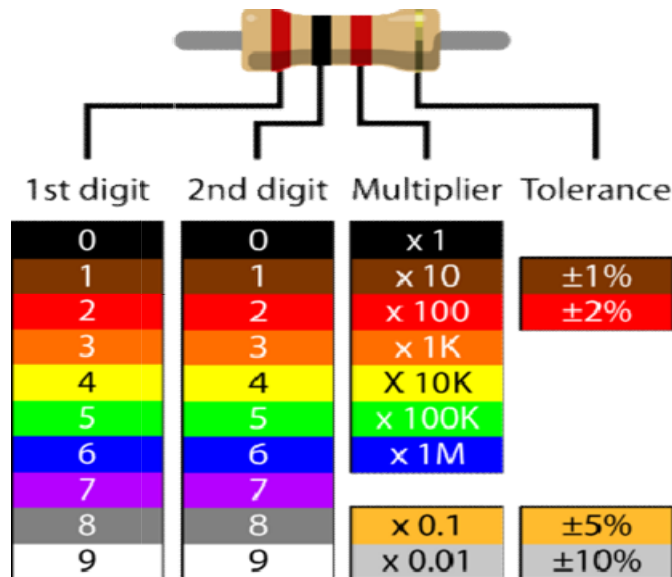




LABORATORY MANUAL OF
ELECTRONIC DEVICES AND CIRCUITS LAB
I YEAR B.TECH II-SEM ECE (R25)



A.Y.2026-27

DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

PREPARED BY

Mr.M.KRANTHI KUMAR, Asst.Prof

Mrs.NAGAJYOTHI, Asst.Prof



MARRI LAXMAN REDDY INSTITUTE OF TECHNOLOGY AND MANAGEMENT

(AN AUTONOMOUS INSTITUTION)

(Approved by AICTE, New Delhi & Affiliated to JNTUH, Hyderabad)

Accredited by NAAC with 'A' Grade & Recognized Under Section 2(f) & 12(B) of the UGC act, 1956

INDEX

S.No	Details	Pg.No
1	Certificate	I
2	Preface	II
3	Acknowledgement	III
4	General Instructions	IV
5	Safety Measures	V
6	Vision and Mission of the Institute and the Department along with PEOs of the Program	VI
7	Course Descriptor	VII
8	Previous co attainment and target for present semester	VIII
9	Academic Calendar	IX
10	Lab Time table	X
11	Syllabus copy	XI
12	Virtual Lab Details (If applicable)	XII
13	Lab Planner	XIII
14	Rubrics used to assess learnings in laboratories	XIV
List of Experiments		
A. Hardware-Based Experiments (7)		
1.	Study the I-V characteristics of a PN junction diode in forward and reverse bias to determine cut in voltage and dynamic resistance.	1
2.	Examine the reverse bias characteristics of a Zener diode and demonstrate its application as a voltage regulator under varying conditions.	6
3.	Design and analyze half-wave and full-wave rectifiers (center-tap and bridge) with and without capacitor filters to evaluate ripple factor and output voltage.	11
4.	Implement clipper and clamper circuits to observe waveform shaping through positive, negative, and biased configurations.	22
5.	Plot the input and output characteristics of a BJT in common emitter configuration to determine input/output resistance and current gain.	32
6.	Design and test fixed bias and voltage divider bias circuits to establish a stable	38



MARRI LAXMAN REDDY INSTITUTE OF TECHNOLOGY AND MANAGEMENT

(AN AUTONOMOUS INSTITUTION)

(Approved by AICTE, New Delhi & Affiliated to JNTUH, Hyderabad)

Accredited by NAAC with 'A' Grade & Recognized Under Section 2(f) & 12(B) of the UGC act, 1956

	operating point for a BJT amplifier and study DC load line behavior	
7.	Construct and analyze a Common Base (CB) configuration of a BJT to study input-output characteristics and determine current gain (α) and input/output resistance	44
	B. Software-Based Simulation Experiments (7)	48
8.	Simulate a full-wave bridge rectifier with capacitor filter to analyze waveform smoothing and ripple reduction in DC power supply design.	49
9.	Simulate a Zener diode-based voltage regulator to study voltage stabilization against varying supply voltages and load resistances.	51
10.	Simulate a common emitter amplifier with and without emitter bypass capacitor to analyze the effect on voltage gain and signal amplification.	53
11.	Simulate BJT operation as a switch and small-signal amplifier to understand its dual functionality in digital and analog applications.	56
12.	Simulate the output and transfer characteristics of a JFET to determine parameters such as pinch-off voltage, drain resistance, and trans conductance.	59
13.	Simulate the characteristics of a MOSFET and design a CMOS inverter to study digital switching behavior and low-power logic design.	64
14.	Simulate the transfer and output characteristics of an enhancement-mode NMOS transistor to analyze threshold voltage, drain current, and switching behavior	68
OPEN ENDED EXPERIMENTS		
1	Measurement of Ripple factor and Efficiency of Rectifiers and filters, leads to designing of voltage regulators.	72
2	Finding the bandwidth of amplifiers leads to the designing of Amplifiers.	78



MARRI LAXMAN REDDY
INSTITUTE OF TECHNOLOGY AND MANAGEMENT

(AN AUTONOMOUS INSTITUTION)

(Approved by AICTE, New Delhi & Affiliated to JNTUH, Hyderabad)

Accredited by NAAC with 'A' Grade & Recognized Under Section 2(f) & 12(B) of the UGC act, 1956

DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

CERTIFICATE

This is to certify that this manual is a bonafide record of the practical work carried out in the **Electronic Devices and Circuits Laboratory** for the **B.Tech. (Electronics and Communication Engineering) II Semester** programme during the **academic year 2026–2027**

This manual has been prepared by **M.Kranthi Kumar (Assistant Professor)**, **Mrs.Nagajyothi (Assistant Professor)**, Department of Electronics and Communication Engineering, with my/our own efforts and to the best of our knowledge.

Signature of Lab Faculty

Signature of HOD



MARRI LAXMAN REDDY **INSTITUTE OF TECHNOLOGY AND MANAGEMENT**

(AN AUTONOMOUS INSTITUTION)

(Approved by AICTE, New Delhi & Affiliated to JNTUH, Hyderabad)

Accredited by NAAC with 'A' Grade & Recognized Under Section 2(f) & 12(B) of the UGC act, 1956

DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

PREFACE

This laboratory lays the foundation for the Electronics and Communication Engineering students during first year of their course.

The Electronic Devices and Circuits (EDC) Laboratory is organized into two cycles to provide students with a balanced understanding of both hardware experimentation and software-based simulation, ensuring strong conceptual clarity and practical competence.

In Cycle-I, these experiments help students build a strong foundation in basic electronic devices and circuits by directly correlating theory with practical behaviour. Overall, these experiments enhance students' ability to analyze real circuits, design stable amplifiers, and apply electronic principles confidently in practical and industrial applications.

In Cycle-II, students focus on simulation-based analysis of electronic circuits using the Multisim tool. These experiments help students visualize circuit operation, study waveform behavior, and analyze device characteristics under different operating conditions without hardware limitations. After performing all the experiments students are expected to gain comprehensive practical and analytical skills in electronic devices and circuits..

By,

Mr. M.Kranthi Kumar

Mrs.Nagajyothi



MARRI LAXMAN REDDY **INSTITUTE OF TECHNOLOGY AND MANAGEMENT**

(AN AUTONOMOUS INSTITUTION)

(Approved by AICTE, New Delhi & Affiliated to JNTUH, Hyderabad)

Accredited by NAAC with 'A' Grade & Recognized Under Section 2(f) & 12(B) of the UGC act, 1956

DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

ACKNOWLEDGEMENT

It was really a good experience, working with Electronic Devices and Circuits lab. First we would like to thank **Dr.N. Srinivas, Professor, HOD** of Department of Electronics and Communication Engineering, Marri Laxman Reddy Institute of technology & Management for his concern and giving the technical support in preparing the document.

I am deeply indebted to **Dr. Ravi Prasad**, Dean (Academics), Marri Laxman Reddy Institute of Technology & Management, and gratefully acknowledge his constant support, valuable patronage, timely corrections, and scholarly guidance.

I also express my heartfelt thanks to **Dr. P. Sridhar**, Director, and **Dr. R. Murali Prasad**, Principal, Marri Laxman Reddy Institute of Technology & Management, for giving me this wonderful opportunity to prepare the **Electronic Devices and Circuits Laboratory Manual**.

Last but not least, I would like to thank all the faculty members of the **Electronics and Communication Engineering Department** who inspired, guided, and supported me in achieving this goal.

By,

Mr. M.Kranthi Kumar

Mrs.Nagajyothi



DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

GENERAL INSTRUCTIONS

1. Students are instructed to come to the Electronic Devices and Circuits Laboratory on time. Latecomers will not be permitted to enter the lab.
2. Students should be punctual to the lab. Experiments once conducted will not be repeated for latecomers.
3. Students are expected to come prepared by studying the experiments that are scheduled to be performed.
4. Students must display their identity cards before entering the laboratory.
5. Students are strictly instructed not to bring mobile phones into the laboratory.
6. Any damage or loss of equipment such as transformers, transistors, CROs, etc., during the lab session will be the responsibility of the student, and a penalty or fine will be imposed.
7. Students should update their lab records and observation books session-wise. Before leaving the lab, students must get their observation books signed by the concerned faculty.
8. Students should submit their lab records to the concerned faculty members in the staff room by the next lab session for correction and return.
9. Students should not move around the laboratory during the lab session.
10. In case of any emergency, students must take written permission from the concerned faculty member.
11. Faculty members reserve the right to suspend any student from the lab session on disciplinary grounds.
12. Students must not copy outputs from other students and should record only their own outputs.



DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

SAFETY MEASURES

1. Always ensure the power supply is switched OFF before making or modifying circuit connections in hardware experiments to avoid electric shock and damage to components.
2. Use proper ratings of components and instruments (diodes, transistors, resistors, CRO probes, etc.) and verify connections to prevent overheating or circuit failure.
3. Avoid short circuits and loose connections on breadboards; double-check wiring before applying power to ensure safe and correct operation.
4. While using Multisim, select correct component models and operating values to prevent simulation errors and to obtain realistic results comparable to practical circuits.
5. Maintain general lab discipline by handling equipment carefully, following instructor guidelines, and saving simulation files properly to avoid data loss and equipment damage.



DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

Vision of the Institute

To be a globally recognized institution that fosters innovation, excellence, and leadership in education, research, and technology development, empowering students to create sustainable solutions for the advancement of society.

Mission of the Institute

- To foster a transformative learning environment that empowers students to excel in engineering, innovation, and leadership.
- To produce skilled, ethical, and socially responsible engineers who contribute to sustainable technological advancements and address global challenges.
- To shape future leaders through cutting-edge research, industry collaboration, and community engagement.

Quality Policy

- Ensure excellence in education through innovative teaching and continuous improvement.
- Promote ethical, skilled, and employable graduates who drive sustainable technologies.
- Encourage research, industry collaboration, and community engagement for societal benefit.



DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

Vision of the Department

To provide quality technical education in Electronics and Communication Engineering through research, innovation, striving for global recognition in specified domain, leadership, and sustainable societal solutions.

Mission of the Department

- To create a transformative learning environment that empowers students in electronics and communication engineering, fostering excellence in technical skills and leadership.
- To drive innovation through research, deliver a transformative education grounded in ethical principles, and nurture the development of professionals
- To cultivate strong industry partnerships, and engaging actively with the community for societal and technological progress.

PROGRAM OUTCOMES (PO's):

PO1: Engineering Knowledge: Apply knowledge of mathematics, natural science, computing, engineering fundamentals and an engineering specialization as specified in WK1 to WK4 respectively to develop to the solution of complex engineering problems.

PO2: Problem Analysis: Identify, formulate, review research literature and analyze complex engineering problems reaching substantiated conclusions with consideration for sustainable development

PO3: Design/Development of Solutions: Design creative solutions for complex engineering problems and design/develop systems/components/processes to meet identified needs with consideration for the public health and safety, whole-life cost, net zero carbon, culture, society and environment as required.

PO4: Conduct Investigations of Complex Problems: Conduct investigations of complex engineering problems using research-based knowledge including design of experiments, modelling, analysis & interpretation of data to provide valid conclusions.

PO5: Engineering Tool Usage: Create, select and apply appropriate techniques, resources and modern engineering & IT tools, including prediction and modelling recognizing their limitations to solve complex engineering problems.

PO6: The Engineer and The World: Analyze and evaluate societal and environmental aspects while solving complex engineering problems for its impact on sustainability with reference to economy, health, safety, legal framework, culture and environment.



MARRI LAXMAN REDDY INSTITUTE OF TECHNOLOGY AND MANAGEMENT

(AN AUTONOMOUS INSTITUTION)

(Approved by AICTE, New Delhi & Affiliated to JNTUH, Hyderabad)

Accredited by NAAC with 'A' Grade & Recognized Under Section 2(f) & 12(B) of the UGC act, 1956

PO7: Ethics: Apply ethical principles and commit to professional ethics, human values, diversity and inclusion; adhere to national & international laws.

PO8: Individual and Collaborative Team work: Function effectively as an individual, and as a member or leader in diverse/multi-disciplinary teams.

PO9: Communication: Communicate effectively and inclusively within the engineering community and society at large, such as being able to comprehend and write effective reports and design documentation, make effective presentations considering cultural, language, and learning differences

PO10: Project Management and Finance: Apply knowledge and understanding of engineering management principles and economic decision-making and apply these to one's own work, as a member and leader in a team, and to manage projects and in multidisciplinary environments.

PO11: Life-Long Learning: Recognize the need for, and have the preparation and ability for i) independent and life-long learning ii) adaptability to new and emerging technologies and iii) critical thinking in the broadest context of technological change.

PROGRAM SPECIFIC OUTCOMES (PSO's)

PSO1	Analyze and design analog & digital circuits or systems for a given specification and function.
PSO2	Implement functional blocks of hardware-software co-design for signal processing and communication applications.
PSO3	Gain the hands-on competency skills in Computing Tools for electronics and communication systems for the entry level position to meet the requirements of the Employer.



MARRI LAXMAN REDDY **INSTITUTE OF TECHNOLOGY AND MANAGEMENT**

(AN AUTONOMOUS INSTITUTION)

(Approved by AICTE, New Delhi & Affiliated to JNTUH, Hyderabad)

Accredited by NAAC with 'A' Grade & Recognized Under Section 2(f) & 12(B) of the UGC act, 1956

Program Educational Objectives (PEOs)

PEO 1: Have Successful career in Industry

Graduates will excel in the Electronics and Communication industry with a strong foundation in technical expertise, continuous learning, and innovation.

PEO 2: Show Excellence in higher studies/Research

Graduates will excel in higher studies and research in Electronics and Communication Engineering (ECE) through a combination of rigorous academic dedication, cutting-edge innovation, and a deep understanding of emerging technologies.

PEO 3: Show Good Competency towards Entrepreneurship

Graduates will have to show good competency towards entrepreneurship in the field of Electronics and Communication Engineering, one must demonstrate an in-depth understanding of emerging technologies, market trends, and the ability to innovate within this rapidly evolving industry.



MARRI LAXMAN REDDY
INSTITUTE OF TECHNOLOGY AND MANAGEMENT

(AN AUTONOMOUS INSTITUTION)

(Approved by AICTE, New Delhi & Affiliated to JNTUH, Hyderabad)

Accredited by NBA and NAAC with 'A' Grade & Recognized Under Section 2(f) & 12(B) of the UGC act, 1956

ELECTRONIC DEVICES AND CIRCUITS LAB

1	Department	ELECTRONICS & COMMUNICATION ENGINEERING							
2	Course Name	ELECTRONIC DEVICES AND CIRCUITS LAB							
3	Course Code	2520471							
4	Year/Semester	I/II							
5	Regulation	MLRS-R25							
6	Structure of the course	Theory				Practical			
		Lecture	Tutorials	Practical	Credit	L	T	P	C
		0	0	0	0	0	0	2	1
7	Type of course	BS	HS	ES	PC	PE	OE	PS	MC
		×	×	×	✓	×	×	×	×
8	Course Offered	Odd Semester			Even Semester				✓
9	Total lecture, tutorial and practical hours for this course Offered (16 weeks of teaching per semester)								
	Lectures: 2 Hours		Tutorials: 0 hours			Practical: 32 hours			
10	Course Coordinator	Mr.M.KRNTHI KUMAR							
11	Date Approved by BOS								
12	Course Webpage	www.mlritm.ac.in/							
13	Prerequisites/	Level	Course Code	Semester	Prerequisites				
	Co-requisites	-	-	-	Nil				

14. Course Overview:

This laboratory course aims to provide hands-on experience and simulation-based learning of semiconductor devices and basic electronic circuits. Students will analyze the characteristics and applications of diodes, BJTs, and FETs, design rectifiers and amplifiers, and simulate modern electronic circuits using software tools. The course bridges theoretical concepts with practical implementation, developing foundational skills essential for analog electronics and circuit analysis.

15. Course Objectives:

The students will try to learn:

- The practical behavior of PN junction and Zener diodes through I–V characteristics, rectifiers, voltage regulation, and waveform shaping applications.
- Biasing and amplification characteristics of BJTs in different configurations, along with their input- output parameters and stability aspects.
- The design, implementation, and evaluation of rectifier circuits with and without filters to understand ripple factor and DC supply performance.
- The simulation and analysis of transistor- and FET-based circuits, including CE amplifiers, JFETs, MOSFETs, and CMOS inverters, for analog and digital applications
- Utilization of modern circuit simulation tools to validate experimental results, analyze device characteristics, and design low-power, high-performance electronic circuits.

16. Course Outcomes:

After successful completion of the course, students should be able to:








CO1	Analyze the I–V characteristics of semiconductor devices such as diodes, BJTs, and FETs.
CO2	Design and evaluate basic rectifier, clipper, clamper, and voltage regulation circuits.
CO3	Demonstrate biasing techniques for BJTs and determine their operating point using DC load line analysis.
CO4	Design and analyze transistor amplifier circuits in various configurations using h-parameter models.
CO5	Simulate and interpret electronic circuits using appropriate simulation tools.

17. Employability Skills:

Example: Communication skills / Programming skills / Project based skills/

The Electronic Devices and Circuits Lab focuses on developing circuit design and analytical skills through hands-on experimentation. Students explore semiconductor device characteristics such as PN Junction and Zener diodes and apply them in rectifiers with and without filters. Waveform shaping techniques are practiced using clippers and clampers, while transistor behavior is studied across CE, CB, CC, and CS configurations. Advanced experiments include analyzing SCR characteristics, amplifier designs, and logic gates using BJTs. Practical applications, such as voltage level indicators, powered backup systems, and transistor switches, enhance problem-solving and project-based skills in electronic circuit implementation.

18. Content Delivery / Instructional Methodologies:

✓	 Day to Day lab evaluation	✓	 Demo Video	✓	 Viva Vocequestions	✓	 Open EndedExperiments
x	 Competitions	x	 Hackathons	x	 Certifications	✓	Probing FurtherQuestions

19. Evaluation Methodology:

Each laboratory will be evaluated for a total of 100 marks consisting of 40 marks for Continuous Internal Evaluation (CIE) and 60 marks for semester end lab examination. Out of 30 marks for internal evaluation:

- A write-up on day-to-day experiment(aim, components/procedure, expected outcome) which shall be evaluated for 10 marks
- 10 marks for viva-voce/ tutorial/ case study/ application/ poster presentation.
- Internal practical examination shall be evaluated for 10 marks.
- The remaining 10 marks are for Laboratory Project (Design/ Software / Hardware Model/ App Development/ Prototype).

Table 1: CIE marks distribution

Component				
Type of Assessment	Day to Day performance and viva voce examination	Final internal lab assessment	Laboratory Report / Project and Presentation	Total Marks
CIE marks	20	10	10	40

Continuous Internal Evaluation (CIE): Two CIE exams shall be conducted at the 8th week and 16th week of the semester; the average of the two CIEs will be taken into account. The CIE exam is conducted for 20 marks.

The external examiner shall be appointed from the other colleges which will be decided by the Head of the institution The Semester End Examination shall be conducted with an external examiner and the laboratory teacher.

In the Semester End Examination held for 3 hours, total 60 marks are divided and allocated as shown below:

- 15 marks for write-up
- 15 for experiment/program
- 10 for evaluation of results
- 10 marks for presentation on another experiment/program in the same laboratory course and
- 10 marks for viva-voce on concerned laboratory course.

20. Course content:

CO 1	<ol style="list-style-type: none"> 1. Understand the V-I Characteristics of PN Junction Diode 2. Measurement of Input and Output characteristics of CB Configuration 3. Measurement of Input and Output characteristics of CE Configuration
CO 2	<ol style="list-style-type: none"> 1. Understand reverse bias characteristics of a Zener diode 2. Design of rectifiers with and without filters 3. Design of Clipper and clamper circuits with and without reference voltage
CO 3	<ol style="list-style-type: none"> 1. Design and test fixed bias and voltage divider bias circuits to establish a stable operating point for a BJT amplifier and study DC load line behaviour.
CO 4	<ol style="list-style-type: none"> 1. Simulate of common emitter amplifier with and without emitter bypass capacitor
CO 5	<ol style="list-style-type: none"> 1. Simulate a Zener diode-based voltage regulator 2. Simulate a full-wave bridge rectifier with capacitor filter 3. Simulate BJT operation as a switch 4. Simulate the output and transfer characteristics of a JFET 5. Simulate the characteristics of a MOSFET 6. Simulate the transfer and output characteristics of an enhancement-mode NMOS transistor.

21. Course Plan:

The course plan is meant as a guideline. Probably there may be changes.

Sino.	Topics to be covered	Cos	Reference
	Discussion on Outcome Based Education, CO, POs and PSOs	-	-
1	Study the I-V characteristics of a PN junction diode in forward and reverse bias to determine cut	CO1	T1: 3.1, T2: 1.3, R1: 2.1, R2: 1.2
2	Design and test fixed bias and voltage divider bias circuits to establish a stable operating point for a BJT amplifier and study DC load line behavior.	CO3	T1: 5.1, T2: 4.1, R1: 4.2, R2: 5.1
3	Examine the reverse bias characteristics of a Zener diode and demonstrate its application as a voltage regulator under varying conditions.	CO2	T1: 4.2, T2: 3.5, R1: 3.6, R2: 4.2

4	Design and analyze half-wave and full-wave rectifiers (center-tap and bridge) with and without capacitor filters to evaluate ripple factor and output voltage.	CO2	T1: 4.2, T2: 3.5, R1: 3.6, R2: 4.2
5	Implement clipper and clamper circuits to observe waveform shaping through positive, negative, and biased configurations	CO2	T1: 4.3, T2: 3.6, R1: 3.6, R2: 4.2
6	Plot the input and output characteristics of a BJT in common emitter configuration to determine input/output resistance and current gain	CO1	T1: 5.3, T2: 4.3, R1: 4.3, R2: 5.4
7	Construct and analyze a Common Base (CB) configuration of a BJT to study input-output characteristics and determine current gain (α) and input/output resistance.	CO1	T1: 4.4, T2: 3.7, R1: 3.6, R2: 4.2
8	Simulate a common emitter amplifier with and without emitter bypass capacitor to analyze the effect on voltage gain and signal amplification.	CO4	T1: 6.7, T2: 7.4, R1: 5.6, R2: 7.5
9	Simulate a full-wave bridge rectifier with capacitor filter to analyze waveform smoothing and ripple reduction in DC power supply design	CO5	T1: 4.2, T2: 3.5, R1: 3.6, R2: 4.2
10	Simulate a Zener diode-based voltage regulator to study voltage stabilization against varying supply voltages and load resistances.	CO5	T1: 4.2, T2: 3.5, R1: 3.6, R2: 4.2
11	Simulate BJT operation as a switch and small-signal amplifier to understand its dual functionality in digital and analog applications.	CO5	T1: 5.4, T2: 4.4, R1: 4.4, R2: 5.4
12	Simulate the output and transfer characteristics of a JFET to determine parameters such as pinch-off voltage, drain resistance, and transconductance.	CO5	T1: 7.1, T2: 6.2, R1: 6.1, R2: 7.2
13	Simulate the characteristics of a MOSFET and design a CMOS inverter to study digital switching behavior and low-power logic design.	CO5	T1: 7.4, T2: 6.5, R1: 6.4, R2: 7.6
14	Simulate the transfer and output characteristics of an enhancement-mode NMOS transistor to analyze threshold voltage, drain current, and switching behavior.	CO5	T1: 7.3, T2: 6.4, R1: 6.3, R2: 7.5
15	Internal Test on Cycle 1 Experiments		
16	Internal Test on Cycle 2 Experiments		

22. Experiments for Enhanced Learning:

S.No	Open Ended Experiments /Design Oriented Experiments
1	Measurement of Ripple factor and Efficiency of Rectifiers and filters, leads to designing of voltage regulators.
2	Finding the bandwidth of amplifiers leads to the designing of Amplifiers.

23. PROGRAM OUTCOMES & PROGRAM SPECIFIC OUTCOMES:

PO1: Engineering Knowledge: Apply knowledge of mathematics, natural science, computing, engineering fundamentals and an engineering specialization as specified in WK1 to WK4 respectively to develop to the solution of complex engineering problems.
PO2: Problem Analysis: Identify, formulate, review research literature and analyze complex engineering problems reaching substantiated conclusions with consideration for sustainable development. (WK1 to WK4)
PO3: Design/Development of Solutions: Design creative solutions for complex engineering problems and design/develop systems/components/processes to meet identified needs with consideration for the public health and safety, whole-life cost, net zero carbon, culture, society and environment as required. (WK5).
PO4: Conduct Investigations of Complex Problems: Conduct investigations of complex engineering problems using research-based knowledge including design of experiments, modelling, analysis & interpretation of data to provide valid conclusions. (WK8).
PO5: Engineering Tool Usage: Create, select and apply appropriate techniques, resources and modern engineering & IT tools, including prediction and modelling recognizing their limitations to solve complex engineering problems. (WK2 and WK6) .
PO6: The Engineer and The World: Analyze and evaluate societal and environmental aspects while solving complex engineering problems for its impact on sustainability with reference to economy, health, safety, legal framework, culture and environment. (WK1, WK5, and WK7).
PO7: Ethics: Apply ethical principles and commit to professional ethics, human values, diversity and inclusion; adhere to national & international laws. (WK9).
PO8: Individual and Collaborative Team work: Function effectively as an individual, and as a member or leader in diverse/multi-disciplinary teams.
PO9: Communication: Communicate effectively and inclusively within the engineering community and society at large, such as being able to comprehend and write effective reports and design documentation, make effective presentations considering cultural, language, and learning differences.
PO10: Project Management and Finance: Apply knowledge and understanding of engineering management principles and economic decision-making and apply these to one's own work, as a member and leader in a team, and to manage projects and in multidisciplinary environments.
PO11: Life-Long Learning: Recognize the need for, and have the preparation and ability for i) independent and life-long learning ii) adaptability to new and emerging technologies and iii) critical thinking in the broadest context of technological change. (WK8).
Program Specific Outcomes
PSO 1: Analyze and design analog & digital circuits or systems for a given specification

and function.

