lendi Institute of Engineering & Technology
An Autonomous Institution
Accredited by NAAC with '' A'' Grade, Accredited by NBA (ECE, CSE.EEE & MECH)
Approved by A.I.C.T.E. & Permanently Affiliated to J. N. T. U. Gurajada, VIZIANAGARAM
Via 5th APSP Battalion, Jonnada (V), Denkada (M), NH-3, Vizianagaram Dist - 535005, A.P. Website : www.lendi.org
Ph : 08922-241111, 241666, Cell No : 9490344747, 9490304747,e-mail : lendi_2008@yahoo.com

Department of Electrical and Electronics Engineering

LAB MANUAL

Name of the laboratory : **ELECTRICAL & ELECTRONICS ENGINEERING WORKSHOP**

Regulation : **R23**

Subject Code : **R23EEE-ES1102**

Branch : **EEE**

Year & Semester : **I B.Tech- I Semester**

INSTITUTE

VISION

Producing globally competent and quality technocrats with human values for the holistic needs of industry and society.

MISSION

- Creating an outstanding infrastructure and platform for enhancement of skills, knowledge and behaviour of students towards employment and higher studies.
- Providing a healthy environment for research, development and entrepreneurship, to meet the expectations of industry and society.
- Transforming the graduates to contribute to the socio-economic development and welfare of the society through value-based education.

DEPARTMENT OF ELECTRICAL AND ELECTRONICS ENGINEERING

VISION

To be a center of excellence in imparting knowledge, skills and ethical values, while fostering innovation, sustainability and globally competent to make exemplary contributions to the field of Electrical and Electronics Engineering.

MISSION

- To impart technical education using state-of-the-art infrastructure, laboratories and instructional methods ensuring students acquire comprehensive knowledge and skills.
- To foster industry-oriented learning by facilitating internships, industrial visits, collaborative projects with industries.
- To create a congenial environment for higher education, employment and entrepreneurship by delivering quality education, enhancing professional skills and promoting research and innovation.
- To promote societal commitment and ethical leadership by instilling moral values and encouraging responsible engineering practices among students.

PROGRAM EDUCATIONAL OBJECTIVES (PEOs)

- Graduates will possess a strong foundation in core and interdisciplinary areas of Electrical and Electronics Engineering along with analytical and computational skills, enabling them to tackle global challenges through innovative and critical problem-solving.
- Graduates will actively engage in research, entrepreneurship, and innovation to address contemporary challenges in Electrical and Electronics Engineering while promoting sustainable and inclusive technological development for the betterment of society.
- Graduates will exhibit effective communication skills, collaborative abilities, and ethical values, preparing them for successful careers, higher education, and leadership roles in a rapidly evolving competitive environment.

PROGRAM OUTCOMES (POs)

PO1: Engineering knowledge: Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.

PO2: Problem analysis: Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.

PO3: Design/development of solutions: Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.

PO4: Conduct investigations of complex problems: Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.

PO5: Modern tool usage: Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.

PO6: The engineer and society: Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.

PO7: Environment and sustainability: Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.

PO8: Ethics: Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.

PO9: Individual and team work: Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.

PO10: Communication: Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.

PO11: Project management and finance: Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.

PO12: Life-long learning: Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

PROGRAM SPECIFIC OUTCOMES (PSOs)

- Capable of design, develop, test, verify and implement electrical and electronics engineering systems and products.
- Succeed in national and international competitive examinations for successful higher studies and employment

I Year I Semester

Course Code	Course Name	L	T	P	Credits
R23EEE-ES1102	Electrical & Electronics Engineering Workshop	0	0	3	1.5

Course Objectives:

- To impart knowledge on the fundamental laws & theorems of electrical circuits, functions of electrical machines and energy calculations.
- To impart knowledge on the principles of digital electronics and fundamentals ofelectron devices & its applications

Course Outcomes:

At the end of the course, the student will be able to

1. Apply theoretical concepts to obtain calculations for the measurement of electrical parameters.
2. Analyse various characteristics of electrical circuits, electrical machines and measuring instruments.
3. Design suitable circuits and methodologies for the measurement of various electrical parameters; Household and commercial wiring.
4. Summarize the characteristics of various electronic devices.
5. Analyze the different digital circuits.
6. Evaluate the electronic devices with simulation

Activities:

1. **Familiarization of commonly used Electrical & Electronic Workshop Tools:** Bread board, Solder, cables, relays, switches, connectors, fuses, Cutter, plier, screwdriver set,wire stripper, flux, knife/blade, soldering iron, de-soldering pump etc.

- Provide some exercises so that hardware tools and instruments are learned to be usedby the students.

2. Familiarization of Measuring Instruments like Voltmeters, Ammeters, multimeter, LCR-Q meter, Power Supplies, CRO, DSO, Function Generator, Frequency counter.

- Provide some exercises so that measuring instruments are learned to be used by the students.

3. **Components:**

- Familiarization/Identification of components (Resistors, Capacitors, Inductors, Diodes, transistors, IC's etc.) – Functionality, type, size, colour coding package, symbol, cost etc.
- Testing of components like Resistor, Capacitor, Diode, Transistor, ICs etc. - Compare values of components like resistors, inductors, capacitors etc with the measured values by using instruments

List of experiments:

PART A: Electrical Engineering Lab

1. Verification of KCL and KVL
2. Verification of Superposition theorem
3. Measurement of Resistance using Wheat stone bridge
4. Magnetization Characteristics of DC shunt Generator
5. Measurement of Power and Power factor using Single-phase wattmeter
6. Measurement of Earth Resistance using Megger
7. Calculation of Electrical Energy for Domestic Premises

PART B: Electronics Engineering Lab

1. Plot V-I characteristics of PN Junction diode A) Forward bias B) Reverse bias.
2. Plot V – I characteristics of Zener Diode and its application as voltage Regulator.
3. Implementation of half wave and full wave rectifiers
4. Plot Input & Output characteristics of BJT in CE and CB configurations
5. Frequency response of CE amplifier.
6. Simulation of RC coupled amplifier with the design supplied
7. Verification of Truth Table of AND, OR, NOT, NAND, NOR, Ex-OR, Ex-NOR gates using ICs.
8. Verification of Truth Tables of S-R, J-K& D flip flops using respective ICs.

Note: *Minimum Six Experiments to be performed in each part. All the experiments shall be implemented using Hardware/ Software.*

References:

1. Basic Electrical Engineering, D. C. Kulshreshtha, Tata McGraw Hill, 2019, First Edition
2. Power System Engineering, P.V. Gupta, M.L. Soni, U.S. Bhatnagar and A. Chakrabarti, Dhanpat Rai & Co, 2013
3. Fundamentals of Electrical Engineering, Rajendra Prasad, PHI publishers, 2014, ThirdEdition
4. R. L. Boylestad & Louis Nashlesky, Electronic Devices & Circuit Theory, Pearson Education, 2021.
5. R. P. Jain, Modern Digital Electronics, 4th Edition, Tata Mc Graw Hill, 2009
6. R. T. Paynter, Introductory Electronic Devices & Circuits – Conventional Flow Version, Pearson Education, 2009.

DEPARTMENT OF ELECTRICAL AND ELECTRONICS ENGINEERING

ELECTRICAL & ELECTRONICS ENGINEERING WORKSHOP LABORATORY

LIST OF EXPERIMENTS WITH CO-PO MAPPING			
EXP. No	**NAME OF THE EXPERIMENT**	**COURSE OUTCOMES**	**PROGRAM OUTCOMES**
1	Verification of KCL and KVL	CO1	PO1, PO2, PO3, PO4, PO9, PO12, PSO1
2	Verification of Superposition theorem	CO1	PO1, PO2, PO3, PO4, PO9, PO12, PSO1
3	Magnetization Characteristics of DC shunt Generator	CO2	PO1, PO2, PO3, PO4, PO5, PO9, PO12, PSO1
4	Measurement of Power and Power factor using Single-phase wattmeter	CO3	PO1, PO2, PO3, PO4, PO5, PO9, PO12
5	Measurement of Earth Resistance using Megger	CO3	PO1, PO2, PO3, PO4, PO5, PO9, PO12
6	Calculation of Electrical Energy for Domestic Premises	CO3	PO1, PO2, PO3, PO4, PO9, PO12, PSO1
7	Plot V-I characteristics of PN Junction diode A) Forward bias B) Reverse bias.	C04	PO1, PO2, PO3, PO4, PO9, PO12, PSO1
8	Plot V – I characteristics of Zener Diode and its application as voltage Regulator.	C04	PO1, PO2, PO3, PO4, PO9, PO12, PSO1
9	Implementation of half wave and full wave rectifiers	C04	PO1, PO2, PO3, PO4, PO9, PO12, PSO1
10	Plot Input & Output characteristics of BJT in CE and CB configurations	C04	PO1, PO2, PO3, PO4, PO9, PO12, PSO1
11	Verification of Truth Table of AND, OR, NOT, NAND, NOR, Ex-OR, Ex-NOR gates using ICs.	C05	PO1, PO2, PO3, PO4, PO5, PO9, PO12, PSO1

12	Frequency response of CE amplifier.	C06	PO1, PO2, PO3, PO4, PO9, PO12, PSO1
ADDITIONAL EXPERIMENTS			
1	Measurement of Resistance using Wheatstone Bridge	CO3	PO1, PO2, PO3, PO4, PO5, PO9, PO12
2	Verification of Truth Tables of S-R, J-K& D flip flops using respective ICs.	CO5	PO1, PO2, PO3, PO4, PO5, PO9, PO12, PSO1
3	Simulation of RC coupled amplifier	C06	PO1, PO2, PO3, PO4, PO5, PO9, PO12, PSO1

COURSE OUTCOMES	
CO1	Apply theoretical concepts to obtain calculations for the measurement of electrical parameters (L3).
CO2	Analyse various characteristics of electrical circuits, electrical machines and measuring instruments(L4).
C03	Design suitable circuits and methodologies for the measurement of various electrical parameters; Household and commercial wiring (L6).
CO4	Summarize the characteristics of various electronic devices (L2).
CO5	Analyze the different digital circuits (L4).
CO6	Evaluate the electronic devices with simulation (L5).

COURSE DATA SHEET

PROGRAM: ELECTRICAL AND ELECTRONIC ENGINEERING	DEGREE: **B. TECH**
COURSE**: Electrical & Electronics Engineering Workshop Lab R23EEE-ES1102**	SEMESTER: **I-I** CREDITS: **1.5**
COURSE CODE: **C108** REGULATION: **R23**	COURSE TYPE: **CORE**
COURSE AREA/DOMAIN: **Core Engineering**	CONTACT HOURS**: 3 hours/Week**.
CORRESPONDING LAB COURSE CODE (IF ANY): -	LAB COURSE NAME (IF ANY): **-**

SYLLABUS:

S.No	DETAILS	HOURS
I	Verification of KCL and KVL	**3**
II	Verification of Superposition theorem	**3**
III	Magnetization Characteristics of DC shunt Generator	**3**
IV	Measurement of Power and Power factor using Single-phase wattmeter	**3**
V	Measurement of Earth Resistance using Megger	**3**
VI	Calculation of Electrical Energy for Domestic Premises	**3**
VII	Plot V-I characteristics of PN Junction diode A) Forward bias B) Reverse bias.	**3**
VIII	Plot V – I characteristics of Zener Diode and its application as voltage Regulator.	**3**
IX	Implementation of half-wave and full-wave rectifiers.	**3**
X	Plot Input & Output characteristics of BJT in CE and CB configurations using ICs.	**3**
XI	Frequency response of CE amplifier.	**3**
XII	Verification of Truth Table of AND, OR, NOT, NAND, NOR, Ex-OR, Ex-NOR gates	**3**
	TOTAL HOURS	**36**

TEXT/REFERENCE BOOKS:

T/R	BOOK TITLE/AUTHORS/PUBLICATION
R	Basic Electrical Engineering, D. C. Kulshreshtha, Tata McGraw Hill, 2019, First Edition
R	Power System Engineering, P.V. Gupta, M.L. Soni, U.S. Bhatnagar and A. Chakrabarti, Dhanpat Rai & Co, 2013
R	Fundamentals of Electrical Engineering, Rajendra Prasad, PHI publishers, 2014, Third Edition
R	R. L. Boylestad & Louis Nashlesky, Electronic Devices & Circuit Theory, Pearson Education, 2021.
R	R. P. Jain, Modern Digital Electronics, 4th Edition, Tata Mc Graw Hill, 2009
R	R. T. Paynter, Introductory Electronic Devices & Circuits – Conventional Flow Version,

	Pearson Education, 2009.

COURSE PRE-REQUISITES:

C.CODE	COURSE NAME	DESCRIPTION	YEAR-SEM
1	INTERMEDIATE PHYSICS	BASIC PHYSICS	INTER

COURSE OBJECTIVES:

1	To impart knowledge on the fundamental laws & theorems of electrical circuits, functions of electrical machines and energy calculations.
2	To impart knowledge on the principles of digital electronics and fundamentals of electron devices & its applications

COURSE OUTCOMES:

S. No.	DESCRIPTION	PO(1..12) MAPPING	PSO(1,2) MAPPING
1	Apply theoretical concepts to obtain calculations for the measurement of electrical parameters.	PO1,PO2, PO3,PO4 ,PO9,PO12	-
2	Analyze various characteristics of electrical circuits, electrical machines and measuring instruments.	PO1, PO2,PO3,PO4,PO5,PO9,PO12	PSO2
3	Design suitable circuits and methodologies for the measurement of various electrical parameters, Household and commercial wiring.	PO1,PO2,PO3,PO4,PO9,PO12	-
4	Summarize the characteristics of various electronic devices.	PO1, PO2,PO3,PO4 ,PO9,PO12	-
5	Analyze the different digital circuits.	PO1, PO2,PO3,PO4,PO5,PO9,PO12	
6	Evaluate the electronic devices with simulation	PO1, PO2,PO3,PO4,PO5,PO9,PO12	PSO2
COURSE OVERALL PO/PSO MAPPING: PO1,PO2,PO3,PO4,PO5,PO9,PO12,PSO2.			

COURSE OUTCOMES VS POs MAPPING (DETAILED; HIGH: 3; MEDIUM: 2; LOW: 1**):**

S.No.	PO 1	PO 2	PO 3	PO 4	PO 5	PO 6	PO 7	PO 8	PO 9	PO 10	PO 11	PO 12	PSO 1	PSO 2
CO1	3	3	1	2	-	-	-	-	3	-	-	2	-	-
CO2	3	3	2	2	1	-	-	-	3	-	-	2	-	1
CO3	3	3	3	2	-	-	-	-	3	-	-	2	-	-
CO4	3	3	1	2	-	-	-	-	3	-	-	2	-	-
CO5	3	3	2	3	2	-	-	-	3	-	-	2	-	-
CO6	3	3	2	3	3	-	-	-	3	-	-	2	-	3
CO*	3	3	2	2	2	-	-	-	3	-	-	2	-	2

** For Entire Course, PO & PSO Mapping*

POs & PSO REFERENCE:

PO1	Engineering Knowledge	PO7	Environment & Sustainability	PSO1	Capable of design, develop, test, verify and implement Electrical and Electronics Engineering systems and products.
PO2	Problem Analysis	PO8	Ethics	PSO2	Succeed in national and international competitive examinations for successful higher studies and employment.
PO3	Design & Development	PO9	Individual & Team Work		
PO4	Investigations	PO10	Communication Skills		
PO5	Modern Tools	PO11	Project Mgt. & Finance		
PO6	Engineer & Society	PO12	Life Long Learning		

COs VS POs MAPPING JUSTIFICATION:

SNO	PO/PSO MAPPED	LEVEL OF MAPPING	JUSTIFICATION
CO 1	PO 1	3	Engineering fundamentals strongly required for verification of KCL, KVL and Superposition Theorem.
	PO 2	3	Analyze the electrical circuit strongly to measure voltage and current in an electric circuit.
	PO 3	1	Design and develop the electric circuit to measure electrical parameters, Voltage and Current.
	PO 4	2	Conduct investigations Somewhat to develop complex circuits and able to measure electrical parameters for large-scale complex circuits.
	PO 9	3	Function effectively as an individual and as a member in a team are highly required for verification of KCL, KVL and Superposition Theorem.
	PO 12	2	Lifelong learning is needed to understand the technological changes.
CO 2	PO 1	3	Engineering fundamentals are highly required to Analyze various characteristics of electrical circuits, electrical machines and measuring instruments.
	PO 2	3	Analyzing the students can Strongly obtain knowledge of characteristics of electrical circuits, electrical machines and measuring instruments.
	PO 3	2	The students can able to Design and Develop electrical circuits, electrical machines and measuring instruments and analyse their characteristics.

