater).

### Precautions:

1. While doing the experiment do not exceed the ratings of the diode. This may lead to damage of the diode.
2. Connect voltmeter and Ammeter in correct polarities as shown in the circuit diagram.
3. Do not switch **ON** the power supply unless you have checked the circuit connections as per the circuit diagram.

### Result:

1. Cut in voltage = ..... V
2. Static forward resistance = .....  $\Omega$
3. ac or Dynamic resistance = .....  $\Omega$
4. LED resistance = .....  $\Omega$

### VIVA QUESTIONS:

1. When diode acts like ideal switch?
2. What is the cut in voltage? Give typical values for Ge and Si.
3. What is reverse saturation current?
4. What is Dynamic and static resistance?
5. What is V-I characteristics equation?
6. Define potential barrier.

7. Define doping.
8. What is the effect of temperature on  $I_{co}$ .
9. Define a Q point.
10. Explain how the diode can act as a capacitor.

**EXP-2: V-I Characteristics and Regulation Characteristics of Zener Diode**  
**Objectives:**

1. To plot Volt-Ampere Characteristics of Zener Diode in reverse bias.
2. To find Zener Breakdown Voltage in reverse biased condition.
3. To find load regulation characteristics of Zener voltage regulator

**Apparatus:**

S.No	Apparatus	Type	Range	Quantity
01	Zener diode	IMZ 5.1V		01
02	Resistance		470Ω	01
03	Regulated Power supply		(0-30V)	01
04	Ammeter		(0-100mA)	02
05	Voltmeter		(0-10V)	01
06	Decade Resistance Box		(0-10K)	01
07	Breadboard and Wires			

**Theory:**

An ideal P-N Junction diode does not conduct in reverse biased condition. A **zener diode** conducts excellently even in reverse biased condition. These diodes operate at a precise value of voltage called break down voltage. A **zener diode** when forward biased behaves like an ordinary P-N junction diode.

A **zener diode** when reverse biased can either undergo **avalanche break down** or **zener break down**.

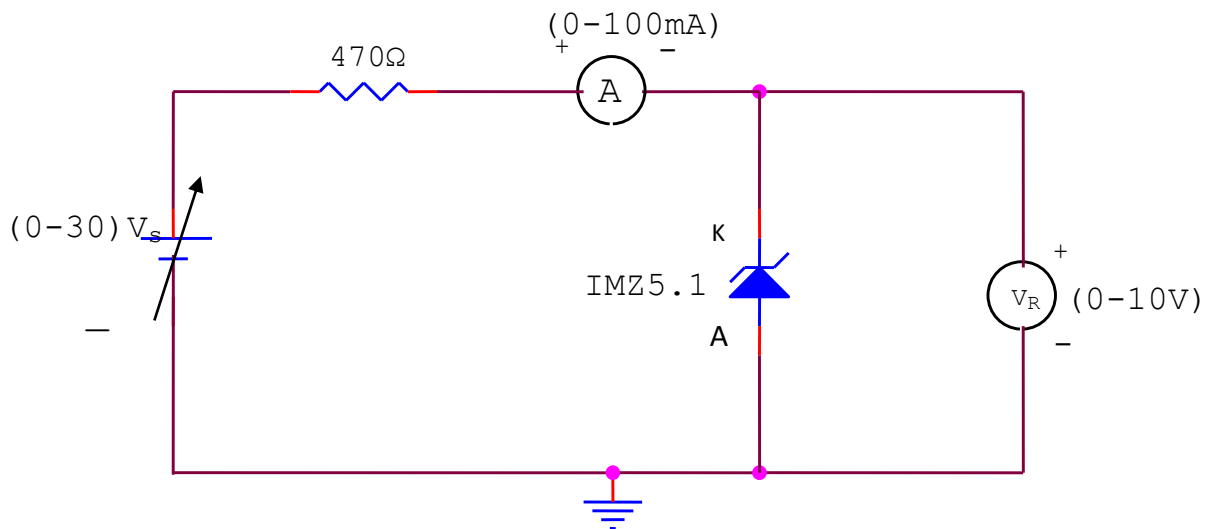
**Avalanche break down:**-If both p-side and n-side of the diode are lightly doped, depletion region at the junction widens. Application of a very large electric field at the junction may

rupture covalent bonding between electrons. Such rupture leads to the generation of a large number of charge carriers resulting in **avalanche multiplication**.

**Zener break down:**-If both p-side and n-side of the diode are heavily doped, depletion region at the junction reduces. Application of even a small voltage at the junction ruptures covalent bonding and generates large number of charge carriers. Such sudden increase in the number of charge carriers results in **zener mechanism**.

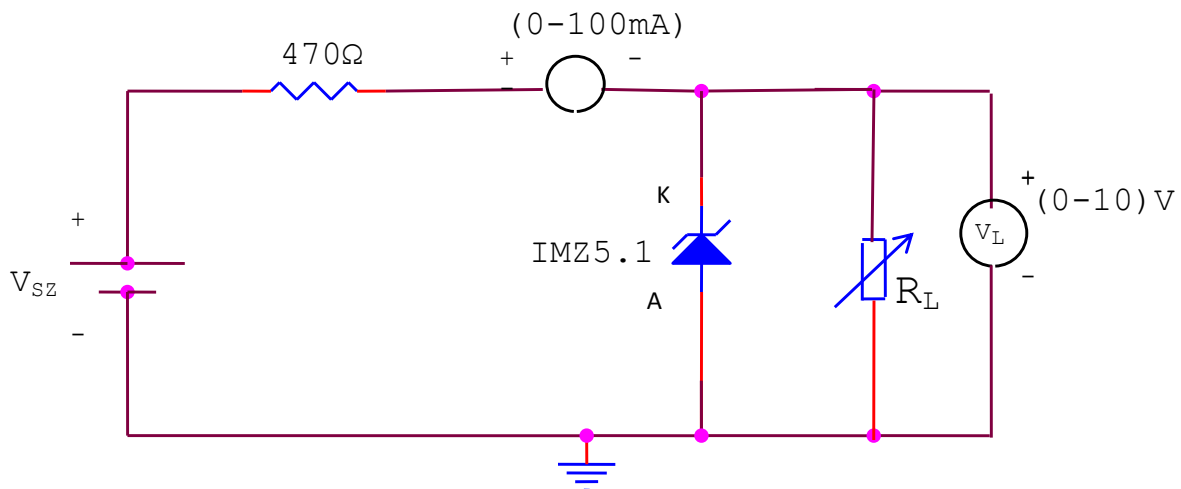
**Circuit Diagram:**

Reverse Biased Zener diode



Zener diode as shunt Voltage Regulator

A



**Procedure:**

**To plot V-I characteristics of Zener diode in reverse bias condition and to find Zener breakdown voltage:**

1. Connect the Zener diode in Reverse bias i.e; anode is connected to negative of the power supply and cathode is connected to positive of the power supply as in circuit.
2. Vary the input voltage in steps of 1V and note down reverse voltage( $V_R$ ) and the corresponding values of reverse current ( $I_R$ ).
3. Plot the graph between reverse voltage ( $V_R$ ) and the reverse current ( $I_R$ ).

**To plot the load regulation characteristics of the Zener voltage regulator.**

1. Connect the Zener diode in Reverse bias i.e; anode is connected to negative of the power supply and cathode is connected to positive of the power supply as in circuit.
2. In finding load regulation , input voltage ( $V_{sz}$ ) is kept constant i.e source voltage is chosen as a voltage at which Zener voltage  $V_Z$  is remaining constant while the current is increasing( $V_{sz}$  from 1<sup>st</sup> circuit characteristics)
3. Measure  $V_{NL}$  (No load voltage) by opening the load resistance.
4. Connect the load resistance, and vary the load resistance from 1100 $\Omega$  to 100 $\Omega$  in steps of 100 $\Omega$  and note down the readings of  $V_L$  and  $I_Z$
5. Calculate % Regulation by using the formula given below.

$$\% \text{ Regulation} = \frac{V_{NL} - V_L}{V_L} \times 100$$



**Observation Table:**

1. To plot V-I characteristics

S.No	$V_s(V)$	$V_R(V)$	$I_R(mA)$

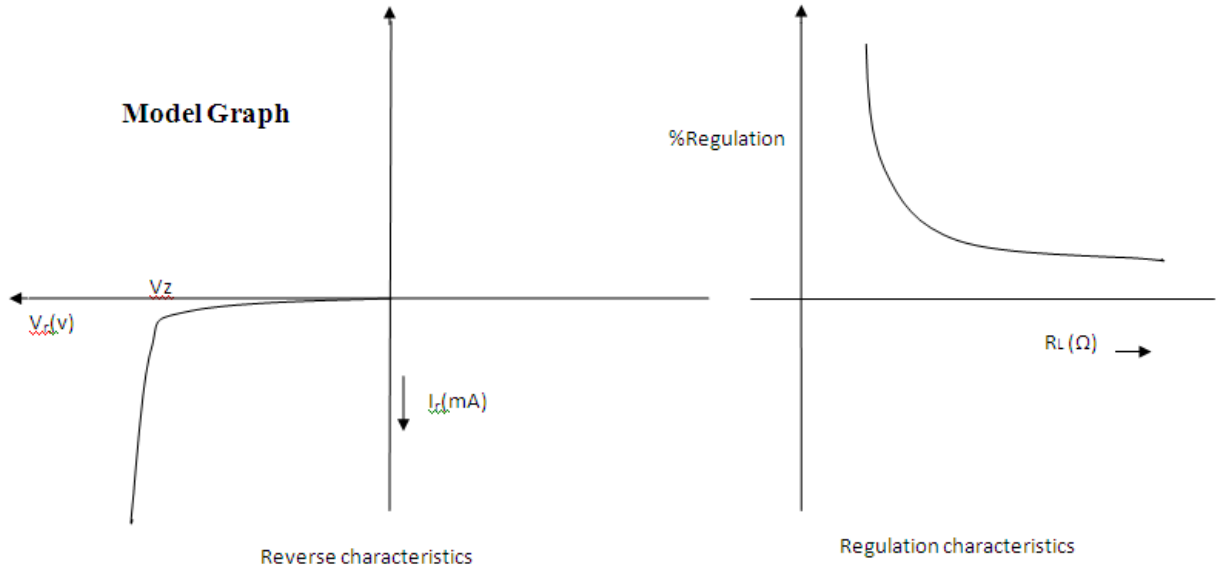
1. Find load regulation characteristics

$$V_{NL} = \underline{\hspace{10em}}$$

S.No	$R_L$	$I_Z(mA)$	$V_L(V)$	% Regulation
	<b>1100</b> <b>To</b> <b>100</b> (in steps of 100)			

**Note:** Keeping the input voltage constant if the load resistance is increased zener current increases so as to make the load voltage to remain constant.

## Model Graphs:



## Calculations from Graph

To find Zener breakdown voltage

1. In the reverse characteristics of Zener diode observe the voltage at which the reverse current is abnormally increasing while the reverse voltage remain constant.
2. That particular reverse voltage is called the breakdown voltage of the Zener diode

## Precautions:

1. While doing the experiment do not exceed the ratings of the diode. This may lead to damage of the diode.
2. Connect voltmeter and Ammeter in correct polarities as shown in the circuit diagram.
3. Do not switch **ON** the power supply unless you have checked the circuit connections as per the circuit diagram.

## Result

1. The V-I characteristics of Zener diode were plotted and the Zener breakdown voltage was determined and is given as .....v
2. Load regulation characteristics were plotted.

### **VIVA QUESTIONS**

1. Difference between Zener and Avalanche breakdown.
2. What is the difference between zener and ordinary diode?
3. Draw equivalent circuit for Zener diode.
4. What is Breakdown voltage?
5. What are the applications of zener diode?
6. How zener acts as a regulator?

### Exp-3. Ripple Factor Calculation of Half Wave Rectifier

#### Objective

1. To plot input and output waveforms of the Half Wave Rectifier with and without Filter
2. To find ripple factor of Half Wave Rectifier with and without Filter
3. To plot input and output waveforms of the Full Wave Rectifier with and without Filter
4. To find ripple factor for Full Wave Rectifier with and without Filter

#### Apparatus

S.No	Apparatus	Type	Range	Quantity
01	Transformer	Center tapped	12-0-12V	01
02	Diode	IN4007		01
03	Resistance		1K $\Omega$	01
04	Capacitor		1000 $\mu$ F/25V	01
05	Multimeter		(0-20V)	01
06	CRO			01
07	Breadboard and Wires			

#### Theory

A device is capable of converting a sinusoidal input waveform into a unidirectional waveform with non zero average component is called a rectifier.

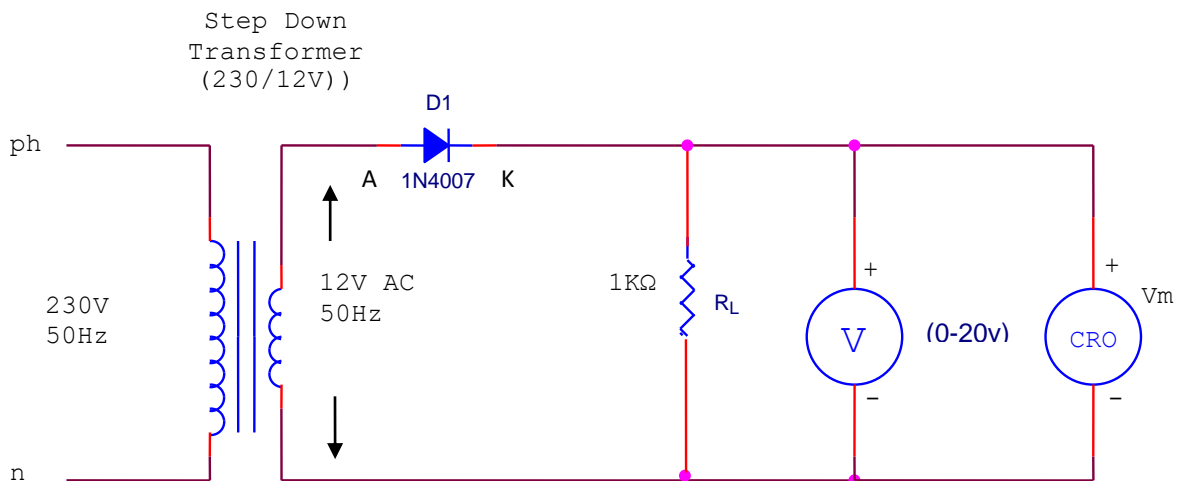
A practical half wave rectifier with a resistive load is shown in the circuit diagram. During the positive half cycle of the input the diode conducts and all the input voltage is dropped across  $R_L$ . During the negative half cycle the diode is reverse biased and it acts as almost open circuit so the output voltage is zero. The filter is simply a capacitor connected from the rectifier output to ground. The capacitor quickly charges at the beginning of a cycle and slowly discharges through  $R_L$  after the positive peak of the input voltage. The variation in the capacitor voltage due to charging and discharging is called ripple voltage. Generally, ripple is undesirable, thus the smaller the ripple, the better the filtering action.