PSO 2: Implement functional blocks of hardware-software co-design for signal processing and communication applications.

PSO 3: Gain the hands-on competency skills in Computing Tools for electronics and communication systems for the entry level position to meet the requirements of the Employer.

24. HOW PROGRAM OUTCOMES ARE ASSESSED:

Program Outcomes		Strength	Proficiency Assessed by
PO1	Engineering Knowledge: Apply knowledge of mathematics, natural science, computing, engineering fundamentals and an engineering specialization as specified in WK1 to WK4 respectively to develop to the solution of complex engineering problems.	3	CIE/PPT/ Objective / quiz /SEE/ Assignments/ Viva-Voce/
PO2	Problem Analysis: Identify, formulate, review research literature and analyze complex engineering problems reaching substantiated conclusions with consideration for sustainable development	3	CIE/PPT/ Objective / quiz /SEE/ Assignments/ Viva-Voce/
PO5	Engineering Tool Usage: Create, select and apply appropriate techniques, resources and modern engineering & IT tools, including prediction and modelling recognizing their limitations to solve complex engineering problems.	2	CIE/PPT/ Objective / quiz /SEE/ Assignments/ Viva-Voce/
PO11	Life-Long Learning: Recognize the need for, and have the preparation and ability for i) independent and life-long learning ii) adaptability to new and emerging technologies and iii) critical thinking in the broadest context of technological change.	2	CIE/PPT/ Objective / quiz /SEE/ Assignments/ Viva-Voce/

25. HOW PROGRAM SPECIFIC OUTCOMES ARE ASSESSED:

Program Outcomes		Strength	Proficiency Assessed by
PSO1	Analyze and design analog & digital circuits or systems for a given specification and function	3	CIE/PPT/ Objective / quiz /SEE/ Assignments/ Viva-Voce
PSO2	Implement functional blocks of hardware-software co-design for signal processing and communication applications.	2	CIE/PPT/ Objective / quiz /SEE/ Assignments/ Viva-Voce
PSO3	Gain the hands-on competency skills in Computing Tools for electronics and communication systems for the entry level position to meet the requirements of the Employer.	2	CIE/PPT/ Objective / quiz /SEE/ Assignments/ Viva-Voce

3 = High; 2 = Medium; 1 = Low

26. MAPPING OF EACH CO WITH PO(s), PSO(s):

Course Outcomes	PROGRAM OUTCOMES											PSOs		
	PO 1	PO 2	PO 3	PO 4	PO 5	PO 6	PO 7	PO 8	PO 9	PO 10	PO 11	PSO 1	PSO 2	PSO 3
CO1	✓	✓	-	-	✓	-	-	-	-	-	✓	✓	✓	✓
CO2	✓	✓	-	-	✓	-	-	-	-	-	✓	✓	✓	✓
CO3	✓	✓	-	-	✓	-	-	-	-	-	✓	✓	✓	✓
CO4	✓	✓	-	-	✓	-	-	-	-	-	✓	✓	✓	✓
CO5	✓	✓	-	-	✓	-	-	-	-	-	✓	✓	✓	✓

27. JUSTIFICATIONS FOR CO – PO / PSO MAPPING - DIRECT:

Course Outcome s(COs)	POs / PSO s	Justification for mapping (Students will be able to)	No. of key competencies
CO 1	PO1	<ol style="list-style-type: none"> 1. Application of scientific principles and methodologies. 2. Utilization of mathematical concepts in problem-solving. 3. Application of specialized engineering knowledge in complex engineering problems. 	3
	PO2	<ol style="list-style-type: none"> 1. Applying mathematical, natural, and engineering sciences in problem-solving. 2. Ensuring accuracy and reliability through validation. 3. Planning and conducting experiments for problem analysis. 4. Implementing and testing solutions through experimentation. 5. Evaluating results to draw meaningful engineering conclusions. 6. Recording findings systematically for future reference and learning 	6
	PO5	<ol style="list-style-type: none"> 1. Develop engineering solutions using modern tools across various disciplines. 2. Implement simulation tools in different engineering fields. 	2
	PO12	<ol style="list-style-type: none"> 1. Pursue professional, Academic, Global certifications. 2. Begin and work towards advanced programs to further deepen knowledge in engineering and related areas. 3. Stay updated on industry trends and emerging technologies to remain relevant in the field. 4. Engage in ongoing self-improvement efforts to enhance both personal and professional growth 	4
	PSO1	<ol style="list-style-type: none"> 1. Analyze response of a circuit or system 2. Design of a circuit or system for a given specification. 3. Understand and apply circuit or system specifications accurately. 	3
	PSO2	<ol style="list-style-type: none"> 1. Develop Operational block diagrams 2. Proficiency in the use of software tools for circuit design. 3. Understanding trade-offs in hardware and software design parameters. 	3
	PSO3	<ol style="list-style-type: none"> 1. Proficiency in Computing and Simulation Tools 2. Professional Ethics and Responsibility 3. Team Collaboration 	3
CO 2	PO1	<ol style="list-style-type: none"> 1. Application of scientific principles and methodologies. 2. Utilization of mathematical concepts in problem-solving. 3. Application of specialized engineering knowledge in complex engineering problems. 	3

	PO2	<ol style="list-style-type: none"> 1. Applying mathematical, natural, and engineering sciences in problem-solving. 2. Ensuring accuracy and reliability through validation. 3. Planning and conducting experiments for problem analysis. 4. Implementing and testing solutions through experimentation. 5. Evaluating results to draw meaningful engineering conclusions. 6. Recording findings systematically for future reference and learning 	6
	PO5	<ol style="list-style-type: none"> 1. Develop engineering solutions using modern tools across various disciplines. 2. Implement simulation tools in different engineering fields. 	2
	PO12	<ol style="list-style-type: none"> 1. Pursue professional, Academic, Global certifications. 2. Begin and work towards advanced programs to further deepen knowledge in engineering and related areas. 3. Stay updated on industry trends and emerging technologies to remain relevant in the field. 4. Engage in ongoing self-improvement efforts to enhance both personal and professional growth 	4
	PSO1	<ol style="list-style-type: none"> 1. Analyze response of a circuit or system 2. Design of a circuit or system for a given specification. 3. Understand and apply circuit or system specifications accurately. 	3
	PSO2	<ol style="list-style-type: none"> 1. Develop Operational block diagrams 2. Proficiency in the use of software tools for circuit design. 3. Understanding trade-offs in hardware and software design parameters. 	3
	PSO3	<ol style="list-style-type: none"> 1. Proficiency in Computing and Simulation Tools 2. Professional Ethics and Responsibility 3. Team Collaboration 	3
CO 3	PO1	<ol style="list-style-type: none"> 1. Application of scientific principles and methodologies. 2. Utilization of mathematical concepts in problem-solving. 3. Application of specialized engineering knowledge in complex engineering problems 	3
	PO2	<ol style="list-style-type: none"> 1. Recognizing and defining complex engineering problems or Applying mathematical, natural, and engineering sciences in problem-solving. 2. Ensuring accuracy and reliability through validation. 3. Planning and conducting experiments for problem analysis. 4. Implementing and testing solutions through experimentation. 5. Evaluating results to draw meaningful engineering conclusions. 6. Recording findings systematically for future reference and learning 	6
	PO5	<ol style="list-style-type: none"> 1. Develop engineering solutions using modern tools across various disciplines. 2. Implement simulation tools in different engineering fields 	2
	PO12	<ol style="list-style-type: none"> 1. Pursue professional, Academic, Global certifications. 2. Begin and work towards advanced programs to further deepen knowledge in engineering and related areas. 3. Stay updated on industry trends and emerging technologies to remain relevant in the field. 	4

		4. Engage in ongoing self-improvement efforts to enhance both personal and professional growth	
	PSO1	1. Analyze response of a circuit or system 2. Design of a circuit or system for a given specification. 3. Understand and apply circuit or system specifications accurately	3
	PSO2	1. Develop Operational block diagrams 2. Proficiency in the use of software tools for circuit design. 3. Understanding trade-offs in hardware and software design parameters	3
	PSO3	1. Proficiency in Computing and Simulation Tools 2. Professional Ethics and Responsibility 3. Team Collaboration	3
CO 4	PO1	1. Application of scientific principles and methodologies. 2. Utilization of mathematical concepts in problem-solving. 3. Application of specialized engineering knowledge in complex engineering problems.	3
	PO2	1. Applying mathematical, natural, and engineering sciences in problem-solving. 2. Ensuring accuracy and reliability through validation. 3. Planning and conducting experiments for problem analysis. 4. Implementing and testing solutions through experimentation. 5. Evaluating results to draw meaningful engineering conclusions. 6. Recording findings systematically for future reference and learning	6
	PO5	1. Develop engineering solutions using modern tools across various disciplines. 2. Implement simulation tools in different engineering fields.	2
	PO12	1. Pursue professional, Academic, Global certifications. 2. Begin and work towards advanced programs to further deepen knowledge in engineering and related areas. 3. Stay updated on industry trends and emerging technologies to remain relevant in the field. 4. Engage in ongoing self-improvement efforts to enhance both personal and professional growth	4
	PSO1	1. Analyze response of a circuit or system 2. Design of a circuit or system for a given specification. 3. Understand and apply circuit or system specifications accurately.	3
	PSO2	1. Develop Operational block diagrams 2. Proficiency in the use of software tools for circuit design. 3. Understanding trade-offs in hardware and software design parameters	3
	PSO3	1. Proficiency in Computing and Simulation Tools 2. Professional Ethics and Responsibility. 3. Team Collaboration	3
CO 5	PO1	1. Application of scientific principles and methodologies. 2. Utilization of mathematical concepts in problem-solving. 3. Application of specialized engineering knowledge in complex engineering problems.	3

	PO2	<ol style="list-style-type: none"> 1. Applying mathematical, natural, and engineering sciences in problem-solving. 2. Ensuring accuracy and reliability through validation. 3. Planning and conducting experiments for problem analysis. 4. Implementing and testing solutions through experimentation. 5. Evaluating results to draw meaningful engineering conclusions. 6. Recording findings systematically for future reference and learning 	6
	PO5	<ol style="list-style-type: none"> 1. Develop engineering solutions using modern tools across various disciplines. 2. Implement simulation tools in different engineering fields 	2
	PO12	<ol style="list-style-type: none"> 1. Pursue professional, Academic, Global certifications. 2. Begin and work towards advanced programs to further deepen knowledge in engineering and related areas. 3. Stay updated on industry trends and emerging technologies to remain relevant in the field. 4. Engage in ongoing self-improvement efforts to enhance both personal and professional growth. 	4
	PSO1	<ol style="list-style-type: none"> 1. Analyze response of a circuit or system 2. Design of a circuit or system for a given specification. 3. Understand and apply circuit or system specifications accurately. 	3
	PSO2	<ol style="list-style-type: none"> 1. Develop Operational block diagrams 2. Proficiency in the use of software tools for circuit design. 3. Understanding trade-offs in hardware and software design parameters 	3
	PSO3	<ol style="list-style-type: none"> 1. Proficiency in Computing and Simulation Tools 2. Professional Ethics and Responsibility 3. Team Collaboration 	3

28. TOTAL COUNT OF KEY COMPETENCIES FOR CO – (PO, PSO) MAPPING:

Course Outcomes	PROGRAM OUTCOMES											PSOs		
	PO 1	PO 2	PO 3	PO 4	PO 5	PO 6	PO 7	PO 8	PO 9	PO 10	PO 11	PSO 1	PSO 2	PSO 3
	4	10	10	10	4	5	4	10	5	10	8	4	4	5
CO1	3	6	-	-	2	-	-	-	-	-	4	3	3	3
CO2	3	6	-	-	2	-	-	-	-	-	4	3	3	3
CO3	3	6	-	-	2	-	-	-	-	-	4	3	3	3
CO4	3	6	-	-	2	-	-	-	-	-	4	3	3	3
CO5	3	6	-	-	2	-	-	-	-	-	4	3	3	3

29. PERCENTAGE OF KEY COMPETENCIES FOR CO – (PO/ PSO):

Course Outcomes	PROGRAM OUTCOMES											PSOs		
	PO 1	PO 2	PO 3	PO 4	PO 5	PO 6	PO 7	PO 8	PO 9	PO 10	PO 11	PSO 1	PSO 2	PSO 3
CO1	75	60	-	-	50	-	-	-	-	-	50	75	75	60
CO2	75	60	-	-	50	-	-	-	-	-	50	75	75	60
CO3	75	60	-	-	50	-	-	-	-	-	50	75	75	60
CO4	75	60	-	-	50	-	-	-	-	-	50	75	75	60
CO5	75	60	-	-	50	-	-	-	-	-	50	75	75	60

30. COURSE ARTICULATION MATRIX (PO – PSO MAPPING):

CO'S and PO'S, CO'S and PSO'S on the scale of 0 to 3, 0 being no correlation, 1 being the low correlation, 2 being medium correlation and 3 being high correlation.

0 - $0 \leq C \leq 5\%$ – No correlation,

2 - $40\% < C < 60\%$ –Moderate

1-5 $< C \leq 40\%$ – Low/ Slight

3 - $60\% \leq C < 100\%$ – Substantial /High

Course Outcomes	PROGRAM OUTCOMES											PSOs		
	PO 1	PO 2	PO 3	PO 4	PO 5	PO 6	PO 7	PO 8	PO 9	PO 10	PO 11	PSO 1	PSO 2	PSO 3
CO1	3	3	-	-	2	-	-	-	-	-	2	3	3	3
CO2	3	3	-	-	2	-	-	-	-	-	2	3	3	3
CO3	3	3	-	-	2	-	-	-	-	-	2	3	3	3
CO4	3	3	-	-	2	-	-	-	-	-	2	3	3	3
CO5	3	3	-	-	2	-	-	-	-	-	2	3	3	3
Total	15	15	-	-	10	-	-	-	-	-	10	15	15	15
Average	3	3	-	-	2	-	-	-	-	-	2	3	3	3

31. ASSESSMENT METHODOLOGY DIRECT:






CIE Exams	✓	SEE	✓	Laboratory Practices	✓
Certification	--	Viva-Voce/PPT/Project	✓	Open Ended Experiments	-










32. ASSESSMENT METHODOLOGY INDIRECT:




✓	Course End Survey (CES)

33. RELEVANCE TO SUSTAINABILITY GOALS:

By focusing on these areas within your studies, you can contribute to achieving sustainability goals, particularly in promoting clean energy solutions and improving energy efficiency.

x	1	 <p>NO POVERTY</p>	
x	2	 <p>ZERO HUNGER</p>	
x	3	 <p>GOOD HEALTH AND WELL-BEING</p>	Good Health and well-being: Different Diodes, Rectifiers and transistors used in various healthcare devices, supporting advancements in medical research and improving healthcare delivery.
✓	4	 <p>QUALITY EDUCATION</p>	Study of semiconductor devices, rectifiers, amplifiers, and simulation-based experiments enhances students' conceptual understanding and practical skills in electronics, promoting experiential learning, technical competency, and preparation for higher studies and industry applications.
x	5	 <p>GENDER EQUALITY</p>	

x	6	<p>CLEAN WATER AND SANITATION</p> 	
✓	7	<p>AFFORDABLE AND CLEAN ENERGY</p> 	Affordable and clean Energy: Different Analog Circuits is useful in optimizing energy systems, analyzing energy consumption patterns, and promoting the use of renewable energy sources.
✓	8	<p>DECENT WORK AND ECONOMIC GROWTH</p> 	Hands-on training in electronic devices, circuit design, and simulation enhances employability, supports skill development for the electronics and semiconductor industries, and contributes to a technically skilled workforce.
✓	9	<p>INDUSTRY, INNOVATION AND INFRASTRUCTURE</p> 	Industry, Innovation and Infrastructure: Various Analog Circuits are used in Industry, Innovation and Infrastructure growth which leads to employment.
x	10	<p>REDUCED INEQUALITIES</p> 	
x	11	<p>SUSTAINABLE CITIES AND COMMUNITIES</p> 	
✓	12	<p>RESPONSIBLE CONSUMPTION AND PRODUCTION</p> 	Use of simulation tools and optimized circuit design practices in this course promotes efficient use of electronic components, minimizes material waste, and encourages sustainable and responsible engineering solutions
✓	13	<p>CLIMATE ACTION</p> 	Climate Action: Analog Circuits consists of FET devices can be used to develop circuits for environmental monitoring, such as tracking climate change indicators, analyzing biodiversity data, or monitoring ocean health. This supports efforts in climate action and environmental conservation.
x	14	<p>LIFE BELOW WATER</p> 	

x	15	 <p>LIFE ON LAND</p>	
x	16	 <p>PEACE, JUSTICE AND STRONG INSTITUTIONS</p>	
x	17	 <p>PARTNERSHIPS FOR THE GOALS</p>	

Signature of Course Coordinator

HOD

Name & Designation

Mr.M.Kranthi Kumar

Assistant Professor



DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

Previous co attainment and target for present semester

1. CO1 Attained-86%
2. CO2 Attained-93%
3. CO3 Attained-87%
4. CO4 Attained-88%
5. CO5 Attained-93%

Target for Present Sem:

1. CO1 Attained-89%
2. CO2 Attained-95%
3. CO3 Attained-90%
4. CO4 Attained-91%
5. CO5 Attained-95%



MARRI LAXMAN REDDY INSTITUTE OF TECHNOLOGY AND MANAGEMENT

(AN AUTONOMOUS INSTITUTION)

(Approved by AICTE, New Delhi & Affiliated to JNTUH, Hyderabad)

Accredited by NAAC with 'A' Grade & Recognized Under Section 2(f) & 12(B) of the UGC act, 1956

2520471: ELECTRONIC DEVICES AND CIRCUITS LABORATORY

I Year B.Tech. ECE II – Sem.

L T P C

0 0 2 1

Course Overview:

This laboratory course aims to provide hands-on experience and simulation-based learning of semiconductor devices and basic electronic circuits. Students will analyze the characteristics and applications of diodes, BJTs, and FETs, design rectifiers and amplifiers, and simulate modern electronic circuits using software tools. The course bridges theoretical concepts with practical implementation, developing foundational skills essential for analog electronics and circuit analysis.

Pre-requisites: Knowledge about electronic devices and circuits is required

Course Objectives:

The students will try to learn

- The practical behavior of PN junction and Zener diodes through I–V characteristics, rectifiers, voltage regulation, and waveform shaping applications.
- Biasing and amplification characteristics of BJTs in different configurations, along with their input-output parameters and stability aspects.
- The design, implementation, and evaluation of rectifier circuits with and without filters to understand ripple factor and DC supply performance.
- The simulation and analysis of transistor- and FET-based circuits, including CE amplifiers, JFETs, MOSFETs, and CMOS inverters, for analog and digital applications.
- Utilization of modern circuit simulation tools to validate experimental results, analyze device characteristics, and design low-power, high-performance electronic circuits.

Course Outcomes:

After successful completion of the course, students shall be able to

- Analyze the I–V characteristics of semiconductor devices such as diodes, BJTs, and FETs.
- Design and evaluate basic rectifier, clipper, clamper, and voltage regulation circuits.
- Demonstrate biasing techniques for BJTs and determine their operating point using DC load line analysis.
- Design and analyze transistor amplifier circuits in various configurations using h-parameter models.
- Simulate and interpret electronic circuits using appropriate simulation tools.

List of Experiments:

A. Hardware-Based Experiments (7):

1. Study the I–V characteristics of a PN junction diode in forward and reverse bias to determine cut in voltage and dynamic resistance.
2. Examine the reverse bias characteristics of a Zener diode and demonstrate its application as a Voltage regulator under varying conditions.
3. Design and analyze half-wave and full-wave rectifiers (center-tap and bridge) with and without Capacitor filters to evaluate ripple factor and output voltage.
4. Implement clipper and clamper circuits to observe waveform shaping through positive, negative, and biased configurations.
5. Plot the input and output characteristics of a BJT in common emitter configuration to determine Input/output resistance and current gain.
6. Design and test fixed bias and voltage divider bias circuits to establish a stable operating point for a BJT amplifier and study DC load line behavior.
7. Construct and analyze a Common Base (CB) configuration of a BJT to study input-output Characteristics and determine current gain (α) and input/output resistance.

B. Software-Based Simulation Experiments (7):

1. Simulate a full-wave bridge rectifier with capacitor filter to analyze waveform smoothing and ripple Reduction in DC power supply design.
2. Simulate a Zener diode-based voltage regulator to study voltage stabilization against varying Supply voltages and load resistances.
3. Simulate a common emitter amplifier with and without emitter bypass capacitor to analyze the Effect on voltage gain and signal amplification.
4. Simulate BJT operation as a switch and small-signal amplifier to understand its dual functionality in digital and analog applications.
5. Simulate the output and transfer characteristics of a JFET to determine parameters such as pinch-off voltage, drain resistance, and transconductance.
6. Simulate the characteristics of a MOSFET and design a CMOS inverter to study digital switching behavior and low-power logic design.
7. Simulate the transfer and output characteristics of an enhancement-mode NMOS transistor to analyze threshold voltage, drain current, and switching behavior.

Open Ended Experiments

1. Measurement of Ripple factor and Efficiency of Rectifiers and filters, leads to designing of voltage regulators.
2. Finding the bandwidth of amplifiers leads to the designing of Amplifiers.



DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING
ELECTRONIC DEVICES AND CIRCUITS LABORATORY

Virtual lab details

Name of the Virtual Lab: Basic Electronics Virtual Laboratory

Virtual Lab Host Institute: IIT Kharagpur

URL/Link to Lab: <https://be-iitkgp.vlabs.ac.in/>

Academic Year: 2025-26

Semester:II

List of Experiments Available in Virtual Lab

1. VI Characteristics of a Diode
2. Half Wave Rectification
3. Full Wave Rectification
4. Zener Diode-Voltage Regulator
5. BJT Common Emitter Characteristics
6. BJT Common Base Characteristics
7. Studies on BJT CE Amplifier



MARRI LAXMAN REDDY INSTITUTE OF TECHNOLOGY AND MANAGEMENT

(AN AUTONOMOUS INSTITUTION)

(Approved by AICTE, New Delhi & Affiliated to JNTUH, Hyderabad)

Accredited by NAAC with 'A' Grade & Recognized Under Section 2(f) & 12(B) of the UGC act, 1956

DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

ELECTRONIC DEVICES AND CIRCUITS LABORATORY

LAB PLANNER

S.No	Experiment	CO	Virtual Lab Availability	Date planned	Date conducted
1	I-V characteristics of a PN junction diode in forward and reverse bias	1	Yes		
2	reverse bias characteristics of a Zener diode	2	Yes		
3	half-wave and full-wave rectifiers with and without capacitor filters	2	Yes		
4	clipper and clamper circuits	2	No		
5	input and output characteristics of a BJT in common emitter configuration	3	Yes		
6	Common Base (CB) configuration of a BJT to study input-output characteristics	3	Yes		
7	MID-I				
8	Simulate a full-wave bridge rectifier with capacitor filter	5	Yes		
9	Simulate a Zener diode-based voltage regulator	5	Yes		
10	Simulate a common emitter amplifier with and without emitter bypass capacitor	4	Yes		



MARRI LAXMAN REDDY INSTITUTE OF TECHNOLOGY AND MANAGEMENT

(AN AUTONOMOUS INSTITUTION)

(Approved by AICTE, New Delhi & Affiliated to JNTUH, Hyderabad)

Accredited by NAAC with 'A' Grade & Recognized Under Section 2(f) & 12(B) of the UGC act, 1956

11	Simulate BJT operation as a switch	5	No		
12	Simulate the output and transfer characteristics of a JFET	5	No		
13	Simulate the characteristics of a MOSFET	5	No		
14	MID-II				



DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

ELECTRONIC DEVICES AND CIRCUITS LABORATORY

RUBRICS USED TO ASSESS LEARNINGS IN LABORATORIES

1. RUBRICS FOR DAY TO DAY EVALUATION

Parameter	Max Marks	Level-1 (Very Poor)	Level-2 (Poor)	Level-3 (Average)	Level-4 (Good)	Level-5 (Excellent)
Observation Book	05	No observations or irrelevant data. (0-1)	Incomplete or incorrect data. (2)	Basic values with some errors. (3)	Mostly correct with good format. (4)	Fully correct, clear, and well-formatted. (5)
Record Writing	05	Not submitted. (0-1)	Submitted but mostly incomplete. (2)	Submitted with some missing/wrong parts. (3)	Submitted with minor issues. (4)	Fully complete, correct algorithm & flowchart. (5)
Result	05	No result or major errors. (0-1)	Result partially obtained. (2)	Acceptable result with limited error. (3)	Near-correct result and reasonable error. (4)	Accurate result. (5)
Viva-Voce	05	Did not answer any questions. (1)	Answered very few questions. (2)	Answered some questions with help. (3)	Answered most questions correctly. (4)	Answered all questions accurately. (5)



MARRI LAXMAN REDDY INSTITUTE OF TECHNOLOGY AND MANAGEMENT

(AN AUTONOMOUS INSTITUTION)

(Approved by AICTE, New Delhi & Affiliated to JNTUH, Hyderabad)

Accredited by NAAC with 'A' Grade & Recognized Under Section 2(f) & 12(B) of the UGC act, 1956

DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

ELECTRONIC DEVICES AND CIRCUITS LABORATORY

2. RUBRICS FOR INTERNAL EVALUATION

Criterion	Max Marks	Level-1 (<i>Very Poor</i>)	Level-2 (<i>Poor</i>)	Level-3 (<i>Average</i>)	Level-4 (<i>Good</i>)	Level-5 (<i>Excellent</i>)
Design/Tool/Apparatus Selection	2 Marks	Incorrect tool/design and no reasoning. (0)	Tool/design selection attempted with unclear logic. (0.5)	Satisfactory selection with partial justification. (1)	Correct selection and proper analysis with few errors. (1.5)	Smart selection with accurate, relevant analysis. (2)
Execution (Code/Debug/Run) /Analysis/Method Used	4 Marks	Did not attempt or completely failed to execute. (0)	Attempted but unable to proceed or with major errors. (1)	Partial execution with some logic/syntax errors. (2)	Mostly correct execution with minimal help. (3)	Fully correct and independently executed program. (4)
Results & Documentation	2 Marks	Incomplete or poorly presented. (0)	Basic structure but lacks clarity or formatting. (0.5)	Complete but generic or with formatting issues. (1)	Well-structured and mostly clear. (1.5)	Well-organized, professional, and engaging documentation. (2)
Viva-Voce (Understanding of Concepts)	2 Marks	No understanding; could not answer questions. (0)	Answered a few with difficulty. (0.5)	Answered half the questions with basic clarity. (1)	Good understanding with confident answers. (1.5)	Answered all questions with clarity and depth. (2)



DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

ELECTRONIC DEVICES AND CIRCUITS LABORATORY

3. RUBRICS FOR SEMESTER END EXAMINATIONS

Criterion	Max Marks	Level-1 (Very Poor) (0–2 marks)	Level-2 (Poor)(3–4 marks)	Level-3 (Average)(5– 6 marks)	Level-4 (Good)(7–9 marks)	Level-5 (Excellent)(10–12 marks)
Preparedness for the Experiment	12 marks	No clarity on objective or procedure. Unable to explain basics.	Limited idea of the objective/procedure. Needed prompting.	Has basic understanding; minor gaps in concept or preparation.	Well-prepared, with clear understanding of steps and background.	Fully prepared with strong conceptual clarity and confident explanation.
Performance in the Laboratory	12 marks	Unable to perform experiment. Relied entirely on examiner's help.	Performed with multiple errors and constant support.	Performed with some errors; required occasional help.	Performed mostly independently with minimal support.	Performed independently, efficiently, and with precision.
Calculations & Graphs	12 marks	No or incorrect calculations. Graphs missing or irrelevant.	Multiple calculation errors. Graphs/plots inaccurate or poorly labeled.	Calculations partially correct. Graphs present but with some flaws.	Correct calculations and graphs with minor errors.	Accurate calculations and well-labeled graphs with proper interpretation.
Results & Error Analysis	12 marks	No result or invalid result. No error analysis attempted.	Incorrect result with vague or no error discussion.	Acceptable result. Error analysis attempted but limited.	Correct result with sound error discussion.	Accurate result with detailed and relevant error analysis.
Viva-Voce (Subject Knowledge)	12 marks	Unable to answer any questions. No conceptual understanding.	Answered few questions with poor logic.	Answered half of the questions with average understanding.	Answered most questions with clarity and confidence.	Answered all questions with depth, clarity, and reasoning.