	PO 4	2	Conduct investigations Somewhat to develop complex electric circuits, electrical machines and measuring instruments and able to analyze their characteristics.
	PO 5	1	Slightly Analyze the electrical machines and measuring instruments and solve the electrical circuits by the usage of modern tools.
	PO 9	3	Function effectively as an individual and as a member in a team are highly required to Analyze various characteristics of electrical circuits, electrical machines, measuring instruments.
	PO 12	2	Lifelong learning is needed to understand the technological changes.
CO 3	PO 1	3	Engineering knowledge is strongly required to Design suitable circuits and methodologies for the measurement of various electrical parameters, Household and commercial wiring.
	PO 2	3	Analyze the suitable circuits and methodologies so that the students can design suitable circuits and methodologies for the measurement of various electrical parameters, Household and commercial wiring
	PO 3	3	Design and Develop suitable circuits and methodologies for the measurement of various electrical parameters, Household and commercial wiring
	PO 4	2	Conduct investigations to design and develop Somewhat suitable circuits and methodologies for measurement of various electrical parameters, Household and commercial wiring.
	PO 9	3	Function effectively as an individual and as a member in a team are highly required to Design and Develop suitable circuits and methodologies for measurement of various electrical parameters, Household and commercial wiring.
	PO 12	2	Lifelong learning is needed to understand the technological changes.
CO 4	PO 1	3	Engineering fundamentals are highly required to Summarize the characteristics of various electronic devices.
	PO 2	3	Analyze the electronic devices to understand their characteristics.
	PO 3	1	Design and Develop electronic devices for their characteristics.
	PO 4	2	Conduct investigations to design and develop electronic circuits to summarize their characteristics.

	PO 9	3	Function effectively as an individual and as a member in a team are highly required to Summarize the characteristics of various electronic devices.
	PO 12	2	Lifelong learning is needed to understand the technological changes.
CO5	PO1	3	Engineering fundamentals are highly required to analyze various digital circuits.
	PO 2	3	Analyze the different digital circuits like AND, OR, NOT, NAND, NOR, Ex-OR, Ex-NOR gates electronic devices to understand their characteristics.
	PO3	2	Design and Develop the different digital circuits using various gates.
	PO4	3	Slightly Conduct investigations to design and develop digital circuits and to verify their truth-tables.
	PO5	2	Digital Circuits can be analyzed by Design and Develop by using the modern tools and verify their truth tables.
	PO9	3	Function effectively as an individual and as a member in a team are highly required for verify the truth tables of various digital circuits.
	PO12	2	Lifelong learning is needed to understand the technological changes.
CO6	PO1	3	Engineering fundamentals are highly required to evaluate the electronic devices with simulation
	PO 2	3	Analyze the electronic devices strongly with simulation to evaluate them effectively.
	PO3	2	Design and Develop the different electronic devices using simulation.
	PO4	3	Conduct investigations Strongly to design and develop digital electronic devices using simulation.
	PO5	3	Electronic devices can be evaluated by usage of modern tools.
	PO9	3	Function effectively as an individual and as a member in a team are highly required to evaluate various electronic devices.
	PO12	2	Lifelong learning is needed to understand the technological changes.
	PSO2	3	Have knowledge and expertise to analyze electronic devices using latest tools and technologies to evaluate them.

WEB SOURCE REFERENCES:

1	www.electrical4u.com
2	www.electricaleasy.com
3	www.learnengineering.org
4	www.studyelectrical.com

DELIVERY/INSTRUCTIONAL METHODOLOGIES:

☑ CHALK & TALK	☐ STUD. ASSIGNMENT	☐ WEB RESOURCES	☐ NPTEL/OTHERS
☐ LCD/SMART BOARDS	☐ STUD. SEMINARS	☐ ADD-ON COURSES	☐ WEBNIARS

ASSESSMENT METHODOLOGIES-DIRECT

☐ ASSIGNMENTS	☐ STUD. SEMINARS	☑ TESTS/MODEL EXAMS	☑ END SEM EXAM
☑STUD. LAB PRACTICES	☑ STUD. VIVA	☐ MINI/MAJOR PROJECTS	☐ CERTIFICATIONS
☐ ADD-ON COURSES	☐ OTHERS		

☑ COURSE EXIT SURVEY			

ASSESSMENT METHODOLOGIES-INDIRECT

Prepared by
Name: A.Anil kumar
Signature:

Course Co-ordinator
Name:
Signature:

Approved by PAC

Name:

Signature:

Head of the Department

EXPERIMENT 1

VERIFICATION OF KCL AND KVL

A) Verification of Kirchhoff Voltage Law

AIM: To verify Kirchhoff Voltage Law practically for the given circuit.

APPARATUS:

S.NO	APPARATUS	RANGE	TYPE	QUANTITY
1.	Rheostats	50 Ω/5A	Wire wound	1
2.	Rheostats	300 Ω/2A	Wire wound	1
3.	Voltmeter	(0-300) V	MC	3
4.	Ammeters	(0-5) A	MC	1
5.	Connecting wires	Required no		

Theory:

Kirchhoff's Voltage Law (KVL) states that "The algebraic sum of all voltages around a closed loop or mesh is equal to zero".
Mathematically, this is expressed as:

$$\sum V=0$$

Where:

- $\sum V$ represents the sum of all voltages in the loop.
- This includes both the voltage drops across resistors (or other components) and the electromotive forces (emf) provided by sources like batteries.

This law is based on the law of conservation of energy, which states that energy supplied to a circuit by a source must be dissipated by the resistive elements of the circuit. In other words, when a current flows through a closed loop, the total energy supplied by the power sources (like batteries or voltage sources) equals the total energy lost in the form of voltage drops across resistive elements (like resistors).

Kirchhoff's Current Law (KCL) is one of the fundamental laws of electrical circuit analysis. It states that:" The total current entering a junction (or node) in a circuit is equal to the total current leaving the junction".

Mathematically, KCL can be expressed as:

$$\sum I_{in} = \sum I_{out}$$

Where:

- $\sum I_{in}$ represents the sum of currents entering the node.
- $\sum I_{out}$ represents the sum of currents leaving the node.

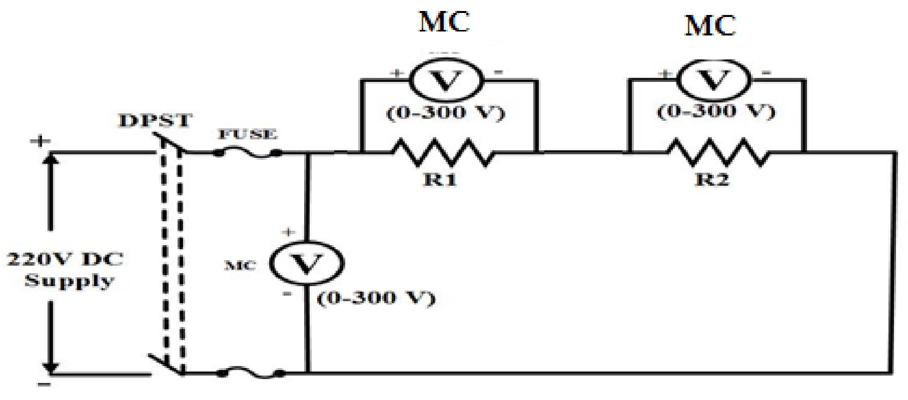
Alternatively, the sum of all currents at a node must be zero:

$$\sum I = 0$$

This means:

- Currents flowing into a node are considered positive.
- Currents flowing out of the node are considered negative.

Circuit Diagram:



VERIFICATION OF KIRCHHOFF'S VOLTAGE LAW (KVL)

For R1= 50 ohm , R2= 300 ohm

PROCEDURE:

1. Make the connections as per circuit diagram
2. Close the DPST Switch and note the voltmeter readings and tabulate them.

3. Switch off the supply and remove the connections.

TABULAR FORM:

V_S (V)	V_1 (V)	V_2 (V)

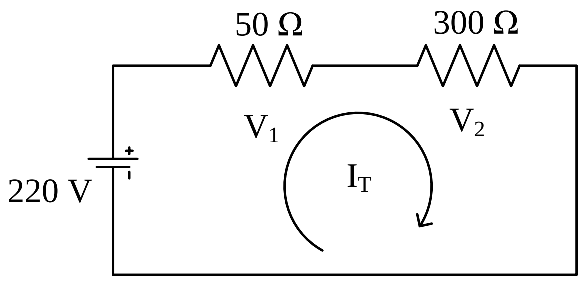
PRECAUTIONS:

1. Avoid loose connections
2. Avoid parallax Error while taking the reading.

RESULT:

OUTCOME:

Theoretical Calculations:



From the Given circuit,

Total Current, I_T =

Voltage across resistor R_1, $V_1 = I*R_1$ =

Voltage across resistor R_2, $V_2 = I*R_2$ =

According to the KVL,

$V = V_1 + V_2$ =

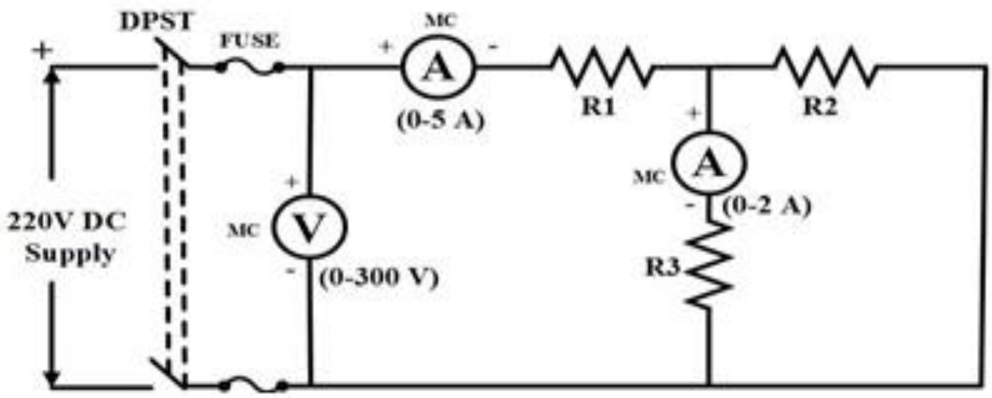
B) VERIFICATION OF KIRCHHOFF CURRENT LAW

AIM: To verify Kirchhoff Current Law practically for the given circuit.

APPARATUS:

S.NO	APPARATUS	RANGE	TYPE	QUANTITY
1.	Rheostats	50 Ω/5A	Wire wound	2
2.	Rheostats	300 Ω/2A	Wire wound	1
3.	Voltmeter	(0-300) V	MC	1
4.	Ammeters	(0-5) A	MC	1
		(0-2) A		1
5.	Connecting wires	Required no		

CIRCUIT DIAGRAM:



VERIFICATION OF KIRCHHOFF'S CURRENT LAW (KCL)

FOR R1= 50Ω , R2= 300Ω , R3= 50Ω

PROCEDURE:

1. Make the connections as per circuit diagram
2. Close the DPST Switch and note the ammeter reading and tabulate them.
3. Switch off the supply and remove the connections.

TABULAR FORM:

I_S (A)	I_1 (A)	I_2 (A)

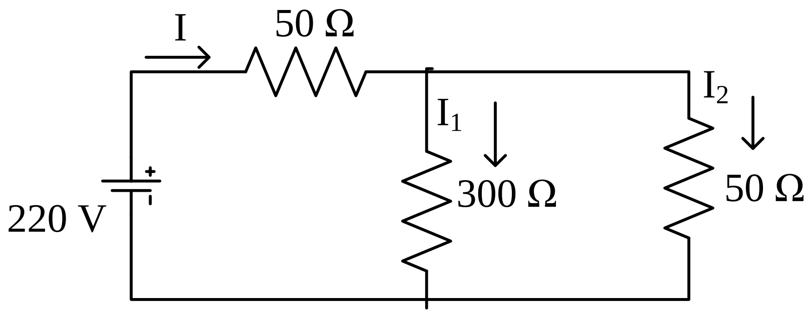
PRECAUTIONS:

1. Avoid loose connections
2. Avoid parallax Error while taking the reading.

RESULT:

OUTCOME:

Theoretical Calculations:



From the Given circuit,

Total Resistance can be calculated as,

R_{Total} =

Total Current can be calculated as,

I_T =

Apply Current Division Rule,

I_1 =

I_2 =

According to KCL, $I = I_1 + I_2$ =

EXPERIMENT 2

VERIFICATION OF SUPERPOSITION THEOREM

AIM: To verify the super position theorem practically.

APPARATUS:

S. No	Equipment	Range	Type	Qty
1	Rheostats	300Ω/2A	W.W	1
2	Rheostats	50Ω/5A	W.W	2
3	Ammeter	(0-2)A	MC	1
4	Voltmeters	(0-300)V	MC	2
5	SPST	---	---	2
5	Connecting wires	---	---	Required

Theory:

Superposition Theorem states that: "In a linear electrical circuit with multiple independent sources, the total response (voltage or current) in any component of the circuit is the algebraic sum of the individual responses caused by each independent source acting alone, with all other sources replaced by their internal impedance". The theorem is based on the principles of linearity (the voltage or current response is proportional to the input) and additivity (the total effect is the sum of individual effects).

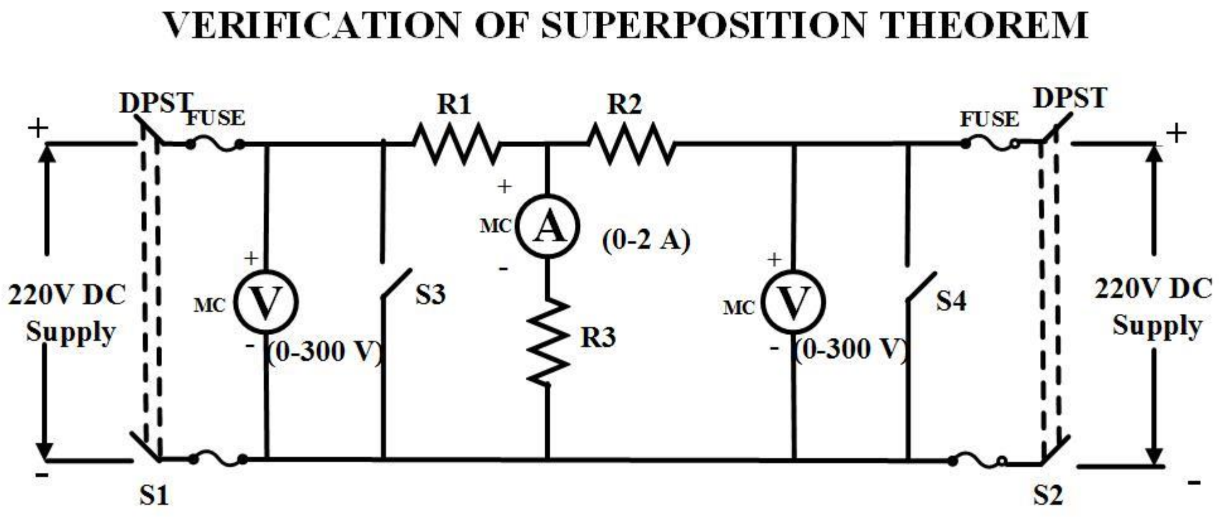
Superposition Theorem Steps:

1. Consider one source at a time:
    - Turn off all the other independent sources.
        - For voltage sources: Replace them with a short circuit.
        - For current sources: Replace them with an open circuit.
2. Solve the circuit with the remaining active source(s) and calculate the current or voltage in the component of interest.
3. Repeat the process for each independent source in the circuit.
4. Sum up all the individual responses (voltages or currents) from each active source. This will give the total response in the component.