### Theory- Full wave rectifier:

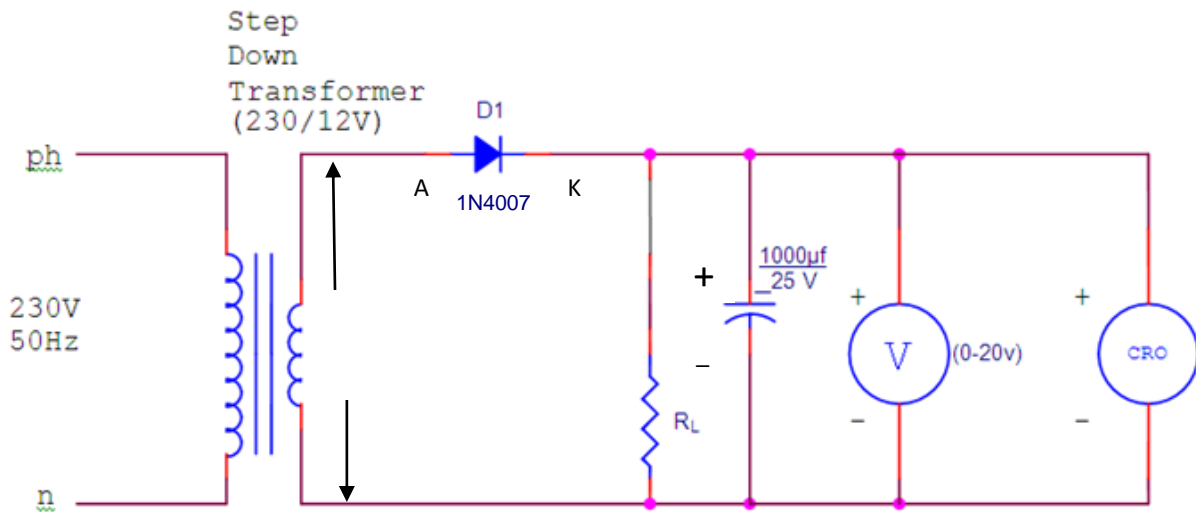
The full wave rectifier consists of two half wave rectifiers connected to a common load. One rectifies during positive half cycle of the input and the other rectifying the negative half cycle. The transformer supplies the two diodes (D1 and D2) with sinusoidal input voltages that are equal in magnitude but opposite in phase. During input positive half cycle, diode D1 is ON and diode D2 is OFF. During negative half cycle D1 is OFF and diode D2 is ON. Generally, ripple is undesirable, thus the smaller the ripple, the better the filtering action.

### Circuit Diagram

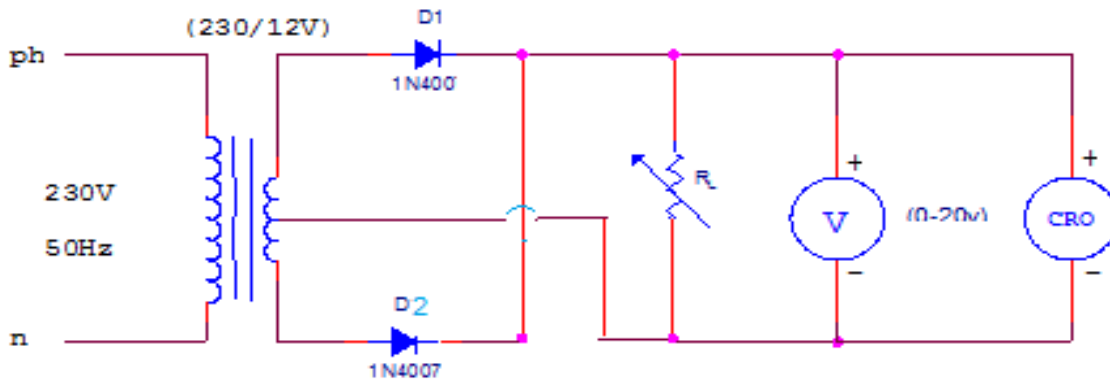
#### Halfwave Rectifier without filter



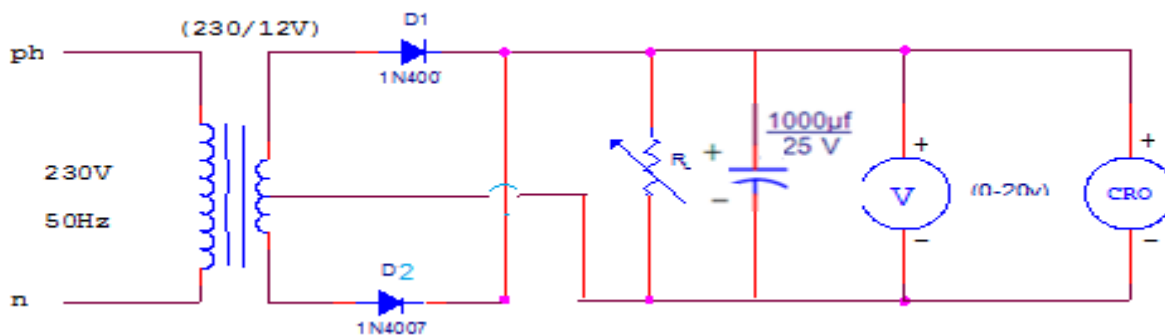
### Half wave Rectifier with Capacitor Filter



### Full wave Rectifier without Filter



### Full wave Rectifier with capacitor Filter



## CALCULATIONS:

### **Theoretical calculations for Ripple factor of Halfwave Rectifier:-**

#### **Without Filter:-**

$$V_{dc} = \frac{V_m}{\pi}$$
$$V_{rms} = \frac{V_m}{2}$$

$$\text{Ripple factor} = \sqrt{\left(\frac{V_{rms}}{V_{dc}}\right)^2} - 1 = 1.21$$

#### **With Capacitor Filter:-**

$$\text{Ripple factor} = \frac{1}{2\sqrt{3}fCR_L}$$

Where  $f = 50\text{Hz}$

$C = 1000\mu\text{F}$

$R_L = 1\text{K}\Omega$

### **Theoretical calculations for Ripple factor of Fullwave Rectifier:-**

#### **Without Filter:-**

$$V_{dc} = \frac{2V_m}{\pi}$$
$$V_{rms} = \frac{V_m}{\sqrt{2}}$$

$$\text{Ripple factor} = \sqrt{\left(\frac{V_{rms}}{V_{dc}}\right)^2} - 1 = 0.482$$

#### **With Capacitor Filter:-**

$$\text{Ripple factor} = \frac{1}{4\sqrt{3}fCR_L}$$

Where  $f = 50\text{Hz}$

$C = 1000\mu\text{F}$

$R_L = 1\text{K}\Omega$

## **Procedure :**

### **HWR Without Filter:**

1. Connections are made as per the circuit diagram of the rectifier without filter.
2. Connect the primary side of the transformer to ac mains and the secondary side to the rectifier input.
3. Note down the no load voltage before applying the load to the Circuit and by using the Multimeter, measure the ac input voltage of the rectifier and its frequency.
4. Now Vary the  $R_L$  in steps of  $100\Omega$  by varying the DRB from  $1100\Omega$  to  $100\Omega$  and note down the load voltage ( $V_L$ ) using the multimeter for each value of  $R_L$  and calculate the percentage regulation.
5. Measure the AC and DC voltage at the output of the rectifier for each value of  $R_L$  using Multimeter.
6. Now Observe the output waveform on CRO across  $R_L$  and find out value of  $V_m$ .
7. Now calculate  $V_{dc}$ ,  $V_{rms}$ , Ripple Factor and other parameters of half wave rectifier according to the given formulae.
8. Measure the amplitude and timeperiod of the transformer secondary(input waveform) by connecting CRO.
9. Feed the rectified output voltage to the CRO and measure the time period and amplitude of the waveform.

### **HWR With Capacitor Filter:**

1. Connections are made as per the circuit diagram of the rectifier with filter.
2. Connect the primary side of the transformer to ac mains and the secondary side to the rectifier input.
3. By the multimeter, measure the ac input voltage of the rectifier and, ac and dc voltage at the output of the rectifier.
4. Measure the amplitude and timeperiod of the transformer secondary(input waveform) by connecting CRO.
5. Feed the rectified output voltage to the CRO and measure the time period and amplitude of the waveform.

### **FWR Without Capacitor Filter:**

1. Connections are made as per the circuit diagram of the rectifier without filter.
2. Connect the primary side of the transformer to ac mains and the secondary side to the rectifier input.
3. By the multimeter, measure the ac input voltage of the rectifier and, ac and dc voltage at the output of the rectifier.
4. Measure the amplitude and time period of the transformer secondary(input waveform) by connecting CRO.



5. Feed the rectified output voltage to the CRO and measure the time period and amplitude of the Waveform.

**FWR With Capacitor Filter:**

1. Connections are made as per the circuit diagram of the rectifier with filter.
2. Connect the primary side of the transformer to ac mains and the secondary side to the rectifier input.
3. By the multimeter, measure the ac input voltage of the rectifier and, ac and dc voltage at the output of the rectifier.
4. Measure the amplitude and time period of the transformer secondary(input waveform) by connecting CRO.
5. Feed the rectified output voltage to the CRO and measure the time period and amplitude of the waveform.

**Observation Table:**

**HWR Without Filter:**

$R_L$ ( $\Omega$ )	$V_L$ (V)	$V_m$ (V)	$V_{dc} = \frac{V_m}{\pi}$ (V)	$V_{rms} = \frac{V_m}{2}$ (V)	$V_r(rms)$ $= \sqrt{V_{rms}^2 - V_{dc}^2}$  (V)	$R.F = \frac{V_r(rms)}{V_{dc}}$

**HWR With Capacitor Filter:**

$R_L$ ( $\Omega$ )	$V_L$ (V)	$V_m$ (V)	$V_r$ (V)	$V_{dc} = V_m - \frac{V_r}{2}$ (V)	$V_r(rms)$ $= \frac{V_r}{2\sqrt{3}}$	$R.F = \frac{V_r(rms)}{V_{dc}}$

**FWR without Filter:**

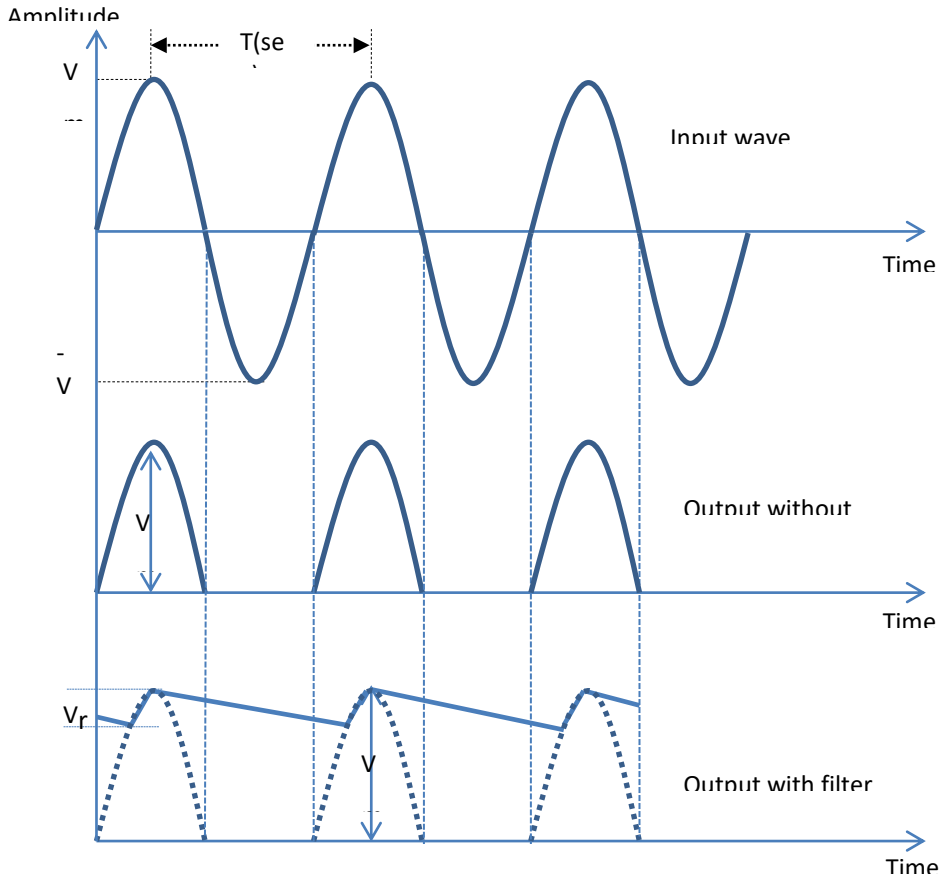
$R_L$ ( $\Omega$ )	$V_L$ (V)	$V_m$ (V)	$V_{dc} = \frac{2V_m}{\pi}$ (V)	$V_{rms} = \frac{V_m}{\sqrt{2}}$ (V)	$V_r(rms) = \sqrt{V_{rms}^2 - V_{dc}^2}$ (V)	$R.F = \frac{V_r(rms)}{V_{dc}}$	% Regulation $\frac{V(NL) - V(L)}{V_L}$

**FWR with capacitor filter**

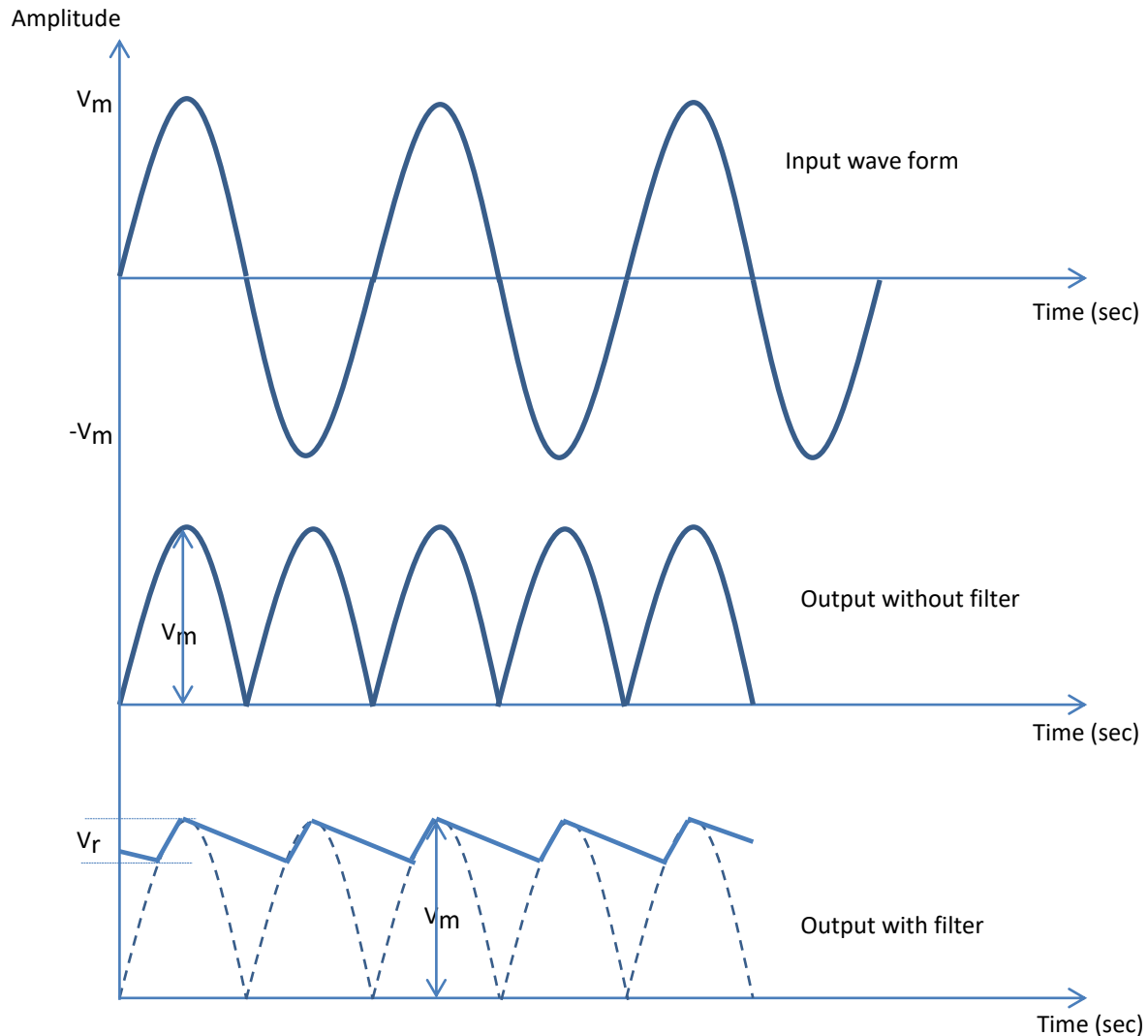
$R_L$ ( $\Omega$ )	$V_L$ (V)	$V_m$ (V)	$V_r$ (V)	$V_{dc} = V_m - \frac{V_r}{2}$ (V)	$V_r(rms) = \frac{V_r}{4\sqrt{3}}$ (V)	$R.F = \frac{V_r(rms)}{V_{dc}}$	% Regulation $\frac{V(NL) - V(L)}{V_L}$

**Model Graphs:**

**Half Wave Rectifier:**



## Fullwave Rectifier:



## PRECAUTIONS

1. The primary and secondary sides of the transformer should be carefully identified.
2. The polarities of the diode should be carefully identified.