Experiment No:1**Study the I–V characteristics of a PN junction diode in forward and reverse bias to determine cut in voltage and dynamic resistance**

AIM: To plot the V-I characteristics of a PN junction diode in both forward and reverse directions.

APPARATUS:

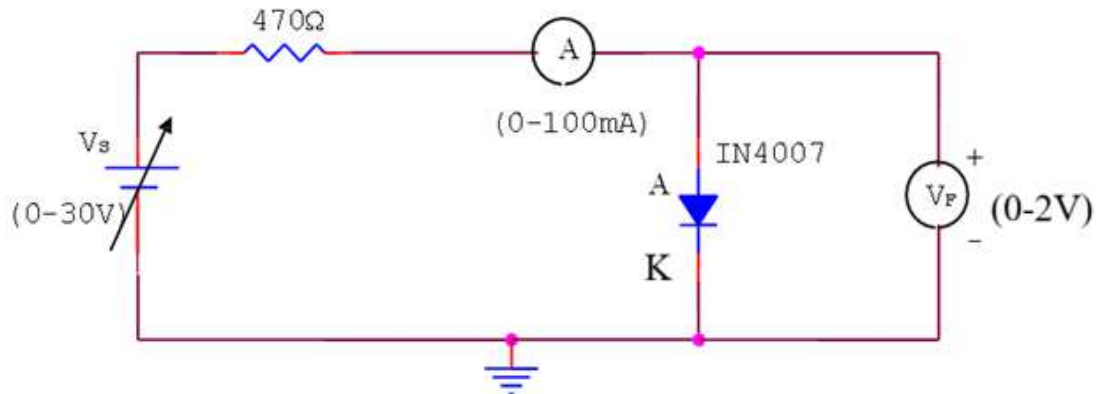
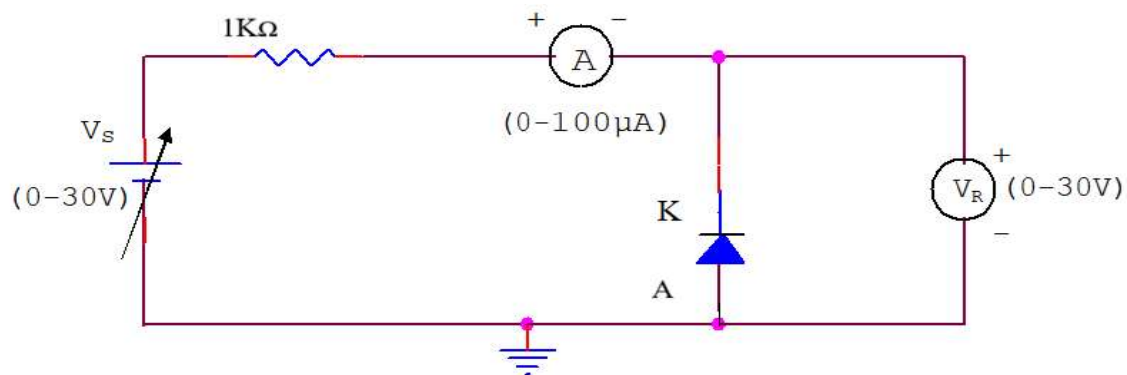
S.No	Apparatus	Type	Range	Quantity
01	PN Junction diode	1N4007		01
02	Resistance		470 Ω ,1K Ω	01
03	Regulated Power supply		(0-30V)	01
04	Ammeter		(0-100mA),(0-100 μ A)	01
05	Voltmeter		(0-2V),(0-30V)	01
06	LED			
07	Bread board and Wires			

THEORY:

A p–n junction diode conducts current only in one direction. The V–I characteristics of a diode represent the relationship between the voltage across the diode and the current flowing through it. When the external voltage is zero, the circuit is open and the potential barrier does not allow current to flow; therefore, the circuit current is zero.

When the p-type region (anode) is connected to the positive terminal and the n-type region (cathode) is connected to the negative terminal of the supply voltage, the diode is said to be forward biased. In the forward biased condition, the potential barrier is reduced. At a certain forward voltage, the potential barrier is completely eliminated and current starts flowing through the diode as well as the circuit. The diode is then said to be in the ON state. The current increases with an increase in forward voltage.

When the n-type region (cathode) is connected to the positive terminal and the p-type region (anode) is connected to the negative terminal of the supply voltage, the diode is said to be reverse biased. In this condition, the potential barrier across the junction increases, resulting in very high junction resistance. Hence, only a very small current known as reverse saturation current flows through the circuit. The diode is said to be in the OFF state. The reverse bias current is due to minority charge carriers

CIRCUIT DIAGRAM:**Forward bias of PN diode:****Reverse Bias of PN diode:****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 $1K\Omega$
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).

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 (μ A)

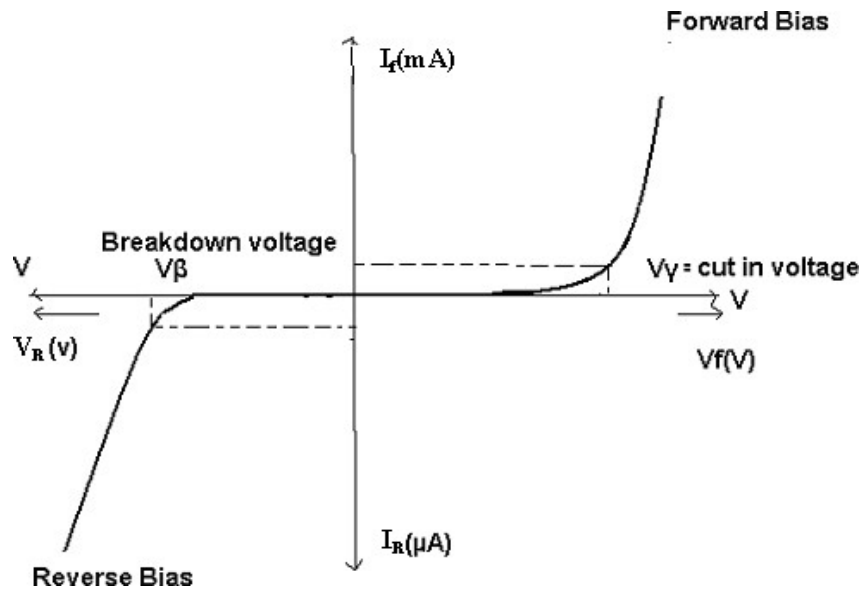
EXPECTED GRAPH:

Fig: Expected wave form of a PN junction Diode characteristics

RESULT:

Thus, the V-I characteristics of a PN junction diode in both forward and reverse bias are studied and plotted.

VIVA QUESTIONS:

1. What is a PN junction diode?
2. How is a PN junction formed?
3. What are P-type and N-type semiconductors?
4. What is the depletion region?
5. Why is the depletion region formed?
6. What is barrier potential?
7. What is the barrier potential of a silicon diode?
8. What is the barrier potential of a germanium diode?
9. What happens when a PN junction is forward biased?
10. What happens when a PN junction is reverse biased?
11. What is cut-in (knee) voltage?
12. What is reverse saturation current?
13. What is breakdown voltage?
14. What is avalanche breakdown?
15. What is Zener breakdown?
16. What is leakage current?
17. Why does a diode conduct only in one direction?
18. What are the V-I characteristics of a PN junction diode?
19. What is the maximum forward current of a diode?
20. What are the applications of a PN junction diode?
21. What is the difference between a PN junction diode and a Zener diode?
22. What is the use of a diode in rectifiers?

23. How is a diode represented in circuit diagrams?
24. What is reverse recovery time in a diode?
25. How does temperature affect a diode's operation?
26. What is the effect of increasing forward voltage on current?
27. What is a junction capacitance?
28. What is the role of minority carriers in reverse bias?
29. What is the difference between Si and Ge diodes?
30. What is the significance of the depletion layer width?

REAL TIME APPLICATIONS:

1. Rectification (AC to DC Conversion)

Used in half-wave, full-wave, and bridge rectifiers

Found in mobile chargers, power adapters, TVs, computers

2. Electronic Switching

Acts as an ON/OFF switch in digital and logic circuits

Used in signal routing and control circuits

3. Signal Detection (Demodulation)

Used in AM radio receivers

Extracts audio signal from modulated carrier wave

4. Clipping Circuits

Used to remove unwanted voltage levels

Applied in wave shaping and signal protection

5. Clamping Circuits

Used to shift voltage level of a signal

Applied in TV receivers and communication systems

6. Protection Circuits

Protects circuits from reverse polarity

Used in power supply inputs

7. Voltage Multipliers

Used in voltage doubler, tripler circuits

Applications in CRT displays, X-ray machines

8. Logic Gates

Used in diode logic (AND, OR gates)

Simple digital circuits

9. Solar Panels (Bypass Diodes)

Prevents reverse current flow

Improves efficiency and safety

Experiment No:2

Examine the reverse bias characteristics of a Zener diode and demonstrate its application as a voltage regulator under varying conditions.

AIM:

Plot the V-I characteristics of a Zener diode, find zener breakdown voltage in reverse bias condition and perform Zener diode voltage regulator.

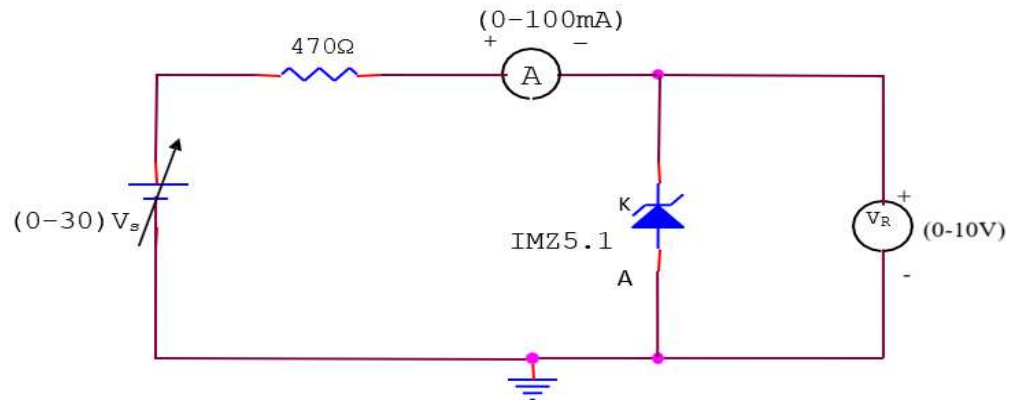
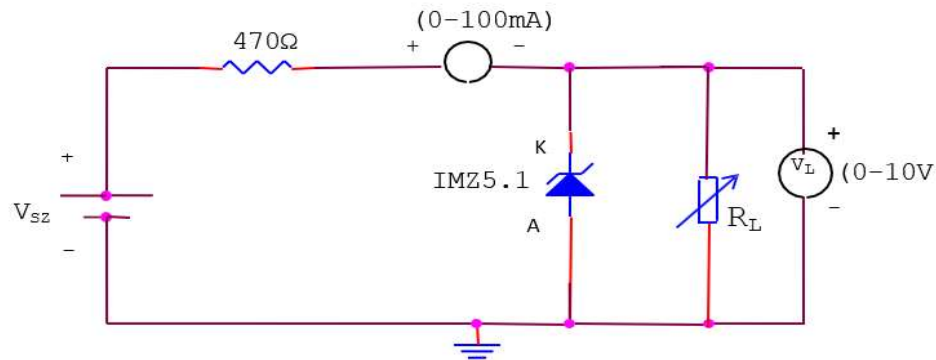
COMPONENTS REQUIRED:

S.No	Apparatus	Type	Range	Quantity
01	Zener diode	IMZ5.1V		01
02	Resistance		470 Ω	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	Bread board and Wires			

THEORY:

A zener diode is heavily doped p-n junction diode, specially made to operate in the break down region. A p-n junction diode normally does not conduct when reverse biased. But if the reverse bias is increased, at a particular voltage it starts conducting heavily. This voltage is called Break down Voltage. High current through the diode can permanently damage the device.

To avoid high current, we connect a resistor in series with zener diode. Once the diode starts conducting it maintains almost constant voltage across the terminals whatever may be the current through it, i.e., it has very low dynamic resistance. It is used in voltage regulators.

CIRCUIT DIAGRAM:**Reverse Biased Zener diode:****Zener diode as shunt Voltage Regulator:****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 1st circuit characteristics)
3. Measure V_{NL} (Noload voltage) by opening the load resistance.

4. Connect the load resistance, and vary the load resistance from 1100Ω to 100Ω in steps of 100Ω 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

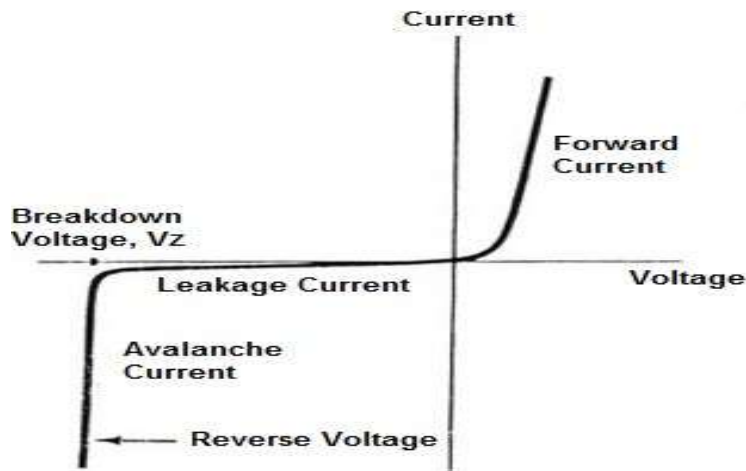
S.No	V_s (V)	V_R (V)	I_R (mA)

2. Find load regulation characteristics

$V_{NL} = \dots\dots\dots$

S.No	R_L	I_Z (mA)	V_L (V)	%Regulation
	1100 To 100 (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.

EXPECTEDGRAPH**Fig: Expected wave form Zener Diode Characteristics**

RESULT: Thus, the V-I characteristics of a Zener diode under forward and reverse bias are studied and plotted.

VIVAQUESTIONS:

1. What is a Zener diode?
2. How is a Zener diode different from a PN junction diode?
3. In which bias does a Zener diode operate?
4. What is Zener breakdown?
5. What is avalanche breakdown?
6. What is Zener voltage?
7. How is Zener voltage controlled during manufacturing?
8. What is the main application of a Zener diode?
9. Can a Zener diode be used in forward bias?
10. What is the role of series resistor in a Zener voltage regulator?
11. What is the reverse current in a Zener diode?
12. What is the maximum power rating of a Zener diode?
13. What happens if the Zener current exceeds the rated value?
14. What is dynamic resistance of a Zener diode?
15. How does a Zener diode maintain constant voltage?
16. What is the effect of temperature on Zener voltage?
17. What are the practical applications of Zener diode?
18. Can Zener diode be used as a rectifier?
19. What is the difference between Zener diode and normal diode in reverse bias?
20. How is a Zener diode represented in circuit diagrams?

REALTIME APPLICATIONS:**1.Voltage Regulation**

Used in power supplies to maintain a constant output voltage despite changes in input voltage or load.

2. Overvoltage Protection

Protects electronic circuits from damage by limiting excess voltage.

3.Voltage Reference Circuits

Provides a fixed reference voltage in operational amplifiers, ADCs, and DACs.

4. Surge and Transient Protection

Protects circuits from sudden voltage spikes caused by lightning or switching.

5. Clipping Circuits

Used to limit the maximum voltage level of a signal.

6. Clamping Circuits

Helps shift voltage levels in waveform shaping applications.

7.Meter Protection

Protects voltmeters and measuring instruments from overload.

8.Switching Circuits

Used as a voltage-controlled switch in electronic circuits.

9.Automotive Electronics

Protects ECUs and sensors from voltage fluctuations.

10.Battery Chargers and UPS Systems

Maintains safe voltage levels during charging and power backup.

Experiment No:3

Design and analyze half-wave and full-wave rectifiers (center-tap and bridge) with and without capacitor filters to evaluate ripple factor and output voltage.

HALF-WAVE RECTIFIER

AIM: To rectify the AC signal and to determine the ripple factor and percentage of regulation of a half-wave rectifier with and without a capacitor filter.

APPARATUS:

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

Theory:

A device that converts a sinusoidal input waveform into a unidirectional waveform with a 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 the entire input voltage appears across the load resistance R_L . During the negative half-cycle, the diode is reverse biased and behaves like an open circuit; hence, the output voltage is zero. The filter used is a capacitor connected from the rectifier output to ground. The capacitor charges quickly during the rising portion of the input voltage and discharges slowly through R_L after the positive peak. The variation in the capacitor voltage due to this charging and discharging is called ripple voltage. Since ripple is undesirable, smaller ripple indicates better filtering action.

A full-wave rectifier consists of two half-wave rectifiers connected to a common load. One rectifies during the positive half-cycle of the input, while the other rectifies during the negative half-cycle. The transformer supplies the two diodes (D1 and D2) with sinusoidal voltages that are equal in magnitude but opposite in phase. During the positive half-cycle, diode D1 conducts and diode D2 remains OFF. During the negative half-cycle, diode D1 is OFF and diode D2 conducts. As in the half-wave rectifier, ripple is undesirable, and better filtering is achieved by reducing the ripple content.

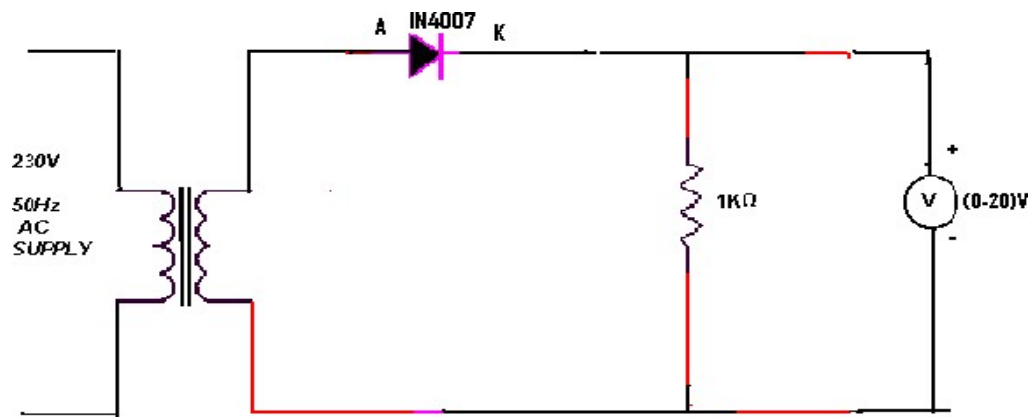
CIRCUITDIAGRAM:**WITHOUT FILTER:**

Fig: Circuit Diagram of Half Wave Rectifier without Filter

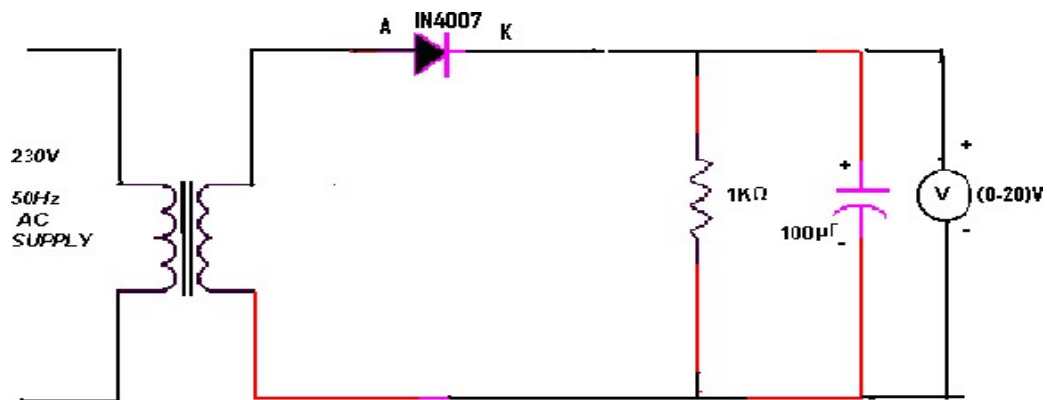
WITH CAPACITOR FILTER:

Fig: Circuit diagram of Half Wave Rectifier With capacitor Filter.

CALCULATIONS:**Theoretical calculations for Ripple factor of Half wave Rectifier: -****Without Filter:**

$$V_{dc} = V_m/\pi$$

$$V_{rms} = V_m/2$$

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

With Capacitor Filter:

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

Where $f=50\text{Hz}$

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

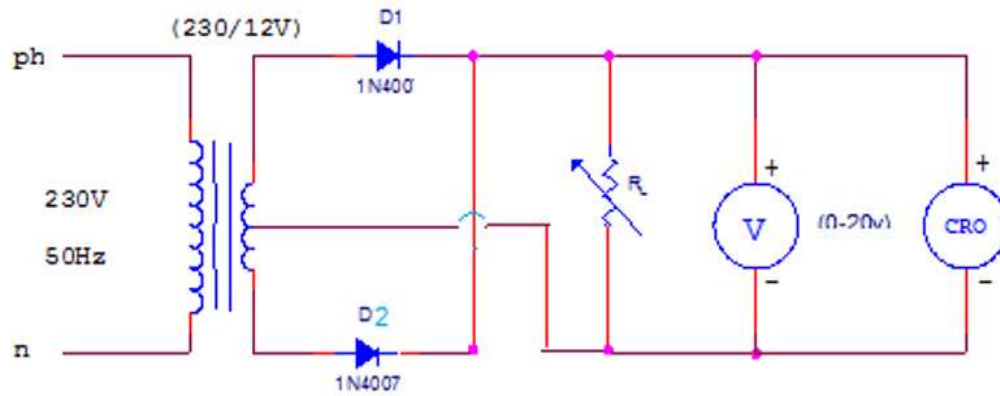
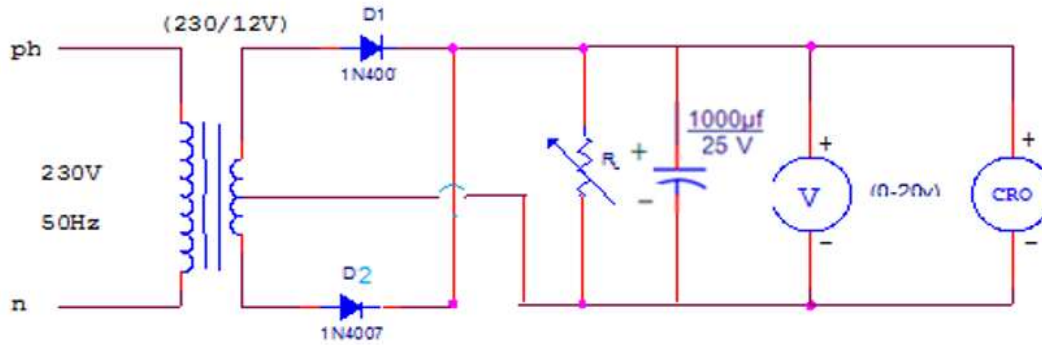
$R_L=1\text{K}\Omega$

PROCEDURE:**HWR Without Filter:**

1. Connections are made as per the circuit diagram of the rectifier with out 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 100Ω by varying the DRB from 1100Ω to 100Ω 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 wave form 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 halfwave rectifier according to the given formulae.
8. Measure the amplitude and time period 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 the AC mains and the secondary side to the rectifier input.
3. Using a multimeter, measure the AC input voltage of the rectifier and the AC and DC voltages at the rectifier output.
4. Measure the amplitude and time period of the transformer secondary (input waveform) by connecting the CRO.
5. Feed the rectified output voltage to the CRO and measure the amplitude and time period of the output waveform.