Applications of Superposition Theorem:

1. **Circuit Analysis:**
    - The Superposition Theorem simplifies the analysis of complex circuits with multiple sources by breaking down the problem into smaller, manageable parts. It helps in finding the voltage or current at any point in the circuit.
2. **Power Systems:**
    - In power systems, the Superposition Theorem is useful for analyzing how different power sources contribute to the total power supplied to a load.
3. **Design and Troubleshooting:**
    - Engineers use the Superposition Theorem when designing circuits or troubleshooting existing ones to determine the effect of individual sources on the overall circuit performance.

Circuit Diagram:



PROCEDURE:

1. Connect the circuit as shown in circuit diagram.
2. Make sure that switch S3 & S4 are in open position initially.
3. Switch ON the DC Supply's by closing DPST S1 & S2.
4. Note down the Ammeter reading **(I_1)** which is the current flowing in Rheostat R3.
5. Switch OFF the DC Supply's by opening DPST switches S1 & S2.
6. Now Deactivate voltage source $V_2 = 0$ by closing Switch S4.
7. Switch ON the DC Supply by closing **DPST S1 only** and ensure Switches S2 & S3 in open position and Note down the Ammeter reading (I^1_1)
8. Switch OFF the DC Supply's by opening DPST switches S1.
9. Now Deactivate voltage source $V_1 = 0$ by closing Switch S3.

10. Switch ON the DC Supply by closing **DPST S2 only** and ensure Switches S1 & S4 are in open position and Note down the Ammeter reading ($\mathbf{I^{11}_1}$)

11. Sum of the Currents in Table 2 and table 3, will verify the superposition theorem in Table 1 reading practically.

OBSERVATION TABLE:

S. No	**Switch Position**	**V_1 (V)**	**V_2(V)**	**I (A)**
1	S1-ON, S2-ON, S3-OFF, S4-OFF	220 V	220 V	
2	S1-ON, S2-OFF, S3-OFF, S4-ON	220 V	0	
3	S1-OFF, S2-ON, S3-ON, S4-OFF	0	220 V	

PRECAUTIONS:

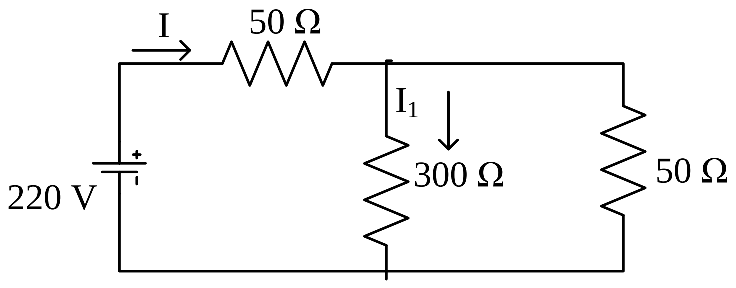
1. Avoid loose connections are allowed.
2. Readings should be taken without parallax error.
3. The apparatus should be handled carefully.
4. Remove the components only after the supply is switched off.

RESULT:

OUTCOME:

Theoretical Calculations:

Case 1: When (A) is activated i.e. Vs = 220 V



From the Given circuit,

Total Resistance can be calculated as,

R_{Total} =

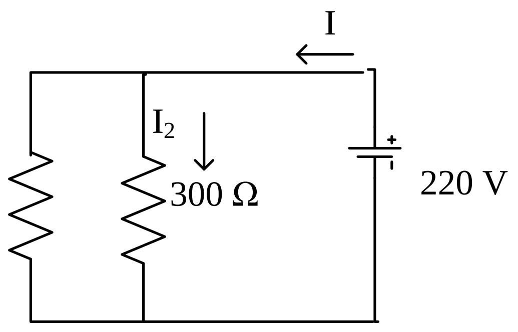
Total Current can be calculated as,

I_T =

Apply Current Division Rule,

I_1 =

Case 2: When (B) is activated i.e. Vs = 220 V



From the Given circuit,

Total Resistance can be calculated as,

R_{Total} =

Total Current can be calculated as,

I_T =

Apply Current Division Rule,

I_1 =

EXPERIMENT 3

MAGNETIZATION CHARACTERISTICS OF A DC SHUNT GENERATOR

AIM: To conduct OCC test on DC Shunt generator to calculate critical Field resistance and critical speed.

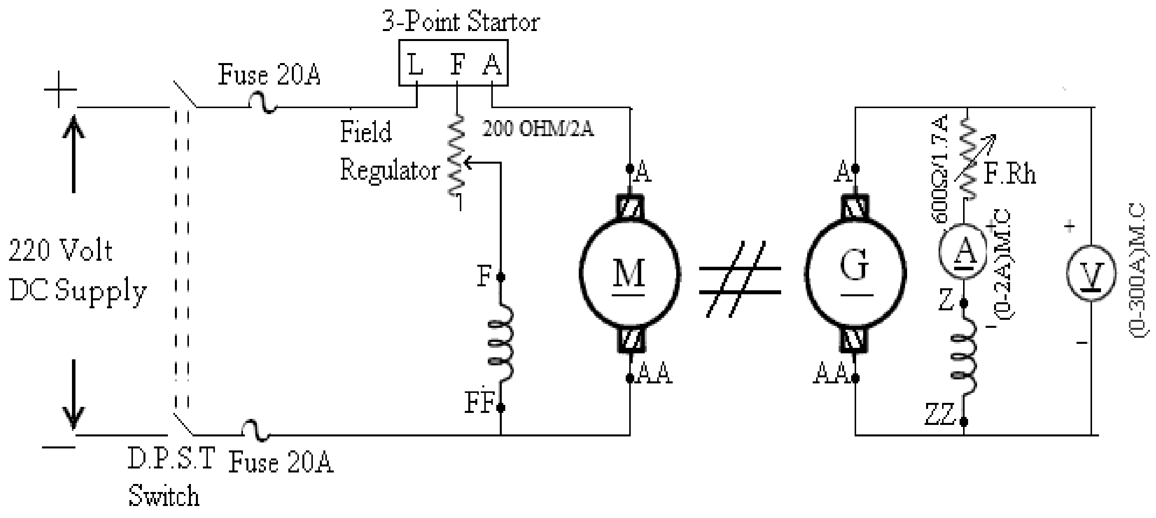
APPARATUS REQUIRED:

S.No.	Name of the Apparatus	Range	Type	Quantity
1	Voltmeters	0-300 V	Moving Coil	1
2	Ammeters	0-2 A	Moving Coil	1
3	Tachometer	0-3000	digital	1
4	Rheostat	300 Ω/2A 600 Ω/2A	Wire Wound	1 1
5	Connecting wires	--	--	As Required

NAME PLATE DETAILS:

S.no	Parameter	Motor	Generator
1	H.P/K.W	3 HP	2 KW
2	Speed	1500	1500
3	Armature Voltage	220 V	220 V
4	Field Voltage	220 V	220 V
5	Armature current	12 A	9 A
6	Field current	0.6 A	0.7 A

CIRCUIT DIAGRAM:



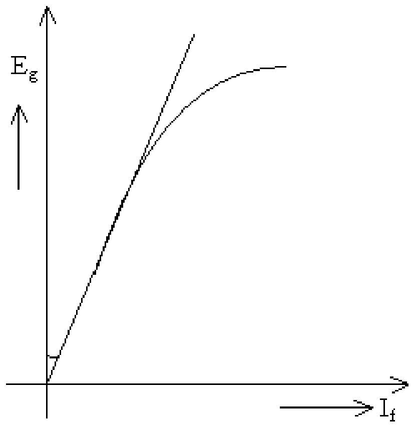
PROCEDURE:

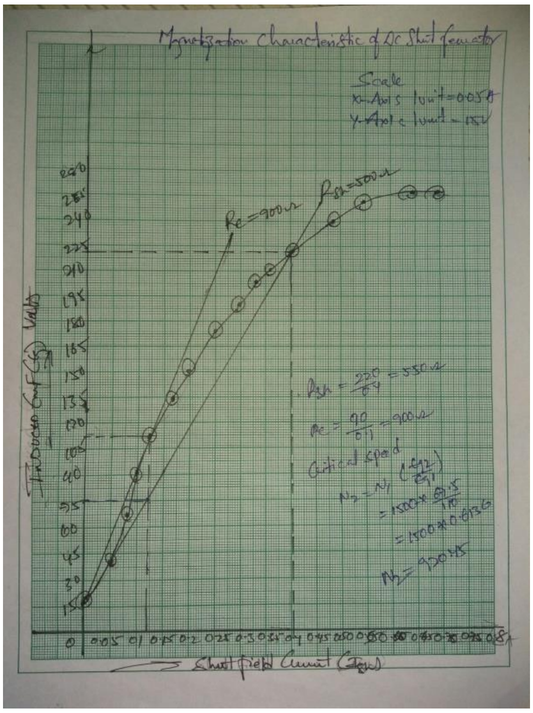
1. Connections are made as per the circuit diagram.
2. Keep the Rheostat Maximum position, Field Regulator at Minimum positions and load at OFF position.
3. Switch on the power supply in the presence of the faculty member. Move the starter from OFF position to ON position.
4. Run the machine at its rated speed with the help of Field Regulator. Note down the Voltmeter and Ammeter readings.
5. Move the Field Rheostat from Maximum to Minimum in terms of Field current up to generate rated Voltage. Note down the Voltmeter and Ammeter readings.
6. Move the Field Rheostat from Minimum to Maximum in terms of Field current up to generate rated Voltage. Note down the Voltmeter and Ammeter readings.
7. Bring the Field Regulator to Minimum position. Switch OFF the supply, Remove the connections.
8. Plot the graph to find the critical resistance, draw a Field resistance line, tangential to OCC curve. The resistance corresponding to the tangent is the Critical resistance.

TABULARFORM:

SNO	INCREASING		INCREASING	
	If (A)	Eg (V)	If (A)	Eg (V)
1				
2				
3				
4				
5				
6				
7				
8				
9				

MODEL GRAPH:





PRECAUTIONS:

1. Give the connections tightly.
2. Circuit connections should not be made while power is ON.
3. Keep the load at open position.
4. Avoid the parallax error to note down the readings.

RESULT:

OUTCOME:

EXPERIMENT 4

MEASUREMENT OF POWER AND POWER FACTOR USING SINGLE-PHASE WATTMETER

AIM: To measure voltage, current and power in single phase circuit using voltmeter, ammeter and wattmeter. Also calculate Power factor of the circuit.

APPARATUS REQUIRED:

Sl.No.	Name of the Apparatus	Range	Type	Quantity
1	Voltmeters	0-300 V	Moving Iron	1
2	Ammeters	0-2 A	Moving Iron	1
3	Wattmeter	300V/2A	UPF	1
4	Choke coil	--	--	1
5	Connecting wires	--	--	As Required

Theory:

The measurement of power and power factor is crucial in electrical engineering to determine the efficiency and energy consumption of electrical systems. A wattmeter is an essential instrument used to measure the electrical power (in watts) consumed by a load. This measurement can be done in both single-phase and three-phase systems, but in this experiment, we focus on the single-phase wattmeter. In a single-phase AC circuit, the power can be calculated as the product of the voltage, current, and the cosine of the phase angle between the current and voltage waveforms (i.e., power factor). The power factor is a dimensionless number that indicates how efficiently the electrical power is being used. A wattmeter measures the instantaneous power, which is the product of the instantaneous voltage and current. The principle of operation of a wattmeter is based on the interaction between the magnetic fields produced by the current in the coils of the wattmeter and the voltage applied to the circuit.

Power Measurement:

The power PPP measured by the wattmeter in a single-phase circuit is given by:

$$P=V\cdot I\cdot\cos\phi$$

where:

- V is the voltage applied across the load,
- I is the current flowing through the load,
- ϕ is the phase angle between the current and the voltage waveforms.

The cosine of the phase angle, cosϕ, is the power factor (PF) of the circuit. The power factor indicates the proportion of the total power that is being effectively converted into useful work. A power factor of 1 means all the power is being used effectively (in purely resistive circuits), while values less than 1 indicate the presence of reactive power (in inductive or capacitive circuits).

Power Factor Measurement:

To measure the power factor, we can use the **wattmeter** and compare the measured power with the apparent power in the circuit. The apparent power SSS is given by:

$$S=V\cdot I$$

The power factor is then calculated as:

$$\textbf{Power Factor}=\mathbf{P/S}$$

Thus, by measuring the power with the wattmeter and calculating the apparent power, the power factor can be determined.

By using a single-phase wattmeter, both the power and power factor of a single-phase AC circuit can be measured accurately. This experiment provides insights into the practical operation of electrical systems and helps in understanding how efficiently electrical energy is being utilized by a load. A good understanding of power and power factor is essential for the design, analysis, and optimization of electrical circuits.

Circuit Diagram:

Procedure:

1. Connections are made as per the circuit diagram.
2. Connect the wattmeter in series with the load and ensure that the potential coil is connected across the load.
3. Apply the rated voltage to the circuit and switch on the power supply.
4. Observe the wattmeter deflection, which corresponds to the power being consumed by the load.
5. Use the measured power (P) and the known values of voltage and current to calculate the power factor.

TABULAR FORM:

Voltage (V)	Current (A)	Power (W)

Calculations:

$$P = V*I* \cos\phi$$

$$P.F., \cos\phi = P/(V*I)$$

PRECAUTIONS:

1. Avoid loose connections
2. Avoid parallax Error while taking the reading.

RESULT:

OUTCOME:

EXPERIMENT 5

MEASUREMENT OF EARTH RESISTANCE USING MEGGER

AIM: To measure the earth's resistance using Megger.

APPARATUS REQUIRED:

S.No.	Name of the Apparatus	Quantity
1	Earth Resistance Tester	1
2	Spikes	2
5.	Connecting wires	As Required

Theory:

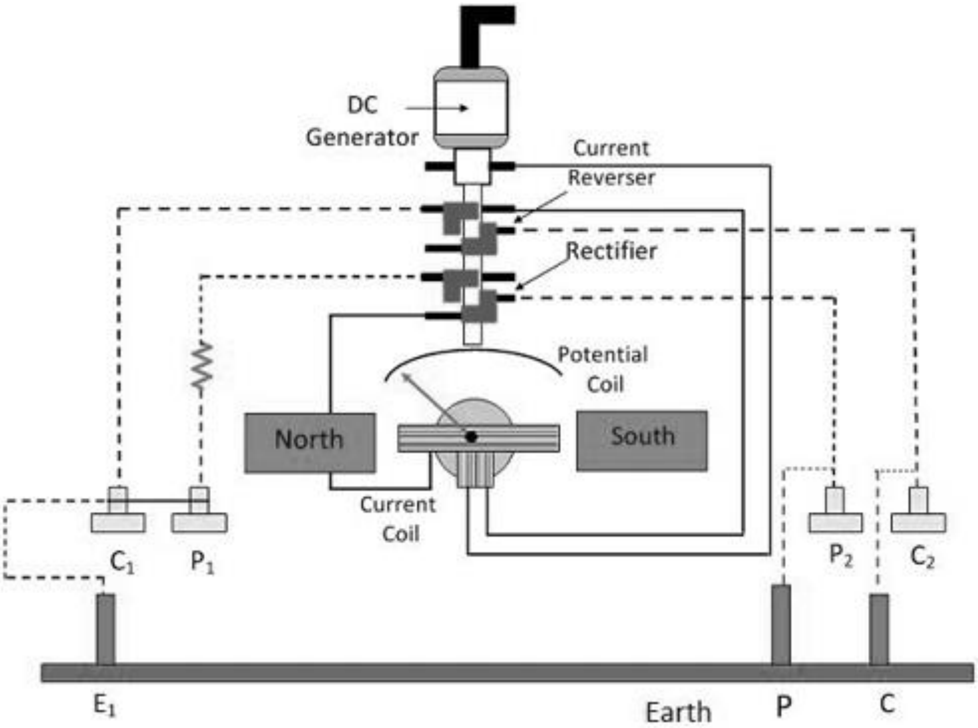
The earth resistance (also known as ground resistance) refers to the resistance encountered by an electrical current when it flows from an electrical system to the earth. Proper grounding is essential for the safety of electrical installations, ensuring that any fault current has a low-resistance path to the earth, thus preventing electrical shocks and protecting electrical equipment. To assess the effectiveness of the grounding system, the earth resistance must be measured. This measurement is typically performed using an instrument known as a megger, also referred to as an earth resistance tester. A megger applies a known test voltage to the earth electrode and measures the resulting current, enabling the calculation of the earth resistance.