### **Result:**

The input and output waveforms of half wave rectifier and full wave rectifier are is plotted .

Ripple factor of HWR without Filter =

Ripple factor of HWR with Capacitor Filter =

The input and output waveforms of full wave rectifier are is plotted .

Ripple factor of FWR without Filter =

Ripple factor of FWR with Capacitor Filter =

**VIVA QUESTIONS:**

1. What is rectifier?
2. What is filter?
3. Define Ripple factor.
4. What is Peak inverse voltage?
5. How capacitor acts as filter.
6. Define regulation.
7. What are the applications of rectifiers?
8. Define transformer utilization factor.

## 4. DIODE CLIPPERS

### Objective:

To observe the waveforms of clipper circuits using

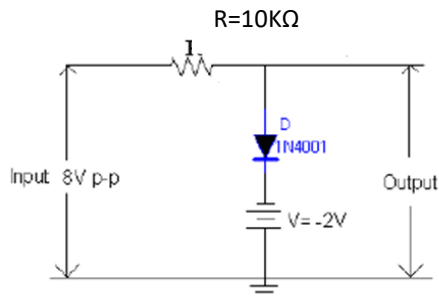
- a. Positive clipper
- b. Negative clipper
- c. Two level clipper or slicer circuit.

### Apparatus:

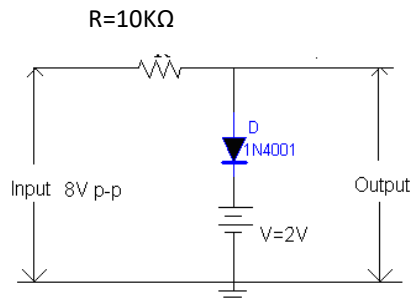
1. 1N 4007 diodes (2)
2. Resistor-10K (1)
3. Capacitor 4.7 $\mu$ f - 1
4. IN4007 diode - 1
5. Resistor 1M  $\Omega$  - 1
6. TRPS
7. Function Generator
8. Bread board and connecting wires
9. CRO with CRO probes.

### Circuit diagram:

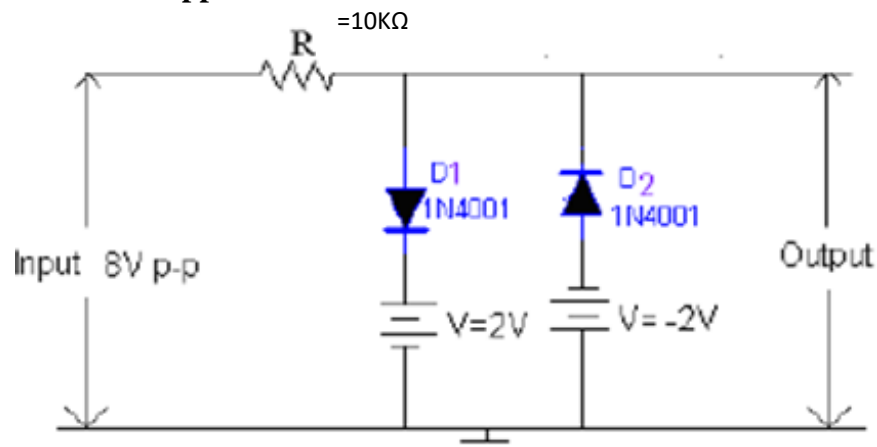
#### **Negative clipper:**



#### **Positive clipper:**



**Two level clipper:**



**Circuit operation:**

Clippers are used to select a part of signal waveform above or below a reference voltage for transmission.

**Negative Clipper:**

For  $V_i < V_R + V_r$ , The diode D is OFF, since it is reverse biased and hence does not conduct. Since no current flows, there is no voltage drop across R.

$V_o = V_i$  for  $V_i < V_R + V_r$  Where  $V_r$  is Cut-in voltage of the diode. For  $V_i > V_R + V_r$ , the diode D is ON, Since it is forward biased and the potential barrier is overcome  $V_o = V_R + V_r$

**Transfer characteristic Equation:**

$$V_o = V_i \text{ for } V_i < V_R + V_r$$

$$V_o = V_R + V_r \text{ for } V_i > V_R + V_r$$

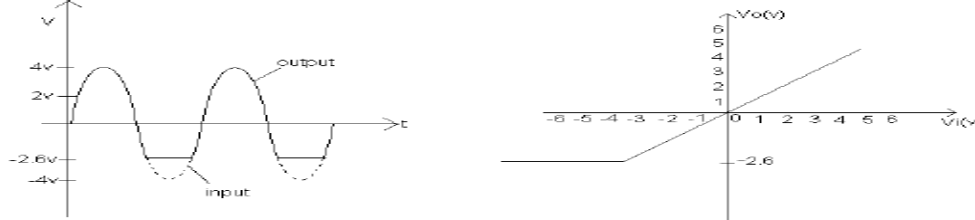
**Positive Clipper:**

When  $V_i > V_R + V_r$  the diode is forward biased and hence it conducts since it is ON it is short circuited. It is obvious that  $V_o = V_R + V_r$  Whatever the comment. When  $V_i < V_R + V_r$  the diode is reverse biased and hence it is OFF. It acts as an open circuit.  $V_o = V_i$

**Transfer Characteristic Equation:**

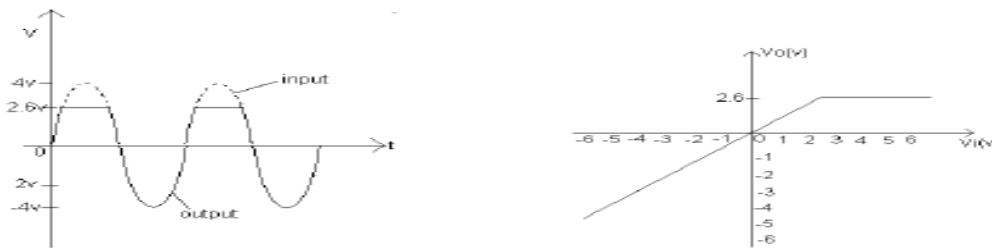
$$V_o = V_i \text{ for } V_i < V_R + V_r; \quad V_o = V_R + V_r \text{ for } V_i > V_R + V_r$$

**Model Graphs:**  
**Negative clipper:**

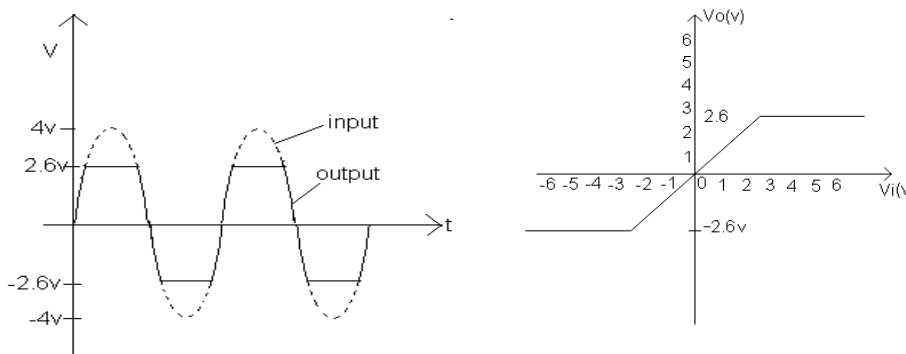


**Positive clipper:**

Positive peak clipper: Reference voltage= 2V



**Two level clipper:**



**Procedure:**

**Clippers:**

1. Connections are made as per the circuit diagram
2. For the positive clipper the diode is connected along with reference voltage as shown by applying the input and the output is observed on the C.R.O.
3. For the negative clipper the directions of diode and the reference voltage are reversed and by giving the input, the output is observed on the C.R.O.
4. For the Slicer Circuit has two Diodes along with reference voltages are connected as shown and output is observed on the C.R.O.
5. A sinusoidal input 10V (p-p) 1KHZ is given to positive clipper, negative clipper and slicer circuit and corresponding output is observed.

**Observation table:**

**Clippers:**

Name Of the Clipper	Negative Clipper		Positive Clipper		2-Level Clipper	
	I/P	O/P	I/P	O/P	I/P	O/P
Wave Form						
Amplitude (p-p) in volts						
Time Period(mSec)						

**Precautions:**

1. Loose and wrong connections are to be avoided.
2. The output waveforms should be obtained without distortion.
3. Parallax error should be avoided

**RESULT**

The output wave forms of diode clippers are plotted.

**Viva questions**

6. What is meant by non linear wave shaping?
7. What is clipper? What are the different types of clippers?
8. What are the different applications of clipper?
9. What is two level clipper
10. Explain the operation of positive and negative clipper?



## 5. CLAMPING CIRCUITS

### Objective:

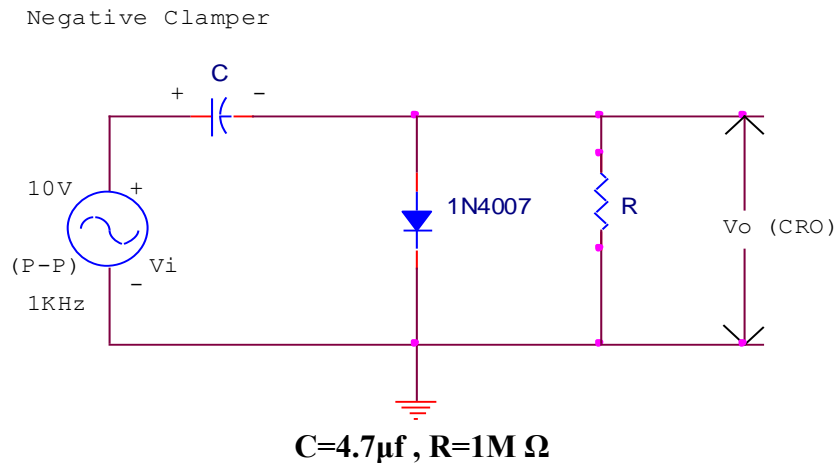
To observe the waveforms of the Positive and Negative clamping circuits.

### Apparatus:

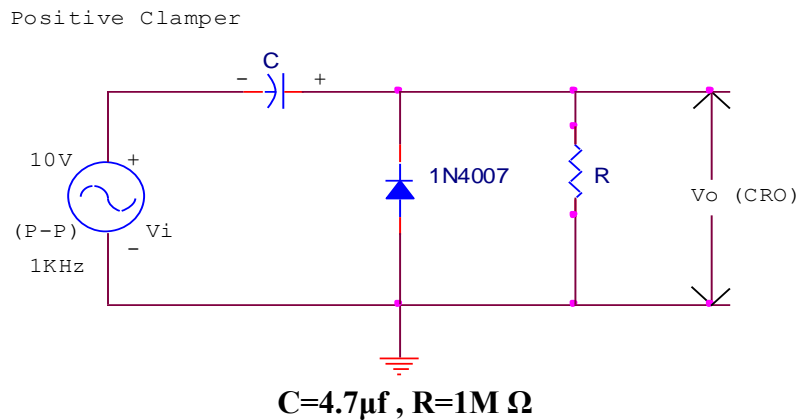
1. 1N 4007 diode
2. Capacitor  $4.7\mu\text{f}$  - 1
3. IN4007 diode - 1
4. Resistor  $1\text{M}\Omega$  - 1
5. TRPS
6. Function Generator
7. Bread board and connecting wires
8. CRO with CRO probes.

### Circuit diagram:

#### Negative Clamper:



#### Positive Clamper



### **Theory:**

A Clamper Circuit is a circuit that adds a DC level to an AC signal. Actually, the positive and negative peaks of the signals can be placed at desired levels using the clamping circuits. As the DC level gets shifted, a clamper circuit is called as a Level Shifter.

Clamper circuits consist of energy storage elements like capacitors. A simple clamper circuit comprises of a capacitor, a diode, a resistor and a dc battery if required.

### **Clamper Circuit**

A Clamper circuit can be defined as the circuit that consists of a diode, a resistor and a capacitor that shifts the waveform to a desired DC level without changing the actual appearance of the applied signal.

In order to maintain the time period of the wave form, the tau must be greater than, half the time period dischargingtimeofthecapacitorshouldbeslow.dischargingtimeofthecapacitorshouldbeslow.

$$\tau = R_c$$

Where

- R is the resistance of the resistor employed
- C is the capacitance of the capacitor used

The time constant of charge and discharge of the capacitor determines the output of a clamper circuit.

- In a clamper circuit, a vertical shift of upward or downward takes place in the output waveform with respect to the input signal.
- The load resistor and the capacitor affect the waveform. So, the discharging time of the capacitor should be large enough.

The DC component present in the input is rejected when a capacitor coupled network is used asacapacitorblocksdcasacapacitorblocksdc. Hence when dc needs to be restored, clamping circuit is used.