Full wave Rectifier without Filter**Full wave Rectifier with capacitor Filter**

Theoretical calculations for Ripple factor of Full wave 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:**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 the 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 wave form) by connecting the 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) byconnecting 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 (Ω)	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:

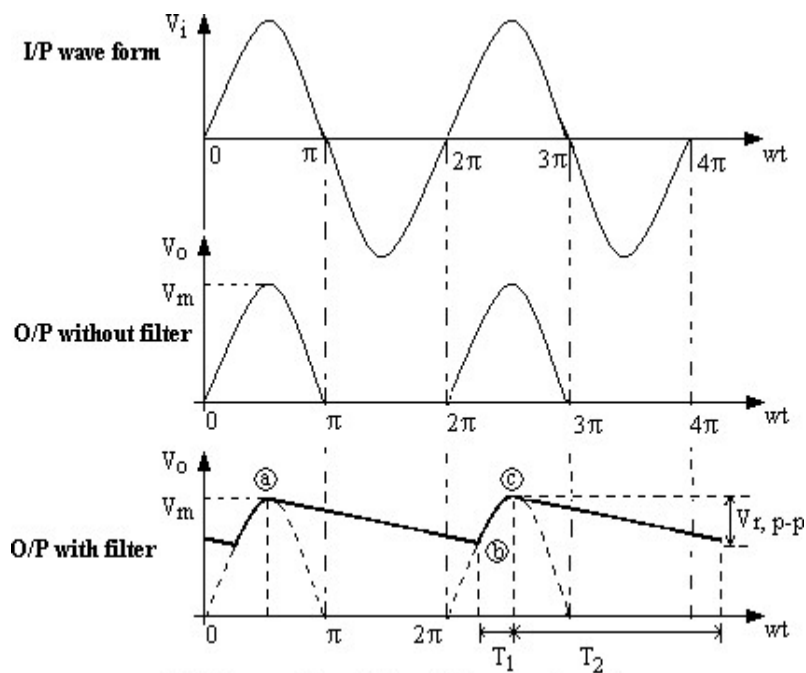
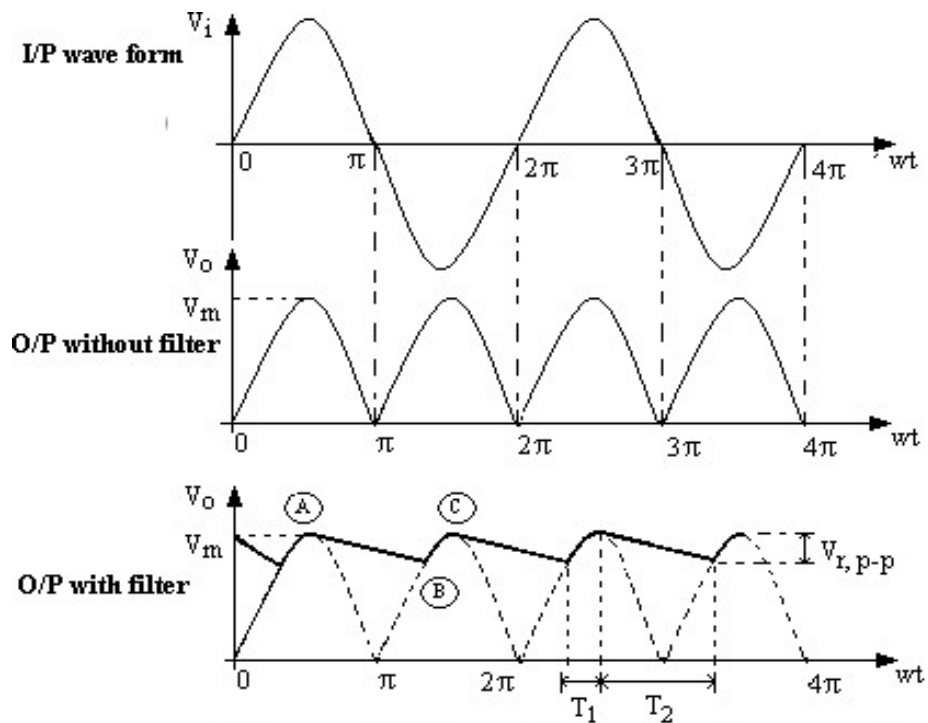
R_L (Ω)	V_L (V)	V_m (V)	V_r (V)	$V_{dc} = V_m \frac{V_r}{2}$ (V)	$V_r(rms)$ V_r $= \frac{V_r}{2\sqrt{3}}$	$R.F = \frac{V_r(rms)}{V_{dc}}$

FWR without Filter:

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

FWR with capacitor filter:

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

EXPECTED WAVE FORMS:**Half-wave Rectifier with capacitor filter wave form****Full-wave Rectifier with capacitor filter wave form**

RESULT:

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

Ripple factor of HWR without filter =

Ripple factor of HWR with capacitor filter =

Ripple factor of FWR without filter =

Ripple factor of FWR with capacitor filter =

VIVAQUESTIONS:

- 1.What is a rectifier?
2. What is the main function of a rectifier?
3. What is a half-wave rectifier?
- 4.How many diodes are used in a half-wave rectifier?
- 5.What are the disadvantages of a half-wave rectifier?
- 6.What is a full-wave rectifier?
- 7.What is the difference between half-wave and full-wave rectifier?
- 8.What is a center-tap full-wave rectifier?
- 9.Why is a center-tapped transformer required?
- 10.What is a bridge rectifier?
- 11.How many diodes are used in a bridge rectifier?
- 12.Which rectifier does not require a center-tapped transformer?
- 13.What is ripple factor?
- 14.What is the ripple factor of a half-wave rectifier?
- 15.What is the ripple factor of a full-wave rectifier?
- 16.Why is ripple factor less in a full-wave rectifier?
- 17.What is the average DC output voltage of a half-wave rectifier?
- 18.What is the average DC output voltage of a full-wave rectifier?
- 19.What is the purpose of a capacitor filter?
- 20.How does a capacitor filter reduce ripple?

REALTIME APPLICATIONS:**1. Power Supplies (AC to DC Conversion)**

Used in TVs, computers, mobile chargers, laptops

Converts AC mains supply into DC required by electronic circuits

2. Battery Charging Circuits

Used in mobile phones, UPS, inverters, electric vehicles

Rectifier converts AC into DC to charge batteries safely

3. DC Motor Drives

Used in electric trains, cranes, lifts, rolling mills

Rectifiers supply controlled DC voltage to DC motors

4. Radio and Communication Systems

Used in AM radio receivers

Detects and converts modulated AC signals into audio signals

5. Electroplating and Electrolysis

Used in gold plating, silver plating, metal purification

Requires DC supply obtained using rectifiers

6. Welding Equipment

Used in arc welding machines

Rectifiers convert AC to high-current DC for stable welding

7. Uninterruptible Power Supply (UPS)

Used in hospitals, data centers, computers

Rectifier charges battery and supplies DC to inverter

8. HVDC Transmission Systems

Used in long-distance power transmission

Rectifiers convert AC to DC at sending end

9. X-Ray Machines

Used in medical imaging

Rectifier converts AC into high-voltage DC

10. Signal Demodulation

Used in communication receivers

Rectifier extracts information signal from carrier wave

Experiment No:4

Implement clipper and clamper circuits and observe waveform shaping for positive, negative, and biased configurations.

AIM: To study the clipping circuits for different reference voltages and to verify their responses.

APPARATUS:

1. Resistors- $1K\Omega$
2. IN4007Diode-2No.
3. Breadboard.
4. CRO(1Hz-20MHz)
5. Function Generator(1Hz-1MHz)
6. Power supply(0-30V)
7. Connecting wires.

THEORY:

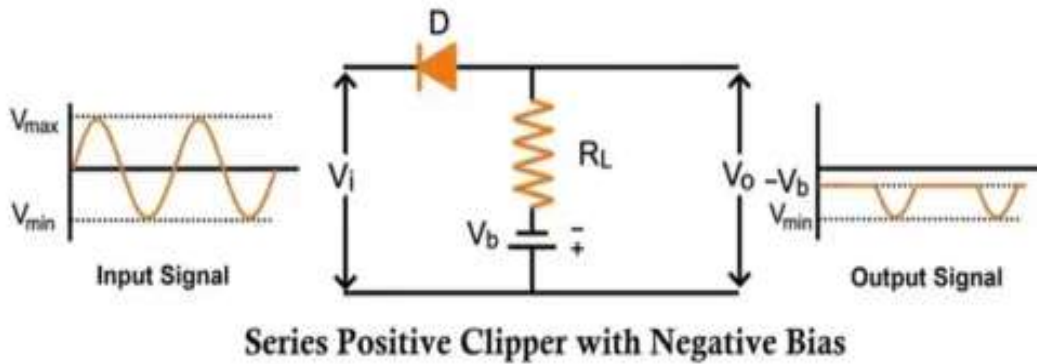
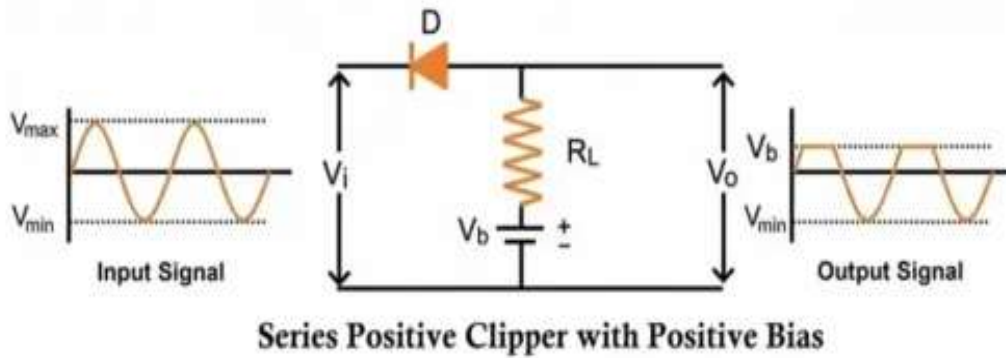
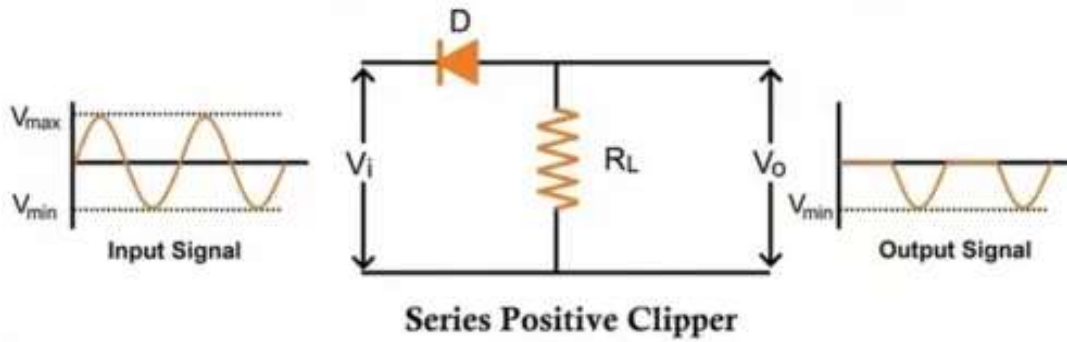
A non-linear semiconductor diode in combination with a resistor can function as a clipper circuit. Energy-storage components are not required in the basic process of clipping. These circuits select a portion of an arbitrary waveform that lies above or below a particular reference voltage level, and the selected portion of the waveform is used for transmission. Hence, they are also referred to as voltage limiters, current limiters, amplitude selectors, or slicers.

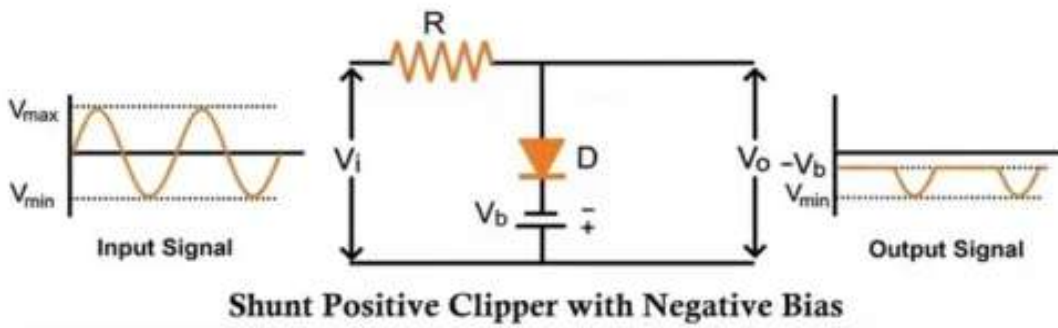
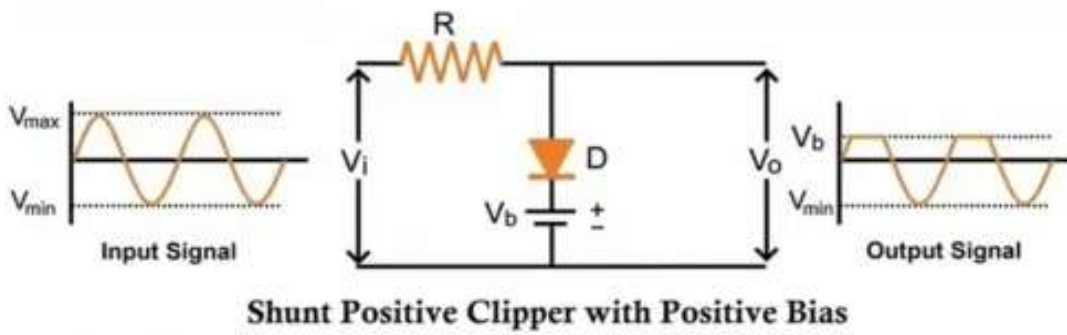
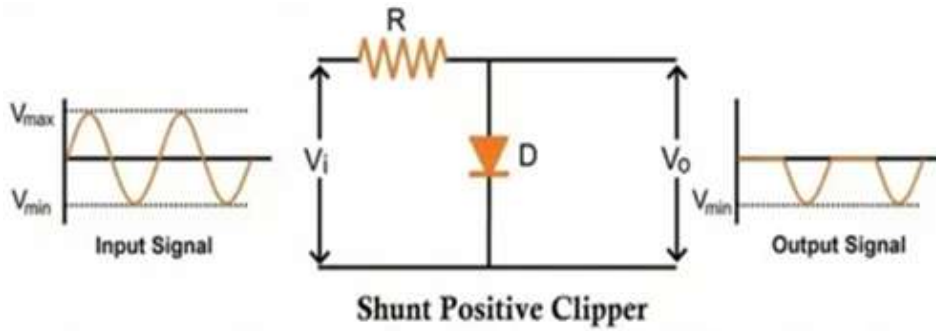
There are three different types of clipping circuits:

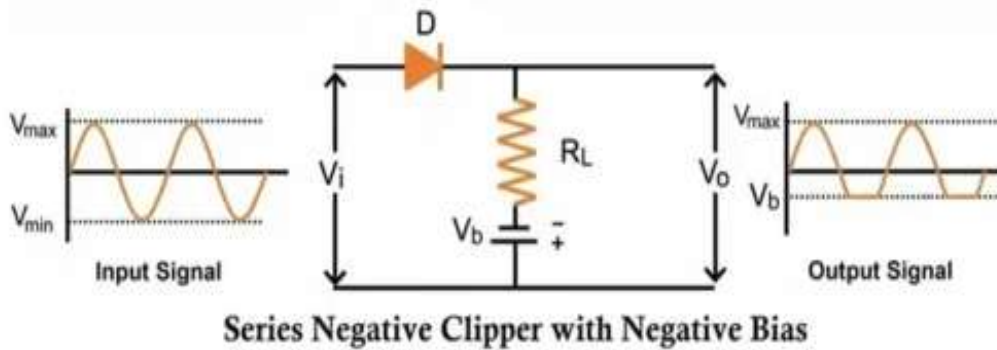
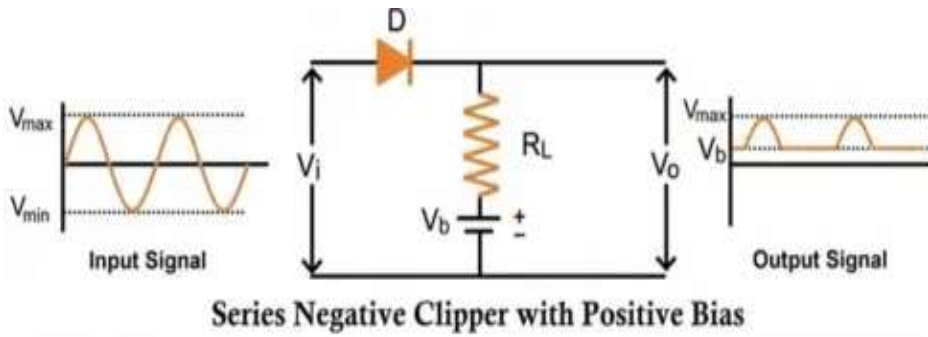
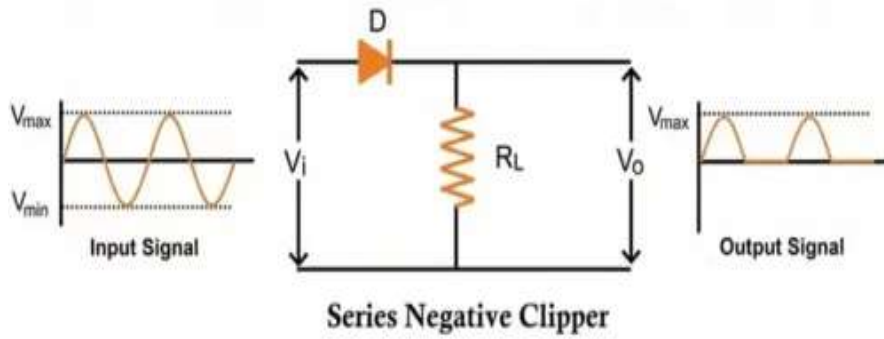
1. Positive clipping circuit
2. Negative clipping circuit
3. Positive and negative clipping circuit (slicer)

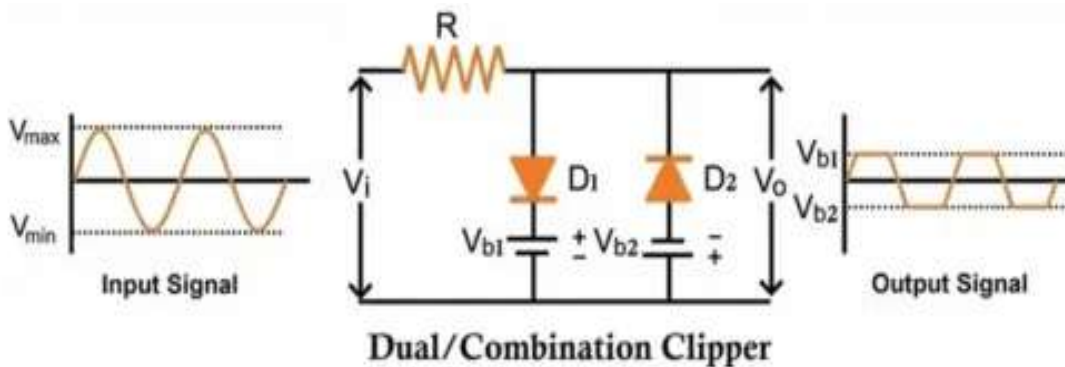
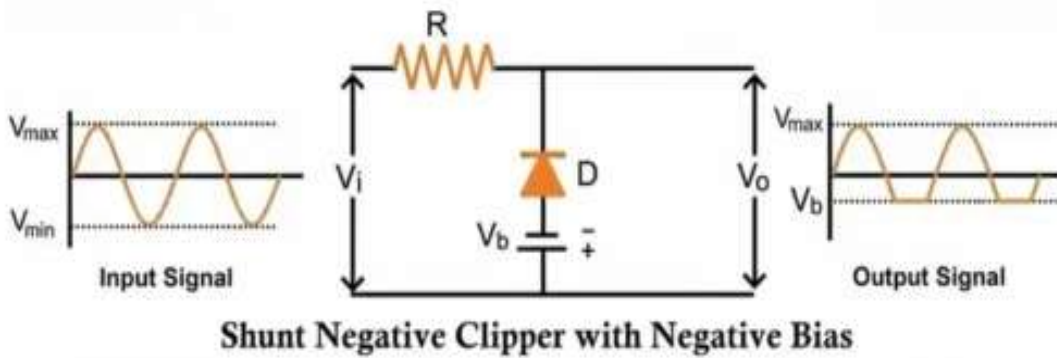
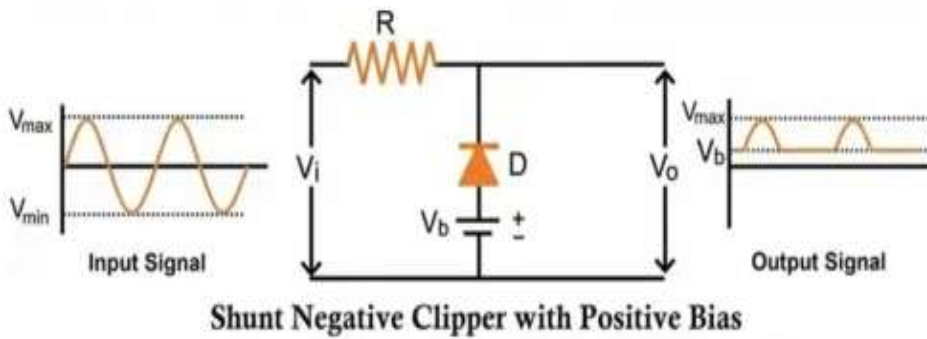
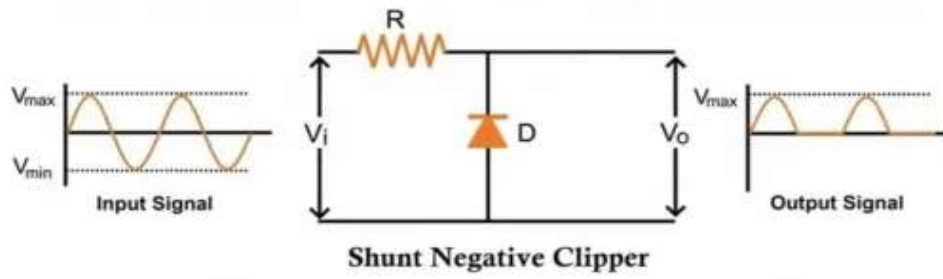
In a positive clipping circuit, the positive portion of the sinusoidal signal is clipped, and only the negative portion appears at the output. When a reference voltage is added, instead of clipping the entire positive cycle, only the portion of the positive cycle that exceeds the reference voltage level is clipped.

In a negative clipping circuit, the negative portion of the sinusoidal signal is clipped, while the positive portion appears at the output.