A megger for earth resistance measurement typically consists of the following components:

1. Test Leads: These are connected to the earth electrode and the surrounding earth for measurement.
2. Current Source: The megger generates a known current, which is applied to the ground system.
3. Current Meter: Measures the current that flows through the earth system when the voltage is applied.
4. Voltage Meter: Measures the potential difference (voltage) applied between the earth electrode and the surrounding earth.
5. Earth Electrodes: The earth electrode is connected to the installation's grounding system, and auxiliary electrodes are placed in the surrounding earth at specific distances from the main electrode to complete the measurement.

The measurement of earth resistance is an important aspect of ensuring the safety and effectiveness of electrical grounding systems. By using a megger, the earth resistance can be easily measured, allowing for proper maintenance and compliance with safety standards. The use of the megger helps ensure that electrical faults are safely directed to the ground, protecting both personnel and equipment.

Megger Diagram:



Procedure:

1. Take the wires which are connected to the earth.
2. Name the terminal wires as 1,2,3.
3. Connect the wires to the assumed pair of terminals (1,2) to the megger and note down the values of earth's resistance. Here terminals of extra pair are (1,3) & (2,3).
4. Connect the wires to the assumed pair of terminals (2,3) to the megger and note down the values of earth's resistance. Here terminals of extra pair are (1,2) & (1,3).
5. Connect the wires to the assumed pair of terminals (3,1) to the megger and note down the values of earth's resistance. Here terminals of extra pair are (1,2) & (2,3).

Tabular Form:

Sl.No.	Assumed Pair of Terminals	Terminals of Extra Pair	Values of Earth's Resistance
1			
2			
3			

RESULT:

OUTCOME:

EXPERIMENT 6

CALCULATION OF ELECTRICAL ENERGY FOR DOMESTIC PREMISES

Aim: To calculate the energy for various domestic loads.

Apparatus Required:

1. Wattmeter or Energy Meter (for measuring power consumption).
2. Multimeter (for measuring voltage and current, if necessary).
3. Stopwatch (for measuring usage time).
4. Various household appliances (e.g., light bulbs, fans, refrigerators, air conditioners).
5. Calculator (for computing the energy consumption).

Theory:

Electrical energy is the energy consumed by electrical devices over a period of time. The energy consumption can be calculated using the formula:

$$\text{Energy (in kWh)} = \text{Power (in kW)} \times \text{Time (in hours)}$$

Where:

Power (P) is the rated power of the appliance (in kilowatts, kW), Time (T) is the duration for which the appliance is used (in hours) and Energy (E) is the electrical energy consumed (in kilowatt-hours, kWh).

Procedure:

1. **List the appliances:**
   - Make a list of electrical appliances in a domestic setting (e.g., refrigerator, fan, light bulb, air conditioner).
   - Record the rated power consumption of each appliance (in watts).
2. **Determine the time of usage:**
   - Measure or estimate the time each appliance is used over a period of time (e.g., per day or per month). This can be done using a stopwatch or based on regular usage patterns.
3. **Measurement of power:**
   - Use a wattmeter to measure the actual power consumption of each appliance. If a wattmeter is unavailable, the power rating is typically given by the manufacturer and can be found on the appliance's label.

4. **Calculate the energy consumption for each appliance:**
   Use the formula:

   Energy (in kWh)=Power (in W)1000×Time (in hours)

5. **Total Energy Consumption:**

- For a monthly energy usage, multiply the daily usage by 30 (or the number of days in the month).
- Sum up the energy consumption for all appliances to get the total energy consumed in the household.

6. **Record the observations:**

- Record the power rating of each appliance.
- Note the time of usage for each appliance.
- Calculate the energy consumed by each appliance.

7. **Compare energy consumption:**

- Compare the energy consumption of different appliances to understand which ones are more energy-hungry.
- You can then calculate the cost of energy consumption using the energy tariff provided by your electricity provider.

Sample Calculations:

Appliance	**Power Rating (W)**	**Time Used (hours)**	**Energy Consumed (kWh)**
Light Bulb			
Fan			
Refrigerator			
Air Conditioner			

Total Energy Consumption:

Total Energy (kWh)=0.3+0.32+4.8+4.5=9.92 kWh

Precautions:

1. Ensure all electrical appliances are properly plugged in before measuring their power consumption.
2. Handle electrical devices carefully to avoid electric shocks.
3. Ensure the wattmeter or multimeter is set correctly to measure power.
4. Always turn off appliances after usage to avoid unnecessary energy consumption.
5. Make sure the time measured for usage is accurate.

RESULT:

OUTCOMES:

EXPERIMENT NO: 7

Plot V-I characteristics of PN Junction diode A) Forward bias B) Reverse bias

Aim: To observe and draw the V-I Characteristics of a P-N Junction diode in Forward and Reverse bias. Calculate Static and Dynamic Resistances.

Apparatus Required:

PN Diode IN4007.

Regulated Power supply (0-30V)

Resistor 1KΩ

Ammeters (0-20 mA, 0-750µA)

Voltmeter (0-1V, 0-20V)

Bread board

Connecting wires

Theory:

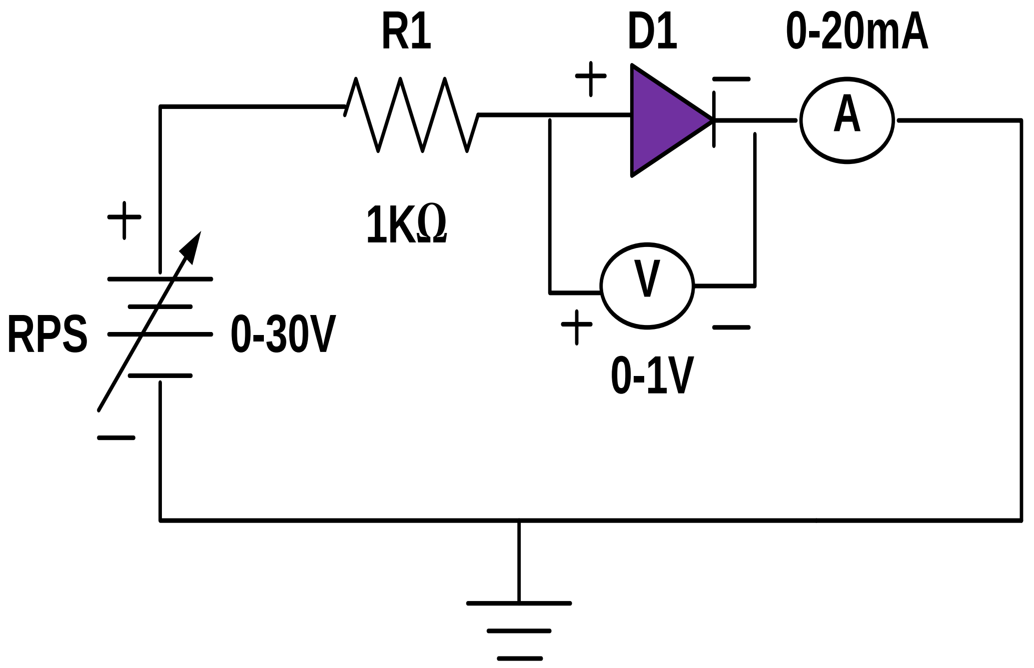
A p-n junction diode conducts only in one direction, the V-I characteristics of p-n diode is a curve plotted between the voltage across diode and current through the diode. When external voltage is zero, circuit is open and the potential barrier does not allow the current to flow. Therefore, the circuit current is zero. When P-type Anode is connected to +ve terminal and N- type Cathode is connected to the –ve terminal of supply voltage the diode is considered to be operating under forward bias condition. When the voltage across the diode is increased in the forward biased condition, the potential barrier is reduced and is altogether eliminated at some forward voltage, resulting in the current to flow through the diode and the circuit. The diode is now said to be in the ON state and the current keeps on increasing with an increase in the forward voltage.

A p-n junction diode is considered to be operating under reverse bias condition when the N-type cathode is connected to +ve terminal and P-type Anode is connected to the –ve terminal of supply voltage. Under reverse bias condition potential barrier across the junction increases with a corresponding rise in the supply voltage, hence junction resistance becomes very high and a very small reverse saturation current flows in the circuit due to minority charge carriers. The diode is now said to be in the OFF state.

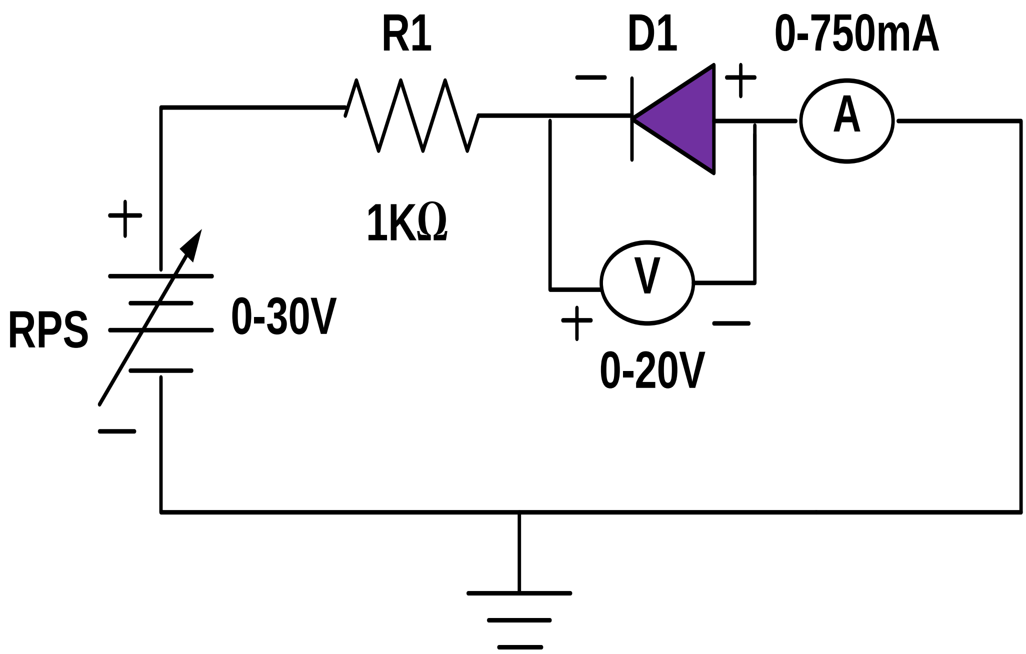
The V-I characteristics of a PN junction diode demonstrate the fundamental behavior of the diode under forward and reverse bias conditions. In forward bias, the diode allows current to flow after a threshold voltage, and in reverse bias, the current remains minimal until breakdown occurs. The characteristics curve is crucial in understanding how diodes function in electronic circuits, including rectifiers, signal clippers, and voltage regulators.

Circuit Diagram:

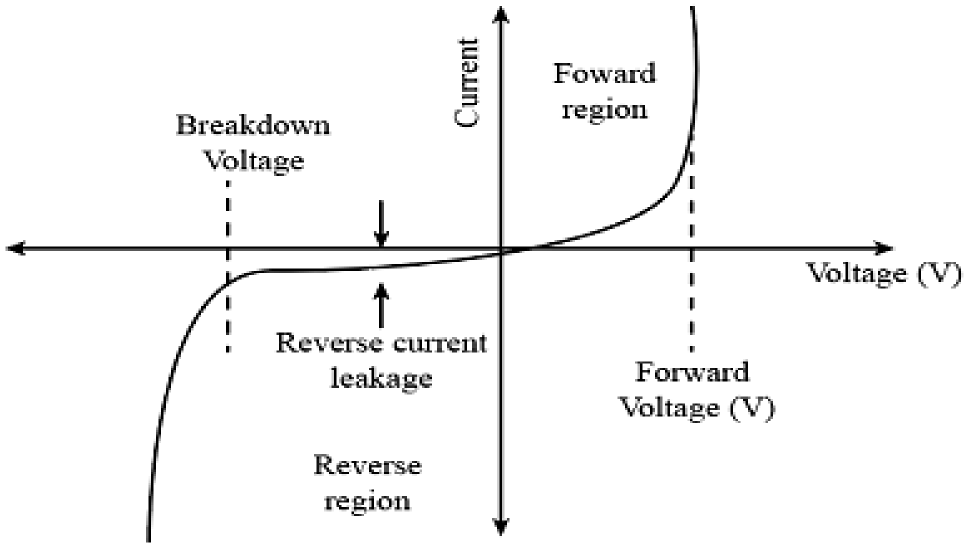
Forward Bias:



Reverse Bias:



Model Waveform:



Procedure:

Forward Bias: -

1. Connections are made as per the circuit diagram.
2. Under forward bias, the RPS +ve terminal is connected to the anode of diode and RPS –ve Terminal is connected to the cathode of the diode.
3. Switch on the power supply and increase the input voltage (supply voltage) in Steps.
4. Note down the corresponding current flowing through the diode and voltage across the Diode for each and every step of the input voltage.
5. Tabulate the readings of voltage and current.

6. Plot the graph between voltage and current.
7. Find the cut-in voltage in forward bias.
8. Calculate the static and dynamic resistances.

Observations: -

Sl. No	Applied Voltage (V)	Voltage across the Diode (V)	Current through the Diode (mA)
1			
2			
3			
4			
5			
6			
7			
8			
9			
10			
11			
12			

Reverse Bias:

1. Connections are made as per the circuit diagram.
2. Under reverse bias, the RPS -ve terminal is connected to the anode of diode and RPS +ve Terminal is connected to the cathode of the diode.
3. Switch on the power supply and increase the input voltage (supply voltage) in Steps.
4. Note down the corresponding current flowing through the diode and voltage across the Diode for each and every step of the input voltage.
5. Tabulate the readings of voltage and current.
6. Plot the graph between voltage and current.
7. Calculate the dynamic resistance.

Observations: -

Sl. No	Applied Voltage (V)	Voltage across the Diode (V)	Current through the Diode (mA)

1			
2			
3			
4			
5			
6			
7			
8			
9			
10			
11			
12			

Calculations:

Forward Bias:

Static resistance $R_d = V_d/I_d$

Dynamic resistance $r_d = \Delta V_d/ \Delta I_d$

Reverse Bias:

Dynamic resistance $r_d = \Delta V_d/ \Delta I_d$

Results: Forward and Reverse Bias characteristics of p-n diode are obtained.

Cut-in voltage =

Static Resistance in Forward Bias =

Dynamic resistance in Forward Bias =

Static Resistance in Reverse Bias =

Dynamic resistance in Reverse Bias =

Viva Questions:

1. Define depletion region of a diode?

2. What is meant by transition & space charge capacitance of a diode?

3. Is the V-I relationship of a diode Linear or Exponential?

4. Define cut-in voltage of a diode and specify the values for Si and Ge diodes?

5. What are the applications of a p-n diode?

6. Draw the ideal characteristics of P-N junction diode?

7. What is the diode equation?
8. What is PIV?

9. What is the break down voltage?

10. What is the effect of temperature on PN junction diodes?

11. How does the diode act as a switch?

12. Differentiate cut-in and cut-off voltages?

Outcome: Students are able to

EXPERIMENT NO: 8

Plot V – I characteristics of Zener Diode and its application as voltage Regulator

Aim: a) To observe and draw the V-I Characteristics of a Zener diode in Forward and Reverse bias conditions.

b) Zener Diode Application - Voltage Regulator.