### **Types of Clampers:**

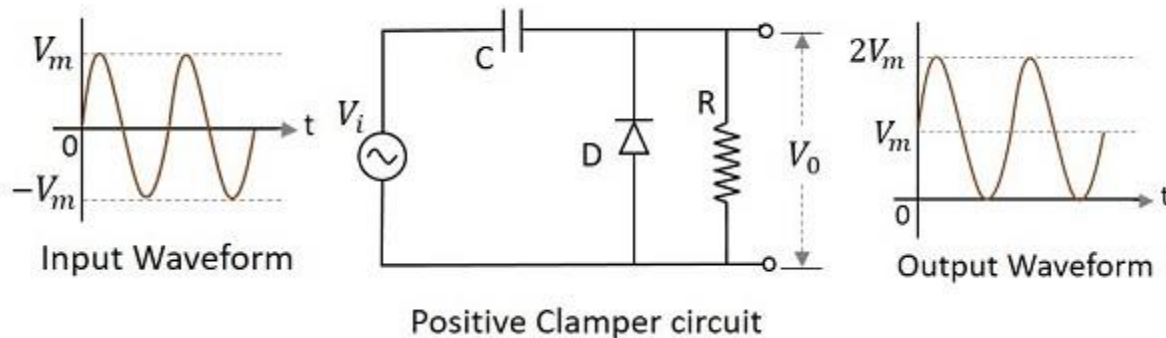
There are few types of clamper circuits, such as

- Positive Clamper
- Positive clamper with positive  $V_r$
- Positive clamper with negative  $V_r$
- Negative Clamper
- Negative clamper with positive  $V_r$
- Negative clamper with negative  $V_r$

### **Positive Clamper Circuit**

A Clamping circuit restores the DC level. When a negative peak of the signal is raised above to the zero level, then the signal is said to be positively clamped.

A Positive Clamper circuit is one that consists of a diode, a resistor and a capacitor and that shifts the output signal to the positive portion of the input signal. The figure below explains the construction of a positive clamper circuit.



Initially when the input is given, the capacitor is not yet charged and the diode is reverse biased. The output is not considered at this point of time. During the negative half cycle, at the peak value, the capacitor gets charged with negative on one plate and positive on the other. The capacitor is now charged to its peak value  $V_m$ . The diode is forward biased and conducts heavily.

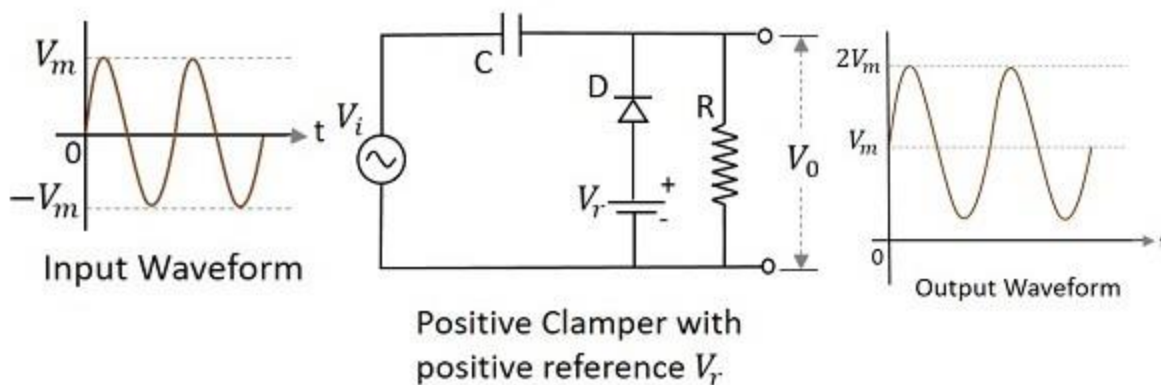
During the next positive half cycle, the capacitor is charged to positive  $V_m$  while the diode gets reverse biased and gets open circuited. The output of the circuit at this moment will be

$$V_0 = V_i + V_m$$

Hence the signal is positively clamped as shown in the above figure. The output signal changes according to the changes in the input, but shifts the level according to the charge on the capacitor, as it adds the input voltage.

### Positive Clamper with Positive Vr

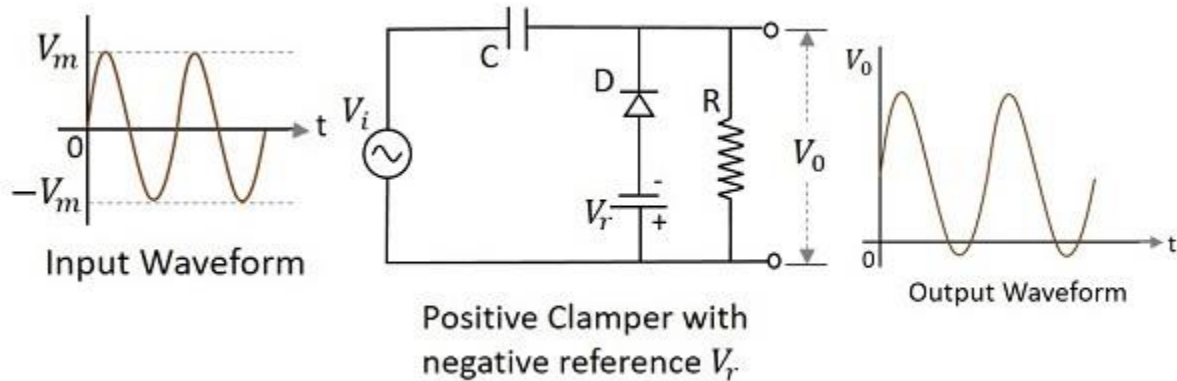
A Positive clamper circuit if biased with some positive reference voltage, that voltage will be added to the output to raise the clamped level. Using this, the circuit of the positive clamper with positive reference voltage is constructed as below.



During the positive half cycle, the reference voltage is applied through the diode at the output and as the input voltage increases, the cathode voltage of the diode increase with respect to the anode voltage and hence it stops conducting. During the negative half cycle, the diode gets forward biased and starts conducting. The voltage across the capacitor and the reference voltage together maintain the output voltage level.

### Positive Clamper with Negative $V_r$

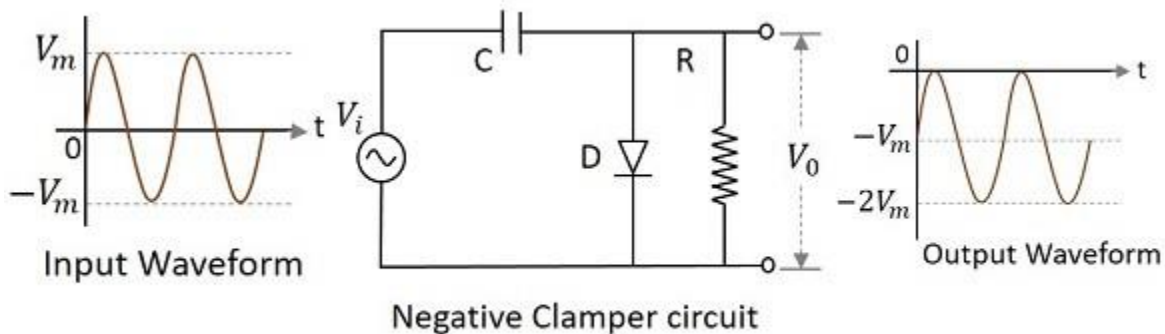
A Positive clamper circuit if biased with some negative reference voltage, that voltage will be added to the output to raise the clamped level. Using this, the circuit of the positive clamper with positive reference voltage is constructed as below.



During the positive half cycle, the voltage across the capacitor and the reference voltage together maintain the output voltage level. During the negative half-cycle, the diode conducts when the cathode voltage gets less than the anode voltage. These changes make the output voltage as shown in the above figure.

### Negative Clamper

A Negative Clamper circuit is one that consists of a diode, a resistor and a capacitor and that shifts the output signal to the negative portion of the input signal. The figure below explains the construction of a negative clamper circuit.



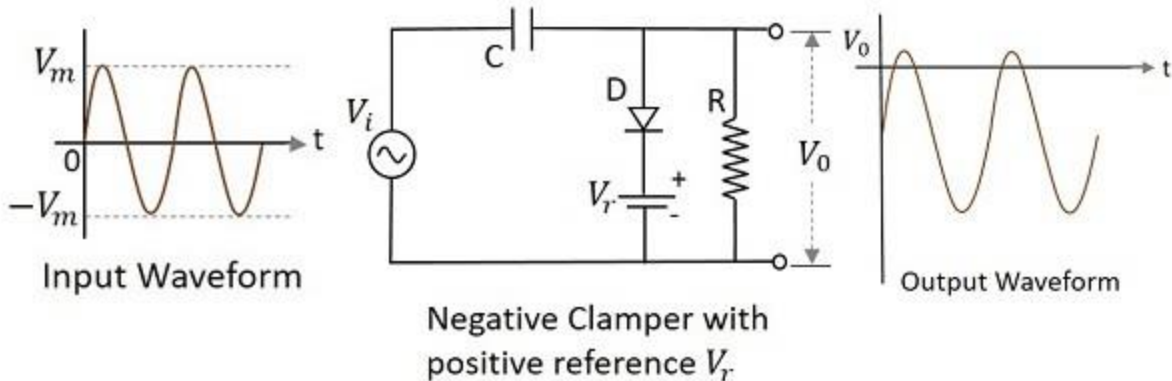
During the positive half cycle, the capacitor gets charged to its peak value  $v_m$ . The diode is forward biased and conducts. During the negative half cycle, the diode gets reverse biased and gets open circuited. The output of the circuit at this moment will be

$$V_0 = V_i + V_m \quad V_0 = V_i + V_m$$

Hence the signal is negatively clamped as shown in the above figure. The output signal changes according to the changes in the input, but shifts the level according to the charge on the capacitor, as it adds the input voltage.

### Negative clamper with positive $V_r$

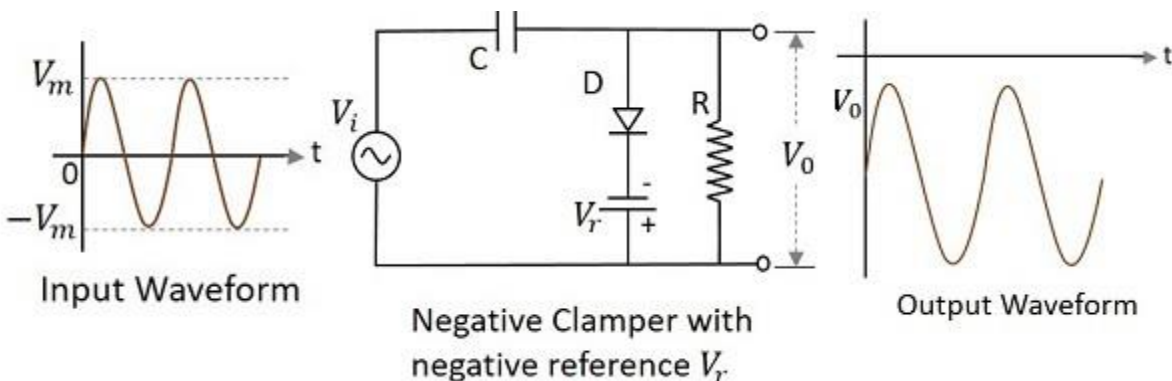
A Negative clamper circuit if biased with some positive reference voltage, that voltage will be added to the output to raise the clamped level. Using this, the circuit of the negative clamper with positive reference voltage is constructed as below.



Though the output voltage is negatively clamped, a portion of the output waveform is raised to the positive level, as the applied reference voltage is positive. During the positive half-cycle, the diode conducts, but the output equals the positive reference voltage applied. During the negative half cycle, the diode acts as open circuited and the voltage across the capacitor forms the output.

### Negative Clamper with Negative $V_r$

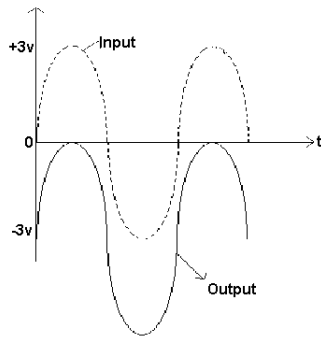
A Negative clamper circuit if biased with some negative reference voltage, that voltage will be added to the output to raise the clamped level. Using this, the circuit of the negative clamper with negative reference voltage is constructed as below.



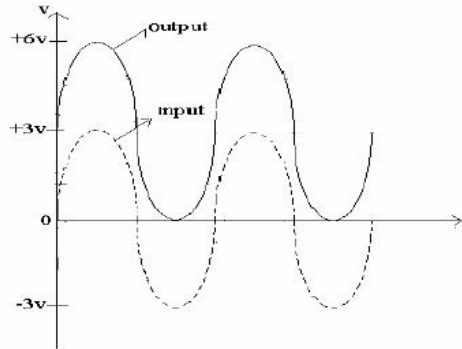
The cathode of the diode is connected with a negative reference voltage, which is less than that of zero and the anode voltage. Hence the diode starts conducting during positive half cycle, before the zero voltage level. During the negative half cycle, the voltage across the capacitor appears at the output. Thus the waveform is clamped towards the negative portion.

**Model Graphs:**

**(A) Negative clamper:**



**(B) Positive clamper:**



**Procedure:**

**Clampers:**

1. The circuits are connected as per the circuit diagram.
2. The input signal  $V_i$  of (10V p-p) frequency (1KHz) is applied to each of the circuits.
3. The corresponding output waveforms are noted from the C.R.O.
4. The input and output waveform are plotted on the graph sheets.

**Observation table:**

Name Of the Clamper	Negative Clamper		Positive Clamper	
Wave Form	I/P	O/P	I/P	O/P
Amplitude (p-p)				
Time Period				

**Precautions:**

1. Loose and wrong connections are to be avoided.
2. The output waveforms should be obtained without distortion.
3. Parallax error should be avoided

**RESULT**

The output wave forms of diode clampers are plotted.

**Viva questions**

1. What is meant by non linear wave shaping?
2. What do mean by clamper?
3. What are the different types of clamping circuits?
4. What are the different applications of clampers?
5. Why clamper is called DC inserter?
6. Explain the operation of positive clamper and negative clamper?

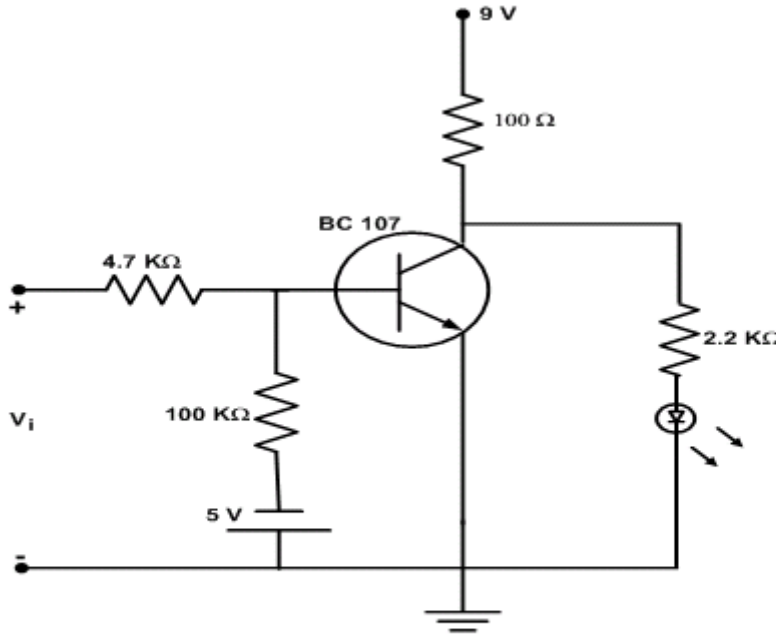
## Exp-6 Transistor as a Switch

**Objective:** To verify the switching action of a transistor and to measure the cut off and saturation voltages.

**Apparatus:**

- |                                     |               |
|-------------------------------------|---------------|
| 1. (0-30) V TRPS                    | -2no.s,       |
| 2. 100K, 2.2K, 100Ω, 4.7K resistors | -1no. (each), |
| 3. (0-2) V & (0-20) V Voltmeters    | -1no. (each), |
| 5. BC 107 transistor                | -1no.,        |
| 6. Bread Board,                     |               |
| 7. Connecting wires.                |               |

**Circuit diagram:**



**Observation Table:**

$V_{CE(SAT)}$	$V_{CB(SAT)}$	$V_{BE(SAT)}$	$V_{CE(cutoff)}$	$V_{CB(cutoff)}$	$V_{BE(cutoff)}$	$Q_1$ ON/OFF	$Q_2$ ON/OFF	<b>LED1</b> ON/OFF	<b>LED2</b> ON/OFF

**Procedure:**

1. Connections are made as per the circuit diagram.  $V_{ce(cutoff)}$ .
2. Set  $V_i=0V$ , Measure  $V_{CE}$ ,  $V_{CB}$ ,  $V_{BE}$  cutoff voltages.
3. Observe the glow of LED.
4. Set  $V_i=5V$ , measure  $V_{CE}$ ,  $V_{CB}$ ,  $V_{BE}$  Saturation voltages. LED doesn't glow

**Precautions:**

1. Connections must be made carefully to avoid short circuit.
2. Readings must be taken without parallel or error.
3. The readings should not exceed the meter range.

**Result:** Switching Action of a Transistor is verified

**Viva Questions:**

1. What are the Modes of Operation of BJT
2. Is BJT is a current controlled device or voltage controlled device?
3. Why phase shift occurs in BJT?
4. What is D.C. Load line?
5. How does a transistor acts as a switch?



## Exp-7 Linear wave shaping

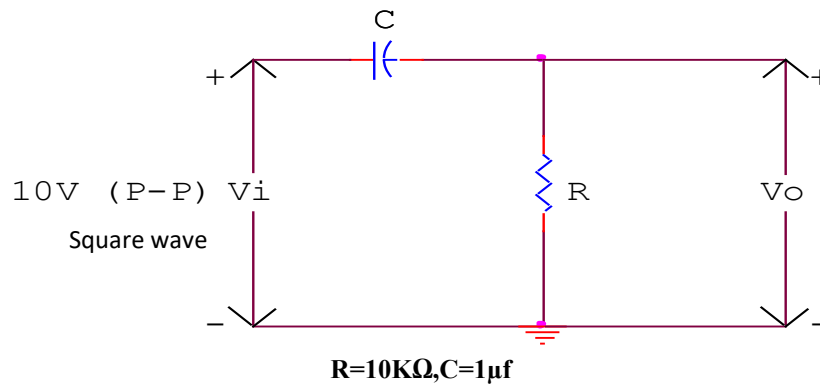
**Objective:** To observe the process of linear wave shaping for square wave input for high pass RC circuit and low pass RC circuit.

**Apparatus :**

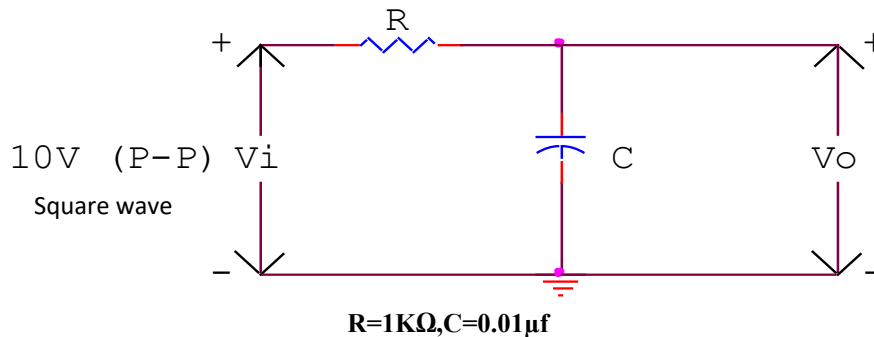
1. Resistors – 1KΩ (1) & 10KΩ (1)
2. Capacitors – 1μf(1) & 0.01μf(1)
3. Function Generator
4. TRPS
5. CRO & CRO probes
6. Bread Board and connecting wires.

**Circuit Diagram:**

High Pass RC Circuit



Low Pass RC Circuit



**High pass RC Circuit:**

The reactance of a capacitor decreases with increasing frequency; the higher frequency components in input signal appear at the output with less attenuation than do the lower frequency components. At very high frequency the capacitor acts almost as a short circuit and virtually all the input appears at the output. This behavior accounts for the designation ‘High Pass Filter ‘.

Square wave responses of a high pass RC circuit. The dashed curve represents the output if  $RC \gg T$ .

$$\begin{aligned} V_1^1 &= V_1 \exp(-T_1/RC) & V_1^1 - V_2 &= V \\ V_2^1 &= V_2 \exp(-T_2/RC) & V_1 - V_2^1 &= V \end{aligned}$$

A symmetrical square wave is one for which  $T_1 = T_2 = T/2$ . Because of symmetry  $V_1 = -V_2$  and  $V_1^1 = -V_2^1$

$$V_1 = V / (1 + \exp^{-T/2RC}) \qquad V_1^1 = V / (1 + \exp^{T/2RC})$$

Peaking of square wave resulting from a time constant small compared with T.

The high pass RC circuit acts as a differentiator if time constant is very small in comparisons with the time required for the input signal to make an appreciable change.

**Low Pass R-C Circuit :**

The low pass RC circuit passes low frequencies readily but attenuates high frequencies because the reactance of capacitor decreases with increasing frequency. At high frequencies, the capacitor acts as a virtual short circuit and output falls to zero. Square wave input (b-d) , output of low pass RC circuit. The time constant is smallest for (b) and largest for (d).

**Equation of rising portion :**

$$V_{01} = V^1 + (V_1 - V^1) \exp(-t/RC)$$

$V_1 =$  initial value of output voltage .

**Equation of falling portion :**

$$V_{02} = V^{11} + (V_2 - V^{11}) \exp-(t - T_1)/RC$$

The low pass RC circuit acts as an integrator if time constant is very large in comparison with time required for the input signal to make an appreciable change.

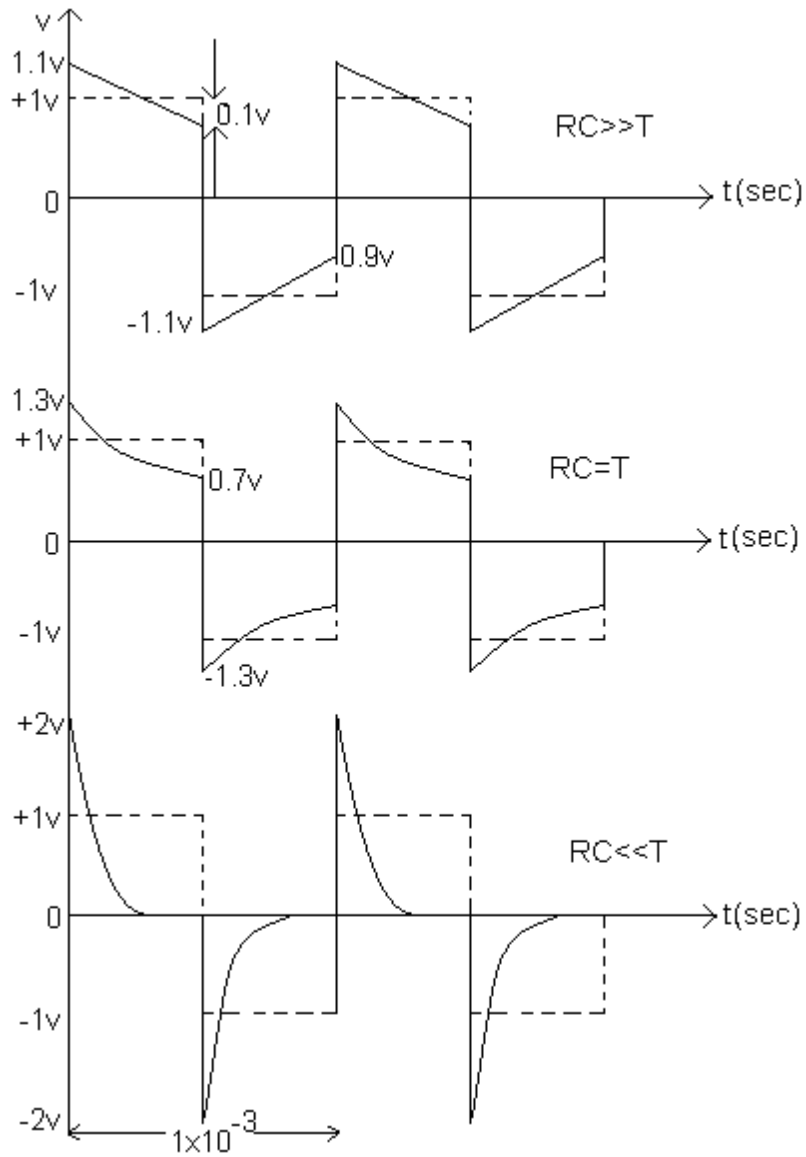
**Procedure:**

1. Connections are made as per the circuit diagram.
2. To the high pass circuit a square wave input of amplitude 10V(p-p) is given.
3. The time period of waveform is adjusted such that  $RC \ll T$ ,  $RC = T$  and  $RC \gg T$  to get spikes and tilted output respectively. The time period and amplitude are noted.
4. Now to the low pass circuit a square wave input of amplitude 10V (p-p) is given.
5. The time period of input signal is adjusted with the help of a function generator such that  $RC \ll T$  and  $RC \gg T$  to get the corresponding waveforms. The time period and amplitude are noted.
6. Graphs are plotted for both input and output waveforms of both the circuits when  $RC \ll T$  and  $RC \gg T$

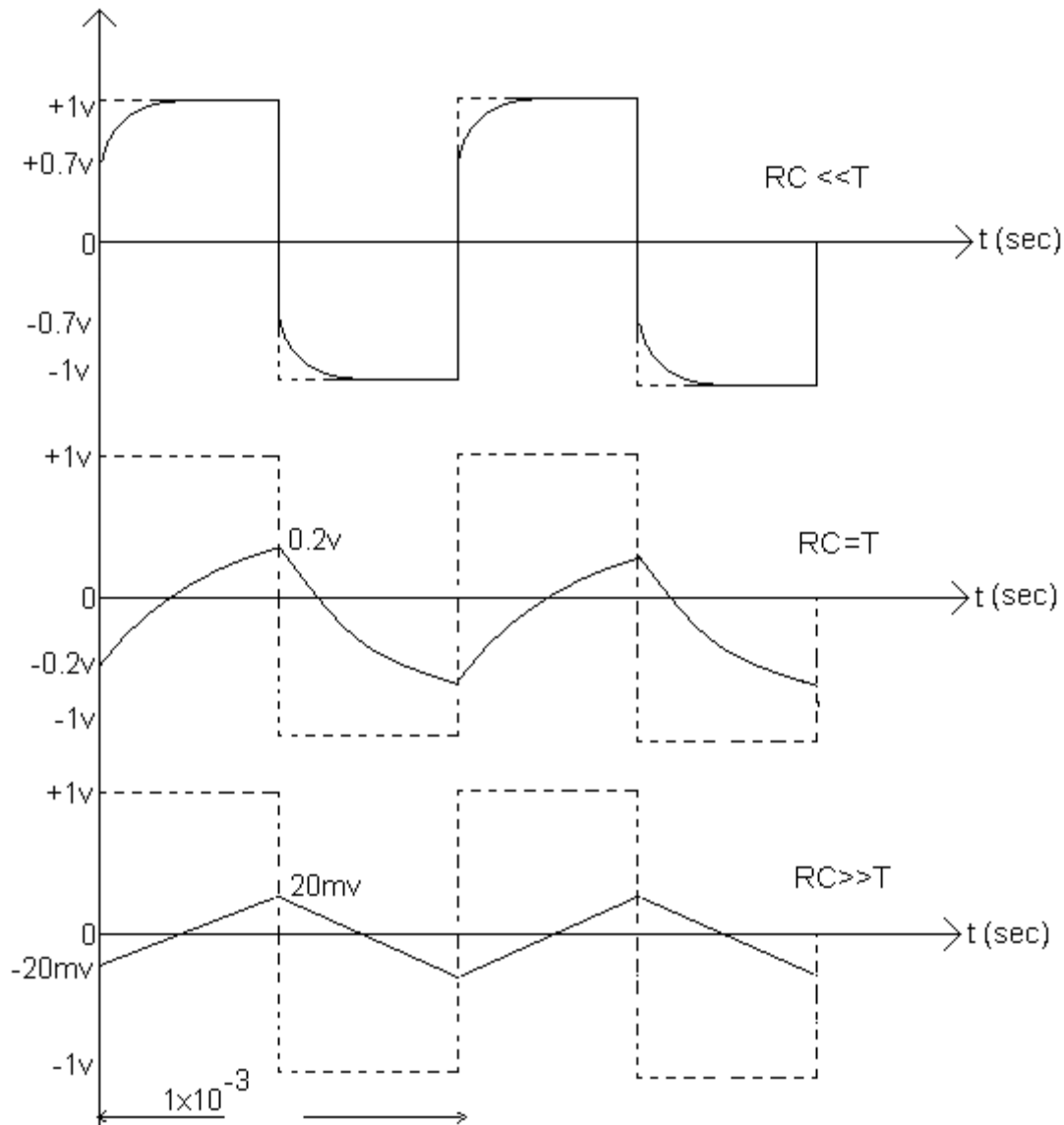
**Observation Table:**

	High Pass RC Circuit			Low Pass RC Circuit		
	$RC \gg T$	$RC = T$	$RC \ll T$	$RC \gg T$	$RC = T$	$RC \ll T$
Theoretical Calculations						
Practical Calculations						

**Model graphs:  
High pass circuit**



### Low Pass Circuit



### Precautions:

1. Loose and wrong connections are to be avoided.
2. The output waveforms should be obtained without and distortion
3. Parallax error should be avoided.

### Result:

Output wave forms of Linear wave shaping circuits are plotted.

### Viva questions:

7. What is linear wave shaping?
8. How low pass RC circuit works as an integrator?
9. How low pass RC circuit works as differentiator?
10. Define time constant?
11. Define tilt?
12. Explain the output wave forms of high pass and low pass circuit for different conditions.

### Exp-8 Common Emitter Configuration

**Objective :** To plot the input and output characteristics of a transistor in CE Configuration and to compute the h – parameters.

#### **Apparatus**

S.No	Apparatus	Type	Range	Quantity
01	Transistor	BC107		01
02	Resistance		300K $\Omega$ ,1K $\Omega$	01
03	Regulated Power supply		(0-30V)	01
04	Ammeter		(0-100mA),(0-100 $\mu$ A)	01
05	Voltmeter		(0-2V),(0-20V)	01
06	Breadboard and Wires			

#### **Theory:**

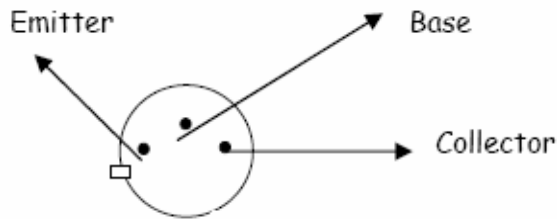
Bipolar junction transistor (BJT) is a 3 terminal (emitter, base, collector) semiconductor device. There are two types of transistors namely NPN and PNP. It consists of two P-N junctions namely emitter junction and collector junction.

In Common Emitter configuration the input is applied between base and emitter and the output is taken from collector and emitter. Here emitter is common to both input and output and hence the name common emitter configuration.

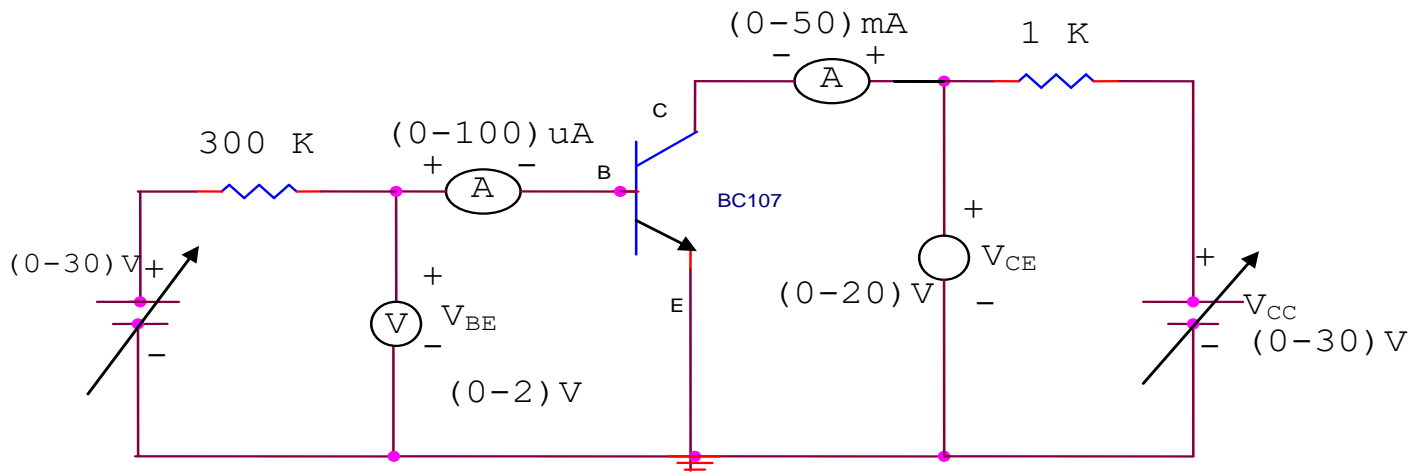
Input characteristics are obtained between the input current and input voltage taking output voltage as parameter. It is plotted between  $V_{BE}$  and  $I_B$  at constant  $V_{CE}$  in CE configuration.

Output characteristics are obtained between the output voltage and output current taking input current as parameter. It is plotted between  $V_{CE}$  and  $I_C$  at constant  $I_B$  in CE configuration.

## PIN Assignment



## Circuit Diagram



### Precautions:

1. While doing the experiment do not exceed the ratings of the transistor. This may lead to damage the transistor.
2. Connect voltmeter and Ammeter in correct polarities as shown in the circuit diagram.
3. Do not switch **ON** the power supply unless you have checked the circuit connections as per the circuit diagram.
4. Make sure while selecting the emitter, base and collector terminals of the transistor.

### Procedure:

#### Input Characteristics

1. Connect the transistor in CE configuration as per circuit diagram
2. Keep output voltage  $V_{CE} = 0V$  by varying  $V_{CC}$ .
3. By varying  $V_{BB}$ , vary  $V_{BE}$  in steps of 0.1V and note down base current  $I_B$ .
4. Repeat above procedure (step 3) for various values of  $V_{CE}$  ( $V_{CE}=5V$  and  $V_{CE}=10V$ )

5. Plot the input characteristics by taking  $V_{BE}$  on X-axis and  $I_B$  on Y-axis at constant  $V_{CE}$ .

**Output Characteristics**

1. Make the connections as per circuit diagram.
2. By varying  $V_{BB}$  keep the base current  $I_B = 0\mu A$ .
3. By varying  $V_{CC}$ , vary  $V_{CE}$  in steps of 1V and note down the readings of collector-current ( $I_C$ )
4. Repeat above procedure (step 3) for different values of  $I_B$
5. Plot the output characteristics by taking  $V_{CE}$  on x-axis and  $I_C$  on y-axis by taking  $I_B$  as a constant parameter.

**Observation Table:**

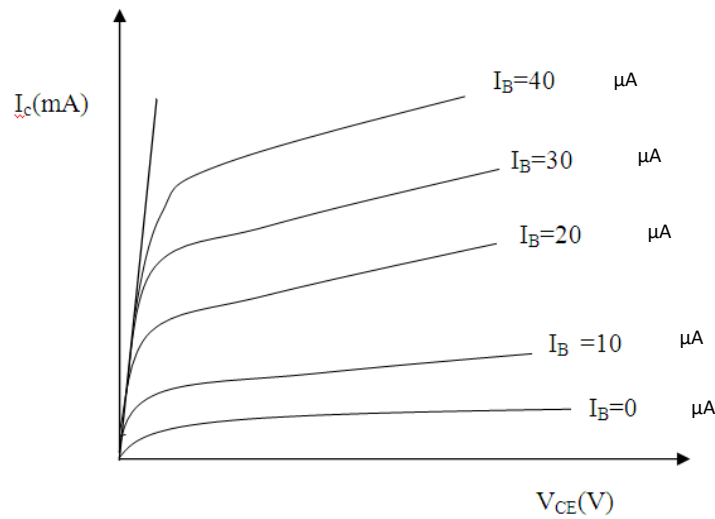
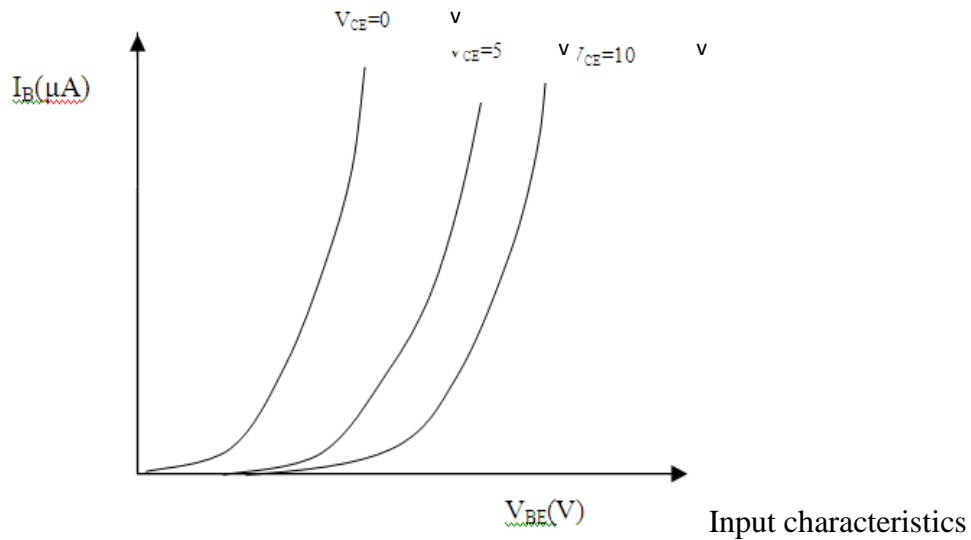
**Input Characteristics**

$V_{CE} = 0\text{ V}$		$V_{CE} = 5\text{ V}$		$V_{CE} = 10\text{ V}$	
$V_{BE}\text{ (V)}$	$I_B\text{ (}\mu\text{A)}$	$V_{BE}\text{ (V)}$	$I_B\text{ (}\mu\text{A)}$	$V_{BE}\text{ (V)}$	$I_B\text{ (}\mu\text{A)}$

**Output Characteristics**

$I_B = 0\ \mu\text{A}$		$I_B = 10\ \mu\text{A}$		$I_B = 20\ \mu\text{A}$	
$V_{CE}\text{ (V)}$	$I_C\text{ (mA)}$	$V_{CE}\text{ (V)}$	$I_C\text{ (mA)}$	$V_{CE}\text{ (V)}$	$I_C\text{ (mA)}$

## Model Graph



### Calculations from graph:

#### Input characteristics

- a) Input impedance( $h_{ie}$ )=  $\Delta V_{BE} / \Delta I_B$  ,  $V_{CE}$  constant.
- b) Reverse voltage gain( $h_{re}$ )=  $\Delta V_{BE} / \Delta V_{CE}$  ,  $I_B$  constant

#### Output characteristics

- a) Output admittance( $h_{oe}$ )=  $\Delta I_C / \Delta V_{CE}$  ,  $I_B$  constant
- b) Forward current gain( $h_{fe}$ )=  $\Delta I_C / \Delta I_B$  ,  $V_{CE}$  constant



**Result:**

Thus the input and output characteristics of CE configuration is plotted.

- a) Input impedance( $h_{ie}$ )=
- b) Forward current gain( $h_{fe}$ )=
- c) Output admittance( $h_{oe}$ )=
- d) Reverse voltage gain( $h_{re}$ )=

**VIVA QUESTIONS:**

1. Why CE configuration is most widely used?
2. Draw the equivalent Circuit of C.E
3. What is the Current Gain, voltage gain, i/p and o/p impedance in CE?.
4. Relation between ' $\alpha$ ' and ' $\beta$ ' and  $\gamma$
5. Give the condition to operate the given Transistor in active, saturation & Cut-off Regions
6. What is Emitter Efficiency?

## Exp-9 Common Base Configuration

**Objective :** To plot the input and output characteristics of a transistor in CB Configuration and to compute the h – parameters.

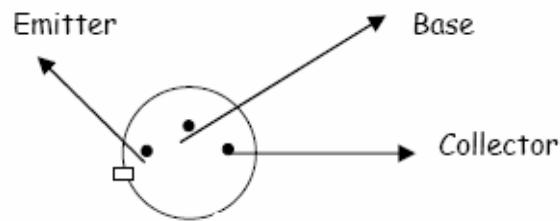
### **Apparatus**

S.No	Apparatus	Type	Range	Quantity
01	Transistor	BC107		01
02	Resistance		1K $\Omega$	02
03	Regulated Power supply		(0-30V)	02
04	Ammeter		(0-100mA)	02
05	Voltmeter		(0-2V),(0-20V)	01
06	Breadboard and Wires			

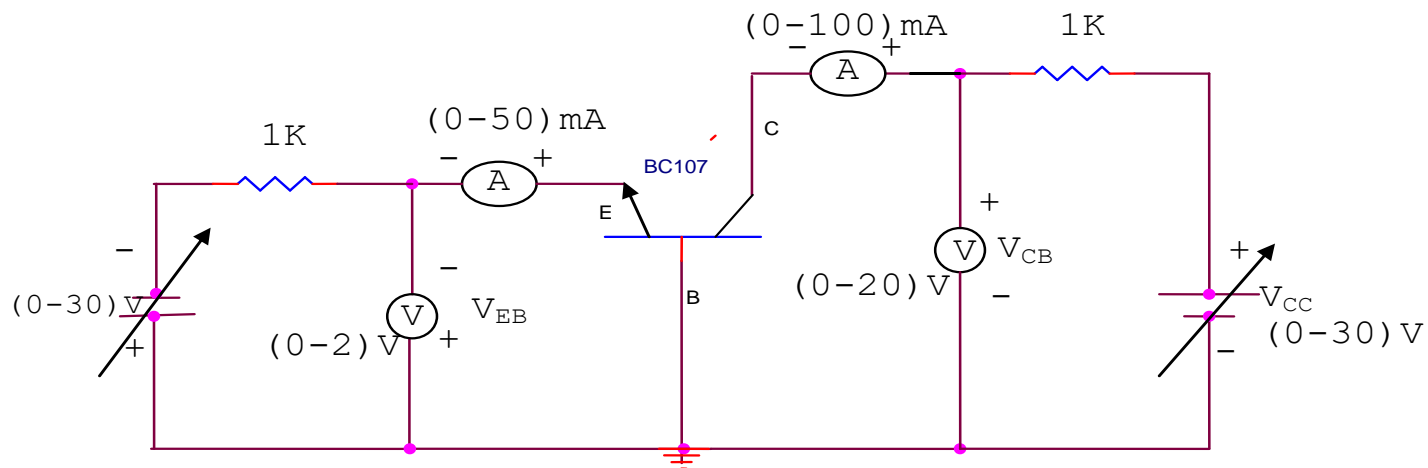
### **Theory:**

Bipolar junction transistor (BJT) is a 3 terminal (emitter, base, collector) semiconductor device. There are two types of transistors namely NPN and PNP. It consists of two P-N junctions namely emitter junction and collector junction. In Common Base configuration the input is applied between emitter and base and the output is taken from collector and base. Here base is common to both input and output and hence the name common base configuration. Input characteristics are obtained between the input current and input voltage taking output voltage as parameter. It is plotted between  $V_{EB}$  and  $I_E$  at constant  $V_{CB}$  in CB configuration. Output characteristics are obtained between the output voltage and output current taking input current as parameter. It is plotted between  $V_{CB}$  and  $I_C$  at constant  $I_E$  in CB configuration.

## PIN Assignment



## Circuit Diagram



### Precautions:

1. While doing the experiment do not exceed the ratings of the transistor. This may lead to damage the transistor.
2. Connect voltmeter and Ammeter in correct polarities as shown in the circuit diagram.
3. Do not switch **ON** the power supply unless you have checked the circuit connections as per the circuit diagram.
4. Make sure while selecting the emitter, base and collector terminals of the transistor.

### Procedure:

#### Input Characteristics

1. Connect the transistor in CB configuration as per circuit diagram
2. Keep output voltage  $V_{CB} = 0V$  by varying  $V_{CC}$ .
3. By varying  $V_{EE}$ , vary  $V_{EB}$  in steps of 0.1V and note down emitter current  $I_E$ .
4. Repeat above procedure (step 3) for various values of  $V_{CB}$  ( $V_{CB}=5V$  and  $V_{CB}=10V$ )

### Output Characteristics

1. Make the connections as per circuit diagram.
2. By varying  $V_{EE}$  keep the base current  $I_E = 10\text{mA}$ .
3. By varying  $V_{CC}$ , vary  $V_{CB}$  in steps of 1V and note down the readings of collector-current ( $I_C$ ).
4. Repeat above procedure (step 3) for different values of  $I_E$  ( $I_E = 15\text{mA}$  &  $I_E = 20\text{mA}$ )

### Observation Table:

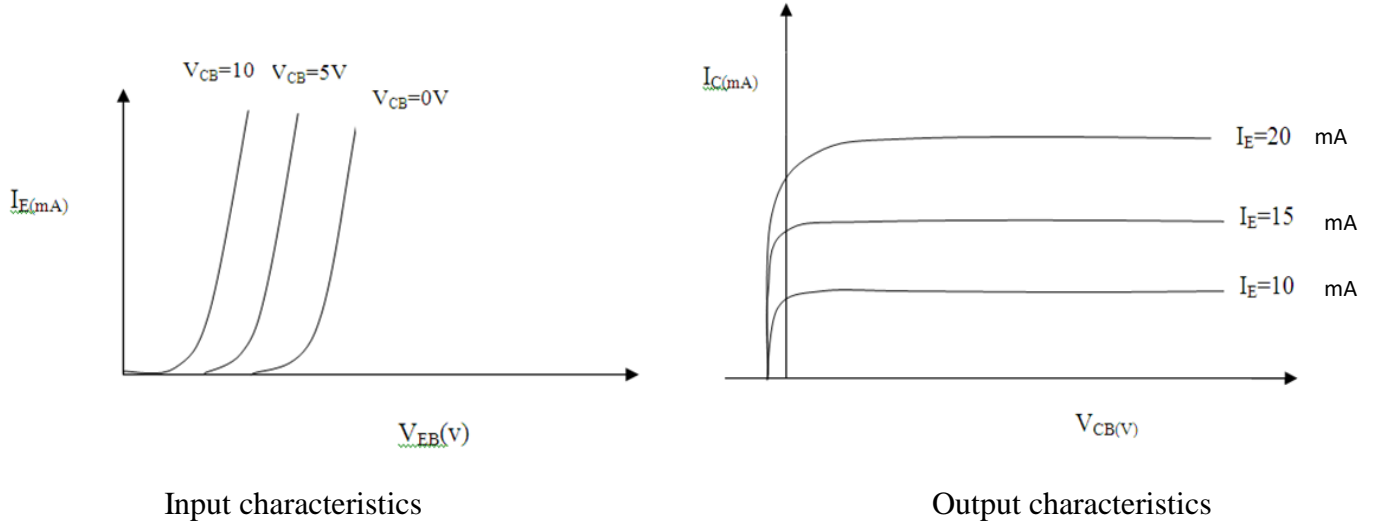
#### Input Characteristics

$V_{CB} = 0\text{ V}$		$V_{CB} = 5\text{V}$		$V_{CB} = 10\text{ V}$	
$V_{EB}$ (V)	$I_E$ (mA)	$V_{EB}$ (V)	$I_E$ (mA)	$V_{EB}$ (V)	$I_E$ (mA)

#### Output Characteristics

$I_E = 10\text{ mA}$		$I_E = 15\text{ mA}$		$I_E = 20\text{ mA}$	
$V_{CB}$ (V)	$I_C$ (mA)	$V_{CB}$ (V)	$I_C$ (mA)	$V_{CB}$ (V)	$I_C$ (mA)

## Model Graph



## Calculations from the Graph

### Input characteristics

- Input impedance( $h_{ib}$ )=  $\Delta V_{EB} / \Delta I_E$  ,  $V_{CB}$  constant.
- Reverse voltage gain( $h_{rb}$ )=  $\Delta V_{EB} / \Delta V_{CB}$  ,  $I_E$  constant

### Output characteristics

- Output admittance( $h_{ob}$ )=  $\Delta I_C / \Delta V_{CB}$  ,  $I_E$  constant
- Forward current gain( $h_{fb}$ )=  $\Delta I_C / \Delta I_E$  ,  $V_{CB}$  constant

## Result:

Thus the input and output characteristics of CB configuration are plotted and h parameters are found.

- Input impedance( $h_{ib}$ )=
- Forward current gain( $h_{fb}$ )=
- Output admittance( $h_{ob}$ )=
- Reverse voltage gain( $h_{rb}$ )=

**Viva questions:**

1. What is Early effect?
2. Draw the small signal model of BJT Common Base Configuration.
3. What is Reach –Through effect?
4. What are the applications of Common Base.
5. What will be the parameters of CB.
6. Explain the Transistor operation?

## Exp-10 Drain and transfer characteristics of JFET

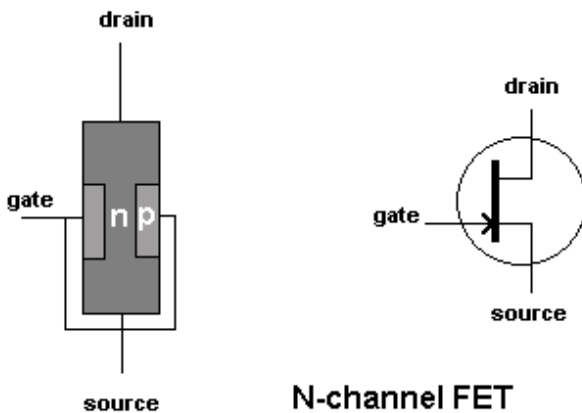
### Objective

- 1.To study Drain characteristics and Transfer characteristics
- 2.To find the Transconductance ,Drain resistance and Amplification factor

### Apparatus

S.No	Apparatus	Type	Range	Quantity
01	JFET	BFW10		01
02	Resistance		1K $\Omega$	01
03	Regulated Power supply		(0-30V)	01
04	Ammeter		(0-100mA)	01
05	Voltmeter		(0-10V),(0-20V)	01
06	Breadboard and Wires			

### Theory:



The field effect transistor (FET) is made of a bar of N type material called the SUBSTRATE with a P type junction (the gate) diffused into it. With a positive voltage on the drain, with respect to the source, electron current flows from source to drain through the CHANNEL.

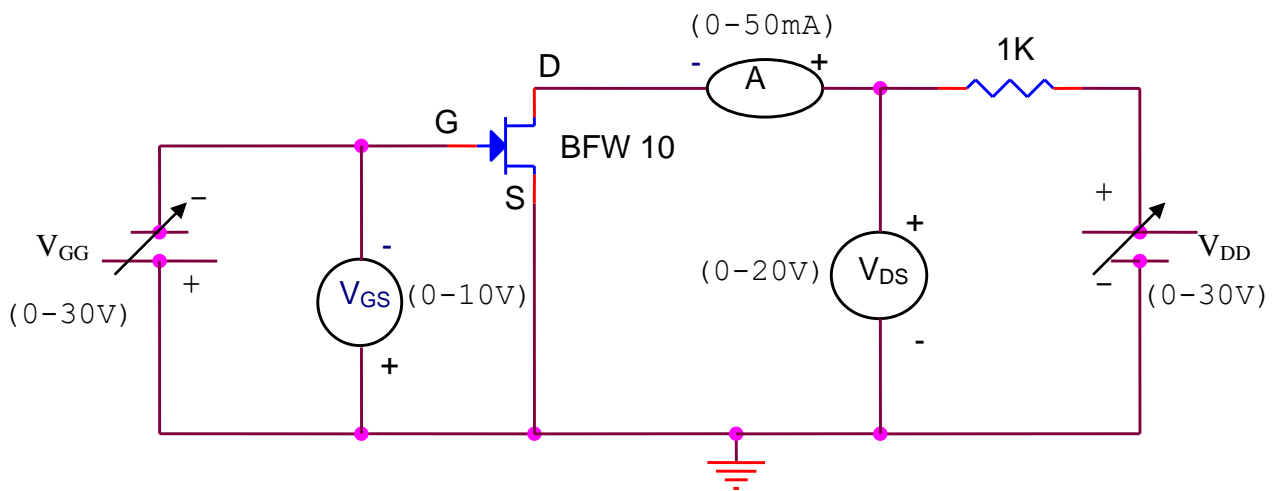
If the gate is made negative with respect to the source, an electrostatic field is created which squeezes the channel and reduces the current. If the gate voltage is high enough the channel will

be "pinched off" and the current will be zero. The FET is voltage controlled, unlike the transistor which is current controlled. This device is sometimes called the junction FET or IGFET or JFET.

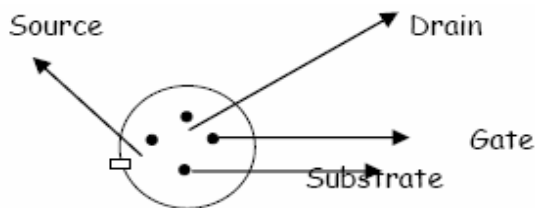
If the FET is accidentally forward biased, gate current will flow and the FET will be destroyed. To avoid this, an extremely thin insulating layer of silicon oxide is placed between the gate and the channel and the device is then known as an insulated gate FET, or IGFET or metal oxide semiconductor FET (MOSFET)

Drain characteristics are obtained between the drain to source voltage ( $V_{DS}$ ) and drain current ( $I_D$ ) taking gate to source voltage ( $V_{GS}$ ) as the parameter. Transfer characteristics are obtained between the gate to source voltage ( $V_{GS}$ ) and Drain current ( $I_D$ ) taking drain to source voltage ( $V_{DS}$ ) as parameter

**Circuit Diagram**



**Pin assignment of FET:**





**Precautions:**

1. While doing the experiment do not exceed the ratings of the FET. This may lead to damage the FET.
2. Connect voltmeter and Ammeter in correct polarities as shown in the Circuit diagram.
3. Do not switch ON the power supply unless you have checked the Circuit connections as per the circuit diagram.
4. Make sure while selecting the Source, Drain and Gate terminals of the FET.

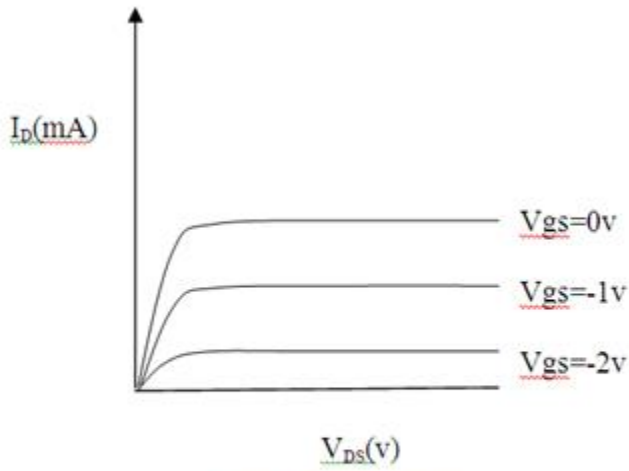
**Procedure:****Drain characteristics**

1. By Varying  $V_{GG}$  keep  $V_{GS} = 0\text{v}$ .
2. By varying  $V_{DD}$ , vary  $V_{DS}$  in steps of  $0.5\text{V}$  and note down corresponding  $I_D$
3. Repeat the above procedure for different values of  $V_{GS}$  (i.e.  $V_{GS} = -1\text{V}$  &  $V_{GS} = -2\text{V}$ )
4. Plot its characteristics with respect to  $V_{DS}$  versus  $I_D$

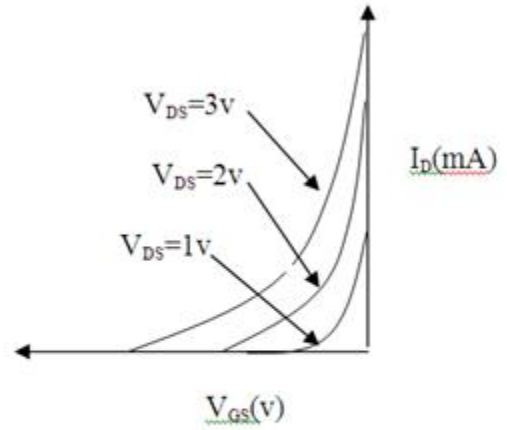
**Transfer characteristics:**

1. By Varying  $V_{DD}$  keep  $V_{DS} = 1\text{v}$ .
2. By varying  $V_{GG}$ , vary  $V_{GS}$  in steps of  $0.5\text{V}$  and note down corresponding  $I_D$
3. Repeat the above procedure for different values of  $V_{DS}$  (i.e.  $V_{DS} = 2\text{V}$  &  $V_{DS} = 3\text{V}$ )
4. Plot its characteristics with respect to  $V_{GS}$  versus  $I_D$

**Model Graph**



Drain characteristics



Transfer Characteristics

**Tabular column**

**Drain Characteristics**

$V_{GS} = 0\text{V}$		$V_{GS} = -1\text{V}$		$V_{GS} = -2\text{V}$	
$V_{DS}(\text{V})$	$I_D(\text{mA})$	$V_{DS}(\text{V})$	$I_D(\text{mA})$	$V_{DS}(\text{V})$	$I_D(\text{mA})$

**Transfer Characteristics**

$V_{DS} = 1\text{V}$		$V_{DS} = 3\text{V}$		$V_{DS} = 5\text{V}$	
$V_{GS}(\text{V})$	$I_D(\text{mA})$	$V_{GS}(\text{V})$	$I_D(\text{mA})$	$V_{GS}(\text{V})$	$I_D(\text{mA})$

### Graph (Instructions):

1. Plot the drain characteristics by taking  $V_{DS}$  on X-axis and  $I_D$  on Y-axis at constant  $V_{GS}$ .
2. Plot the Transfer characteristics by taking  $V_{GS}$  on X-axis and  $I_D$  on Y-axis at constant  $V_{DS}$ .

### Calculations from graph:

#### Drain characteristics

Drain resistance is given by the ratio of small change in drain to source voltage ( $\Delta V_{DS}$ ) to the corresponding change in Drain current ( $\Delta I_D$ ) for a constant gate to source voltage ( $V_{GS}$ ), when the JFET is operating in pinch-off or saturation region.

$$\text{Drain resistance } (r_d) = \frac{\Delta V_{DS}}{\Delta I_D} \text{ } (\Omega) \text{ at constant } V_{GS}$$

#### Transfer characteristics

Ratio of small change in drain current ( $\Delta I_D$ ) to the corresponding change in gate to source voltage ( $\Delta V_{GS}$ ) for a constant  $V_{DS}$ .  $g_m = \Delta I_D / \Delta V_{GS}$  at constant  $V_{DS}$ . (from transfer characteristics) The value of  $g_m$  is expressed in mho's or siemens (s).

$$\text{Mutual conductance } (g_m) = \frac{-\Delta I_D}{\Delta V_{GS}}$$

#### Amplification Factor ( $\mu$ ) :

It is given by the ratio of small change in drain to source voltage ( $\Delta V_{DS}$ ) to the corresponding change in gate to source voltage ( $\Delta V_{GS}$ ) for a constant drain current.

$$\mu = \Delta V_{DS} / \Delta V_{GS}.$$

$$\begin{aligned} \mu &= (\Delta V_{DS} / \Delta I_D) \times (\Delta I_D / \Delta V_{GS}) \\ \mu &= r_d \times g_m. \end{aligned}$$

#### Result

Drain resistance ( $r_d$ ) =

Mutual conductance ( $g_m$ ) =

Amplification factor( $\mu$ )=

**Viva Questions:**

1. What is meant by Field Effect Transistor?
2. What is meant by Unipolar and bipolar?
3. What is the difference between BJT and FET?
4. What are the characteristics of FET
5. What is Pinch Off Voltage?
6. Why FET is called Voltage controlled Device?
7. Draw Small Signal model of FET.
8. What are the advantages of FET?

## Exp-11 TWO STAGE R-C COUPLED AMPLIFIER

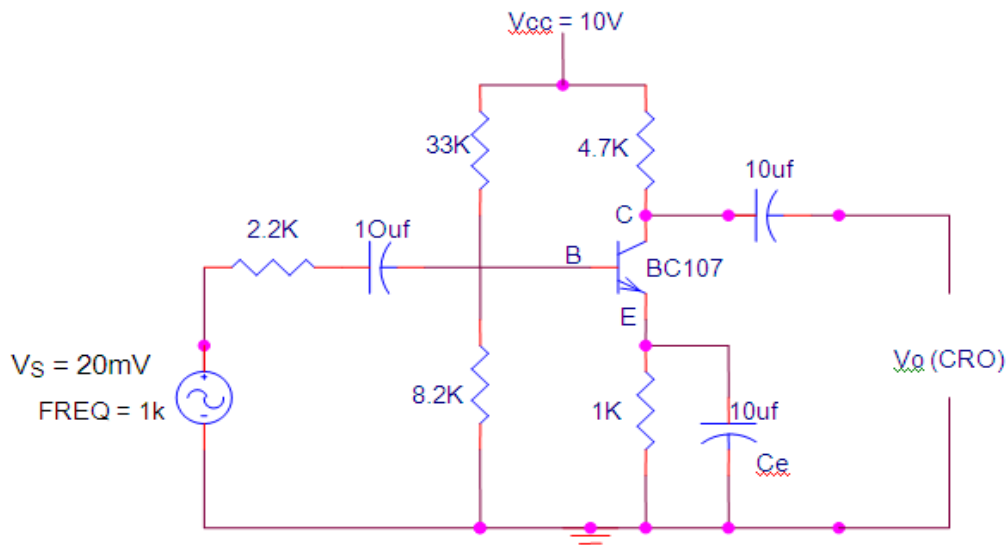
### Objective:

1. To observe the frequency response of RC coupled amplifier and to find the bandwidth of the amplifier.
2. To observe that the total voltage gain is equal to the product of the individual gains.

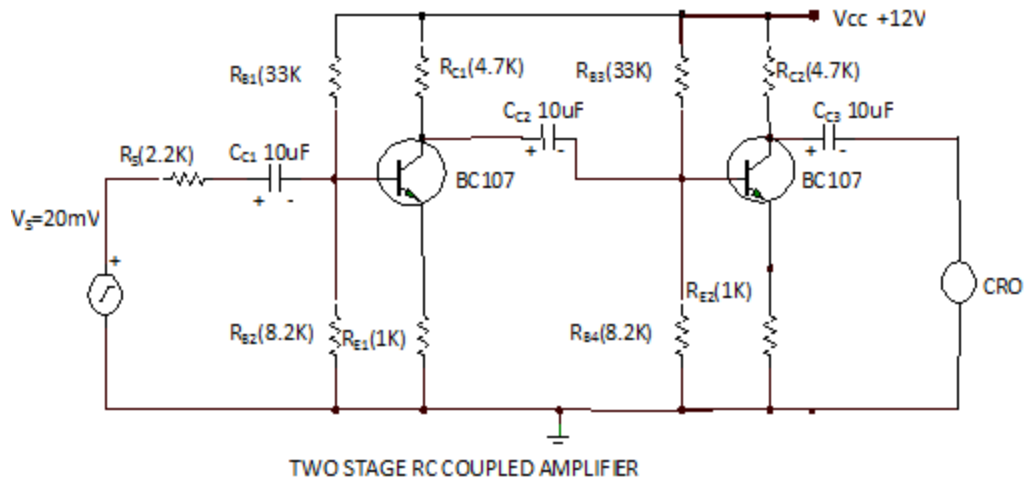
### Apparatus:

Power supply	0-30V	1No.
CRO	20MHz	1No.
Signal generator	1-1MHz	1 No
Resistors	1k $\Omega$ , 4.7k, 8.2k, 33k	2 No
and other designed values.	2.2k	1 No
Capacitors	10 $\mu$ F	3 No,
	100 $\mu$ F	1 No
Transistors	BC107	2 No

### Circuit Diagram:



**Fig: Single stage CE amplifier**



**Procedure:**

1. Connect the circuit as shown in the figure.
2. A 10V DC supply is given to the circuit and a certain amplitude of input signal is kept constant using signal generator.
3. Measure the output voltage for both single stage and multi stage amplifier using CRO.
4. Calculate total voltage gain and also individual voltage gain.
5. Now, by varying the input frequency note the output voltages from CRO and calculate the gain.

**Tabular Form:**  $V_{in} = 20 \text{ mV}$

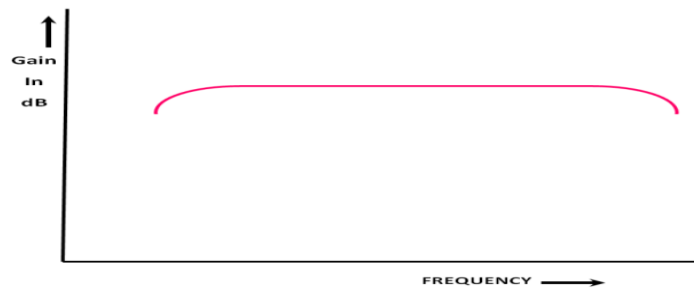
Single Stage Frequency Response:

INPUT FREQUENCY (Hz)	O/P Voltage( $V_o$ ) (V)	Voltage gain $AV = V_o/V_i$	Gain in dB = $20 \log AV$
100			
200			
300			
500			
700			
1k			
2k			
3k			
5k			
7k			
10k			
20k			
30k			
50k			
70k			
100k			
200k			
300k			
500k			
700k			
1M			

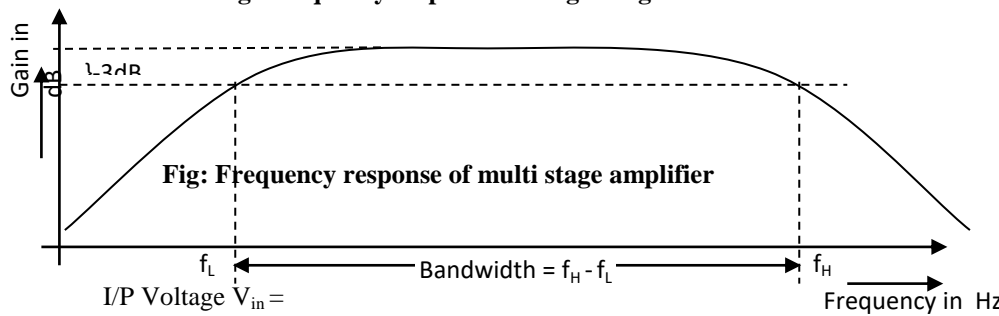
**Multi Stage Frequency Response: Vin=20mV**

INPUT FREQUENCY (Hz)	O/P Voltage(V <sub>o<sub>f</sub></sub> ) (V)	Voltage gain A <sub>V<sub>f</sub></sub> =V <sub>o<sub>f</sub></sub> /V <sub>i</sub>	Gain in dB = 20 log A <sub>V<sub>f</sub></sub>
100			
200			
300			
500			
700			
1k			
2k			
3k			
5k			
7k			
10k			
20k			
30k			
50k			
70k			
100k			
200k			
300k			
500k			
700k			
1M			

Model graphs:



**Fig: Frequency response of Single stage**



**Fig: Frequency response of multi stage amplifier**

Calculations:

I/P Voltage V<sub>in</sub> =

O/P Voltage V<sub>o2</sub> =

O/P Voltage V<sub>o1</sub> =

$$1^{st} \text{ Stage voltage gain} = \frac{V_{o1}}{V_{in}} =$$

$$\text{II}^{\text{nd}} \text{ Stage voltage gain } \frac{V_{o2}}{V_{o1}} =$$

$$\text{Overall voltage gain } \frac{V_{o2}}{V_{in}} =$$

$$\text{Bandwidth} = f_h - f_l =$$

**Graph**

:

A graph is plotted between gain (dB) and frequency (Hz) for both single stage and multi stage amplifiers.

**Precautions:**

1. Connections must be made with proper polarity.
2. Avoid loose and wrong connections.

**Result:**

The frequency response characteristics of two stage RC coupled Amplifier is obtained and total voltage gain is equal to the product of the individual gains.

**Viva- questions:**

1. what are half power points in the frequency response of an amplifier?
  - A. Low frequency component  $F_L$  and high frequency component  $F_H$
2. What is the effect of coupling capacitor on output of amplifier.
  - A. The gain increases with increase in frequency
3. Define cascading of amplifier
  - A. A cascade amplifier is any amplifier constructed from a series of amplifiers, where each amplifier sends its output to the input of the next amplifier



## Exp-12 Bistable Multivibrator

**Objectives:** To observe the operation of fixed bias binary.

**Apparatus:**

- 1) Transistors BC 107 -2
- 2) Resistors 100K $\Omega$  -2  
2.2K $\Omega$  -2  
15k $\Omega$  -2
- 3) Light emitting diodes -2
- 4) Regulated power supply-1
- 5) Bread board and connecting wires

**Design of a bistable multivibrator:**

For the given  $V_{CC}$ ,  $V_{BB}$ ,  $h_{fe(\min)}$ ,  $I_{C(\text{sat})}$  it is possible to compute the values of  $R_{C1}$ ,  $R_1$  and  $R_2$ . The following assumptions are made in order to design the bistable fixed bias multivibrator.

1. If  $Q_1$  and  $Q_2$  are identical silicon transistors, the junction voltages are assumed as  $V_{CE(\text{sat})} = 0.3 \text{ V}$  and  $V_{BE(\text{sat})} = 0.7 \text{ V}$ .
2. The base current of the ON transistor is taken as 1.5 times of the minimum value of base current.  
 $I_B = 1.5 I_{B(\min)}$  Where  $I_{B(\min)} = I_{C(\text{sat})} / h_{fe(\min)}$
3. The current through  $R_2$  of the ON transistor is taken as one tenth of  $I_C$ . If  $Q_2$  ON,  $I_4 = I_{C2}/10$ .
- 4 The current through  $R_1$  is ignored since it is quite small in comparison with the collector current of ON transistor.

**To find  $R_C$ :**

$$R_C = V_{CC} - V_D / I_2 = V_{CC} - V_{CE(\text{sat})} / I_{C(\text{sat})}$$
$$R_{C1} = R_{C2}$$

**To find  $R_2$ :**

The current through  $R_2$  is  $I_4$ , where  $I_4 = I_{C2}/10 = I_{C(\text{sat})}/10$ .

$$R_2 = [V_B - (-V_{BB})] / I_4$$

**To find  $R_1$ :**

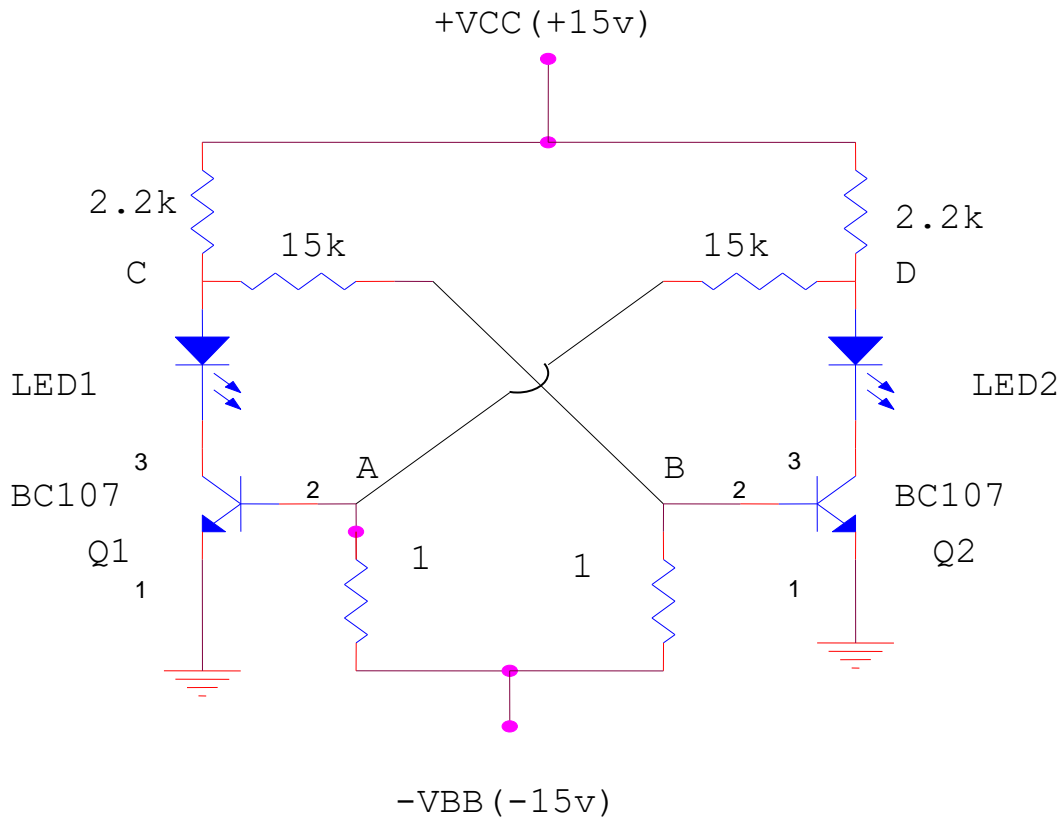
**The current through  $R_1$  is  $I'$**

$$I' = I_{B2} + I_4$$
$$I_{B2} = 1.5 I_{B(\min)}$$

Where  $I_{B(\min)} = I_{C(\text{sat})} / h_{fe(\min)}$ .

$$I_1 = I'$$
$$I_1 = V_{CC} - V_B / R_{C1} + R_1$$
$$R_1 = [V_{CC} - V_{BE(\text{SAT})} / I_1] - R_{C1}$$

**Circuit diagram:**



**Procedure:**

1. The connections are made as per the circuit diagram.
2. The supply is switched on and it is observed that one LED is ON whereas the other is OFF.
3. Now the base voltages of both the transistors  $V_{B1}$  and  $V_{B2}$  and collector voltages  $V_{C1}$  and  $V_{C2}$  are noted.
4. A negative trigger is given at the base of the ON transistor to change the states of the transistors.
5. In this steady state the base voltages of both the transistors  $V_{B1}$  and  $V_{B2}$  and also the collector voltages  $V_{C1}$  and  $V_{C2}$  are noted.

**Precautions:**

1. Loose and wrong connections should be avoided.
2. Parallax error should be avoided.

**Result:**

Working of Bistable Multivibrator circuit is verified.

**Viva questions**

1. What is stable state?
2. Name the types of multivibrators?
3. What is quasi stable state?
4. How many stable states are there in binary?
5. What is the need of triggering?
6. What are the types of triggering are there in multivibrator?