CIRCUIT DIAGRAM:**SERIES POSITIVE CLIPPER WITH AND WITHOUT BIAS:**

SHUNT POSITIVE CLIPPER WITH AND WITHOUT BIAS:

SERIES NEGATIVE CLIPPER WITH AND WITHOUT BIAS:

SHUNT NEGATIVE CLIPPER WITH AND WITHOUT BIAS:

Negative Clipper:

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

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

where V_r is the **cut-in (threshold) 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. Hence, the output voltage is limited to

$$V_o = V_R + V_r$$

Transfer Characteristic Equation:

$$V_o = \begin{cases} V_i, & \text{for } V_i < V_R + V_r \\ V_R + V_r, & \text{for } V_i > V_R + V_r \end{cases}$$

Positive Clipper:

When $V_i > V_R + V_r$, the diode is forward biased and conducts. Since the diode is **ON**, it behaves like a short circuit, and therefore,

$$V_o = V_R + V_r$$

When $V_i < V_R + V_r$, the diode is reverse biased and hence **OFF**. It acts as an open circuit, and therefore,

$$V_o = V_i$$

Transfer Characteristic Equation:

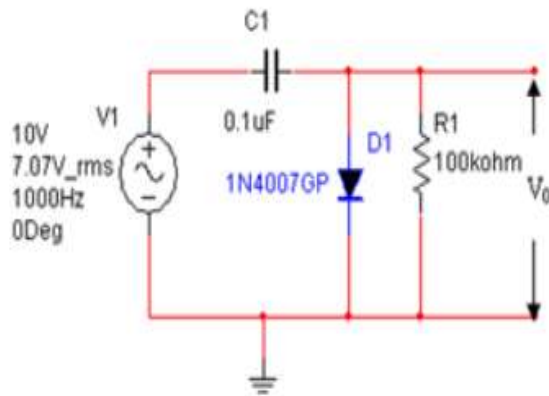
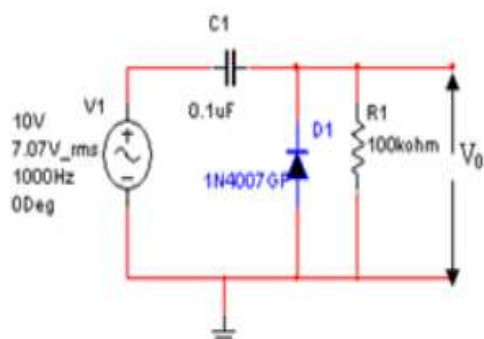
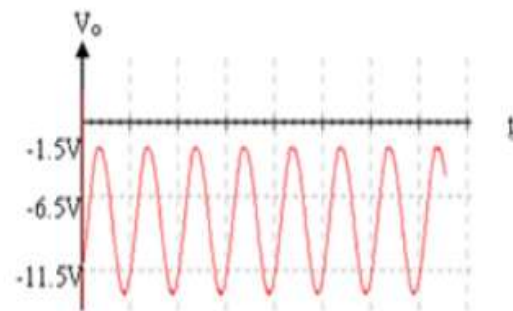
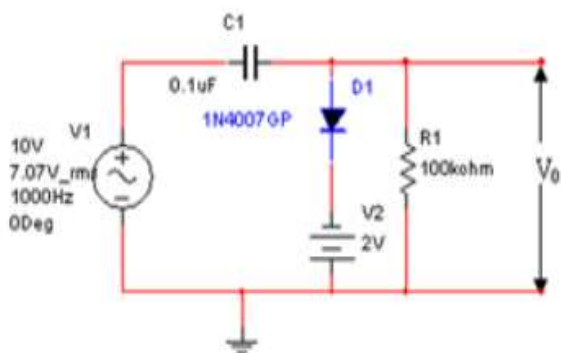
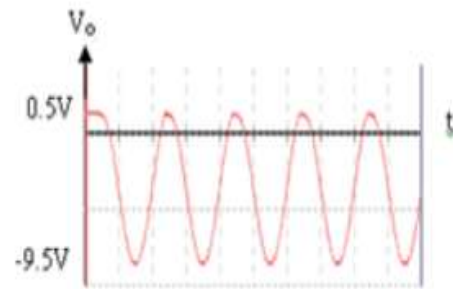
$$V_o = \begin{cases} V_i, & \text{for } V_i < V_R + V_r \\ V_R + V_r, & \text{for } V_i > V_R + V_r \end{cases}$$

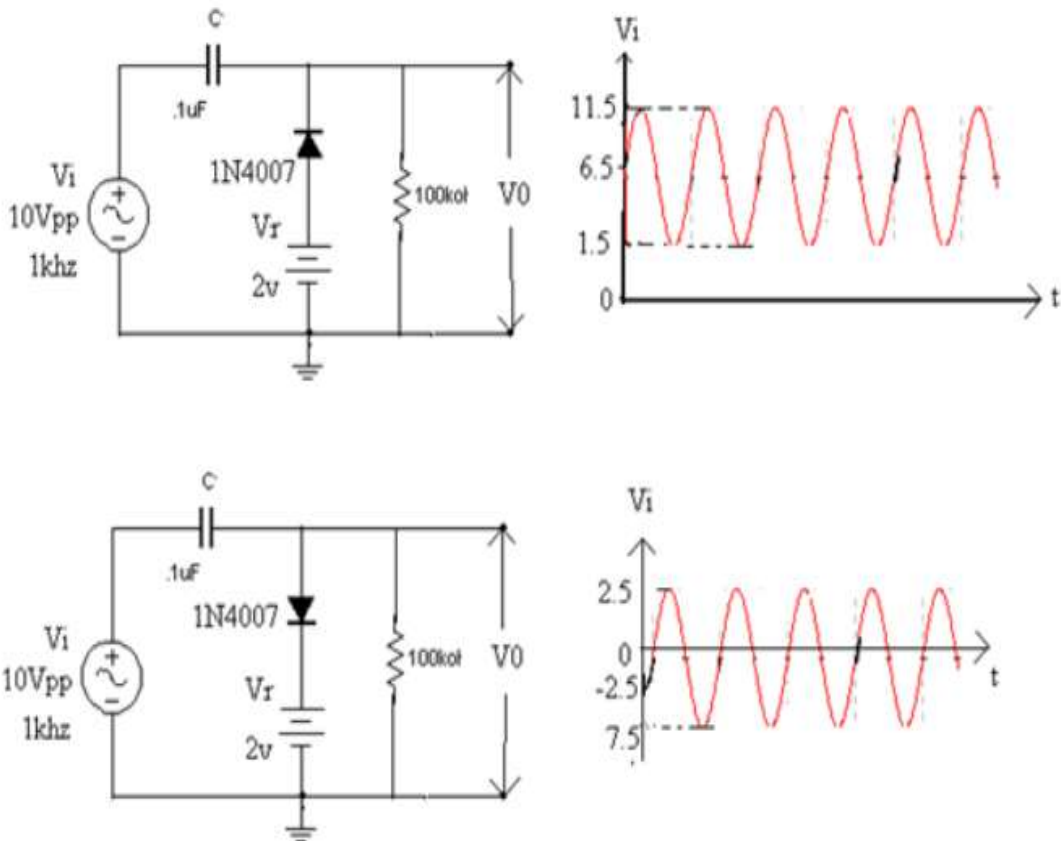
PROCEDURE:

1. Connect the circuit as shown in the figures.
2. In each case, by applying different value of reference voltage change the peak voltage of input signal and note the theoretical and practical clipping values.
3. Observe the Output waveform (V_O in the circuit) on the CRO and compare it with theoretical values.
4. Sketch the Input as well as Output waveforms and mark the voltage levels.
5. Obtain the transfer characteristics of Clipper circuit, by keeping CRO in X-Y mode.

OBSERVATIONTABLE:

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)						

CLAMPERS:**CIRCUIT DIAGRAM****O/P WAVEFORMS**

**RESULT:**

Thus, Different types of Clippers and clamping circuits are studied and observed the response for different combinations of V_R and diodes.

VIVA QUESTIONS:

1. What is a clipper circuit?
2. What is the main function of a clipper?
3. What are the types of clippers?
4. What is a series clipper?
5. What is a shunt (parallel) clipper?
6. How does a positive clipper work?
7. How does a negative clipper work?
8. What is the effect of adding a bias voltage in a clipper?
9. What are the practical applications of clippers?
10. Can clippers be used with AC and DC signals?
11. What is the difference between a biased and unbiased clipper?
12. What happens when a diode in a clipper is reversed?

13. What is the voltage level at which clipping occurs?
14. How can you change the clipping level?
15. Why are clippers called “limiter circuits”?
16. What is a clamper circuit?
17. What is the main function of a clamper?
18. What is the difference between a clipper and a clamper?
19. What are the types of clamper circuits?
20. What is a positive clamper?
21. What is a negative clamper?
22. How does a capacitor work in a clamper circuit?
23. What is the effect of adding a DC bias in a clamper?
24. What are the practical applications of clampers?
25. Can a clamper shift an AC waveform up or down?
26. What is the difference between series and shunt clamper?
27. How do you choose the capacitor and resistor values in a clamper?
28. Why is a diode used in a clamper circuit?
29. What is the output voltage level of a positive clamper?
30. How can a clamper be used in TV and signal processing circuits?

REAL-TIME APPLICATIONS OF CLIPPERS:

Clippers are used to remove or limit unwanted portions of a signal without distorting the remaining part. In real-time applications, they are used in wave-shaping circuits to protect electronic devices from voltage spikes by limiting peak voltage levels. Clippers are commonly used in TV and radio receivers to remove noise, in signal processing circuits to extract information signals, and in over-voltage protection circuits for sensitive components. They are also used in digital circuits to produce square waveforms from sinusoidal inputs, making them essential in pulse shaping and logic circuits.

REAL-TIME APPLICATIONS OF CLAMPERS:

Clampers are used to shift the DC level of a waveform without changing its shape. In practice, they are used in television and video circuits to restore the required DC level of video signals. They are also applied in communication systems to adjust signal levels, in oscilloscope circuits to display signals above or below the reference level, and in analog signal processing where waveform translation is required. Clampers are widely used in pulse circuits, voltage level shifters, and AC signal conditioning.

Experiment No:5**Plot the input and output characteristics of a BJT in common emitter configuration to determine input/output resistance and current gain****AIM:-**

1. To study the input and output characteristics of a transistor (BJT) connected in common emitter Configuration.
2. To calculate current gain β .
3. To calculate input resistance R_i & output resistance R_o .

EQUIPMENTS & COMPONENTS REQUIRED:

S.No	Device	Range/Rating	Qty
1.	Regulated DC supply voltage (RPS)	0-30V	1
2.	Voltmeter	0-1V or 0-10v, 0-20V	1
3.	Ammeter	0-10mA, 200mA	1
4.	Connecting wires & breadboard		
5	Transistor BC 107 or 2n2222 or BC547	NPN	1
6	Resistor	1K, 100K	1

THEORY:

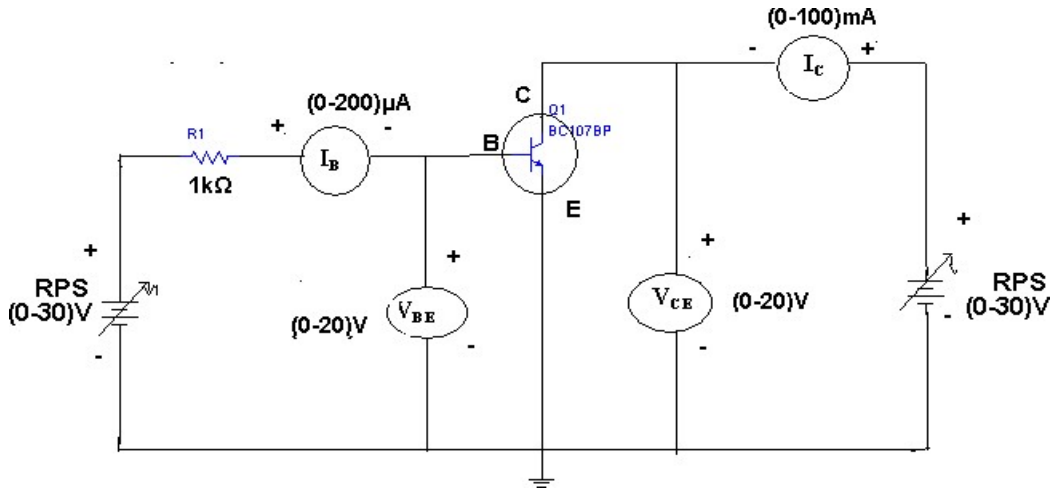
A transistor is a three-terminal device. The terminals are **emitter, base, and collector**. In the **common emitter (CE) configuration**, the input voltage is applied between the **base and emitter** terminals, and the output is taken across the **collector and emitter** terminals. Hence, the emitter terminal is common to both input and output circuits.

The **input characteristics** of a CE transistor resemble that of a **forward-biased diode**, since the base-emitter junction of the transistor is forward biased. Compared to the **common base (CB) configuration**, the base current I_B increases less rapidly with V_{BE} . Therefore, the **input resistance** of the CE circuit is higher than that of the CB circuit.

The **output characteristics** are plotted between the **collector current I_C** and **collector-emitter voltage V_{CE}** at constant I_B . The collector current varies with V_{CE} only up to a few volts. Beyond this, the collector current becomes almost constant and is independent of V_{CE} . The value of V_{CE} up to which the collector current changes is known as the **knee voltage**.

The transistor is normally operated in the region **above the knee voltage**, where I_C remains nearly constant and is approximately proportional to I_B . The **current amplification factor** of the CE configuration is given by:

$$\beta = \frac{\Delta I_C}{\Delta I_B}$$

CIRCUIT DIAGRAM:**Fig 1 Input and Output Characteristics of a CE****PROCEDURE:-****Input Characteristics:**

1. Connect the circuit as per the circuit diagram for input characteristics.
2. Keep the collector-to-emitter voltage $V_{CE} = 0\text{ V}$ by adjusting the collector supply voltage V_{CC} . Increase the base supply voltage V_{BB} gradually from 0 V onwards (0.1 V, 0.2 V ... 0.75 V) and observe the base current I_B for different values of base-to-emitter voltage V_{BE} .
3. Repeat Step 2 for different collector-to-emitter voltages V_{CE} , such as 3 V and 6 V.
4. Tabulate the observed readings and plot the corresponding graph.

Output Characteristics:

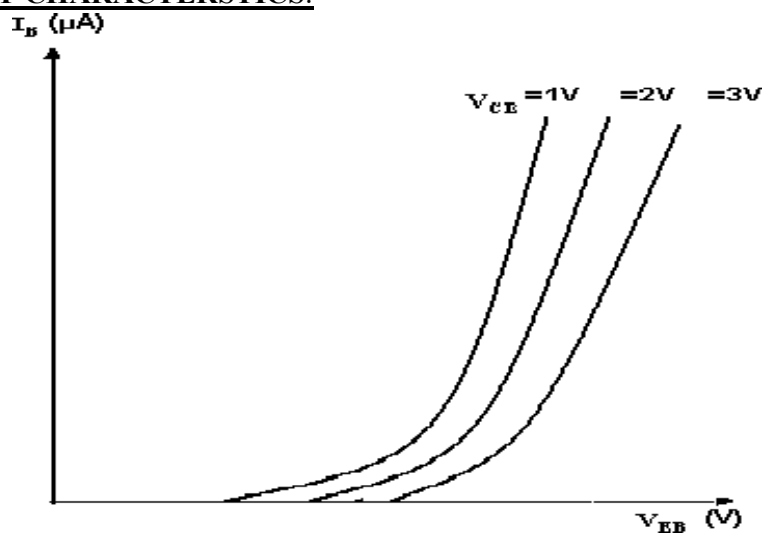
1. Connect the circuit as per the circuit diagram for output characteristics.
2. Keep the collector supply voltage $V_{CC} = 0\text{ V}$. Increase the base supply voltage V_{BB} to set the base current $I_B = 3\text{ }\mu\text{A}$.
3. Now increase the collector supply voltage V_{CC} gradually from 0 V onwards and observe the collector current I_C for different values of collector-to-emitter voltage V_{CE} , ensuring that the rated value of collector current ($I_C = 15\text{ mA}$) is not exceeded.
4. Tabulate the readings in tabular form and plot the corresponding graph.

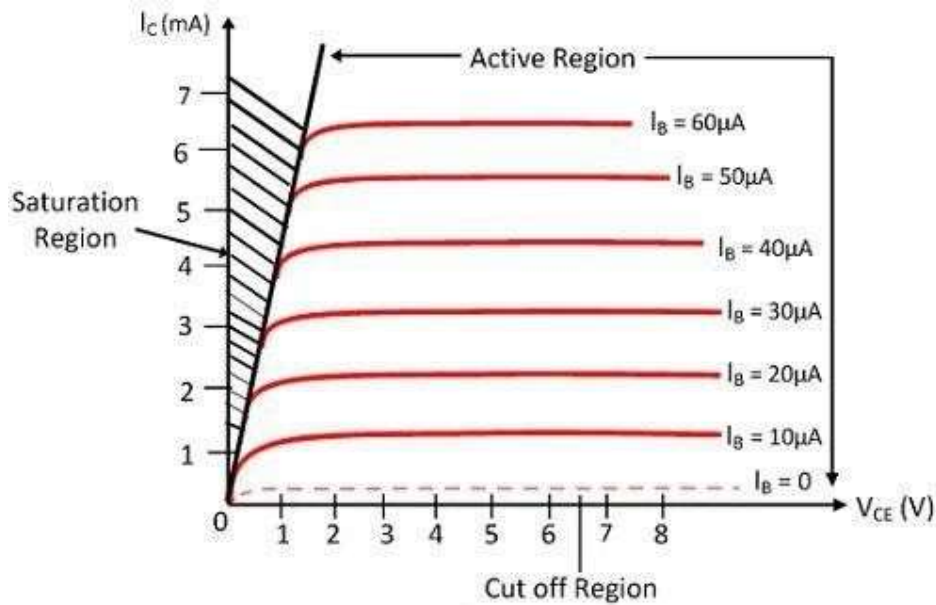
OBSERVATIONS:**INPUT CHARACTERISTICS:**

S.NO	VCE=1V		VCE=2V		VCE=4V	
	V _{BE} (V)	I _B (μ A)	V _{BE} (V)	I _B (μ A)	V _{BE} (V)	I _B (μ A)

OUTPUT CHARACTERISTICS:

S.NO	I _B =50 μ A		I _B =75 μ A		I _B =100 μ A	
	V _{CE} (V)	I _C (mA)	V _{CE} (V)	I _C (mA)	V _{CE} (V)	I _C (mA)

EXPECTED GRAPHS:**INPUT CHARACTERISTICS:****Fig: Input Characteristics of a CE**

OUTPUT CHARACTERISTICS:**Fig: Output Characteristics of a CE configuration****Calculations from Graph:****1. Input Characteristics:**

To obtain the **input resistance**, determine the change in base-emitter voltage ΔV_{BE} and the corresponding change in base current ΔI_B at a **constant collector-emitter voltage** V_{CE} from one of the input characteristic curves.

$$\text{Input resistance} = h_{ie} = R_i = \frac{\Delta V_{BE}}{\Delta I_B} \quad (V_{CE} = \text{constant})$$

The **reverse voltage gain** is obtained by determining the change in base-emitter voltage ΔV_{BE} for a change in collector-emitter voltage ΔV_{CE} at **constant base current** I_B .

$$\text{Reverse voltage gain} = h_{re} = \frac{\Delta V_{BE}}{\Delta V_{CE}} \quad (I_B = \text{constant})$$

2. Output Characteristics:

To obtain the **output resistance**, determine the change in collector current ΔI_C and the corresponding change in collector-emitter voltage ΔV_{CE} at a **constant base current** I_B from one of the output characteristic curves.

$$\text{Output resistance} = R_o = \frac{1}{h_{oe}} = \frac{\Delta V_{CE}}{\Delta I_C} \quad (I_B = \text{constant})$$

The **forward current gain** is obtained by determining the ratio of change in collector current to change in base current at **constant collector-emitter voltage** V_{CE} .

$$\text{Forward current gain} = h_{fe} = \frac{\Delta I_C}{\Delta I_B} \quad (V_{CE} = \text{constant})$$

RESULT:

Input and Output characteristics of a Transistor in Common Emitter Configuration are studied and find out the input resistance, output resistance, β .

VIVA QUESTIONS:

1. What is a BJT?
2. What is the common-emitter configuration?
3. What are the real-time applications of CE BJT?
4. What is the input characteristic of a CE transistor?
5. What is the output characteristic of a CE transistor?
6. What is the cut-off region of a CE transistor?
7. What is the active region of a CE transistor?
8. What is the saturation region of a CE transistor?
9. What is the current gain (β) in CE configuration?
10. What is the phase relationship between input and output in CE BJT?
11. What is the typical input impedance of a CE transistor?
12. What is the typical output impedance of a CE transistor?
13. Why is CE configuration widely used for amplification?
14. Why is CE configuration used in switching applications?
15. How does collector current vary with base current in the active region?
16. How does V_{CE} affect the input characteristics?
17. How is a CE BJT represented in a circuit diagram?
18. What happens when a CE transistor is in saturation?
19. What happens when a CE transistor is in cut-off?
20. What is the significance of the active region for linear amplification?

REAL-TIME APPLICATIONS:

The Bipolar Junction Transistor (BJT) in common-emitter (CE) configuration is widely used in real-time applications such as audio and **RF amplifiers, electronic switches, oscillators, signal modulation circuits, voltage regulators, and pulse shaping circuits**, because it provides high current, voltage, and power gain. Its input characteristics show the base current (I_B) versus base-emitter voltage (V_{BE}), where the input behaves like a forward-biased diode, while the output characteristics plot collector current (I_C) versus collector-emitter voltage (V_{CE}) for different base currents, showing cut-off, active, and saturation regions. In the active region, $I_C \approx \beta \times I_B$, providing linear amplification, and the output voltage is 180° out of phase with the input, making CE configuration ideal for amplification and switching in practical electronic circuits

Experiment No:6**Design and test fixed bias and voltage divider bias circuits to establish a stable operating point for a BJT amplifier and study DC load line behaviour**

AIM:To design and test fixed bias and voltage divider bias circuits to establish a stable operating point (Q-point) for a BJT amplifier and to study the DC load line behavior.

APPARATUS:

1. DC Power Supply
2. BJT NPN Transistor
3. Resistors (Various values)
4. Digital Multimeter
5. Breadboard
6. Connecting Wires

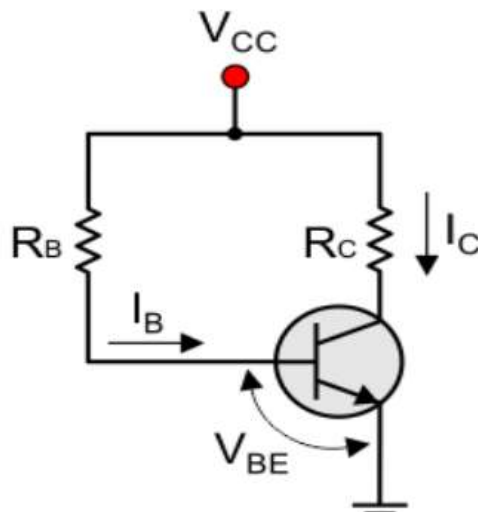
CIRCUIT DIAGRAM:

Fig:Fixed Bias

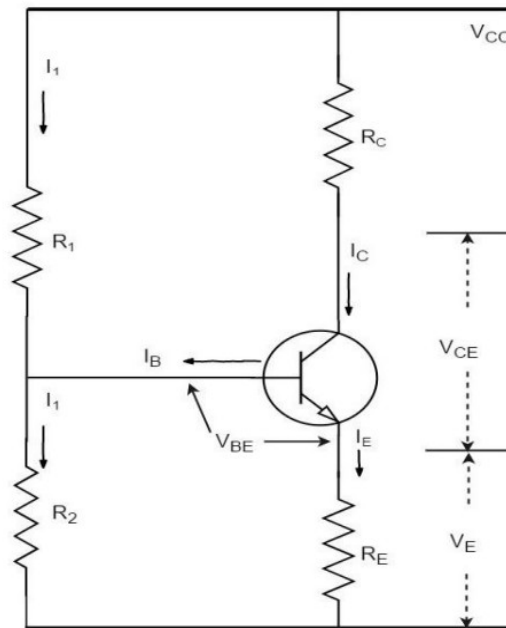


Fig: Voltage divider bias

THEORY: -

Biasing is the process of providing a DC voltage to a transistor to ensure proper operation of the circuit. A transistor is biased so that the emitter–base junction is forward biased and the collector–base junction is reverse biased, thereby maintaining the transistor in the active region for amplifier operation.

Transistor Biasing:-

The proper flow of zero-signal collector current and the maintenance of an appropriate collector–emitter voltage during the passage of a signal is known as **transistor biasing**. The circuit that provides the required biasing to the transistor is called a **biasing circuit**.

Methods of Transistor Biasing:

Biasing in transistor circuits is usually achieved using two DC sources, V_{BB} and V_{CC} . However, to simplify the circuit and make it more economical, it is preferable to use a single DC supply instead of two.

The commonly used methods of transistor biasing are:

- **Base Resistor Method (Fixed Bias)**
- **Emitter Bias Method**
- **Voltage Divider Bias Method**

Base Resistor Method (Fixed Bias):

In the base resistor (fixed bias) method, the biasing circuit consists of a base resistor R_B connected between the base terminal of the transistor and the supply voltage V_{CC} . The base–emitter junction of the

Transistor is forward biased by the voltage drop across R_B which is caused by the base current I_B flowing through it.

From the circuit, the mathematical expression for the base current I_B is given by:

$$I_B = \frac{V_{CC} - V_{BE}}{R_B}$$

Here the values of V_{CC} and V_{BE} are fixed while the value for R_B is constant once the circuit is designed. This leads to a constant value for I_B resulting in a fixed operating point due to which the circuit is named as fixed base bias. This kind of bias, results in a stability factor of $(\beta+1)$ which leads to very poor thermal stability. The reason behind this is the fact the β parameter of a transistor is unpredictable and varies up to a large extent even in the case of transistor with the same model and type. This variation in β results in large changes in I_C which cannot be compensated by any means in the proposed design. Hence it can be concluded that this kind of β dependent bias is prone to the changes in operating point brought about by the variations in transistor characteristics and temperature. However it is to be noted that fixed base bias is most simple and uses less number of components. Moreover it offers the chance for the user to change the operating point anywhere in the active region just by changing the value of R_B in the design. Further it offers no load on the source as there is no resistor across base-emitter junction. Due to these factors this kind of biasing is used in switching applications and to achieve automatic gain control in the transistors. Here, the expressions for other voltages and currents are given as

$$\begin{aligned} V_B &= V_{BE} = V_{CC} - I_B R_B \\ V_C &= V_{CC} - I_C R_C = V_{CC} - V_{CE} \\ I_C &= \beta I_B \\ I_E &\approx I_C \end{aligned}$$

Voltage-divider Bias Method: - This type of biasing network employs a voltage divider formed by the resistors R_1 and R_2 to bias the transistor. This means that here the voltage developed across R_2 will be the base voltage of the transistor which forward biases its base-emitter junction. In general, the current through R_2 will be fixed to be 10 times required base current, I_B (i.e. $I_2 = 10I_B$). This is done to avoid its effect on the voltage divider current or on the changes in β . Further, from the circuit, one gets

$$I_B = V_{CC} \frac{R_2}{R_1 + R_2}$$

$$V_E = I_E R_E$$

$$V_B = I_2 R_2 = V_{BE} + V_E$$

In this kind of biasing, I_C is resistant to the changes in both β as well as V_{BE} which results in a stability factor of 1 (theoretically), the maximum possible thermal stability. This is because, as I_C increases due to a rise in temperature, I_E also increases causing an increase in the emitter voltage V_E which in turn reduces the base-emitter voltage, V_{BE} . This results in the decrease of base current I_B which restores I_C to its original value. The higher stability offered by this biasing circuit makes it to be most widely used in spite of providing a decreased amplifier gain due to the presence of R_E .