Apparatus Required:

Zener diode- Z5.1

Regulated Power Supply - (0-30V). Voltmeter (0-1V, 0-20V)

Ammeter - (0-20mA) Resistor -1KΩ

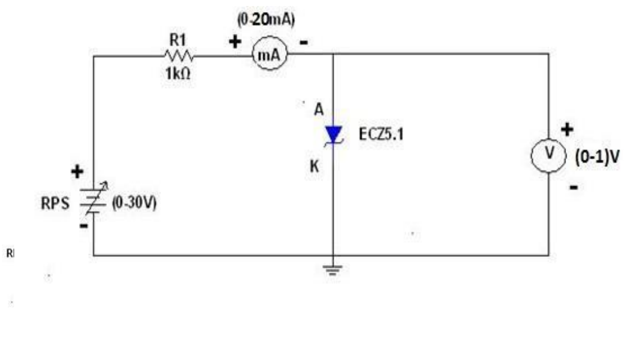
Bread Board Connecting wires

Theory:

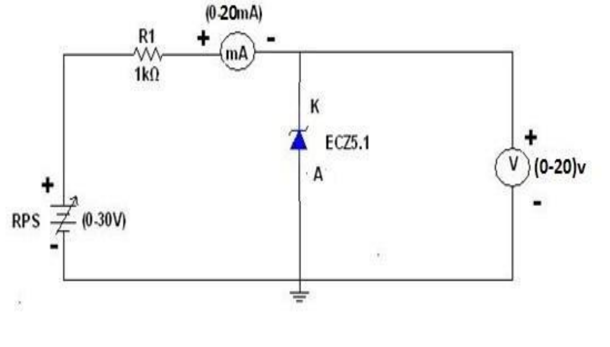
A Zener diode is a heavily doped p-n junction diode specially made to operate in the break down region. Under forward bias condition Zener diode operates similar to an ordinary p-n junction diode. A p-n junction diode normally does not conduct under reverse biased condition, whereas a Zener diode starts conducting heavily, when the reverse bias voltage across it is increased to a particular voltage called break down voltage (VZ). To avoid the passage of high current a resistor known as current limiting resistor is connected in series with Zener diode.

Once the Zener diode starts conducting it maintains constant voltage across the terminals irrespective of the amount of current passing through it, i.e., it has very low dynamic resistance. Hence Zener diode is mostly preferred for operation in voltage regulator circuits.

Circuit Diagram:



Forward Bias



Reverse Bias

Procedure:

Forward Bias:

1. Connections are made as per the circuit diagram.
2. Under forward bias, the RPS +ve terminal is connected to the anode of diode and RPS –ve Terminal is connected to the cathode of the diode.
3. Switch on the power supply and increase the input voltage (supply voltage) in Steps.
4. Note down the corresponding Zener current (lz), and the Zener voltage (Vz) across the diode for each and every step of the input voltage.
5. Tabulate the readings of voltage and current.
6. Plot the graph between Zener current (Iz) and Zener voltage (Vz).
7. Find the cut-in voltage in forward bias.
8. Now calculate the static and dynamic resistances.

Observations:

Forward Bias:

Sl. No	Applied Voltage (V)	Voltage across the Diode (V)	Current through the Diode (mA)
1			
2			
3			
4			
5			
6			
7			
8			
9			
10			
11			
12			

Reverse Bias:

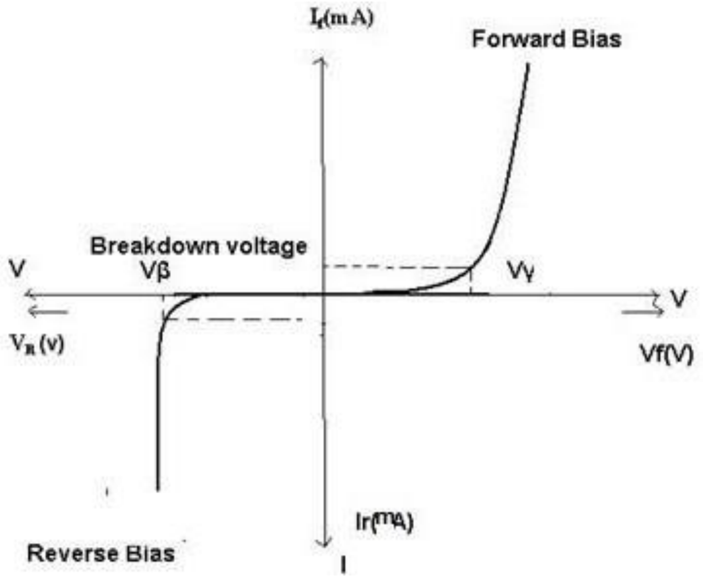
1. Connections are made as per the circuit diagram.
2. For reverse bias, the RPS +ve is connected to the cathode of the diode and RPS –ve is connected to the anode of the diode.
3. Switch on the power supply and increase the input voltage (supply voltage) in Steps.
4. Note down the corresponding current flowing through the diode and voltage across the diode for each and every step of the input voltage.
5. The readings of voltage and current are tabulated.
6. Graph is plotted between voltage and current.
7. Find the Zener breakdown voltage.
8. Now calculate the static and dynamic resistance.

Observations:

Reverse Bias:

Sl. No	Applied Voltage (V)	Voltage across the Diode (V)	Current through the Diode (mA)
1			
2			
3			
4			
5			
6			
7			
8			
9			
10			
11			
12			

Model Waveforms:



<u>Applications:</u>

- Shunt Regulator
- Meter protection
- Peak clipper
- Switching operation
- Controlled comparator
- Power supplies

Viva Questions:

1. What type of temperature Coefficient does the zener diode have?

2. If the impurity concentration is increased, how the depletion width effected?

3. Does the dynamic impendence of a zener diode vary?

4. Explain briefly about avalanche and zener breakdowns?

5. Draw the zener equivalent circuit?

6. In which region Zener diode can be used as a regulator?

EXPERIMENT NO: 9

IMPLEMENTATION OF HALF WAVE AND FULL WAVE RECTIFIERS

(A) HALF WAVE RECTIFIER

Aim: To Rectify the AC signal and then to find out Ripple factor and percentage of Regulation in Half wave and Full wave rectifiers with and without Capacitor filter.

Apparatus Required:

Bread Board
Multimeter
Transformer (9-0-9)
Diode 1N4007
Capacitor 100µf
Resistor 1KΩ.
CRO

Connecting wires

Theory:

A rectifier is an electronic circuit used to convert alternating current (AC) into direct current (DC). The half-wave rectifier is one of the simplest types of rectifiers that allows only one half-cycle of the AC input signal to pass through, blocking the other half. This results in a pulsating DC output.

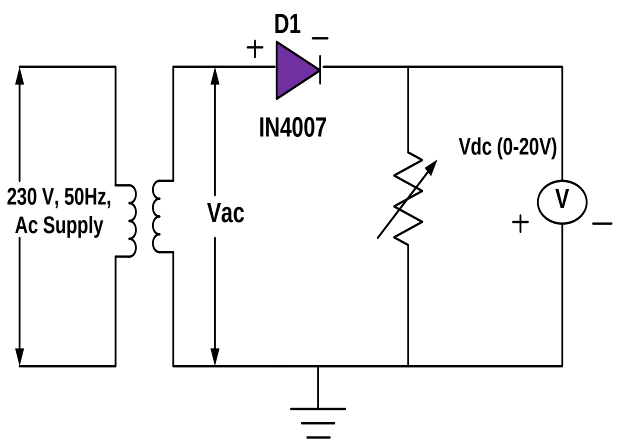
In a half-wave rectifier, the diode conducts during the positive half-cycle of the AC input and blocks the current during the negative half-cycle. As a result, the output consists of only the positive portion of the AC waveform.

The basic working principle of a half-wave rectifier is that the diode allows current to pass in one direction only (forward bias), and blocks current in the opposite direction (reverse bias). Here's the step-by-step operation:

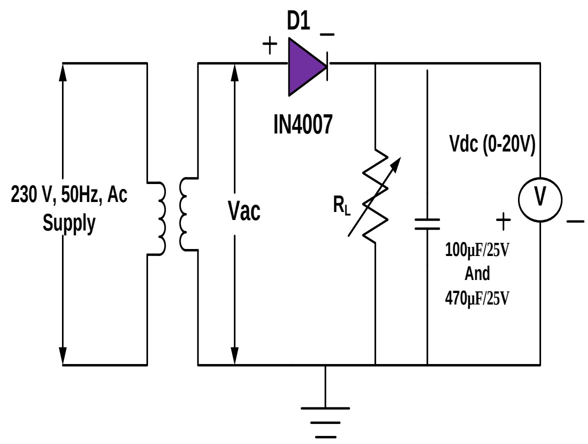
1. Positive Half-Cycle: During the positive half-cycle of the AC input (when the anode of the diode is positive relative to the cathode), the diode becomes forward-biased and conducts current. The output voltage follows the input AC voltage during this phase.
2. Negative Half-Cycle: During the negative half-cycle of the AC input (when the anode of the diode is negative relative to the cathode), the diode becomes reverse-biased and does not conduct any current. As a result, the output voltage is zero during the negative half-cycle.

The output waveform thus consists only of the positive half-cycles of the input AC signal, resulting in a pulsating DC voltage. The half-wave rectifier is a basic and simple circuit used to convert AC to DC. Although it provides a low average DC voltage and is inefficient, it is useful for simple applications where high efficiency is not critical. The half-wave rectifier is often used in low-power applications or as a preliminary step in more complex rectification circuits.

Circuit Diagram:



Half wave Rectifier without Filter



Half wave Rectifier with Filter

Procedure: -

1. Connections are made as per the circuit diagram.

2. Connect the primary of the transformer to ac mains and the secondary to the rectifier input.

3. Measure the ac input voltage of rectifier and dc voltage at the output of the rectifier using multimeter.

4. Find the theoretical value of dc voltage by using the formula,

$$Vdc=V_m/\pi$$

The Ripple factor is calculated by using the formula

r=ac output voltage (Vrms)/dc output voltage (Vdc)

<u>Regulation Characteristics:-</u>

1. Connections are made as per the circuit diagram.

2. By increasing the value of the DRB (Decade Resistance Box), the voltage across the load and current flowing through the load are measured.

3. The reading is tabulated.

4. Draw a graph between load voltage (V_L) and load current (I_L) taking V_L on X-axis and I_L on y-axis.

5. From the value of no-load voltages, the %regulation is calculated using the formula.

<u>Observations:</u>

Half wave rectifier without Filter:

V_{NL} = **5.2 V**

Sl.No.	I_{dc}(mA)	V_{dc}(V)	V_{ac}(V)	Ripple factor = r = V_{ac}/V_{dc}	%Reg = (V_{NL}- V_{FL})/ V_{FL}*100
1					
2					
3					
4					
5					
6					
7					

Half wave rectifier with Filter:

V_{NL} = **11.4 V**

Sl.No.	I_{dc}(mA)	V_{dc}(V)	V_{ac}(V)	Ripple factor = r = V_{ac}/V_{dc}	%Reg = (V_{NL}- V_{FL})/ V_{FL}*100
1					
2					
3					
4					
5					
6					
7					

Theoretical Calculations:

Without Filter:-

Vrms=Vm/2

Vdc=Vm/π

Ripple factor r=$\sqrt{(Vrms/ Vdc)^2 -1}$ =1.21

% Efficiency = P_{dc}/P_{ac} *100

$P_{dc} = V_{dc}^2/R_L$

$P_{ac} = V_{rms}^2/R_L$

With Filter:

Ripple factor r=1/ (2√3 f C R)

Where f =50Hz

C =100μF

R_L=1KΩ

Results: -

Viva Questions: -

1. What is the rectifier?
2. What is the PIV of Half wave rectifier?
3. What is the o/p frequency of Bridge Rectifier?
4. What is meant by a ripple?
5. What is the function of the filters?
6. What is TUF?
7. What is the average value of o/p voltage for HWR?
8. What is the peak factor?

Outcome: Students are able to

b) FULL-WAVE RECTIFIER

Aim: -To Rectify the AC signal and then to find out Ripple factor and percentage of Regulation in Half wave and Full wave rectifiers with and without Capacitor filter.

Apparatus Required: -

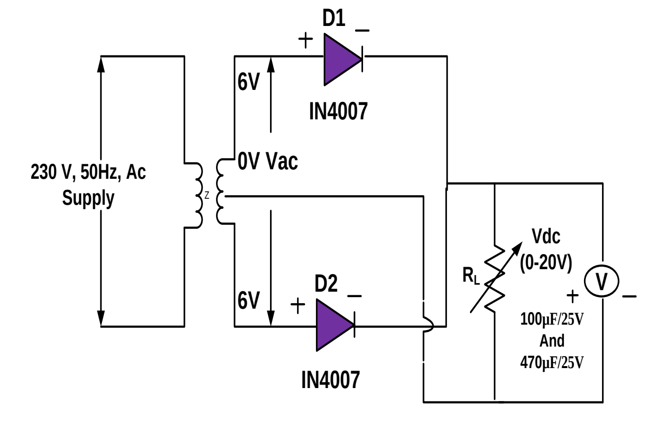
Bread Board
Transformer (9-0-9)

P-n Diodes, (lN4007) - 2 No's
Multimeter
Filter Capacitor (100µF/25v) – 1No
Connecting Wires
CRO
Load resistor -1KΩ

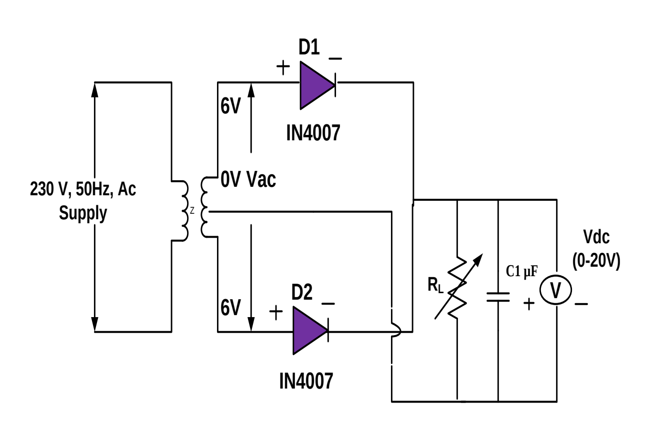
Theory:-

The circuit of a centre-tapped full wave rectifier uses two diodes D1&D2. During positive half cycle of secondary voltage (input voltage), the diode D1 is forward biased and D2 is reverse biased. The diode D1 conducts and current flows through load resistor R_L. During negative half cycle, diode D2 becomes forward biased and D1 reverse biased. Now, D2 conducts and current flows through the load resistor R_L in the same direction. There is a continuous current flow through the load resistor R_L, during both the half cycles making the direction of current unidirectional as show in the model graph.

The difference between full wave and half wave rectification is that a full wave rectifier allows unidirectional (one way) current to the load during the entire 360 degrees of the input signal and half-wave rectifier allows the current only during one half cycle (180 degree).



Full wave Rectifier without Filter



Full wave Rectifier with Filter

<u>Procedure:</u>

1. Connections are made as per the circuit diagram. e
2. Connect the primary of the transformer to ac mains and the secondary to the rectifier input.
3. Measure the ac input voltage of rectifier and dc voltage at the output of the rectifier using multimeter.
4. Find the theoretical value of dc voltage by using the formula,

$$V_{dc} = 2V_m/\pi$$

The Ripple factor is calculated by using the formula

r=ac output voltage (Vrms)/dc output voltage (Vdc).

Regulation Characteristics: -

1. Connections are made as per the circuit diagram.
2. By increasing the value of the rheostat, the voltage across the load and current flowing through the load are measured.
3. The reading is tabulated.
4. Draw a graph between load voltage (V_L) and load current (I_L) taking V_L on X-axis and I_L on y-axis.
5. From the value of no-load voltages, the %regulation is calculated using the formula.

Observations: -

Without Filter: -

V_{NL} = 15.04 V

Sno.	I_{dc}(mA)	V_{dc}(V)	V_{ac}(V)	Ripple factor r = V_{ac}/V_{dc}	%Reg = (V_{NL}- V_{FL})/ V_{FL}*100
1					
2					
3					
4					
5					
6					

With Filter: -

V_{NL} = 10.61 V

Sno.	I_{dc}(mA)	V_{dc}(V)	V_{ac}(V)	Ripple factor r = V_{ac}/V_{dc}	%Reg = (V_{NL}- V_{FL})/ V_{FL}*100
1					
2					
3					
4					
5					
6					

Theoretical Calculations: -

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

$$V_{dc}=2V_m/\pi$$

Without filter: -

Ripple factor, $r = \sqrt{(V_{rms}/V_{dc})^2 - 1}$

With filter: -

Ripple factor, $r = 1/(4\sqrt{3}\, f\, C\, R_L)$

Where f =50Hz

C =100µF

R_L=1KΩ

Practical Calculations: -

Without Filter:-

Vac=

Vdc=

Ripple factor, r=Vac/Vdc

With Filter:-

Vac=

Vdc=

Ripple factor, r=Vac/Vdc

Result:-

Viva Questions:-

1. Define regulation of the full wave rectifier?
2. Define peak inverse voltage (PIV)? And write its value for Full-wave rectifier?
3. If one of the diode is changed in its polarities what wave form would you get?
4. Does the process of rectification alter the frequency of the waveform?
5. What is ripple factor of the Full-wave rectifier?
6. What is the necessity of the transformer in the rectifier circuit?
7. What is meant by ripple and define Ripple factor?
8. Explain how capacitor helps to improve the ripple factor?

Outcome: Students are able to

EXPERIMENT NO: 10

Plot Input & Output characteristics of BJT in CE and CB configurations

(a) BJT- CE Configuration

Aim: To draw the input and output characteristics of BJT transistor connected in CE Configuration.

Apparatus Required:

Transistor (BC 107)
R.P.S (0-30V) - 2Nos
Voltmeters (0-20V) - 2Nos
Ammeters (0-20mA, 0-750µA)
Resistors 1KΩ
Bread board

Theory:

The BJT Common Emitter (CE) configuration is one of the most widely used amplifier configurations in analog electronics. In this configuration, the emitter terminal of the Bipolar Junction Transistor (BJT) is common to both the input and output. The CE configuration is known for providing both current gain and voltage gain, making it ideal for amplification purposes.

The CE amplifier is commonly used in applications like audio amplification, signal processing, and in general-purpose amplification.

In the Common Emitter (CE) configuration, the input signal is applied to the base of the transistor, while the collector provides the output signal. The emitter is common to both the input and output, which is why it is called the common-emitter configuration.

Here's a step-by-step breakdown of the working principle:

1. Base-Emitter Junction: The input signal is applied to the base of the transistor. The base-emitter junction of the transistor is forward biased, and the small input voltage causes a small current (called the base current, IBI_BIB) to flow through it.
2. Collector-Emitter Junction: The collector-emitter junction is reverse biased, and the larger current (collector current, I_C) flows through the collector resistor RCR_CRC. The collector current is proportional to the base current, as determined by the current gain (β\betaβ) of the transistor. Specifically:

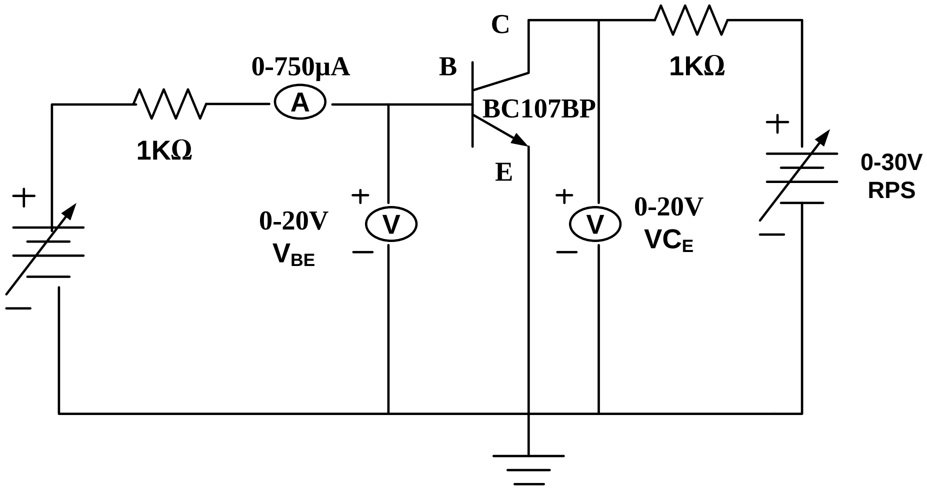
   $$I_C \approx \beta I_B$$

   Voltage Gain: The change in the collector current causes a voltage drop across the collector resistor RCR_CRC, which results in an amplified output voltage. This amplification leads to the voltage gain of the amplifier.
3. Phase Shift: A significant feature of the CE amplifier is that it produces an inverted

output, meaning the output signal is 180 degrees out of phase with the input signal.

Circuit Diagram:

Input Characteristics:



Procedure: -

Input Characteristics:

1. Connect the circuit as per the circuit diagram.
2. For plotting the input characteristics, the output voltage V_{CE} is kept constant and for different values of V_{BE}, note down the values of I_C.
3. Repeat the above step by keeping V_{CE} at 2V and 4V.
4. Tabulate all the readings.
5. Plot the graph between V_{BE} and I_B for constant VCE.

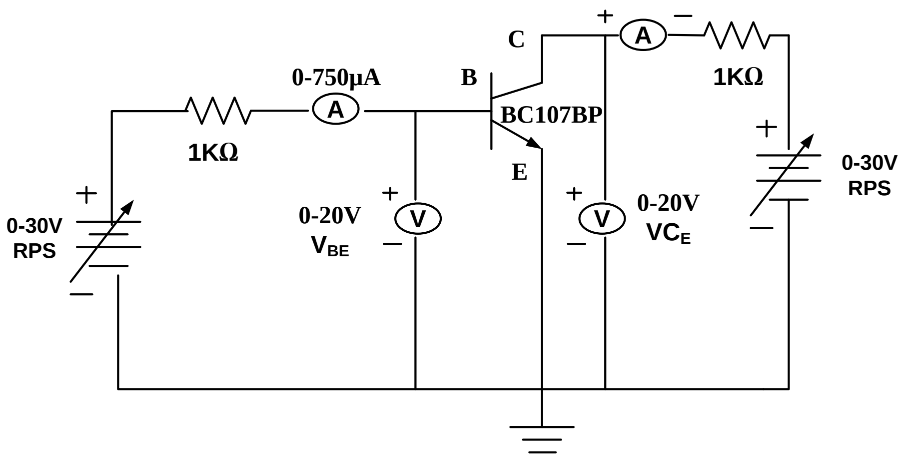
Observations:

Input Characteristics:

S.NO	V_{CE} = V		V_{CE} = V	
	V_{BE}(V)	I_B(μA)	V_{BE}(V)	I_B(μA)
1				
2				
3				
4				
5				
6				
7				

Circuit Diagram:

Output Characteristics:



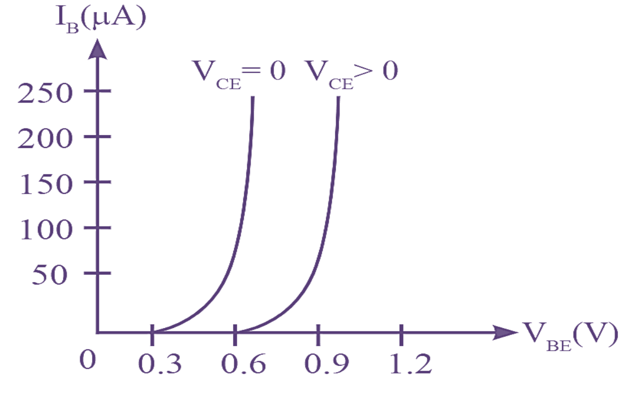
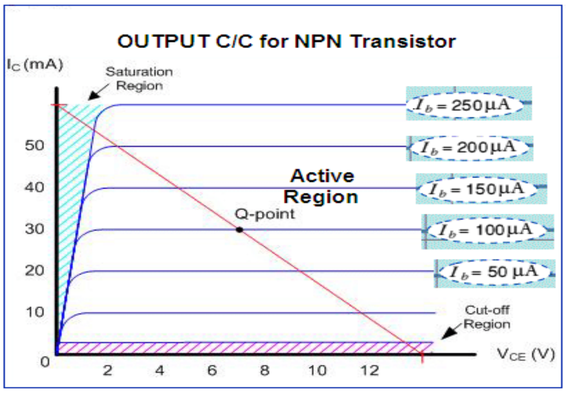
Output characteristics:

1. Connect the circuit as per the circuit diagram.
2. For plotting the output characteristics, the input current I_B is kept constant at 10µA and for different values of V_{CE}, note down the values of I_C.
3. Repeat the above step by keeping I_B at 75 µA and 100 µA.
4. Tabulate the all the readings.
5. Plot the graph between V_{CE} and I_C for constant I_B.

Output Characteristics:

S.NO	I_B = µA		I_B = µA	
	V_{CE}(V)	I_C(mA)	V_{CE}(V)	I_C(mA)
1				
2				
3				
4				
5				
6				
7				
8				
9				
10				

Model Graphs:

	
Input Characteristics	**Output Characteristics**

Applications: -

- Acts as a switch
- As an amplifier
- As an inverter
- In oscillators

Result: -The input and output characteristics of a transistor in CE configuration are plotted.

Viva Questions:

1. What are the input and output impedances of CE configuration?
2. Identify various regions in the output characteristics?
3. Define current gain in CE configuration?
4. Why CE configuration is preferred for amplification?
5. What is the phase relation between input and output?
6. Draw diagram of CE configuration for PNP transistor?
7. What is the power gain of CE configuration?

Outcome: by the completion of this experiment student can

1. Analyze the characteristics of BJT in Common Emitter Configuration.
2. Identify different operating regions of transistor.

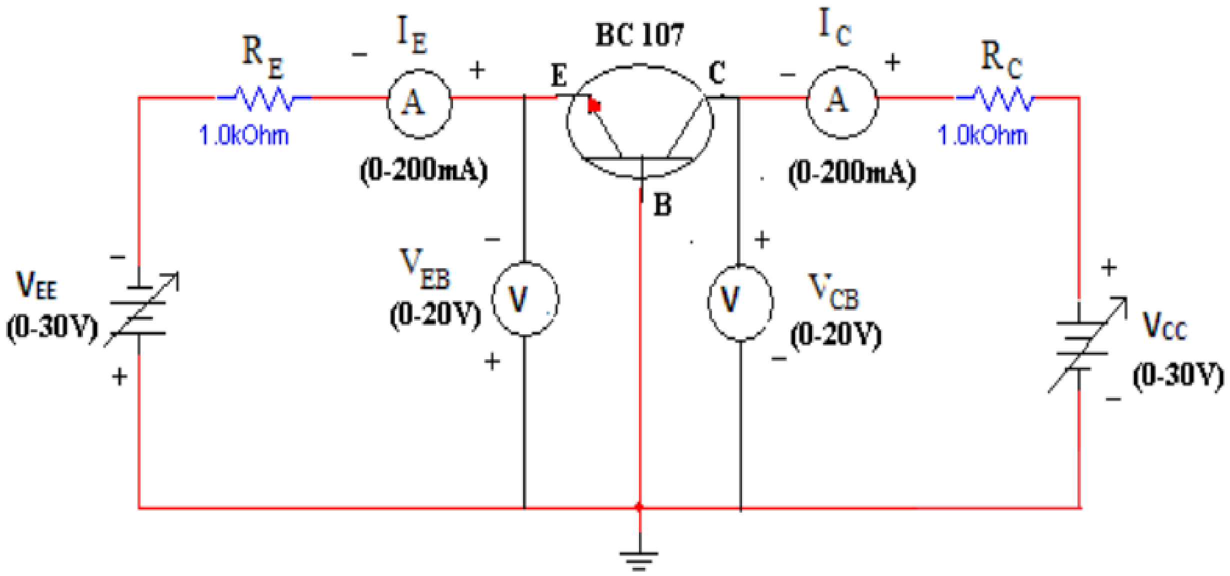
(b) BJT- CB Configuration

Aim: To draw the input and output characteristics of BJT transistor connected in CB Configuration.

Apparatus Required:

Transistor (BC 107)
R.P.S (0-30V) - 2Nos
Voltmeters (0-20V) - 2Nos
Ammeters (0-20mA, 0-750µA)
Resistors 1KΩ
Bread board

Circuit Diagram:



Procedure:

Input Characteristics:

1. Connect the circuit as in the circuit diagram.

2. Keep V_{EE} and V_{CC} in zero volts before giving the supply

3. Set V_{CB} = 1 volt by varying V_{CC} and vary the VEE smoothly with fine control such that emitter current I_E varies in steps of 0.2mA from zero upto 20mA, and note down the corresponding voltage V_{EB} for each step in the tabular form.

4. Repeat the experiment for V_{CB} =2 volts and 3 volts.

5. Draw a graph between V_{EB} Vs I_E against V_{CB} = Constant.

Output Characteristics:

1. Start VEE and VCC from zero Volts.

2. Set the I_E = 1mA by using V_{EE} such that, V_{CB} changes in steps of 1.0 volts from zero up to 20 volts, note down the corresponding collector current IC for each step in the tabular form.

3.Repeat the experiment for I_E = 3mA and I_E = 5mA, tabulate the readings.

4. Draw a graph between V_{CB} vs I_C against I_E = Constant.

Procedure: -

Input Characteristics:

1. Connect the circuit as in the circuit diagram.
2. Keep V_{EE} and V_{CC} in zero volts before giving the supply
3. Set V_{CB} = 1 volt by varying V_{CC}. and vary the V_{EE} smoothly with fine control such that emitter current I_E varies in steps of 0.2mA from zero upto 20mA, and note down the corresponding voltage V_{EB} for each step in the tabular form.
4. Repeat the experiment for V_{CB} =2 volts and 3 volts.
5. Draw a graph between V_{EB} Vs I_E against V_{CB} = Constant.

Observations:

Input Characteristics:

S.NO	V_{CB} = 1V		V_{CB}= 2V	
	V_{BE}(V)	I_E(mA)	V_{BE}(V)	I_E(mA)
1				
2				
3				
4				
5				
6				
7				
8				
9				
10				

Procedure: -

Output Characteristics:

1. Start VEE and VCC from zero Volts.
2. Set the IE = 1mA by using VEE such that, VCB changes in steps of 1.0 volts from zero upto 20 volts, note down the corresponding collector current IC for

each step in the tabular form.

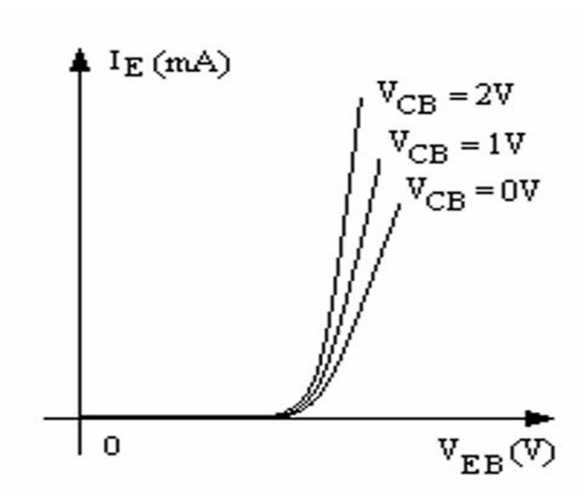
3. Repeat the experiment for IE = 3mA and IE = 5mA, tabulate the readings.

4. Draw a graph between VCB Vs IC against IE = Constant.

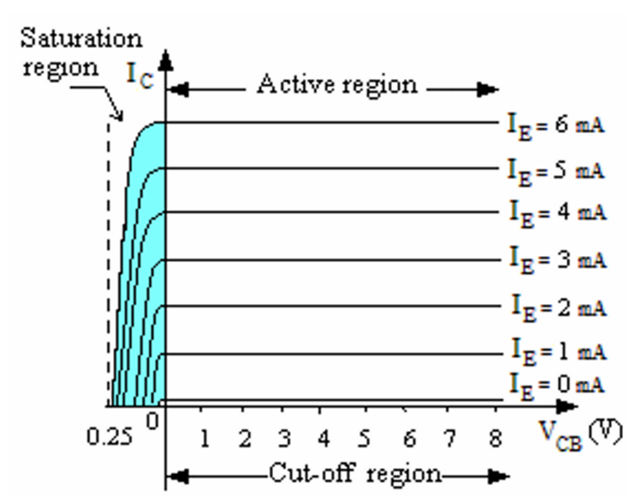
Observations:

Output Characteristics:

S.NO	I_E = 3mA		I_E= 5mA	
	V_{CB}(V)	I_C(mA)	V_{CB}(V)	I_C(mA)
1				
2				
3				
4				
5				
6				
7				
8				
9				
10				



Input Characteristics



Output Characteristics

Result: -The input and output characteristics of a transistor in CB configuration are plotted.