PROCEDURE:

Fixed Bias Circuit

1. Connect the circuit as per the fixed bias diagram on the breadboard.
2. Set the DC power supply to V_{CC} (e.g., 10 V).
3. Switch ON the supply.
4. Measure and note:
5. Base current I_B
6. Collector current I_C
7. Collector-Emitter voltage V_{CE}
8. Ensure the transistor operates in the active region.
9. Switch OFF the supply after measurements.

Voltage Divider Bias Circuit:

1. Connect the circuit as per the voltage divider bias diagram.
2. Apply the same V_{CC} .
3. Switch ON the power supply.
4. Measure and record:
 - I_B
 - I_C
 - V_{CE}
5. Observe that V_{CE} is more stable compared to fixed bias.
6. Switch OFF the supply

OBSERVATION TABLE:

Bias Type	I_B (μA)	I_C (mA)	V_{CE} (V)
Fixed bias			
Voltage Divider Bias			

RESULT: The fixed bias and voltage divider bias circuits were designed and tested successfully.

VIVA QUESTIONS:

1. What is biasing in a BJT?
2. Why is biasing required in a transistor amplifier?
3. What is the operating point or Q-point?
4. What happens if the Q-point shifts?
5. What is a fixed bias circuit?
6. What are the disadvantages of fixed bias?
7. What is voltage divider bias?
8. Why is voltage divider bias more stable than fixed bias?
9. What is thermal stability in a BJT?
10. What is the role of emitter resistor in biasing circuits?
11. What is the DC load line?
12. Why is the DC load line important?
13. How is the DC load line drawn?
14. What are the intercepts of the DC load line?
15. What does the slope of the DC load line represent?
16. In which region should a BJT operate for amplification?
17. What are the three regions of BJT operation?
18. What happens if the transistor enters saturation?
19. What happens if the transistor enters cut-off?
20. What is β (beta) of a transistor?

REAL TIME APPLICATIONS:

BJT biasing circuits are used in **real-time electronic systems** to ensure that transistors operate in a **stable active region** for proper amplification. **Voltage divider bias** is widely used in **audio amplifiers, radio and TV receivers, public address systems, and sensor signal conditioning circuits** because it provides good thermal stability and prevents distortion due to Q-point shift. **Fixed bias** circuits are used in **simple switching applications, educational circuits, and low-cost devices** where stability Requirements are minimal.

The concept of the **DC load line** is applied in **amplifier design, power electronics, and communication systems** to select the correct operating point, avoid cut-off and saturation, and achieve maximum undistorted output. Overall, these biasing techniques are essential in **consumer electronics, industrial control systems, and instrumentation**, where reliable and stable transistor operation is required.

Experiment No:7**Construct and analyze a Common Base (CB) configuration of a BJT to study input-output characteristics and determine current gain (α) and input/output resistance****AIM:**

1. To observe and draw the input and output characteristics of a transistor connected in common base configuration.
2. To find α of the given transistor and also its input and output Resistances.

APPARATUS:

1. Transistor, BC107 -1No.
2. Regulated power supply (0-30V) -1No.
3. Voltmeter (0-20V) - 2No.
4. Ammeters (0-10mA) - 2No.
5. Resistor, $1K\Omega$ - 2No
6. Bread board
7. Connecting wire

THEORY:

A transistor is a three terminal active device. The terminals are emitter, base, collector. In CB configuration, the base is common to both input (emitter) and output (collector). For normal operation, the E-B junction is forward biased and C-B junction is reverse biased. In CB configuration, I_E is +ve, I_C is -ve and I_B is -ve. So

$$V_{EB} = F1 (V_{CB}, I_E) \text{ and}$$

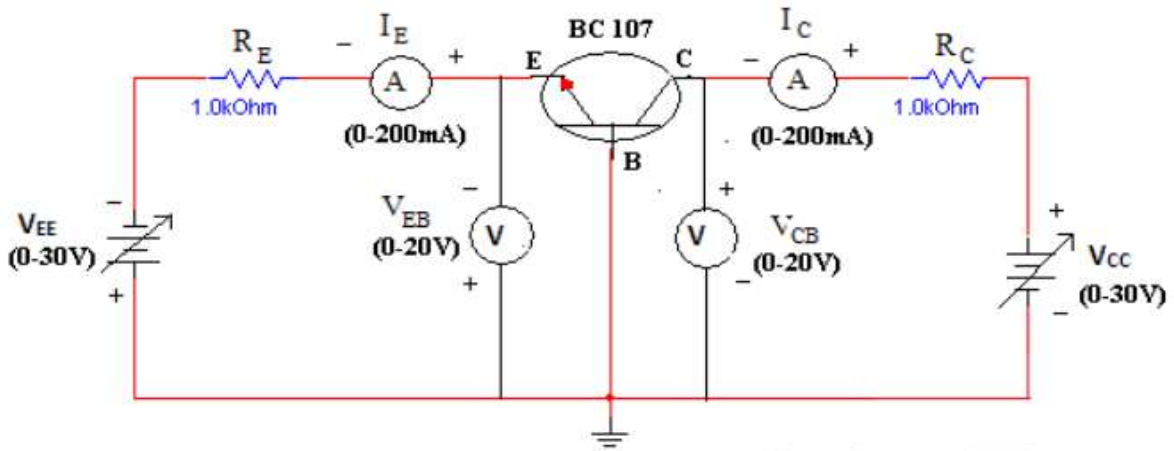
$$I_C = F2 (V_{EB}, I_B)$$

With an increasing the reverse collector voltage, the space-charge width at the output junction increases and the effective base width 'W' decreases. This phenomenon is known as "Early effect". Then, there will be less chance for recombination within the base region. With increase of charge gradient within the base region, the current of minority carriers injected across the emitter junction increases. The current amplification factor of CB configuration is given by,

$$\alpha = \Delta I_C / \Delta I_E$$

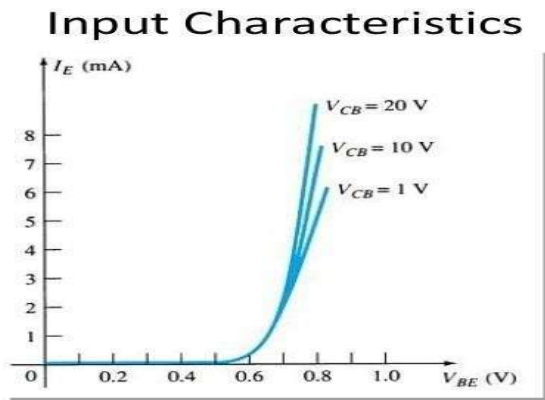
Input Resistance, $r_i = \Delta V_{BE} / \Delta I_E$ at Constant V_{CB}
 Output Resistance, $r_o = \Delta V_{CB} / \Delta I_C$ at Constant I_E

CIRCUIT DIAGRAM:

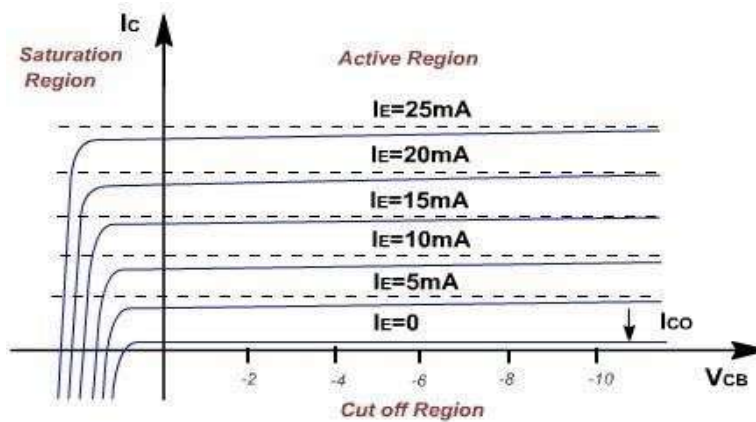


MODEL GRAPHS:

A) INPUT CHARACTERISTICS



B) OUTPUT CHARACTERISTICS



OBSERVATIONS:**A. INPUT CHARACTERISTICS:**

$V_{EE}(V)$	$V_{CB}=1V$		$V_{CB}=2V$		$V_{CB}=4V$	
	$V_{EB}(V)$	$I_E(mA)$	$V_{EB}(V)$	$I_E(mA)$	$V_{EB}(V)$	$I_E(mA)$

B. OUTPUT CHARACTERISTICS:

$V_{cc}(V)$	$I_E=10mA$		$I_E=20mA$		$I_E=30mA$	
	$V_{CB}(V)$	$I_C(mA)$	$V_{CB}(V)$	$I_C(mA)$	$V_{CB}(V)$	$I_C(mA)$

PROCEDURE:**A) INPUT CHARACTERISTICS:**

1. Connections are made as per the circuit diagram.
2. For plotting the input characteristics, the output voltage V_{CE} is kept constant at 0V and for different values of V_{EE} note down the values of I_E and V_{BE}
3. Repeat the above step keeping V_{CB} at 2V, 4V, and 6V and all the readings are tabulated.
4. A graph is drawn between V_{EB} and I_E for constant V_{CB} .

B) OUTPUT CHARACTERISTICS:

1. Connections are made as per the circuit diagram.
2. For plotting the output characteristics, the input I_E is kept constant at 0.5mA and for different values of V_{CC} , note down the values of I_C and V_{CB} .
3. Repeat the above step for the values of I_E at 1mA, 5mA and all the readings are tabulated.
4. A graph is drawn between V_{CB} and I_C for constant I_E

RESULT:

Input and Output characteristics of a Transistor in Common base Configuration are studied and find out the input resistance, output resistance, α .

VIVA QUESTIONS:

1. What is the common base configuration of a BJT?
2. Why is the base terminal common in CB configuration?
3. What are the applications of CB configuration?
4. What are the input and output terminals in CB configuration?
5. What are the input characteristics of a CB transistor?
6. What are the output characteristics of a CB transistor?
7. What is the input resistance of a CB transistor?
8. Why is the input resistance of CB configuration very low?
9. What is the output resistance of CB configuration?
10. Why is the output resistance of CB configuration high?
11. What is current gain (α) in CB configuration?
12. What is the typical value of α ?
13. What is the relation between α and β ?
14. Why is α always less than 1?
15. In which region should the CB transistor operate for amplification?
16. What are the regions of operation in CB configuration?
17. What is the phase relationship between input and output in CB configuration?
18. What is the significance of input characteristics in CB configuration?
19. How is α determined from characteristics?
20. How is input resistance calculated in CB configuration?
21. How is output resistance calculated in CB configuration?
22. What is the DC load line in CB configuration?
23. What happens if the transistor enters saturation in CB mode?
24. What happens if the transistor enters cut-off in CB mode?

REALTIME APPLICATIONS:

The common base (CB) configuration of a BJT is mainly used in high-frequency and RF applications because it has low input resistance, high output resistance, and good frequency response. In real-time systems, CB configuration is used in radio transmitters and receivers, RF amplifiers, impedance-matching circuits, oscillators, and wideband amplifiers, where a low-resistance signal source needs to drive a high-resistance load efficiently. Since the collector current closely follows the emitter current (current gain α is close to 1), it is also useful in current amplification and signal conditioning circuits. Due to the absence of phase reversal and reduced Miller effect, CB configuration performs well in communication and instrumentation applications, though it is rarely used in low-frequency audio amplifiers.

Software-Based Simulation Experiments

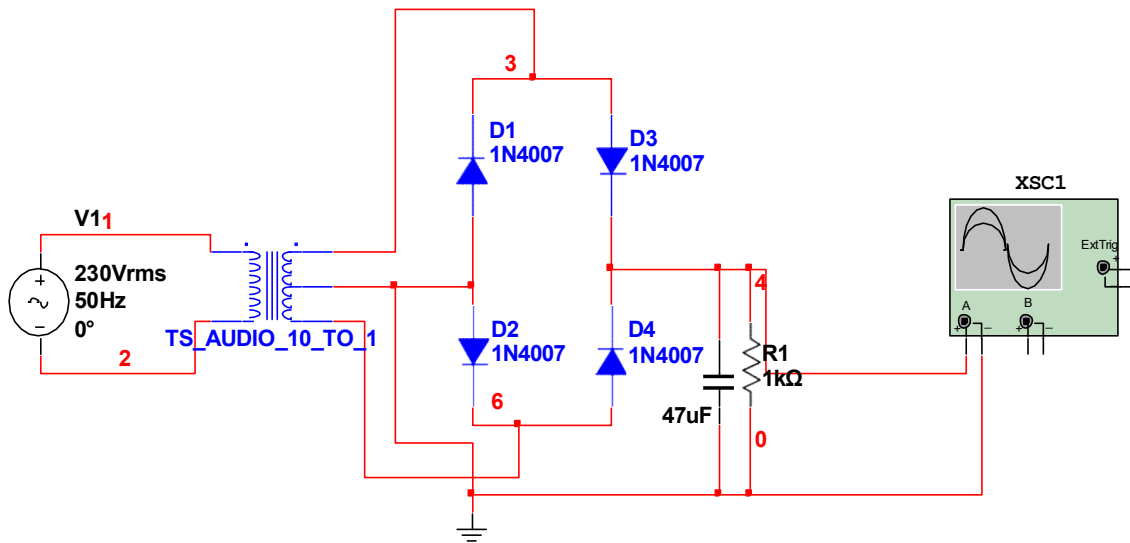
Experiment No:1

Simulate a full-wave bridge rectifier with capacitor filter to analyze waveform smoothing and ripple reduction in DC power supply design

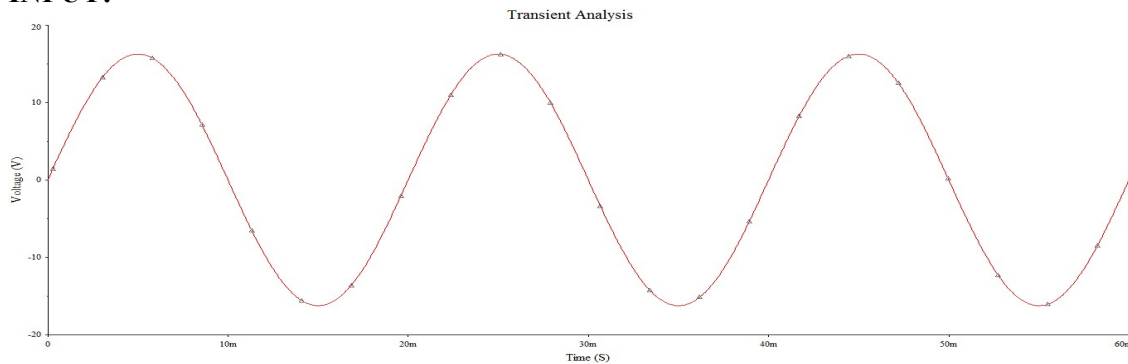
AIM: To study the operation of full-wave rectifier and find its ripple factor using Multisim.

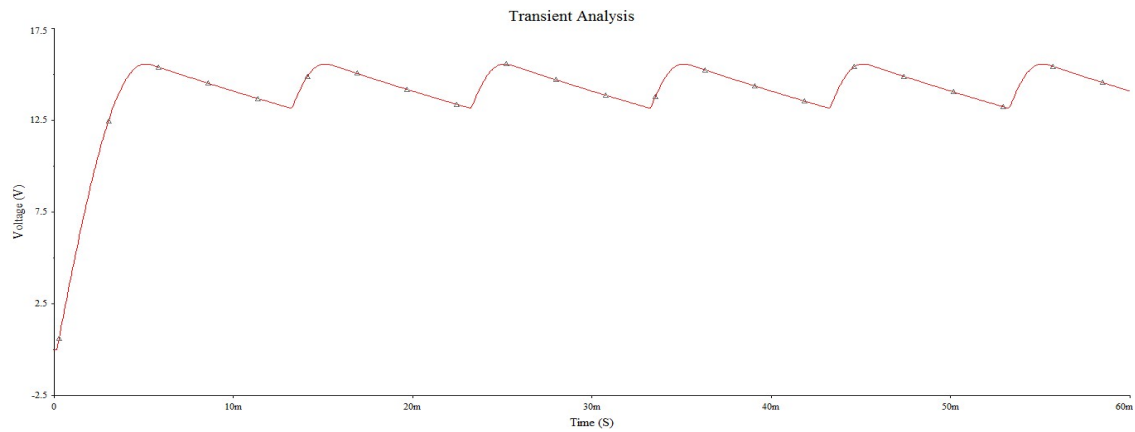
SOFTWARE: Multisim

CIRCUIT DIAGRAM:



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, 230 Vrms across primary coil of the transformer.
4. RUN the circuit and observe the output wave form using oscilloscope.

RESULT:

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

Experiment No: 2

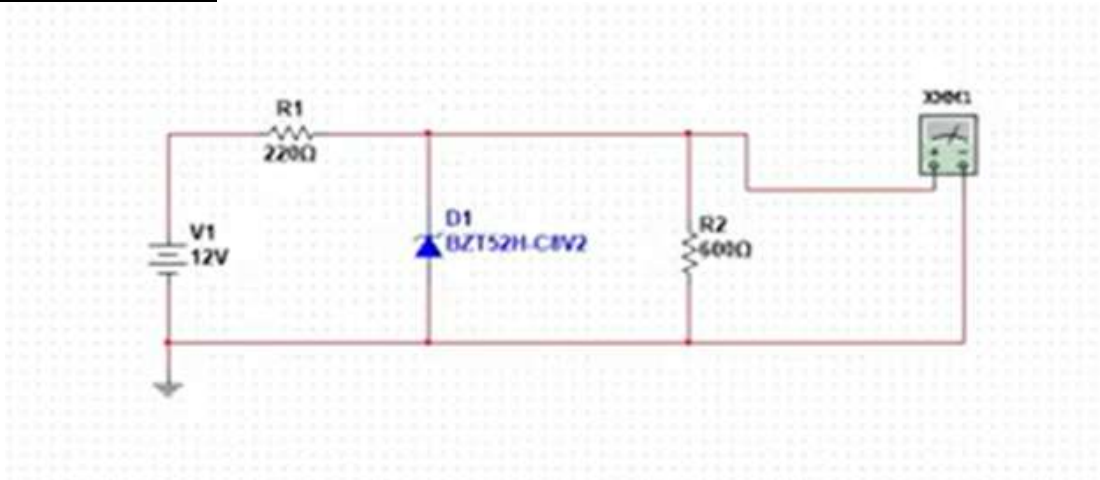
Simulate a Zener diode-based voltage regulator to study voltage stabilization against varying supply voltages and load resistances

AIM: To simulate a Zener diode voltage regulator and study the voltage stabilization against varying supply voltages and load resistances.

APPARATUS / COMPONENTS (IN MULTISIM):

1. DC voltage source
2. Series resistor
3. Zener diode
4. Load resistor
5. Multimeter / Voltage probe
6. Multisim software

CIRCUIT DIAGRAM:



PROCEDURE:

Voltage Stabilization with Varying Input Voltage

1. Open Multisim and place the components as shown in the circuit diagram.
2. Set the **Zener diode breakdown voltage** (V_z) according to your specification (e.g., 5.6 V).
3. Set $R_s = 470 \Omega$ and $R_L = 1 \text{ k}\Omega$.
4. Connect the **DC voltage source** V_{in} starting slightly above V_z (e.g., 6 V).
5. Simulate the circuit and **measure** V_{out} across the load resistor.
6. Gradually increase V_{in} (e.g., 6 V \rightarrow 12 V) and record V_{out} for each value.
7. Observe and note how V_{out} remains nearly constant, demonstrating voltage regulation.

Voltage Stabilization with Varying Load Resistance

1. Keep V_{in} constant (e.g., 12 V).
2. Vary R_L from 500 Ω to 5 k Ω .
3. Measure and record V_{out} for each R_L .
4. Observe that V_{out} remains nearly constant until the current through the Zener falls below its minimum regulation current.

OBSERVATION TABLE:

Input Voltage V_{in} (V)	Load Resistance R_L (Ω)	Output Voltage(V)

Result: Thus, voltage regulation is successfully verified using Multisim

Experiment No: 3

Simulate a common emitter amplifier with and without emitter bypass capacitor to analyze the effect on voltage gain and signal amplification

AIM: To simulate a Common Emitter amplifier using Multisim and study the effect of an emitter bypass capacitor on voltage gain and signal amplification.

APPARATUS / COMPONENTS (MULTISIM)

1. NPN Transistor
2. DC Power Supply (V_{CC})
3. Signal Generator (AC source)
4. Resistors:
 - a. Collector resistor (R_C)
 - b. Emitter resistor (R_E)
 - c. Base bias resistors (R_1, R_2)
5. Capacitors:
 - a. Input coupling capacitor (C_{in})
 - b. Output coupling capacitor (C_{out})
 - c. Emitter bypass capacitor (C_E)
6. Ground
7. Oscilloscope / Multimeter

THEORY:

A CE amplifier provides high voltage gain and 180° phase shift between input and output.

Role of Emitter Bypass Capacitor (C_E)

Without C_E :

Emitter resistor provides negative feedback

Gain is lower but more stable

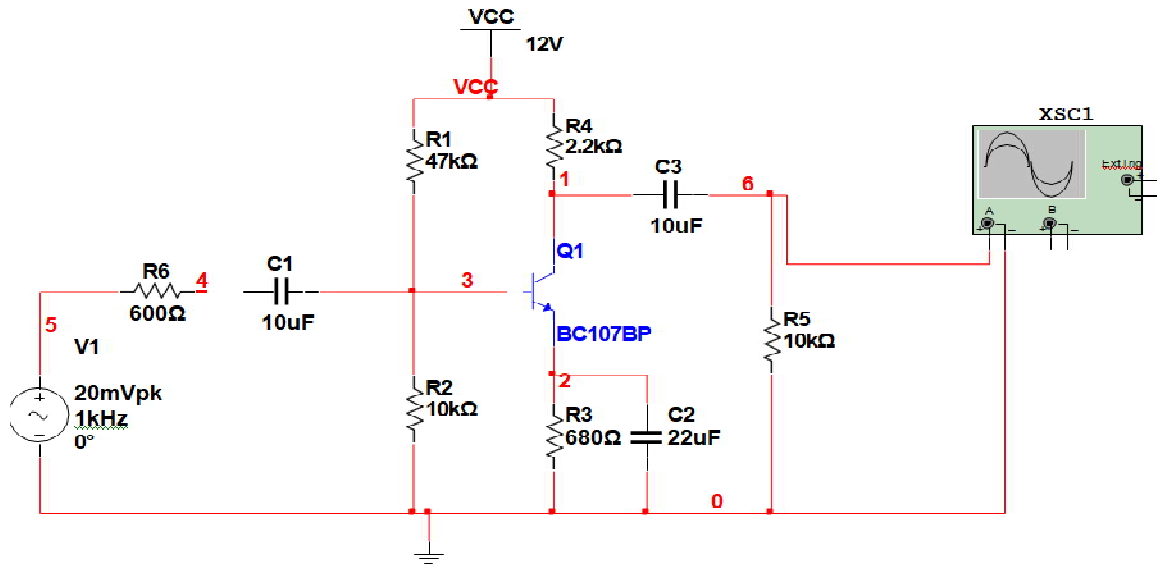
With C_E :

C_E bypasses R_E for AC signals

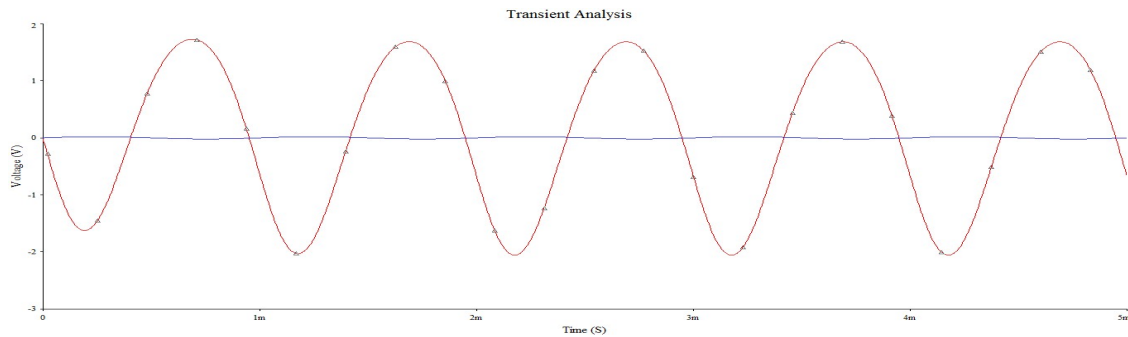
Reduces AC negative feedback

Voltage gain increases significantly

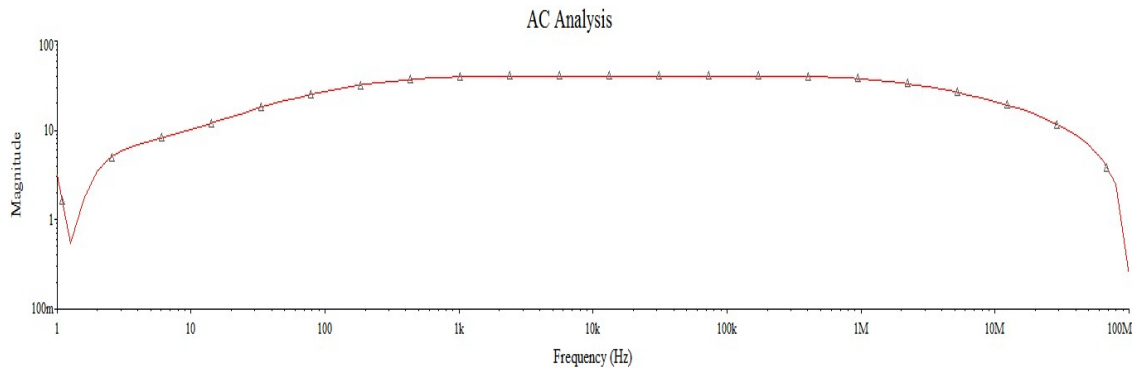
CIRCUIT DIAGRAM:



OUTPUT WAVE FORM:



FREQUENCY RESPONSE:



PROCEDURE:

1. Connect the circuit diagram as shown in the given figure.
2. Apply a sine wave input signal of 1 kHz frequency and 20 mV (peak) amplitude.
3. Select Simulate and then click on Run to start the simulation.
4. Observe the amplified output signal using the oscilloscope.
5. To obtain the frequency response, go to Simulate → Analyses → AC Analysis.
6. Enter the start frequency and stop frequency as required.
7. Write the appropriate expression for voltage gain and click on Simulate.
8. Observe the magnitude response and phase response of the amplifier.
9. From the magnitude response, calculate the voltage gain and bandwidth of the amplifier.