Outcome: By the completion of this experiment student can

1. Analyze the characteristics of BJT in Common Emitter Configuration.
2. Identify different operating regions of transistor.

EXPERIMENT 11

Verification of Truth Table of AND, OR, NOT, NAND, NOR, Ex-OR, Ex-NOR gates using ICs

Aim: Study and Verification of Truth Tables for Various 2 input logic gates like input (i) AND (ii) OR (iii) NOT (iv) NAND (v) NOR (vi) Ex-OR (vii) Ex-NOR using Digital Circuit Trainer board.

Apparatus Required:

S.NO	Components / Equipment's	Quantity
1	Digital Circuit Trainer Board	1
2	Patch Chords/ Connecting wires	1
3	IC 7432 - OR	1
4	IC 7408 - AND	1
5	IC 7402 - NOR	1
6	IC 7400 - NAND	1
7	IC 7404 - NOT	1
8	IC 7486 - Ex-OR	1
9	IC 74266 - Ex-NOR	1

Theory:

The basic logic gates are the building blocks of more complex logic circuits. These logic gates perform the basic Boolean functions, such as AND, OR, NAND, NOR, Inversion, Exclusive- OR, Exclusive-NOR. Fig. below shows the circuit symbol, Boolean function, and truth. It is seen from the Fig that each gate has one or two binary inputs, A and B, and one binary output,

C. The small circle on the output of the circuit symbols designates the logic complement. The AND, OR, NAND, and NOR gates can be extended to have more than two inputs. A gate can be extended to have multiple inputs if the binary operation it represents is commutative and associative.

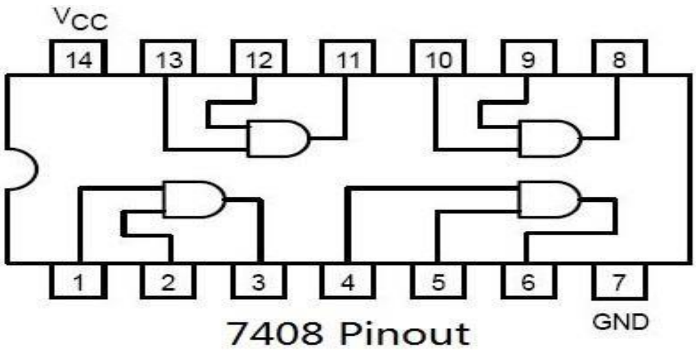
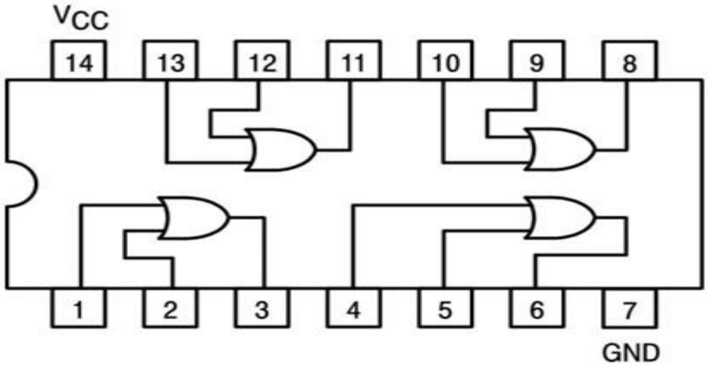
These basic logic gates are implemented as small-scale integrated circuits (SSICs) or as part of more complex medium scale (MSI) or very large-scale (VLSI) integrated circuits.

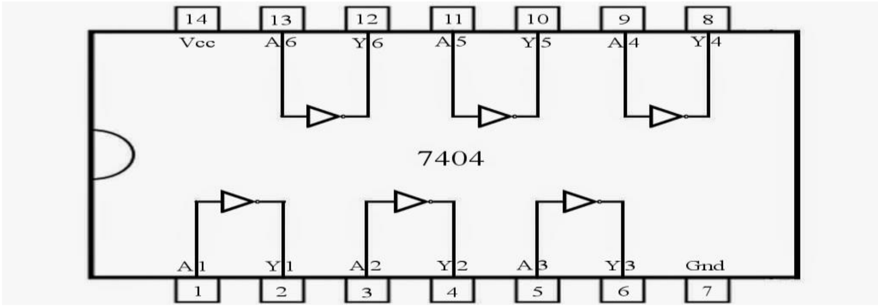
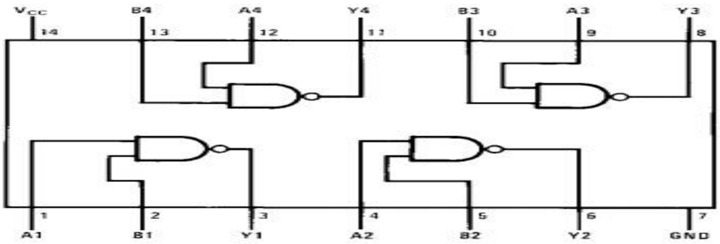
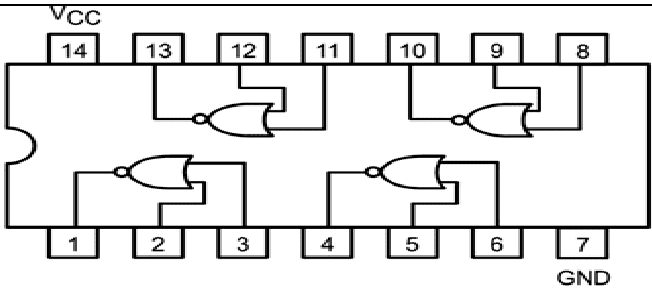
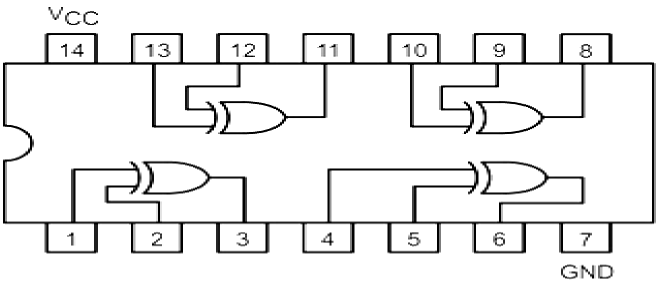
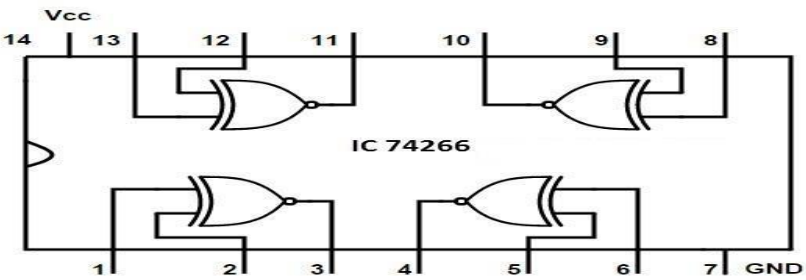
Digital IC gates are classified not only by their logic operation, but also the specific logic-circuit family to which they belong. Each logic family has its own basic electronic circuit upon which more complex digital circuits and functions are developed. The following logic families are the most frequently used.

TTL-Transistor-Transistor Logic ECL -Emitter-coupled logic MOS -Metal-oxide semiconductor

CMOS -Complementary metal-oxide semiconductor

TTL and ECL are based upon bipolar transistors. TTL has a well-established popularity among logic families. ECL is used only in systems requiring high-speed operation. MOS and CMOS, are based on field effect transistors. They are widely used in large scale integrated circuits because of their high component density and relatively low power consumption. CMOS logic consumes far less power than MOS logic. There are various commercial integrated circuit chips available. TTL ICs are usually distinguished by numerical designation as the 5400 and 7400 series.

AND GATE	
OR GATE	

NOT GATE	
NAND GATE	
NOR GATE	
XOR GATE	
XNOR GATE	

Procedure:

1. Place the breadboard gently on the table.
2. Fix the IC which is under observation between the half shadow line of breadboard, so there is no shortage of voltage.
3. Connect the wire to the main voltage source (V_{cc}) whose other end is connected to last pin of the IC (14 place from the notch).
4. Connect the ground of IC (7^{th} place from the notch) to the ground terminal provided on the digital lab kit.
5. Give the input at any one of the gate of the ICs i.e. 1^{st}, 2^{nd}, 4^{th}, 5^{th} …. gate by using connecting wires. (In accordance to IC provided).
6. Connect output pins to the led from 3^{rd}, 6^{th} …. From IC on digital lab kit.
7. Switch on the power supply.
8. If LED glows red then output is true, if it glows green output is false, which is numerically denoted as 1 and 0 respectively. The Colour can change based on the IC manufacturer it's just verification of the Truth Table not the colour change.

Precautions:

1. All connections should be made neat and tight.
2. Digital lab kits and ICs should be handled with utmost care.
3. While making connections main voltage should be kept switched off.
4. Never touch live and open wires.

Observations:

INPUTS		**OUTPUTS**					
A	**B**	**AND**	**OR**	**NAND**	**NOR**	**Ex-OR**	**Ex-NOR**
0	0						
0	1						
1	0						
1	1						

Result: Study and Verification of Truth Tables for Various 2 input logic gates like input (i) AND (ii) OR (iii) NOT (iv) NAND (v) NOR (vi) Ex-OR (vii) Ex-NOR using Digital Circuit

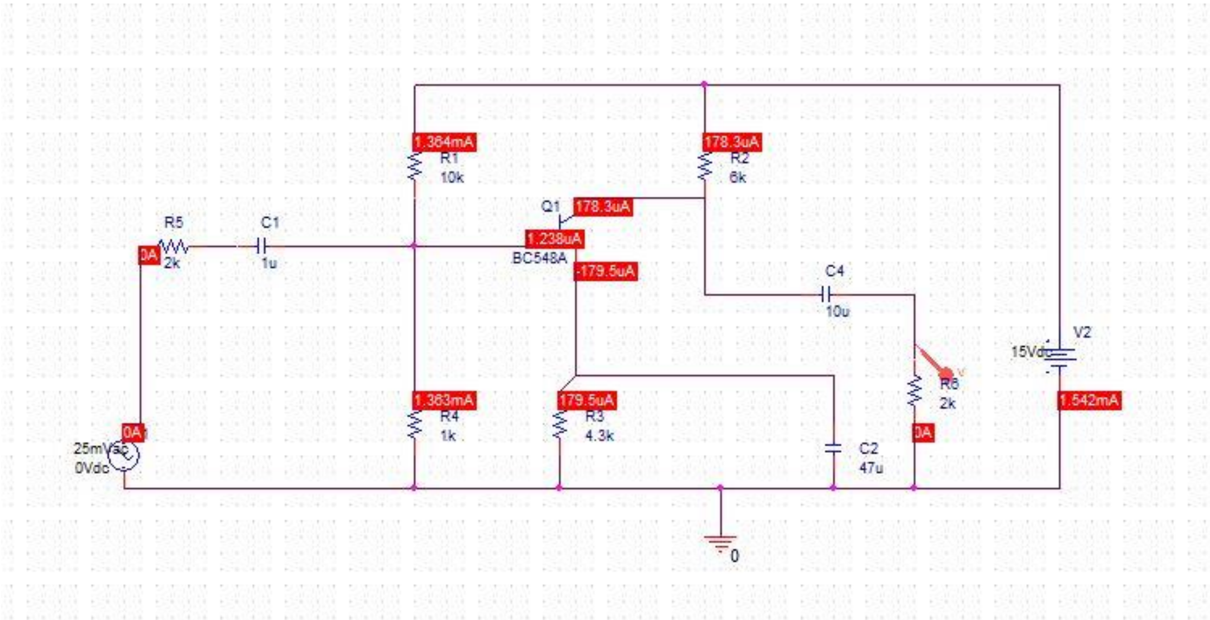
Trainer board was done.

EXPERIMENT 12
FREQUENCY RESPONSE OF CE AMPLIFIER

Aim : To simulate and observe the frequency response of common emitter amplifier using PSpice- Orcad software.

Software Tool: PC with Orcad 16.0 Pspice software.

Circuit Diagram:



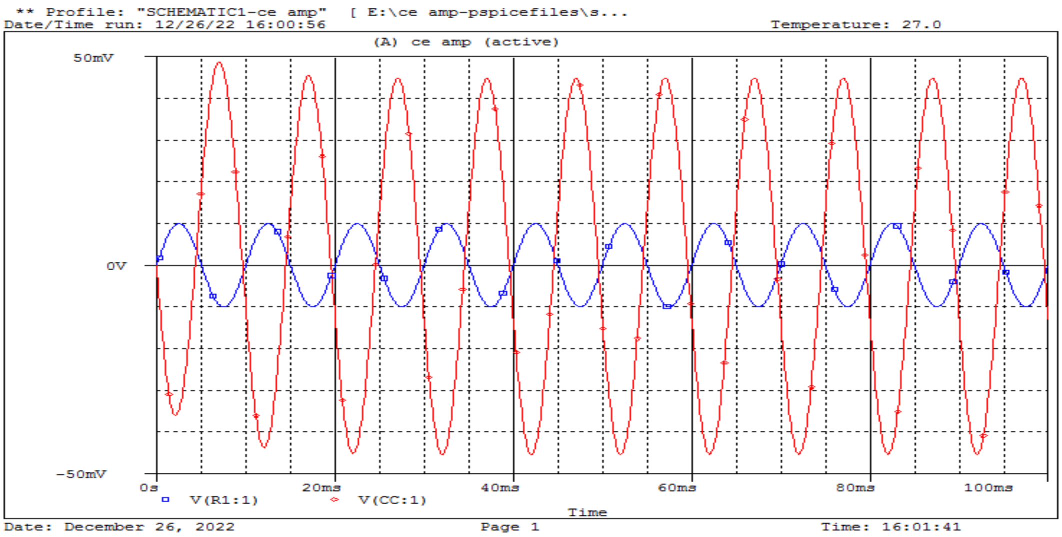
Procedure:

1. Click on File to create new project and assign a name.

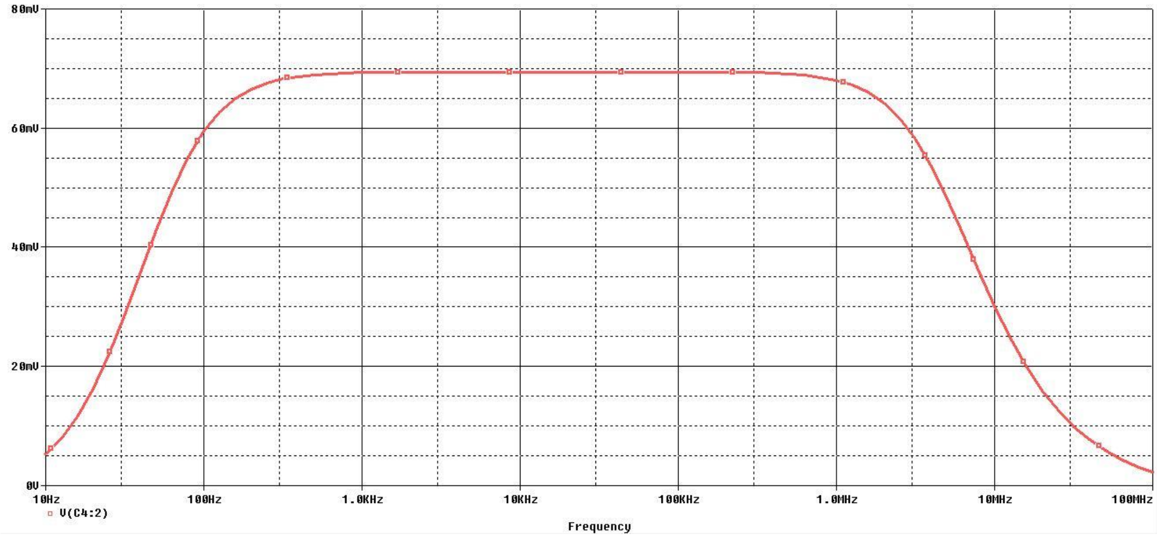
2. Click on “Create Blank Project”.
3. Click on “Place” to get components to create the diagram.
4. Click on “Pspice” and then click on “New Simulation File”.
5. Click on “Create” so that all other components will be activated.
6. The “Analysis setup” process has to be done before performing the simulation.
7. For “Analysis setup”, click on “Pspice” and then click on “Edit Simulation Profile” to get simulation settings block.
8. Click on “RUN” button to perform the simulation.
9. Click on “Trace” and then click on “Add Trace”.
10. Click on the generated output waveform and click trace properties.
11. Perform a DC Sweep Analysis by varying the input voltage (Vin_{in}in) over a suitable range (e.g., 0V to 5V) with a step increment of 0.2V, to observe how the transistor responds in active mode.