Without Emitter Bypass Capacitor:

1. Remove or disconnect C_E .
2. Run the simulation.
3. Observe input and output waveforms.
4. Measure output amplitude.

OBSERVATION TABLE:

Condition	Input (mV)	Output (V)	Voltage Gain $A_v = V_{out}/V_{in}$
Without C_E			
With C_E			

RESULT:

The Common Emitter amplifier shows higher voltage gain when an emitter bypass capacitor is used.

Without the bypass capacitor, gain is reduced due to emitter degeneration, improving stability but reducing amplification.

Experiment No: 4

Simulate BJT operation as a switch and small-signal amplifier to understand its dual functionality in digital and analog applications.

AIM: To simulate the operation of a BJT

1. As a switch (cut-off and saturation regions) and
2. As a small-signal amplifier (active region)
using Multisim, and to study its application in digital and analog circuits.

APPARATUS / COMPONENTS (MULTISIM):

1. NPN BJT
2. DC Power Supply
3. AC Signal Generator
4. Resistors (Base, Collector, Emitter)
5. Load (LED / Resistor)
6. Capacitors (for amplifier circuit)
7. Oscilloscope
8. Ground

CIRCUIT DIAGRAM:

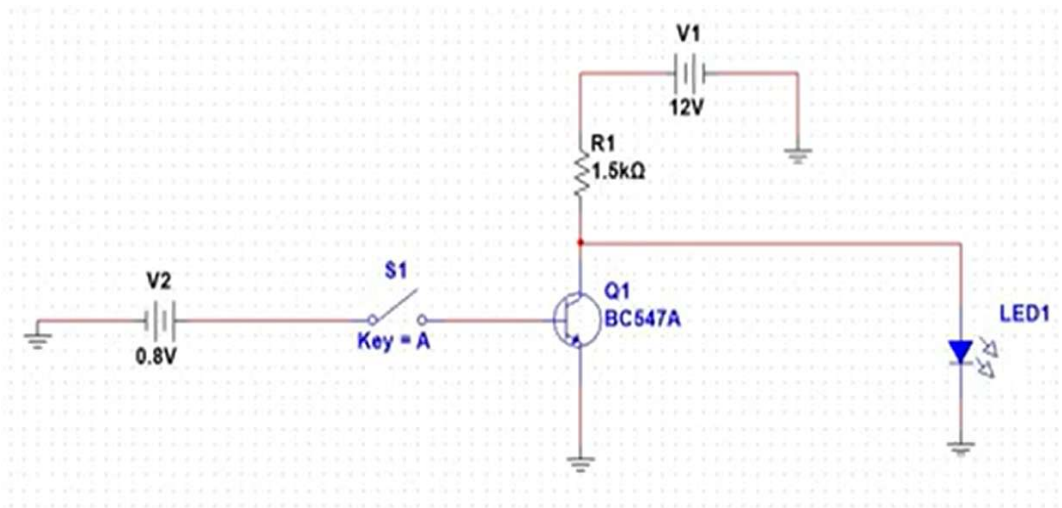


Fig: Transistor as a switch

PROCEDURE (SWITCH MODE – MULTISIM)

1. Connect the BJT circuit as shown in the figure.
2. Apply $V_{CC} = 5\text{ V}$.
3. Apply a square wave / DC input at the base through a resistor.
4. Set input:
 - 0 V → Cut-off
 - 5 V → Saturation
5. Run the simulation.
6. Observe the collector voltage and load (LED/glow or voltage change) using a multimeter or oscilloscope.

OBSERVATION (SWITCH):

Input LOW → BJT OFF → Output HIGH

Input HIGH → BJT ON → Output LOW

RESULT: The BJT operates as an electronic switch, suitable for digital logic and control applications

VIVA QUESTIONS:

1. What is a BJT?
2. Why is BJT called a current-controlled device?
3. What are the regions of operation of a BJT?
4. In which region does a BJT act as a switch?
5. In which region does a BJT act as an amplifier?
6. What is cutoff region?
7. What is saturation region?
8. What is active region?
9. Why is biasing required in a BJT amplifier?
10. What happens to collector current in cutoff region?
11. What happens to collector current in saturation region?
12. What is the function of base resistor in a BJT switch?
13. What is small-signal amplification?
14. Why is CE configuration preferred for amplification?
15. What is voltage gain?
16. What is current gain (β)?
17. What happens if BJT is improperly biased?
18. Why does a BJT produce amplification?
19. How does a BJT act as an ON/OFF switch?
20. What are the digital applications of a BJT?

REAL TIME APPLICATIONS:

In real-time applications, a BJT is widely used as a **switch** in digital systems such as relay drivers, LED control circuits, alarm systems, motor ON/OFF control, and microcontroller interface circuits, where it operates in cutoff and saturation regions to provide reliable switching, while in **analog applications** it functions as a **small-signal amplifier** in audio amplifiers, microphone pre-amplifiers, radio and communication systems, sensor signal conditioning circuits, and voltage amplification stages, where it operates in the active region to amplify weak signals with good linearity, demonstrating the BJT's dual functionality in both digital and analog electronic systems.

Experiment No: 5

Simulate the output and transfer characteristics of a JFET to determine parameters such as pinch-off voltage, drain resistance, and transconductance

AIM:

To simulate the output characteristics and transfer characteristics of an N-channel JFET using Multisim, and to determine:

1. Pinch-off voltage (V_P)
2. Drain resistance (r_d)
3. Tran conductance (g_m)

APPARATUS / COMPONENTS (MULTISIM):

1. N-channel JFET
2. DC Power Supplies (V_{DD} and V_{GG})
3. Drain resistor (optional)
4. Multimeter
5. DC Voltmeter / Ammeter
6. Ground

THEORY:

Output Characteristics

A plot of drain current (I_D) versus drain-source voltage (V_{DS}) for different gate-source voltages (V_{GS}) with V_{GS} negative.

Transfer Characteristics

A plot of drain current (I_D) versus gate-source voltage (V_{GS}) at a constant V_{DS} (saturation region).

Important Parameters

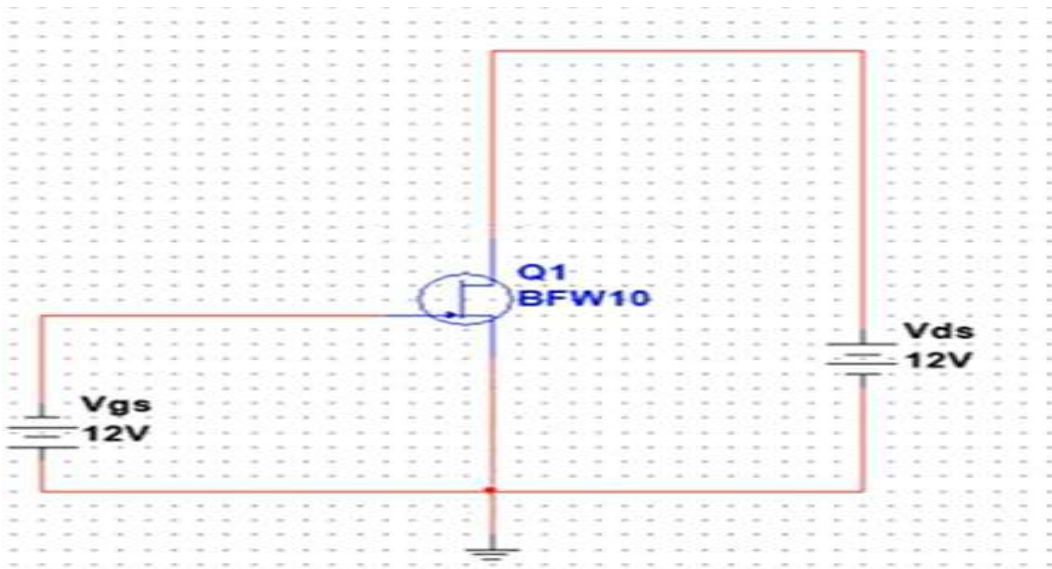
- Pinch-off voltage (V_P): The value of V_{GS} at which I_D becomes approximately zero.
- Drain resistance (r_d):

$$r_d = \frac{\Delta V_{DS}}{\Delta I_D} \quad (\text{in saturation region})$$

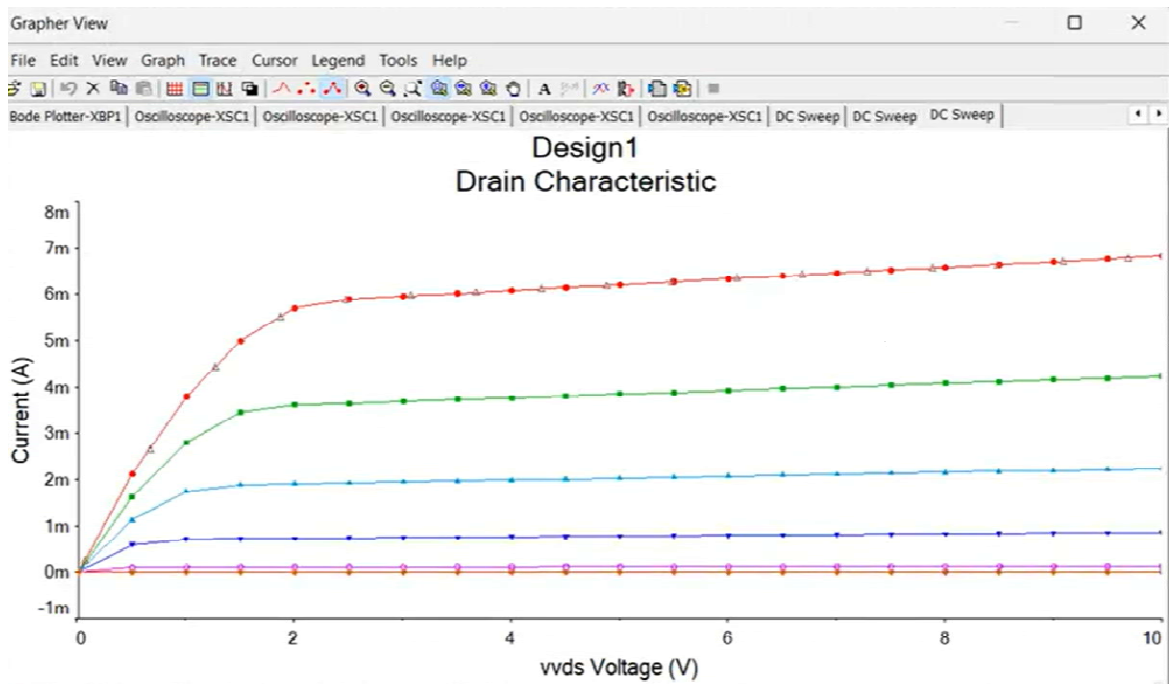
- Transconductance (g_m):

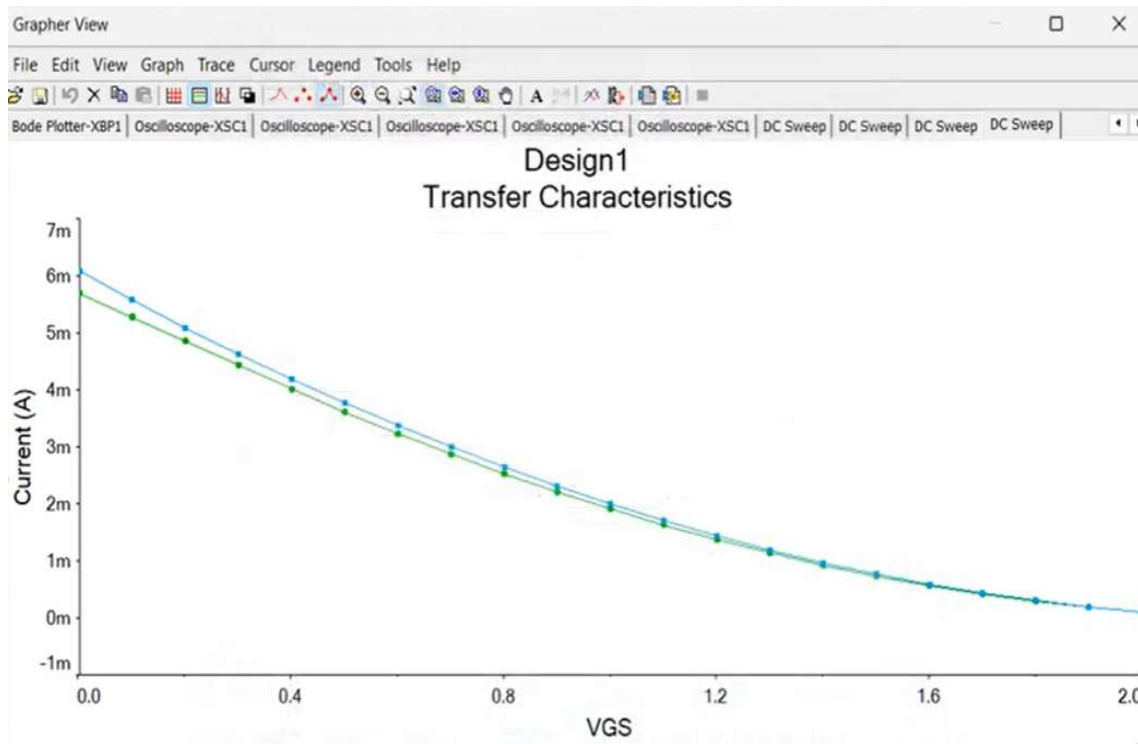
$$g_m = \frac{\Delta I_D}{\Delta V_{GS}}$$

CIRCUIT DIAGRAM:



DRAIN CHARACTERISTICS:



TRANSFER CHARACTERISTICS:**PROCEDURE:****OUTPUT CHARACTERISTICS**

1. Open NI Multisim
2. Place components:
 - JFET
 - DC voltage source
 - DC voltage source for gate
 - Resistor
 - Ground
3. Connect the circuit as shown.
4. Set $V_{GS} = 0$ V
5. Vary V_{DS} from 0 to 15 V in steps
6. Measure and note drain current (I_D).
7. Repeat steps 4–6 for:
 - $V_{GS} = -1$ V
 - $V_{GS} = -2$ V
 - $V_{GS} = -3$ V
8. Plot I_D vs V_{DS} for each V_{GS}

TRANSFER CHARACTERISTICS:

1. Fix $V_{DS} = 10\text{ V}$
2. Vary V_{GS} from 0 V to cutoff (negative values).
3. Measure corresponding I_D .
4. Plot I_D vs V_{GS} .

OBSERVATIONS:**Output Characteristics Table:**

V_{GS} (V)	V_{DS} (V)	I_D (mA)
0	2	
0	4	
-1	2	
-1	4	
-2	2	

Transfer Characteristics Table:

V_{GS} (V)	I_D (mA)
0	
-0.5	
-1	
-2	
-3	

RESULT: Thus, the output and transfer characteristics of a JFET were successfully simulated using Multisim, and the parameters pinch-off voltage, drain resistance, and transconductance were determined.

VIVA QUESTIONS:

1. What is a JFET?
2. Why is JFET called a voltage-controlled device?
3. What are the terminals of a JFET?
4. What is pinch-off voltage?
5. How is pinch-off voltage determined from characteristics?
6. What is drain resistance (r_{dr_drd})?

7. How is drain resistance calculated from output characteristics?
8. What is transconductance (g_m)?
9. How is transconductance obtained from transfer characteristics?
10. What happens to drain current when V_{GS} becomes more negative?
11. Why is gate current nearly zero in a JFET?
12. What is the saturation region in a JFET?
13. What is cutoff region?
14. What is the relation between I_{DID} and V_{GS} ?
15. Why is JFET normally ON?
16. Difference between pinch-off and cutoff?
17. What is the significance of drain resistance?
18. Why are JFETs preferred in low-noise applications?
19. Compare JFET and BJT.
20. What are the advantages of JFET?

REAL-TIME APPLICATIONS:

FETs are widely used in real-time electronic applications such as **low-noise amplifiers**, **audio pre-amplifier stages**, **RF amplifiers**, **signal buffering circuits**, and **impedance-matching networks** due to their high input impedance and low noise, and they are also employed as **voltage-controlled resistors** in automatic gain control circuits, analog switches, mixers, and sensor signal conditioning systems, making JFETs highly suitable for both analog signal processing and communication applications.

Experiment No: 6

Simulate the characteristics of a MOSFET and design a CMOS inverter to study digital switching behavior and low-power logic design

AIM:

1. To simulate the transfer and output characteristics of NMOS and PMOS transistors using NI Multisim.
2. To design and simulate a CMOS inverter.
3. To study digital switching behavior, voltage transfer characteristics (VTC), and low-power operation of CMOS logic.

APPARATUS:

1. NI Multisim
2. NMOS transistor (2N7000 / BS170)
3. PMOS transistor (BS250 or PMOS from Multisim library)
4. DC voltage sources
5. Pulse voltage source
6. Resistors
7. Oscilloscope

THEORY:

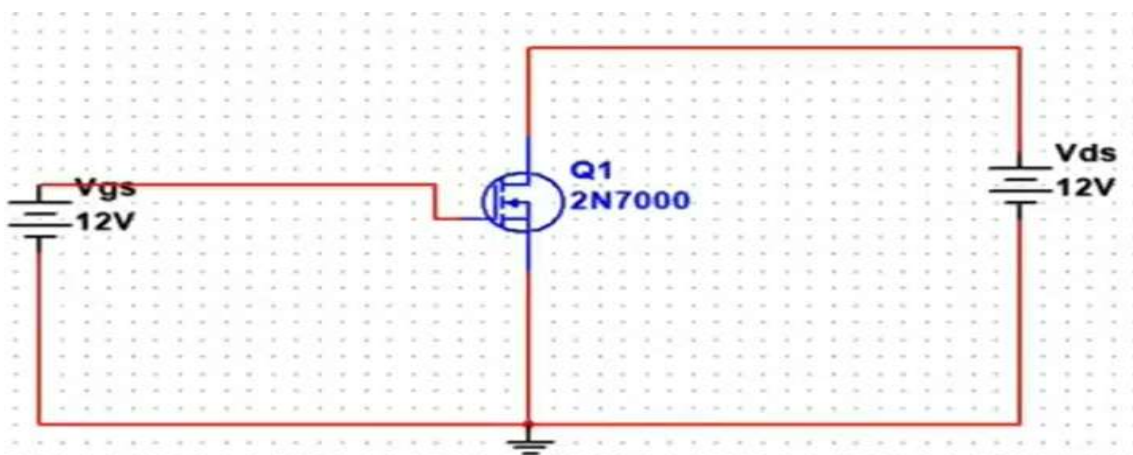
A MOSFET (Metal Oxide Semiconductor Field Effect Transistor) is a voltage-controlled device where drain current is controlled by gate-to-source voltage.

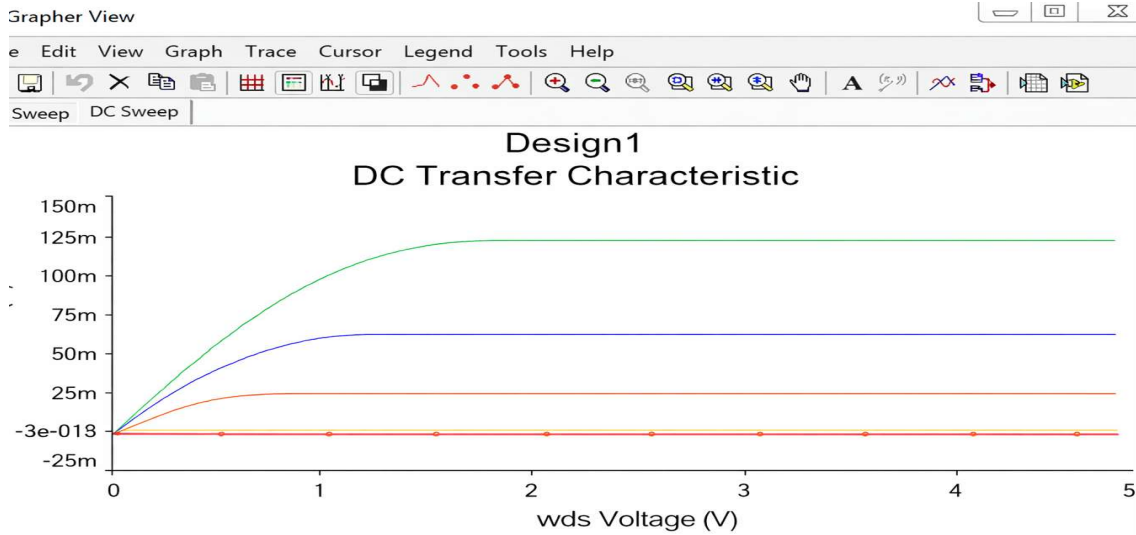
- NMOS conducts when $V_{GS} > V_{TH}$
- PMOS conducts when $V_{SG} > |V_{TH}|$

A CMOS inverter uses:

- PMOS as pull-up device
- NMOS as pull-down device

CIRCUIT DIAGRAM:

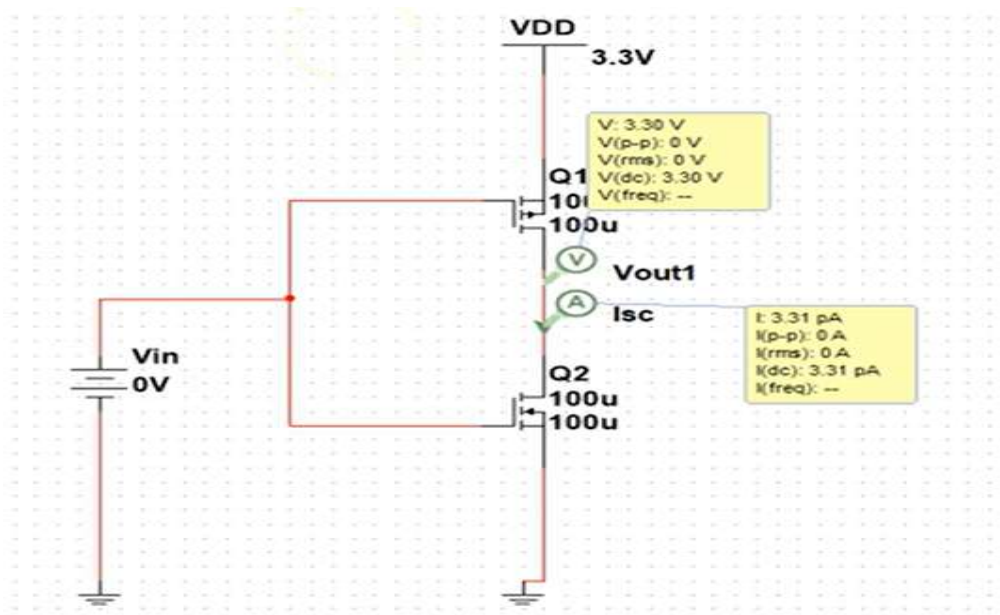




PROCEDURE:

1. Open NI Multisim.
2. Place NMOS transistor, DC supplies, and resistor.
3. Set $V_{DD} = 10\text{ V}$, $R_D = 1\text{ k}\Omega$.
4. Apply $V_{GS} = 2\text{ V}$.
5. Vary V_{DS} from 0 to 10 V and note I_D .
6. Repeat for $V_{GS} = 3\text{ V}$, 4 V, 5 V.
7. Plot I_D vs V_{DS} (output characteristics).
8. Fix $V_{DS} = 5\text{ V}$, vary V_{GS} from 0 to 6 V.
9. Plot I_D vs V_{GS} (transfer characteristics).
10. Repeat similarly for PMOS.

CMOS INVERTER:



PROCEDURE:

1. Place NMOS and PMOS transistors in Multisim.
2. Connect:
 - PMOS source to VDD = 5 V
 - NMOS source to GND
3. Connect drains together to form output (Vout).
4. Apply pulse voltage (0–5 V) to the gates.
5. Observe Vin and Vout on oscilloscope.
6. Perform DC sweep of Vin from 0 to 5 V.
7. Plot Voltage Transfer Characteristics (VTC).

CMOS Inverter Truth Table:

Vin	NMOS	PMOS	Vout
0	OFF	ON	1
1	ON	OFF	0

RESULT:

Thus, the MOSFET characteristics were successfully simulated and a CMOS inverter was designed using Multisim. The digital switching behavior and low-power logic operation of CMOS circuits were verified.

VIVA QUESTIONS:

1. What is a MOSFET?
2. Why is MOSFET called a voltage-controlled device?
3. What are the terminals of a MOSFET?
4. What is threshold voltage (VTH)?
5. What is enhancement-mode MOSFET?
6. Difference between NMOS and PMOS?
7. What are the regions of operation of a MOSFET?
8. When does an NMOS transistor turn ON?
9. When does a PMOS transistor turn ON?
10. What is the saturation region in a MOSFET?
11. What is the transfer characteristic of a MOSFET?
12. What is the output characteristic of a MOSFET?
13. What is a CMOS inverter?
14. Why is CMOS inverter called low-power logic?
15. What happens when input to CMOS inverter is LOW?
16. What happens when input to CMOS inverter is HIGH?
17. What is voltage transfer characteristic (VTC)?
18. Why is static power dissipation low in CMOS circuits?
19. What is switching power dissipation?
20. Why are MOSFETs preferred in digital circuits?

REAL-TIME APPLICATIONS:

MOSFETs and CMOS inverters are extensively used in real-time applications such as **microprocessors, microcontrollers, memory chips, and digital ICs**, where fast switching speed and low power consumption are essential, and CMOS logic forms the backbone of **computers, mobile phones, embedded systems, and IoT devices**, while MOSFETs are also used in **power management circuits, switching regulators, motor drivers, and signal processing systems**, making CMOS technology ideal for high-density, high-speed, and low-power digital system design.

Experiment No: 7

Simulate the transfer and output characteristics of an enhancement-mode NMOS transistor to analyze threshold voltage, drain current, and switching behavior.

AIM:

To simulate the transfer characteristics and output characteristics of an enhancement-mode NMOS transistor using Multisim and to analyze:

1. Threshold voltage (V_{TH})
2. Drain current (I_D) variation
3. Switching behavior of NMOS transistor

APPARATUS:

1. NI Multisim
2. Enhancement-mode NMOS transistor (2N7000 / BS170)
3. DC voltage sources
4. Ammeter / Current probe
5. Voltmeter
6. Ground

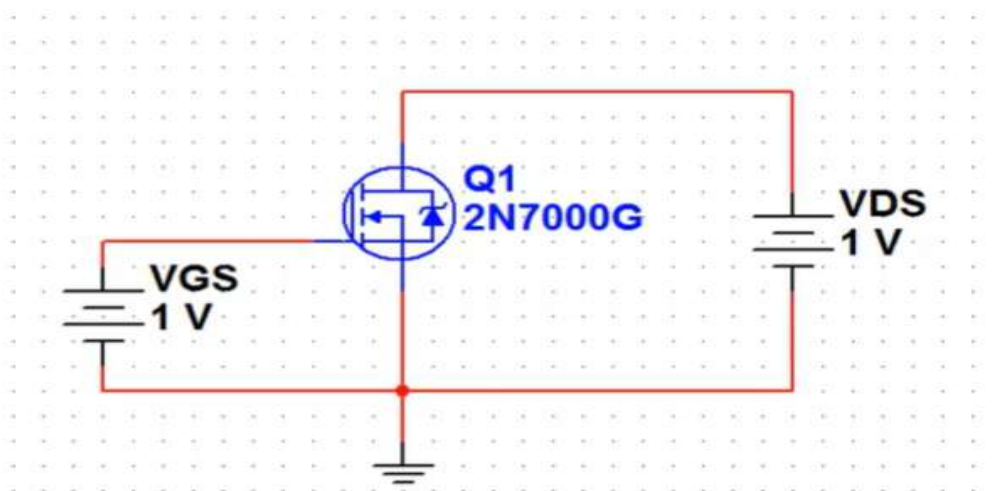
THEORY:

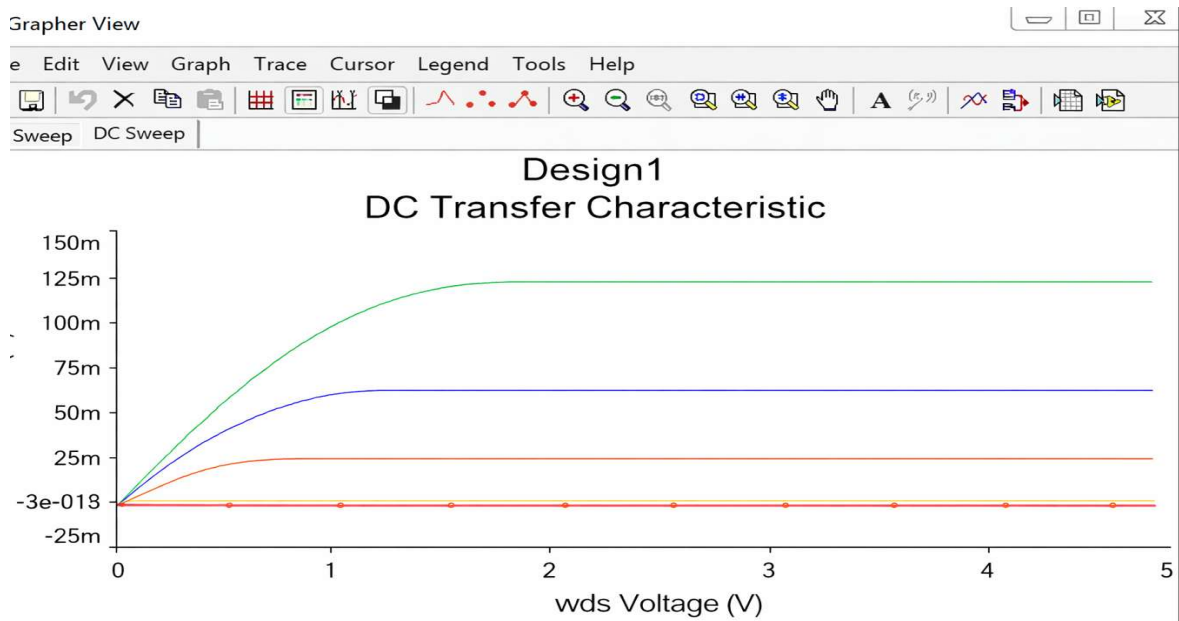
An enhancement-mode NMOS transistor remains OFF when $V_{GS}=0$. Conduction begins only when the gate-to-source voltage exceeds the threshold voltage (V_{TH}).

In analog operation, NMOS controls drain current.

In digital operation, NMOS acts as a switch (ON/OFF).

CIRCUIT DIAGRAM:





PROCEDURE:

OUTPUT CHARACTERISTICS:

1. Open NI Multisim.
2. Place NMOS transistor (2N7000).
3. Set $V_{DD} = 10\text{ V}$, $R_D = 1\text{ k}\Omega$.
4. Apply $V_{GS} = 2\text{ V}$.
5. Vary V_{DS} from 0 to 10 V.
6. Measure drain current (I_D).
7. Repeat for:
 - $V_{GS} = 3\text{ V}$
 - $V_{GS} = 4\text{ V}$
 - $V_{GS} = 5\text{ V}$
8. Plot I_D vs V_{DS} for different V_{GS} .

TRANSFER CHARACTERISTICS:

1. Fix $V_{DS} = 5\text{ V}$.
2. Vary V_{GS} from 0 V to 6 V.
3. Measure I_D for each V_{GS} .
4. Plot I_D vs V_{GS} .

SWITCHING BEHAVIOR:

1. Replace DC gate supply with pulse voltage source.
2. Set: Pulse amplitude: 0–5 V, Frequency: 1 kHz

3. Observe drain voltage on oscilloscope.
4. Note: NMOS OFF when $V_{GS} < V_{TH}$,
NMOS ON when $V_{GS} > V_{TH}$

RESULT:

Thus, the transfer and output characteristics of an enhancement-mode NMOS transistor were successfully simulated using Multisim, and the threshold voltage, drain current, and switching behavior were analyzed.

OPEN ENDED EXPERIMENTS

Experiment No:1**Measurement of Ripple factor and Efficiency of Rectifiers and filters, leads to designing of voltage regulators****AIM:**

To construct half-wave and full-wave rectifier circuits with and without filters, measure their ripple factor and efficiency, and study how filtering improves the DC output, thereby understanding the need for voltage regulators.

APPARATUS:

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

Theory:

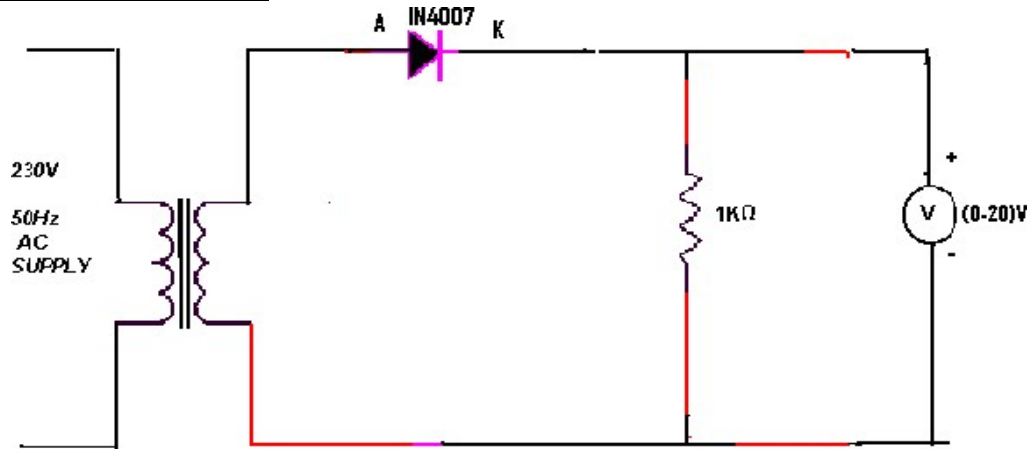
A device that converts a sinusoidal input waveform into a unidirectional waveform with a non-zero average component is called a rectifier.

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 the entire input voltage appears across the load resistance R_L . During the negative half-cycle, the diode is reverse biased and behaves like an open circuit; hence, the output voltage is zero. The filter used is a capacitor connected from the rectifier output to ground. The capacitor charges quickly during the rising portion of the input voltage and discharges slowly through R_L after the positive peak. The variation in the capacitor voltage due to this charging and discharging is called ripple voltage. Since ripple is undesirable, smaller ripple indicates better filtering action.

A full-wave rectifier consists of two half-wave rectifiers connected to a common load. One rectifies during the positive half-cycle of the input, while the other rectifies during the negative half-cycle. The transformer supplies the two diodes (D1 and D2) with sinusoidal voltages that are equal in magnitude but opposite in phase. During the positive half-cycle, diode D1 conducts and diode D2 remains OFF.

During the negative half-cycle, diode D1 is OFF and diode D2 conducts. As in the half-wave rectifier, ripple is undesirable, and better filtering is achieved by reducing the ripple content.

CIRCUIT DIAGRAM:



WITH CAPACITOR FILTER:

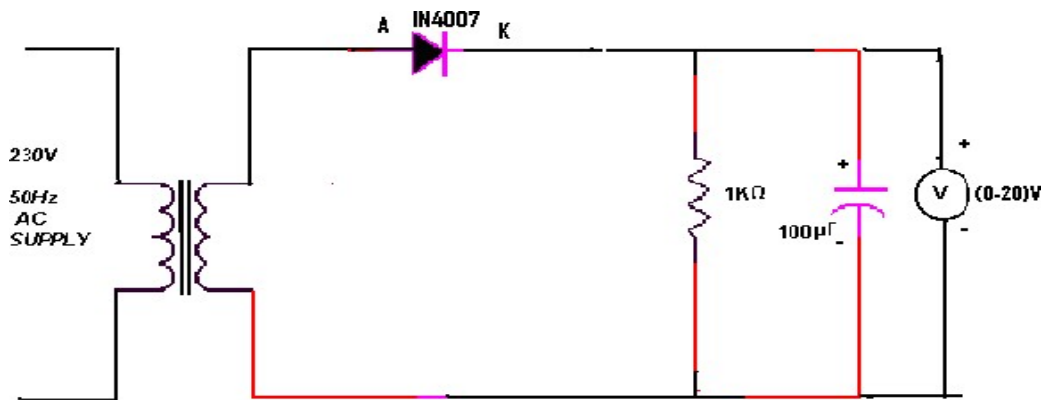
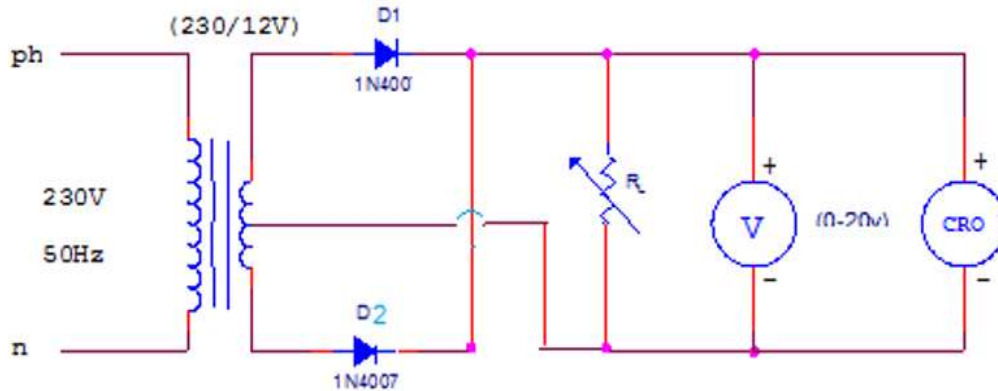
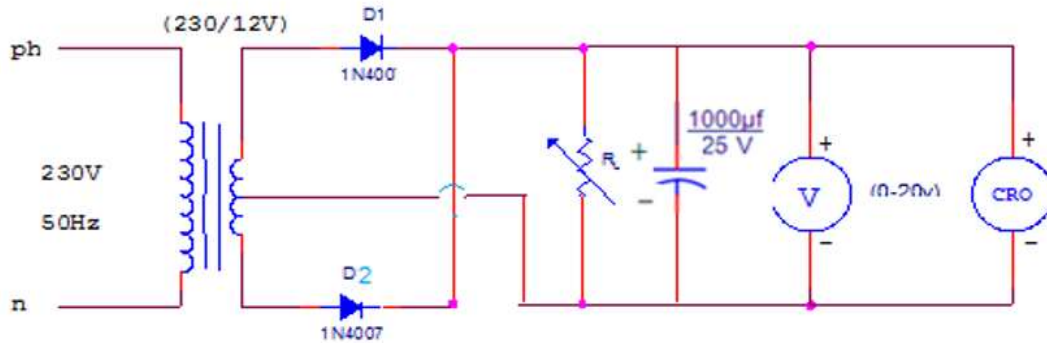


Fig: Circuit diagram of Half Wave Rectifier With capacitor Filter

Full wave Rectifier without Filter**Full wave Rectifier with capacitor Filter****PROCEDURE:**

1. Connect the rectifier circuit as per the circuit diagram without filter.
2. Switch ON the AC supply.
3. Measure the DC output voltage across the load using a DC voltmeter.
4. Measure the AC ripple voltage using an AC voltmeter or CRO.
5. Calculate ripple factor and efficiency.
6. Switch OFF the supply and connect the capacitor filter across the load.
7. Switch ON the supply again and measure DC and ripple voltages.
8. Observe the reduction in ripple and improvement in output voltage.

OBSERVATION TABLES:

Rectifier Output WITHOUT Filter

Sl. No	Rectifier Type	RL (Ω)	Vdc (V)	Vac (V)	Ripple Factor (r)	Efficiency (%)
1	Half Wave Rectifier					
2	Full Wave Rectifier					

Rectifier Output WITH Capacitor Filter

Sl. No	Rectifier Type	RL (Ω)	C (μ F)	Vdc (V)	Vac (V)	Ripple Factor (r)
1	Half Wave Rectifier					
2	Full Wave Rectifier					

Efficiency Calculation

Sl. No	Rectifier Type	Input AC Power (Pin)	Output DC Power (Pdc)	Efficiency (%)
1	Half Wave Rectifier			
2	Full Wave Rectifier			

1. Ripple Factor (r)

$$r = \frac{V_{ac}}{V_{dc}}$$

Where:

- V_{ac} = RMS value of AC ripple voltage
- V_{dc} = DC output voltage

2. DC Output Power (P_{dc})

$$P_{dc} = \frac{V_{dc}^2}{R_L}$$

3. AC Input Power (P_{in})

$$P_{in} = V_{ac(in)} \times I_{ac(in)}$$

4. Rectifier Efficiency (η)

$$\eta = \frac{P_{dc}}{P_{in}} \times 100\%$$

RESULT: The ripple factor and rectification efficiency of half-wave and full-wave (bridge) rectifiers were measured with and without filters.

VIVA QUESTIONS:

1. What is a rectifier?
2. Define ripple factor.
3. What is the ideal value of ripple factor?
4. Define rectification efficiency.
5. Why is ripple present in rectifier output?
6. Which rectifier has higher efficiency: HWR or FWR?
7. What is the maximum efficiency of a half-wave rectifier?
8. What is the maximum efficiency of a full-wave rectifier?
9. What is the purpose of a filter in rectifier circuits?
10. Name different types of filters used with rectifiers.
11. How does a capacitor filter reduce ripple?
12. What is the effect of load resistance on ripple factor?
13. Why is ripple factor less in full-wave rectifiers?
14. What is the significance of efficiency in rectifiers?
15. Why are filters necessary before voltage regulators?

16. What happens if a rectifier is used without a filter?
17. What is the role of a voltage regulator?
18. How does ripple affect electronic circuits?
19. Why is bridge rectifier preferred in power supplies?
20. How does ripple factor measurement help in designing regulators?

REAL-TIME APPLICATIONS:

Rectifiers and filters are widely used in power supply units of electronic devices such as televisions, computers, mobile chargers, and adapters to convert AC mains into usable DC voltage. Measurement of ripple factor and efficiency helps in selecting suitable rectifier and filter combinations for designing stable voltage regulators used in communication systems, medical equipment, and industrial control circuits. Low-ripple DC supplies are essential for microprocessors, amplifiers, and digital circuits to prevent noise and malfunction, making rectifiers with efficient filtering the foundation of reliable voltage regulation systems.

Experiment No: 2**Finding the bandwidth of amplifiers leads to the designing of Amplifiers.****AIM:**

To determine the bandwidth of a given amplifier by experimentally finding its lower cut-off frequency (f_L) and upper cut-off frequency (f_H), and to study its frequency response for amplifier design.

APPARATUS:

S.No	Device	Range/Rating	Qty
1.	(a) Regulated DC supply voltage	0-30V	1
2	Function generator	1MHz	1
2.	Dual trace CRO(oscilloscope)	25MHz	1
3.	BJT	BC107	1
4.	Connecting wires		
5.	Capacitor	10 μ f=2,100 μ f	
6.	Resistor	220. Ω ,5.6k Ω ,22K,1K,10K	

THEORY:

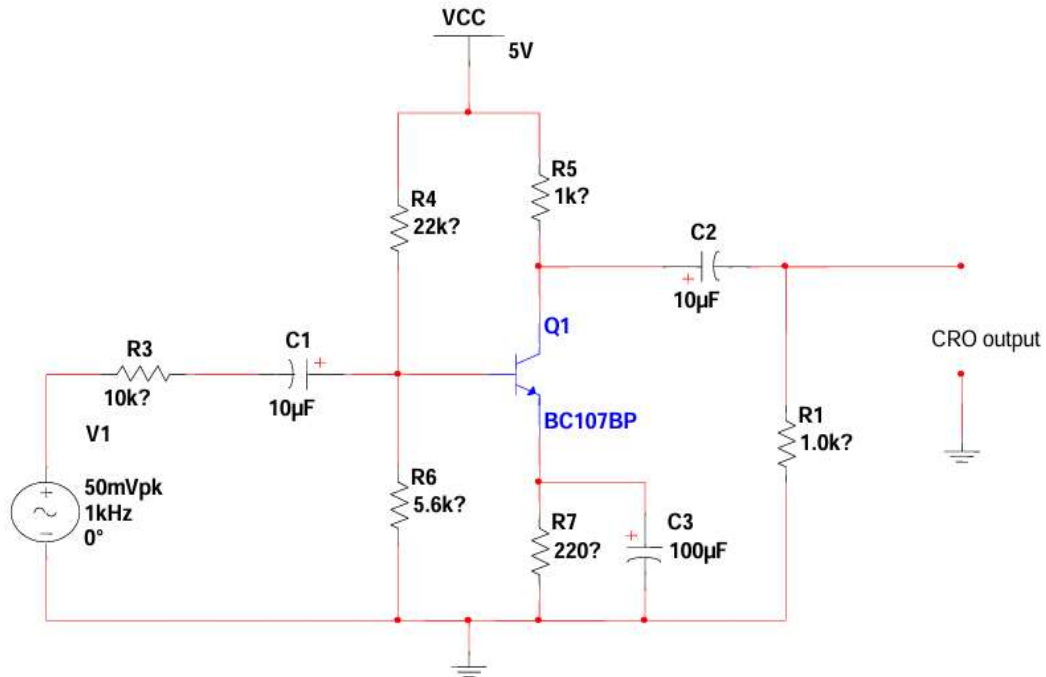
An amplifier increases the amplitude of an input signal. However, practical amplifiers do not amplify all frequencies equally. The frequency response of an amplifier shows the variation of gain with frequency.

The bandwidth (BW) of an amplifier is defined as the range of frequencies between the lower cut-off frequency (f_L) and upper cut-off frequency (f_H), over which the gain remains nearly constant.

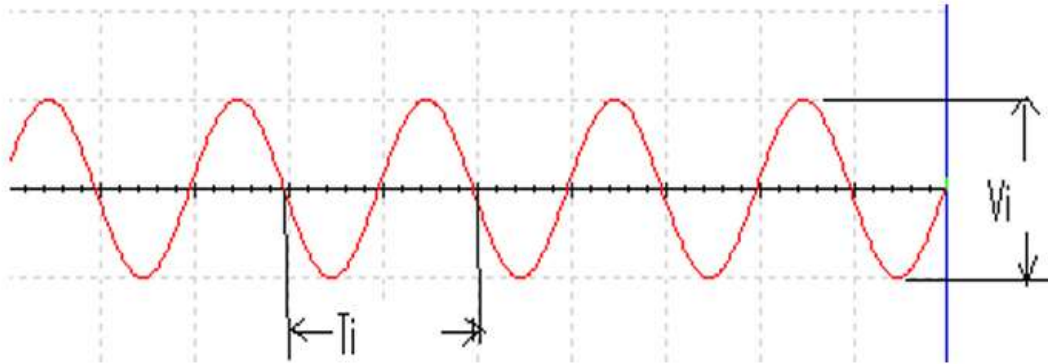
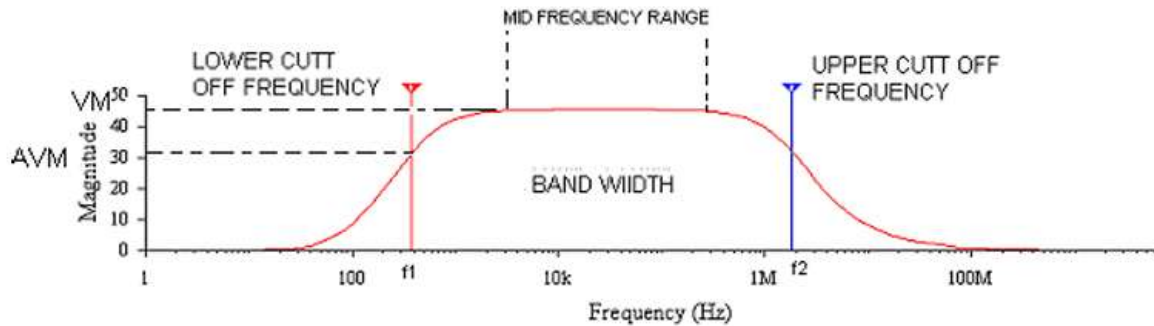
At the cut-off frequencies, the gain falls to 0.707 times (-3 dB) of the maximum gain.

$$\text{Bandwidth (BW)} = f_H - f_L$$

A wider bandwidth indicates better amplifier performance for signal amplification without distortion. Knowledge of bandwidth is essential in designing audio, RF, and communication amplifiers.

CIRCUIT DIAGRAM:**PROCEDURE: -**

1. Connect the circuit according to the circuit diagram as shown in figure on breadboard.
2. Set the power supply at 12V and function generator signal amplitude (20 to 50mV) for (sine wave) 1 KHz frequency on CH-1 knob to minimum position.
3. FEED the signal sine wave (20 to 50mV) to the input of CE amplifier and observe the V_i voltage on Ch-1 & output V_o voltage on Ch-2.
4. keeping the input signal unchanged select the range switch (10Hz-1MHz) in steps.
5. Note down the V_o output voltage amplitude for different frequency {15Hz, 25Hz, 100Hz...1MHz}
6. Tabulate the results in tabular form.
7. After calculation A_v and gain in dB using semi-logarithm sheet plot the curve

EXPECTED WAVE FORMS:**INPUT WAVE FORM:****OUTPUT WAVE FORM****FREQUENCY RESPONSE****CALCULATIONS:**

- Lower cut-off frequency, $f_L = \underline{\hspace{2cm}}$ Hz
- Upper cut-off frequency, $f_H = \underline{\hspace{2cm}}$ Hz

$$\text{Bandwidth} = f_H - f_L = \underline{\hspace{2cm}} \text{ Hz}$$

RESULT:

The frequency response of the given amplifier was studied and the lower cut-off frequency (f_L), upper cut-off frequency (f_H), and bandwidth were determined experimentally.

VIVA QUESTIONS:

1. What is an amplifier?
2. What is meant by frequency response of an amplifier?
3. Define bandwidth of an amplifier.
4. What are lower and upper cut-off frequencies?
5. Why is the gain constant in the mid-band region?
6. What is the significance of -3 dB point?
7. Why does gain decrease at low frequencies?
8. Why does gain decrease at high frequencies?
9. How is bandwidth mathematically expressed?
10. What is meant by mid-band gain?
11. What instruments are used to find the bandwidth of an amplifier?
12. Why is a sinusoidal input signal used in this experiment?
13. What is the effect of coupling capacitors on bandwidth?
14. How does bandwidth affect amplifier performance?
15. What is the relation between gain and bandwidth?
16. What happens to bandwidth when feedback is applied?
17. Why is bandwidth important in audio amplifiers?
18. Why is high bandwidth required in communication amplifiers?
19. What are the limitations of practical amplifiers?
20. How does bandwidth help in designing amplifiers?

REAL TIME APPLICATIONS:

Amplifier bandwidth plays a crucial role in real-time applications such as audio systems, where a wide and flat bandwidth ensures faithful amplification of sound without distortion, and in communication systems like radio, television, and mobile networks, where sufficient bandwidth is required to accurately amplify modulated signals. In medical electronics, amplifiers with controlled bandwidth are used in ECG and EEG machines to amplify bio-signals while rejecting noise. Instrumentation and sensor circuits rely on proper bandwidth selection to accurately amplify fast-changing signals, while in control systems and signal processing circuits, bandwidth determines system stability and response speed, making bandwidth analysis essential for practical amplifier design.