Result:

Output Waveforms:



Voltage Waveform



Frequency Response of CE Amplifier

<u>Viva Questions</u>:

1. What is phase difference between input and output waveforms of CE amplifier?
2. What type of biasing is used in the given circuit?
3. If the given transistor is replaced by a p-n-p, can we get output or not?
4. What is effect of emitter-bypass capacitor on frequency response?
5. What is the effect of coupling capacitor?
6. What is region of the transistor so that it is operated as an amplifier?
7. How does transistor acts as an amplifier?
8. Draw the h-parameter model of CE amplifier?
9. What type of transistor configuration is used in intermediate stages of a multistage amplifier?
10. What is early effect?

Outcome: By the completion of this experiment student can

EXPERIMENT 1

MEASUREMENT OF RESISTANCE USING WHEAT STONE BRIDGE

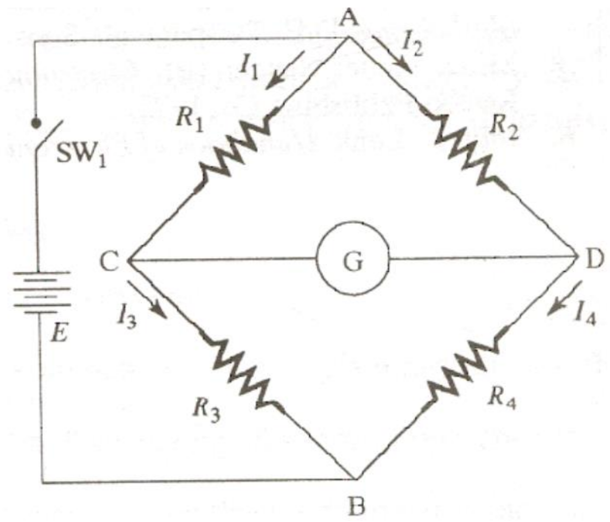
Aim: To study and perform an experiment to measure the unknown Resistance by Wheatstone's bridge.

Apparatus Required:

Light Spot DC Galvanometer, Various Medium Resistors, Multimeter, Portable Wheatstone bridge.

Theory:

A bridge circuit in its simplest form consists of network of four resistance arms forming a closed circuit, with a dc source of current applied to two opposite junctions and a current detector connected to the other two junctions. Wheatstone's bridge is used for accurate measurement of resistance.



When SW1 is closed, current flows and divides into the two arms at point A, i.e. I_1 and I_2. The bridge is balanced when there is no current through the galvanometer, or when the potential difference at points C and D is equal, i.e. the potential across the galvanometer is zero.
To obtain the bridge balance equation,

$$I_1R_1 = I_2R_2 \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots(1)$$

For the galvanometer current to be ze ro, the following conditions should be satisfied.

$$I_1 = I_3 = E/(R_1 + R_3)$$

$$I_2 = I_4 = E/(R_2 + R_4)$$

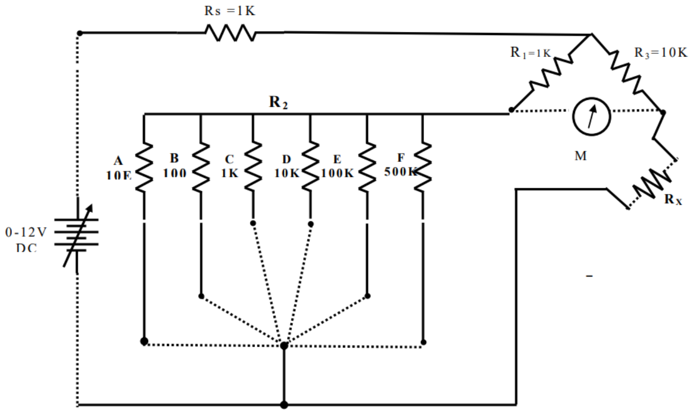
Substitute in equation 1,

$$(E*R_1) / (R_1 + R_3) = (E*R_2) /(R_2 + R_4)$$

$$= R_4 = (R_2*R_3) / R_1$$

This is the equation for bridge to be balanced.

Circuit Diagram:



Procedure:

1. Take The Trainer kit. Measure resistors A, B, C, D, E, F, R1 and the variable pot R_3 by adjusting "ADJ R_3". Note down the values of each resistor.

2. Now insert its mains cord in mains 230 V supply plug and switch it 'ON‟. Measure the DC supply volt age. (It should be 12V DC).

3. Select the unknown resistor and measure its resistance Rx and note it down.

4. Connect the resistor to the terminal (Rx), and connect the power supply into the circuit. Connect the galvanometer to M of the bridge with the help of jumper.

5. Connect the S1 terminal to any resistor A, B, C, D, E, F. Adjust pot „R_3 ‟ to get a null reading on the galvanometer.

6. Once the „Null, reading is found, remove all the jumpers and measure the value of R_3. Put the value of R_3 in the formula given be low and calculate Rx practically.

$$\mathbf{R_X = (R_2 * R_3) / R_1 \text{ .... } (R_2 = A \text{ or } B \text{ or } C \text{.... or } F)}$$

7. Match the practical „Rx‟ with that of the Rx directly measured on multimeter.

8. Take four to five reading to find the unknown resistance i.e. Rx with different resistors.

Observation Table:

R_2	R_3	R_1	R_X

Result:

Outcomes:

EXPERIMENT NO 2

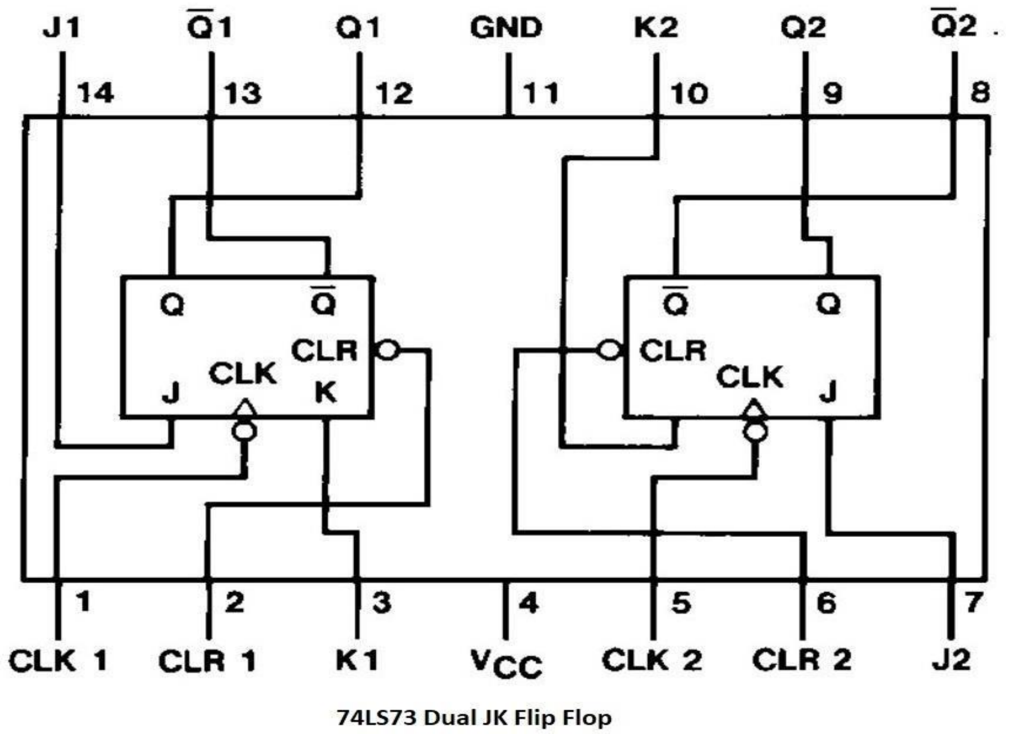
Verification of Truth Tables of S-R, J-K& D flip flops using respective ICs

Aim: To Design and verification of Truth tables of SR, JK and D Flip –Flop using Digital Trainer kit.

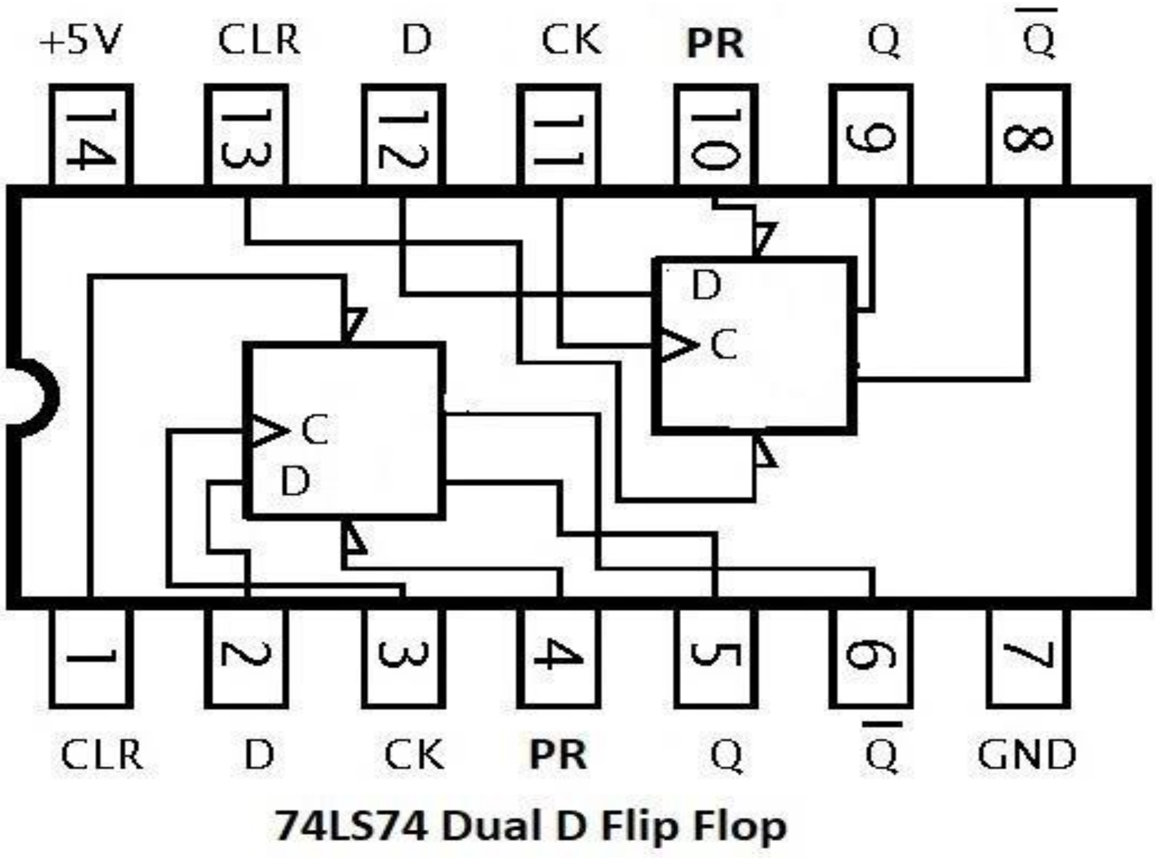
Apparatus Required:

S.NO	Name of the Apparatus	Quantity
1	Digital Circuit Trainer Board	1
2	Patch Chords/ Connecting wires	1
3	IC 7473- JK FF	2
4	IC 7474 D FF	1

Circuit Diagram:



IC7473 JK Flip-flop connection diagram



IC7474 D Flip-flop connection diagram

Procedure:

- Made Connection as per the circuit diagram given above.
- Connect the particular input pins to the logic input section using a connecting wire.
- Similarly connect the output pin to the logic output section of the trainer kit.
- Verify the functionality of JK Flip flop and D Flip flop.
- Write the truth-table for each.

Precautions:

- All connections should be made neat and tight.
- Digital lab kits and ICs should be handled with utmost care.
- While making connections main voltage should be kept switched off.
- Never touch live and open wires.

<u>Observations:</u>

SR Flip Flop:

Clock	S	R	Q_{n+1}

JK Flip Flop:

Clock	S	R	Q_{n+1}

D Flip Flop:

Input	Output

Result:

Outcome:

EXPERIMENT 3

Simulation of RC Coupled Amplifier

Aim: To simulate RC coupled amplifier using PSpice.

Apparatus Required:

PC with PSpice-orcad Software

Theory:

The RC coupled amplifier is a type of amplifier that uses resistors (R) and capacitors (C) for coupling and biasing. It's commonly used in analog signal amplification in radio-frequency circuits, audio amplifiers, and other electronic systems.

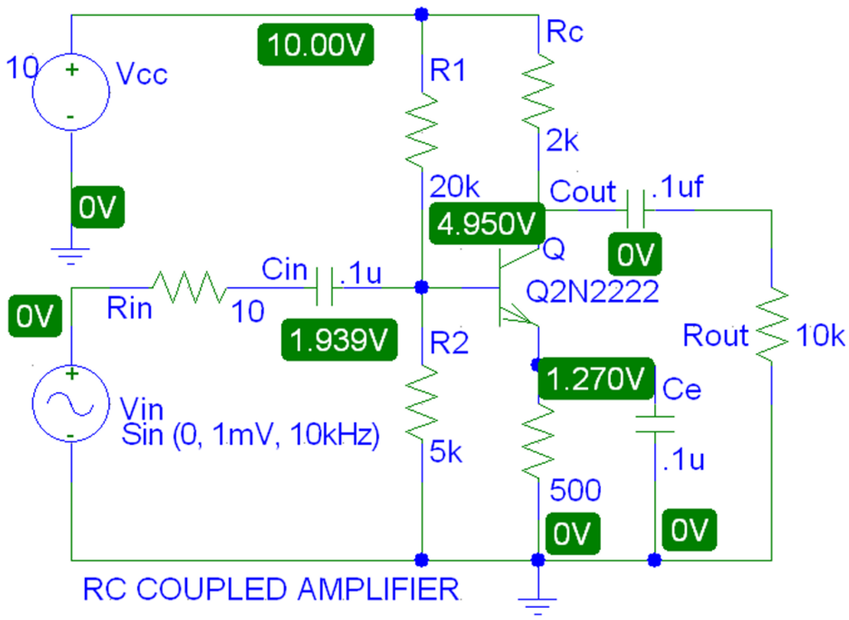
An RC coupled amplifier typically consists of the following components:

1. Transistor: A common-emitter configuration (although other configurations such as common-collector or common-base may also be used).
2. Resistors: To provide biasing for the transistor and load resistance.
3. Capacitors: Used for signal coupling between stages of amplification (to block DC components and only pass the AC signal) and to bypass certain frequencies.

The basic configuration for an RC coupled amplifier can be broken down into three main parts:

- Input stage: The signal is fed into the base of the transistor.
- Amplification stage: The transistor amplifies the input signal.
- Output stage: The amplified signal is taken from the collector of the transistor and is passed to the next stage via an output coupling capacitor.

Circuit Diagram:



Procedure:

1. Open PSpice and Create a New Project
2. Add the Components
3. Build the Circuit
4. Choose the Type of Simulation: For this type of amplifier, you can run a Transient Analysis to see how the amplifier responds to a time-varying input signal. Alternatively, you can perform AC Sweep analysis to determine the frequency response of the amplifier.
5. Set the frequency range from 1Hz to several MHz, depending on the expected bandwidth of your amplifier. Logarithmic sweep is usually recommended for AC analysis.
6. Run the Simulation by clicking on the Run button in the toolbar.

Result:

Outcomes: