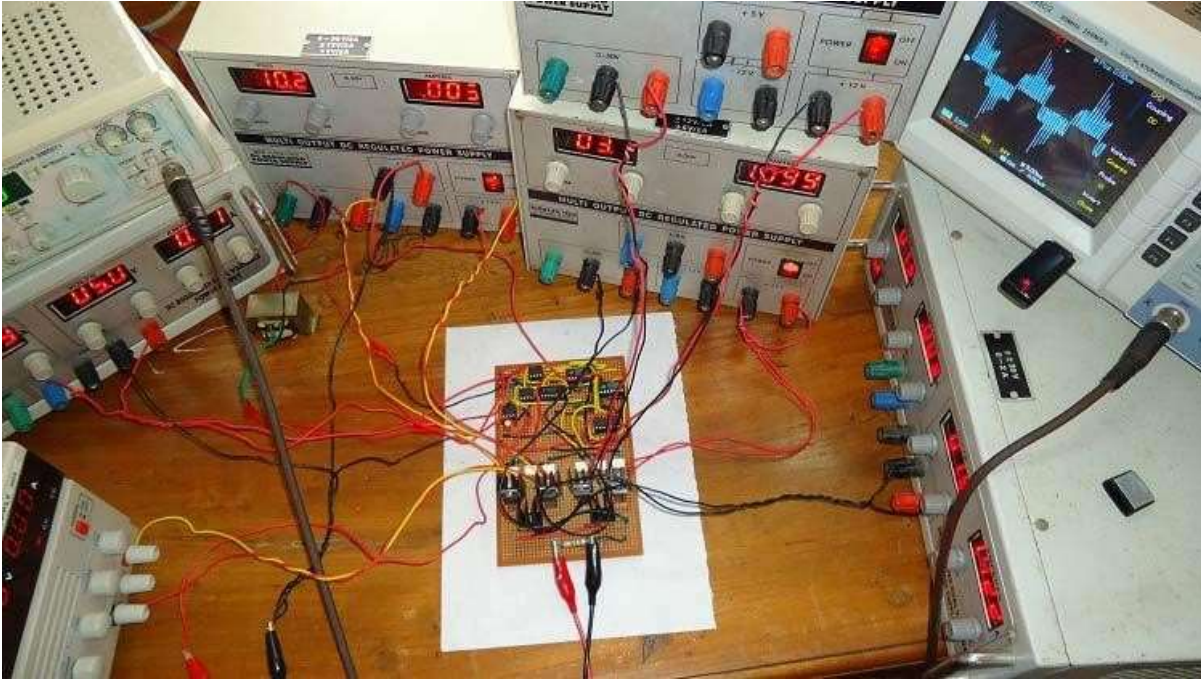


POWER ELECTRONICS & SIMULATION LABORATORY

II B.Tech EEE-II Sem



**DEPARTMENT
OF
ELECTRICAL AND ELECTRONICS ENGINEERING**

VEMU INSTITUTE OF TECHNOLOGY::P.KOTHAKOTA

NEAR PAKALA, CHITTOOR-517112

(Approved by AICTE, New Delhi & Affiliated to JNTUA, Anantapuramu
Accredited by NAAC, NBA (EEE, ECE & CSE), Recognized under 2(f) &12(B) of UGC Act)

VEMU INSTITUTE OF TECHNOLOGY

DEPT.OF ELECTRICAL AND ELECTRONICS ENGINEERING

VISION OF THE INSTITUTE

- ✦ To be a premier institute for professional education producing dynamic and vibrant force of technocrats with competent skills, innovative ideas and leadership qualities to serve the society with ethical and benevolent approach.

MISSION OF THE INSTITUTE

- ✦ To create a learning environment with state-of-the art infrastructure, well equipped laboratories, research facilities and qualified senior faculty to impart high quality technical education.
- ✦ To facilitate the learners to foster innovative ideas, inculcate competent research and consultancy skills through Industry-Institute Interaction.
- ✦ To develop hard work, honesty, leadership qualities and sense of direction in rural youth by providing value based education.

VISION OF THE DEPARTMENT

- ✦ To produce professionally deft and intellectually adept Electrical and Electronics Engineers and equip them with the latest technological skills, research & consultancy competencies along with social responsibility, ethics, Lifelong Learning and leadership qualities.

MISSION OF THE DEPARTMENT

- ✦ To produce competent Electrical and Electronics Engineers with strong core knowledge, design experience & exposure to research by providing quality teaching and learning environment.
- ✦ To train the students in emerging technologies through state - of - the art laboratories and thus bridge the gap between Industry and academia.
- ✦ To inculcate learners with interpersonal skills, team work, social values, leadership qualities and professional ethics for a holistic engineering professional practice through value based education.

PROGRAM EDUCATIONAL OBJECTIVES(PEOs)

Programme Educational Objectives (PEOs) of B.Tech (Electrical and Electronics Engineering) program are:

Within few years of graduation, the graduates will

PEO 1: Provide sound foundation in mathematics, science and engineering fundamentals to analyze, formulate and solve complex engineering problems.

PEO 2: Have multi-disciplinary Knowledge and innovative skills to design and develop Electrical & Electronics products and allied systems.

PEO 3: Acquire the latest technological skills and motivation to pursue higher studies leading to research.

PEO 4: Possess good communication skills, team spirit, ethics, modern tools usage and the life-long learning needed for a successful professional career.

PROGRAM OUTCOMES (POs)

PO-1	Engineering knowledge: Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
PO-2	Problem analysis: Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
PO-3	Design/development of solutions: Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
PO-4	Conduct investigations of complex problems: Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
PO-5	Modern tool usage: Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
PO-6	The engineer and society: Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
PO-7	Environment and sustainability: Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
PO-8	Ethics: Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
PO-9	Individual and team work: Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.

PO-10	Communication: Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
PO-11	Project management and finance: Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.
PO-12	Life-long learning: Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

PROGRAM SPECIFIC OUTCOMES (PSOs)

On completion of the B.Tech. (Electrical and Electronics Engineering) degree, the graduates will be able to

PSO-1:Higher Education: Apply the fundamental knowledge of Mathematics, Science, Electrical and Electronics Engineering to pursue higher education in the areas of Electrical Circuits, Electrical Machines, Electrical Drives, Power Electronics, Control Systems and Power Systems.

PSO-2:Employment: Get employed in Public/Private sectors by applying the knowledge in the domains of design and operation of Electronic Systems, Microprocessor based control systems, Power systems, Energy auditing etc.

POWER ELECTRONICS LABORATORY(20A02401P)

COURSE OUTCOMES

C227.1	Test the turn on – turn off characteristics of various power electronic devices.
C227.2	Test different types of voltage controllers ,converters and inverters with R and RL loads
C227.3	Test and analyze firing circuit of SCR
C227.4	Analyze the TPS7A4901, TPS7A8300 And TPS54160 buck regulators
C227.5	Analyze the performance characteristics of a single phase ac voltage controller, DC jones chopper and series inverter using MATLAB



VEMU INSTITUTE OF TECHNOLOGY::P.KOTHAKOTA

NEAR PAKALA, CHITTOOR-517112

(Approved by AICTE, New Delhi & Affiliated to JNTUA, Anantapuramu)

List of Experiments to be conducted

(19A02506) POWER ELECTRONICS & SIMULATION LABORATORY

1. Study of Characteristics of SCR, MOSFET & IGBT
2. Gate firing circuits for SCR's: (a) R triggering (b) R-C triggering
3. Forced Commutation circuits (Class A, Class B, Class C, Class D & Class E)
4. DC Jones chopper with R and RL Loads
5. Single Phase Parallel, inverter with R and RL loads
6. Single Phase Cycloconverter with R and RL loads
7. Single Phase series inverter with R and RL loads
8. Single Phase dual converter with RL loads

Simulation experiments with PSPICE/PSIM

9. PSPICE simulation of single-phase full converter using RLE loads and single-phase AC voltage controller using RLE loads.
10. PSPICE simulation of resonant pulse commutation circuit and Buck converters and chopper.
11. PSPICE simulation of single phase Inverter with PWM control.

Additional Experiments:

12. Study of dc to dc buck converter TPS54160
13. PSPICE Simulation of Buck Chopper

GENERAL INSTRUCTIONS FOR LABORATORY CLASSES

DO'S

1. Without Prior permission do not enter into the Laboratory.
2. While entering into the LAB students should wear their ID cards.
3. The Students should come with proper uniform.
4. Students should sign in the LOGIN REGISTER before entering into the laboratory.
5. Students should come with observation and record note book to the laboratory.
6. Students should maintain silence inside the laboratory.
7. Circuit connections must be checked by the lab-in charge before switching the supply

DONT'S

8. Students bringing the bags inside the laboratory..
9. Students wearing slippers/shoes insides the laboratory.
10. Students scribbling on the desk and mishandling the chairs.
11. Students using mobile phones inside the laboratory.
12. Students making noise inside the laboratory.
13. Students mishandle the devices.
14. Students write anything on the devices

EXPERIMENT N0: 01

STUDY OF CHARACTERISTICS OF SCR, MOSFET & IGBT

AIM:

To plot the V - I Characteristics of SCR, MOSFET & IGBT.

SCR CHARACTERISTICS

APPARATUS:

S. No	Equipment	Range	Type	Quantity
1	SCR characteristics Trainer			
2	Patch chords			
3	DC Voltmeter			
4	DC Ammeter			

CIRCUITDIAGRAM:

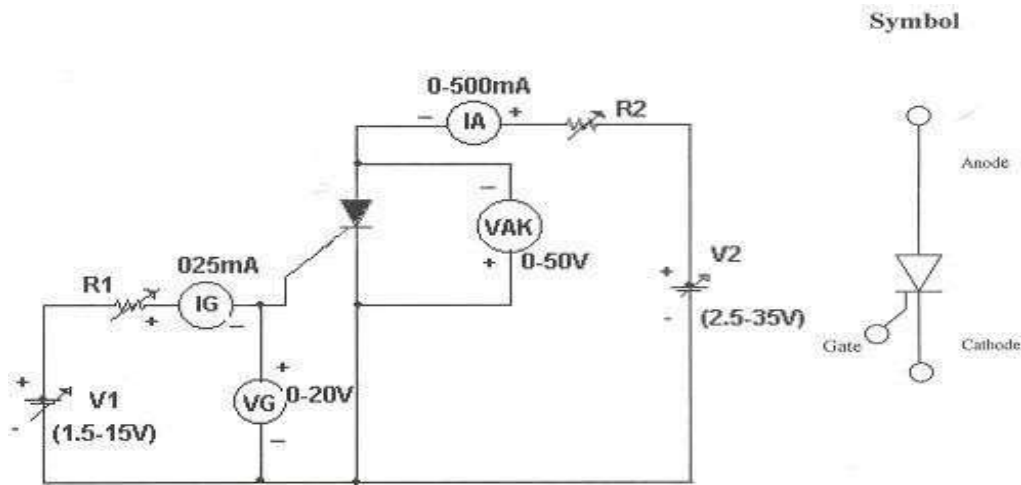


Fig – 1.1 Study of Characteristics of SCR

PROCEDURE:

V - I CHARACTERISTICS:

1. Make all connections as per the circuit diagram.
2. Initially keep V_1 & V_2 at minimum position and R_1 & R_2 maximum position.
3. Adjust Gate current I_g to some constant by varying the V_1 or R_1 .
4. Now slowly vary V_2 and observe Anode to Cathode voltage V_{AK} and Anode current I_A .
5. Tabulate the readings of Anode to Cathode voltage V_{AK} and Anode current I_A .
6. Repeat the above procedure for different Gate current I_g .

GATE TRIGGERING AND FINDING V_G AND I_G :-

1. Keep all positions at minimum.
2. Set Anode to Cathode voltage V_{AK} to some volts say 15V.
3. Now slowly vary the V_1 voltage till the SCR triggers and note down the reading of gate current (I_G) and Gate Cathode voltage (V_{GK}) and rise of anode current I_A .
4. Repeat the same for different Anode to Cathode voltage and find V_{AK} and I_G values.

TO FIND LATCHING CURRENT:

1. Keep R_2 at middle position.
2. Apply 20V to the Anode to cathode by varying V_2 .
3. Rise the V_g voltage by varying V_1 till the device turns ON indicated by sudden rise in I_A . At what current SCR trigger it is the minimum gate current required to turn ON the SCR.
4. Now set R_2 at maximum position, then SCR turns OFF, if it is not turned off reduce V_2 up to turn off the device and put the gate voltage.
5. Now decrease the R_2 slowly, to increase the Anode current gradually in steps.
6. At each and every step, put OFF and ON the gate voltage switches V_1 . If the Anode current is greater than the latching current of the device, the device stays ON even after switch OFF S_1 , otherwise device goes to blocking mode as soon as the gate switch is put OFF.
7. If $I_A > I_L$ then, the device remains in ON state and note that anode current as latching current.
8. Take small steps to get accurate latching current value.

TO FIND HOLDING CURRENT:

1. Now increase load current from latching current level by varying R_2 & V_2 .
2. Switch OFF the gate voltage switch S_1 permanently (now the device is in ON state).
3. Now increase load resistance (R_2), so that anode current reducing, at some anode current the device goes to turn off. Note that anode current as holding current.
4. Take small steps to get accurate holding current value.
5. Observe that $I_H < I_L$.

TABULAR COLUMN:

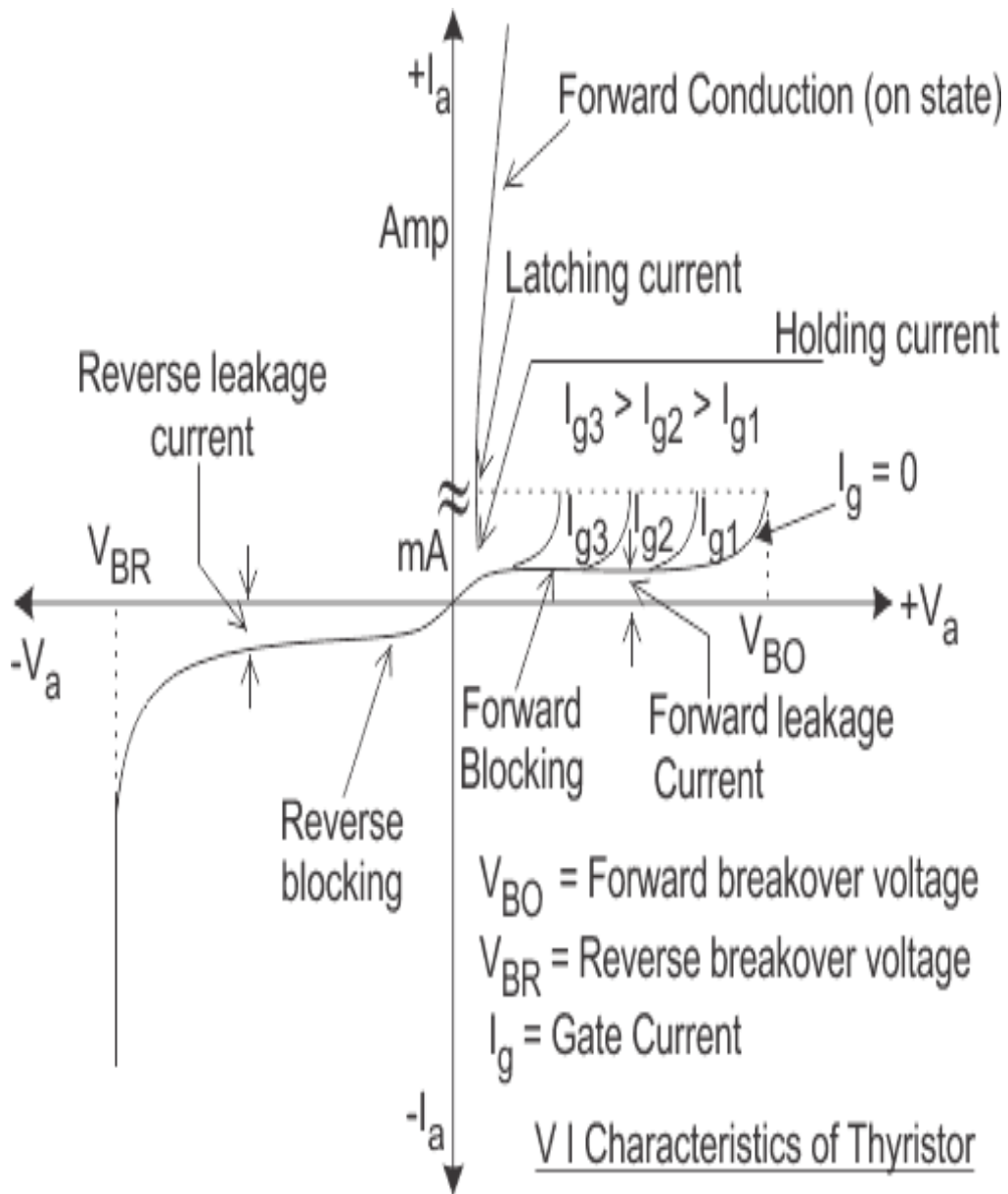
S. No	$I_G = \text{ A}$	
	V_{AK} (Volts)	I_A (Amps)
1		
2		
3		
4		
5		

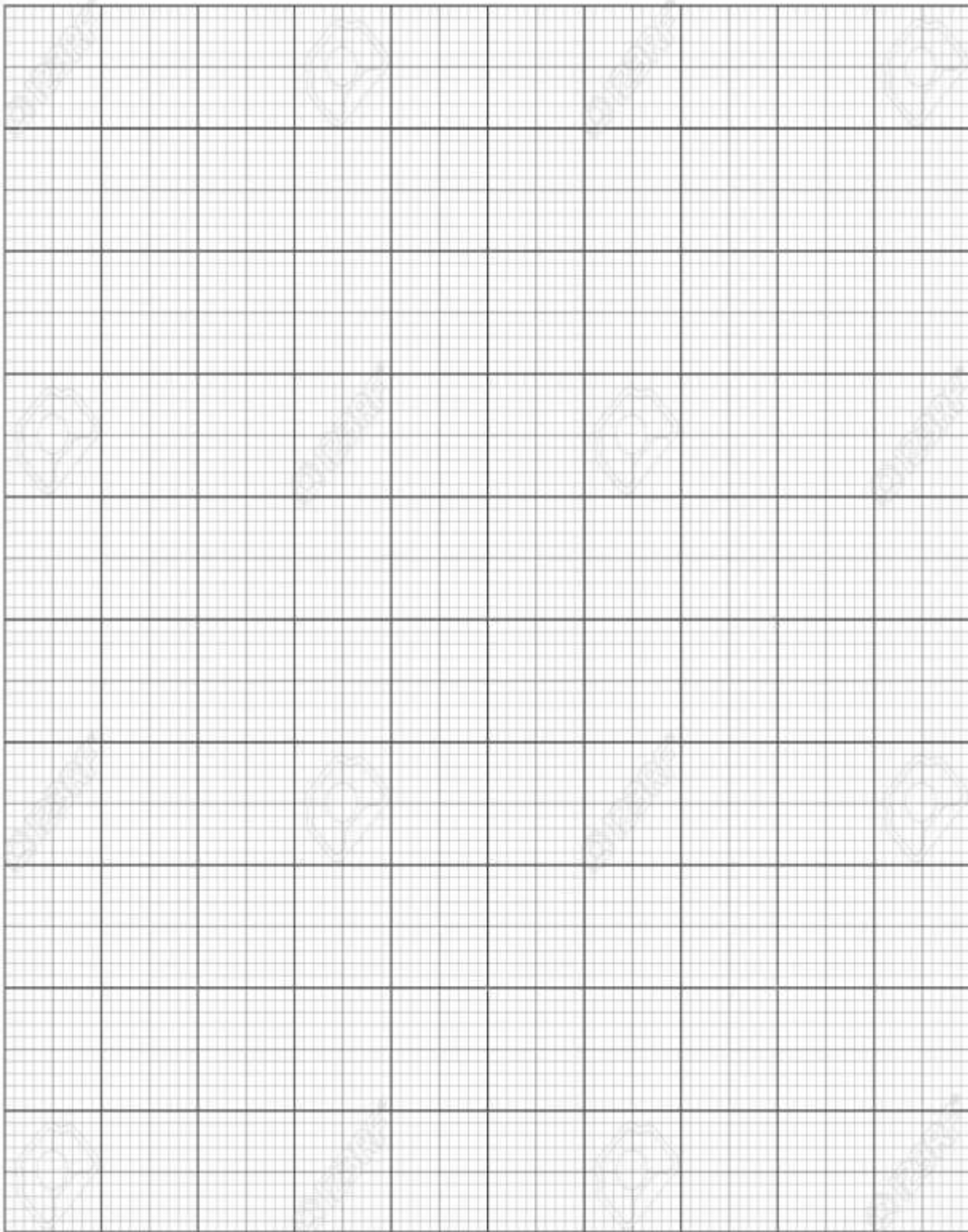
S. No	$I_G = \text{ A}$	
	V_{AK} (Volts)	I_A (Amps)
1		
2		
3		
4		
5		

S. No	$V_{AK} = \text{ (Volts)}$	
	$V_{GK} = \text{V}$	$I_G = \text{ A}$
1		
2		
3		
4		
5		

S. No	$V_{AK} = \text{ (Volts)}$	
	$V_{GK} = \text{V}$	$I_G = \text{ A}$
1		
2		
3		
4		
5		

V- I Characteristics of SCR





MOSFET CHARACTERISTICS**AIM:**

To study the output and transfer characteristics of MOSFET

APPARATUS:

S. No	Equipment	Range	Type	Quantity
1	MOSFET characteristics Trainer			
2	Patch chords			
3	DC Voltmeter			
4	DC Ammeter			

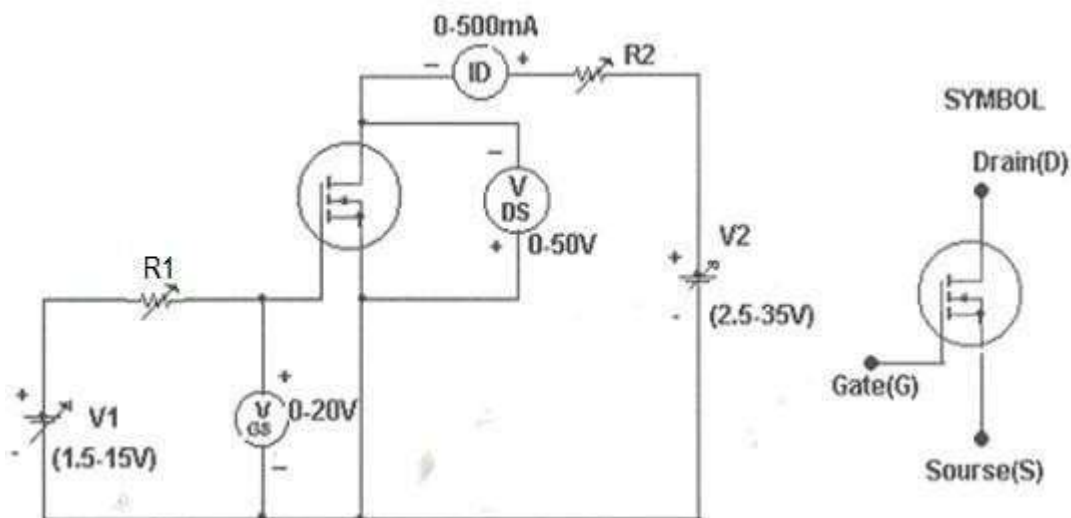
CIRCUIT DIAGRAM:

Fig – 1.2.1 Study of Characteristics of MOSFET

PROCEDURE:**TRANSFER CHARACTERISTICS:**

1. Make all connections as per the circuit diagram.
2. Initially keep V_1 & V_2 at minimum position and R_1 & R_2 middle position.
3. Set V_{DS} to some say 10V.
4. Slowly vary Gate source voltage V_{GS} by varying V_1 .
5. Note down I_D and V_{GS} readings for each step.
6. Repeat above procedure for 20V & 30V of V_{DS} . Draw Graph between I_D & V_{GS} .

OUTPUT CHARACTERISTICS:

1. Initially set V_{GS} to some value say 3V by varying V_1 .
2. Slowly vary V_2 and note down I_D and V_{DS} .
3. At particular value of V_{GS} there a pinch off voltage between drain and source.
If $V_{DS} < V_P$ device works in the constant resistance region and I_D is directly proportional to V_{DS} . If $V_{DS} > V_P$ device works in the constant current region.
4. Repeat above procedure for different values of V_{GS} and draw graph between I_D vs V_{DS} .

TABULAR COLUMN:

S. No.	$V_{GS} = V$	
	V_{DS} (Volts)	I_D (Amps)
1		
2		
3		
4		
5		

S. No	$V_{GS} = V$	
	V_{DS} (Volts)	I_D (Amps)
1		
2		
3		
4		
5		

S. No	$V_{DS} =$ (Volts)	
	V_{GS} (V)	I_D (A)
1		
2		
3		
4		
5		

S. No	$V_{DS} =$ (Volts)	
	V_{GS} (V)	I_D (A)
1		
2		
3		
4		
5		

MODEL GRAPH:

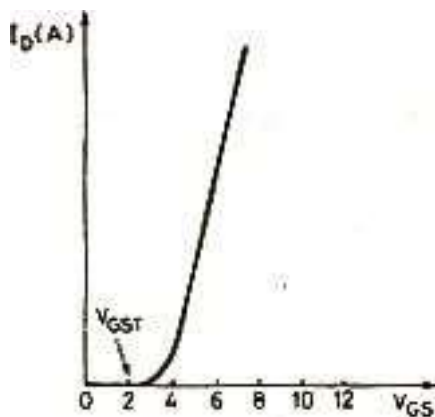


Fig - 1.2.2 Transfer Characteristic of MOSFET

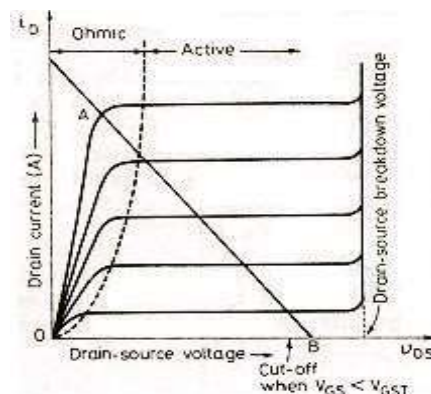
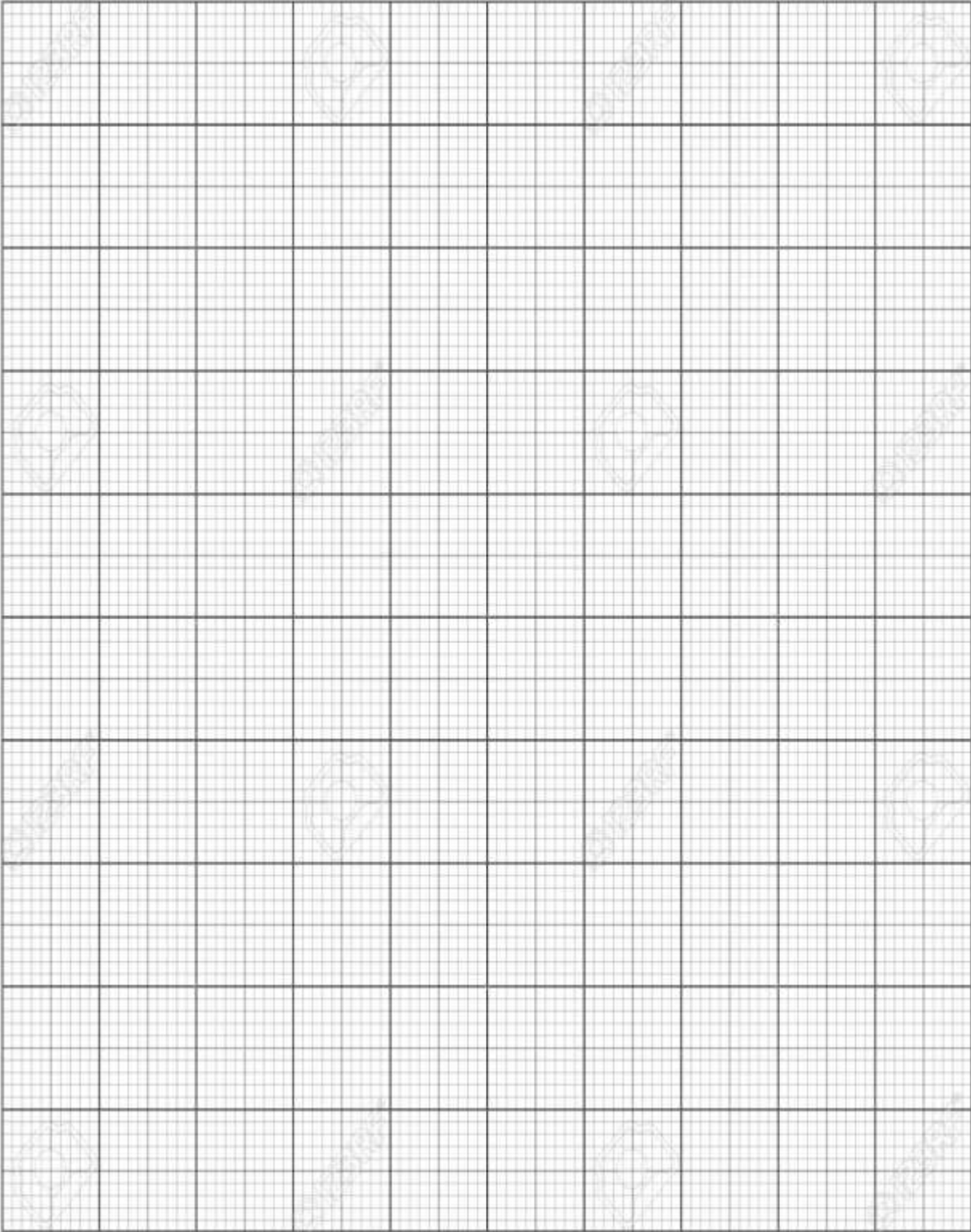


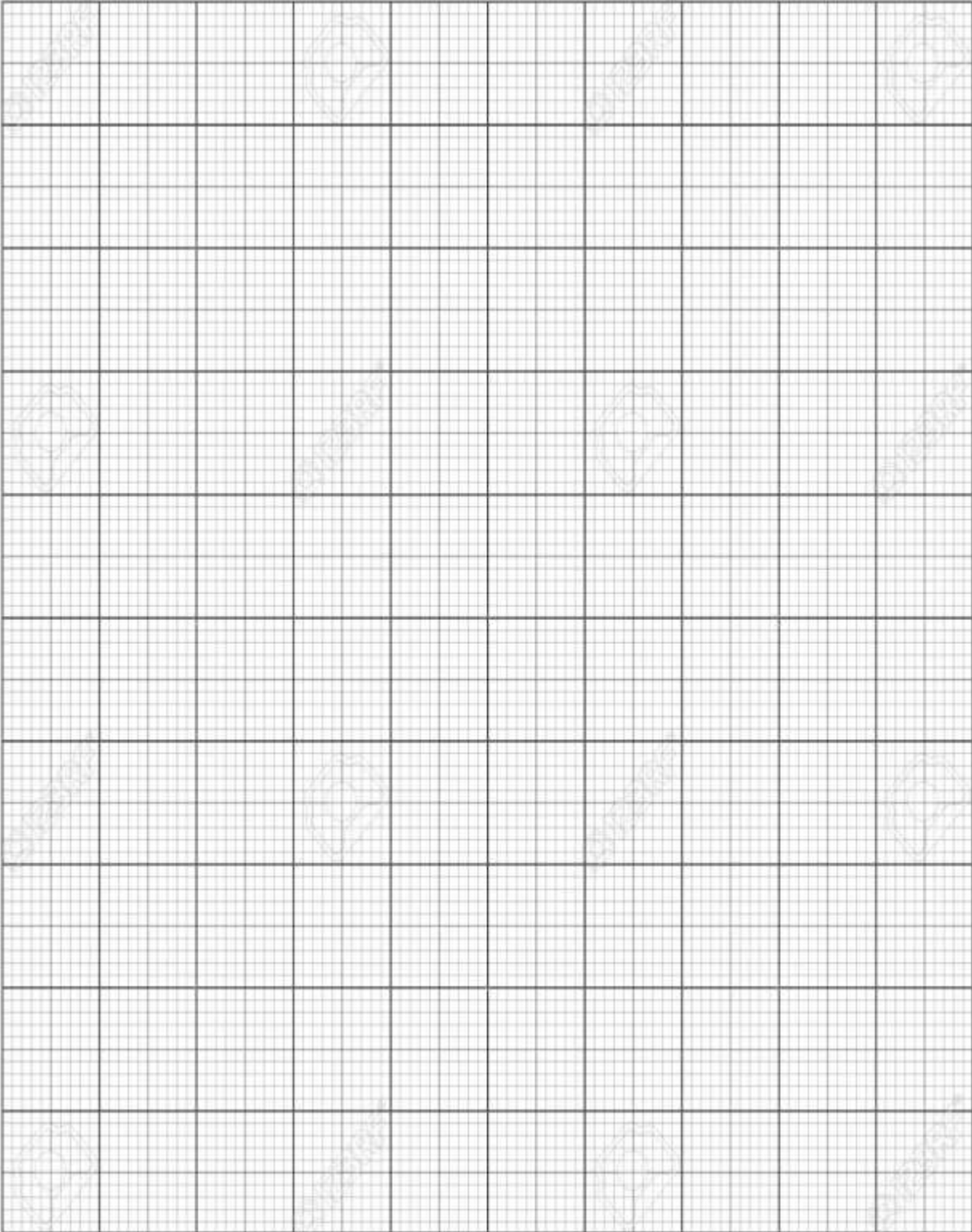
Fig - 1.2.3 Output Characteristics of MOSFET

RESULT:

POST LAB VIVA QUESTIONS:

1. How does n - drift region affect MOSFET?
2. How MOSFET are suitable for low power high frequency applications?
3. What are the requirements of gate drive in MOSFET?
4. Draw the switching model of MOSFET.
5. What is rise time and fall time?
6. In which region does the MOSFET used as a switch?
7. Why are MOSFET's mainly used for low power applications?
8. How is MOSFET turned off?
9. What are the advantages of vertical structure of MOSFET?
10. What are the merits of MOSFET?





EXPERIMENT NO: 1(C)

DATE:

IGBT CHARACTERISTICS

AIM:

To study the output and transfer characteristics of IGBT.

APPARATUS:

S. No	Equipment	Range	Type	Quantity
1	IGBT characteristics Trainer			
2	Patch chords			
3	DC Voltmeter			
4	DC Ammeter			

CIRCUIT DIAGRAM:

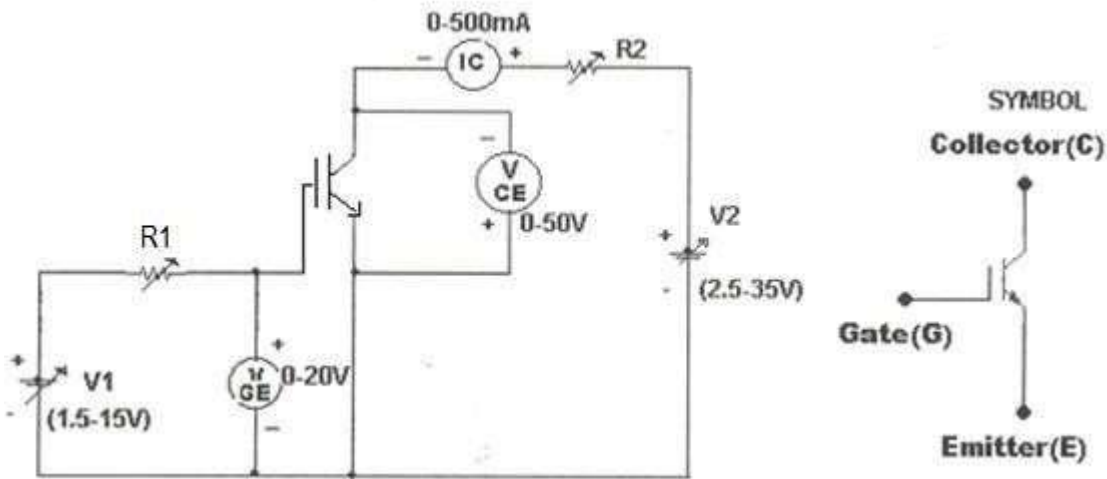


Fig – 1.3.1 Study of Characteristics IGBT

PROCEDURE:

TRANSFER CHARACTERISTICS:

1. Make all connections as per the circuit diagram.
2. Initially keep V_1 & V_2 at minimum position and R_1 & R_2 middle position.
3. Set V_{CE} to some say 10V.
4. Slowly vary Gate Emitter voltage V_{GE} by varying V_1 .
5. Note down I_C and V_{GE} readings for each step.
6. Repeat above procedure for 20V & 25V of V_{DS} . Draw Graph between I_D & V_{GS} .

OUTPUT CHARACTERISTICS:

1. Initially set V_{GE} to some value say 5V by varying V_1 .
2. Slowly vary V_2 and note down I_C and V_{CE} readings.
3. At particular value of V_{GS} there a pinch off voltage V_P between Collector and Emitter.
If $V_{CE} < V_P$ device works in the constant resistance region and I_C is directly proportional to V_{CE} . If $V_{CE} > V_P$ device works in the constant current region.
4. Repeat above procedure for different values of V_{GE} and draw graph between I_C vs V_{GE} .

TABULAR COLUMN:

S. No	$V_{CE} = V$	
	V_{GE} (Volts)	I_C (Amps)
1		
2		
3		
4		
5		

S. No	$V_{CE} = V$	
	V_{GE} (Volts)	I_C (Amps)
1		
2		
3		
4		
5		

S. No	$V_{GE} =$ (Volts)	
	V_{CE} (V)	I_C (A)
1		
2		
3		
4		
5		

S. No	$V_{GE} =$ (Volts)	
	V_{CE} (V)	I_C (A)
1		
2		
3		
4		
5		

MODEL GRAPH:

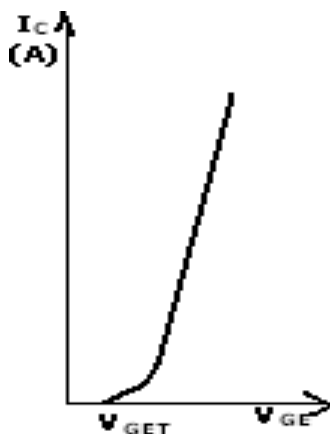


Fig – 1.3.2 Transfer Characteristics of IGBT

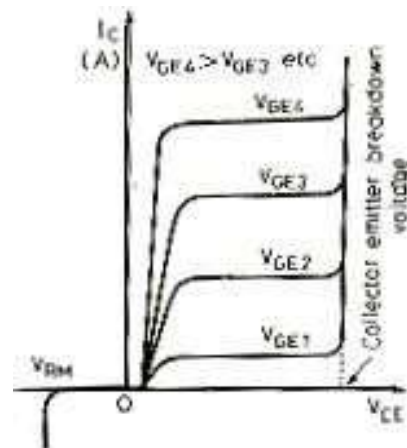
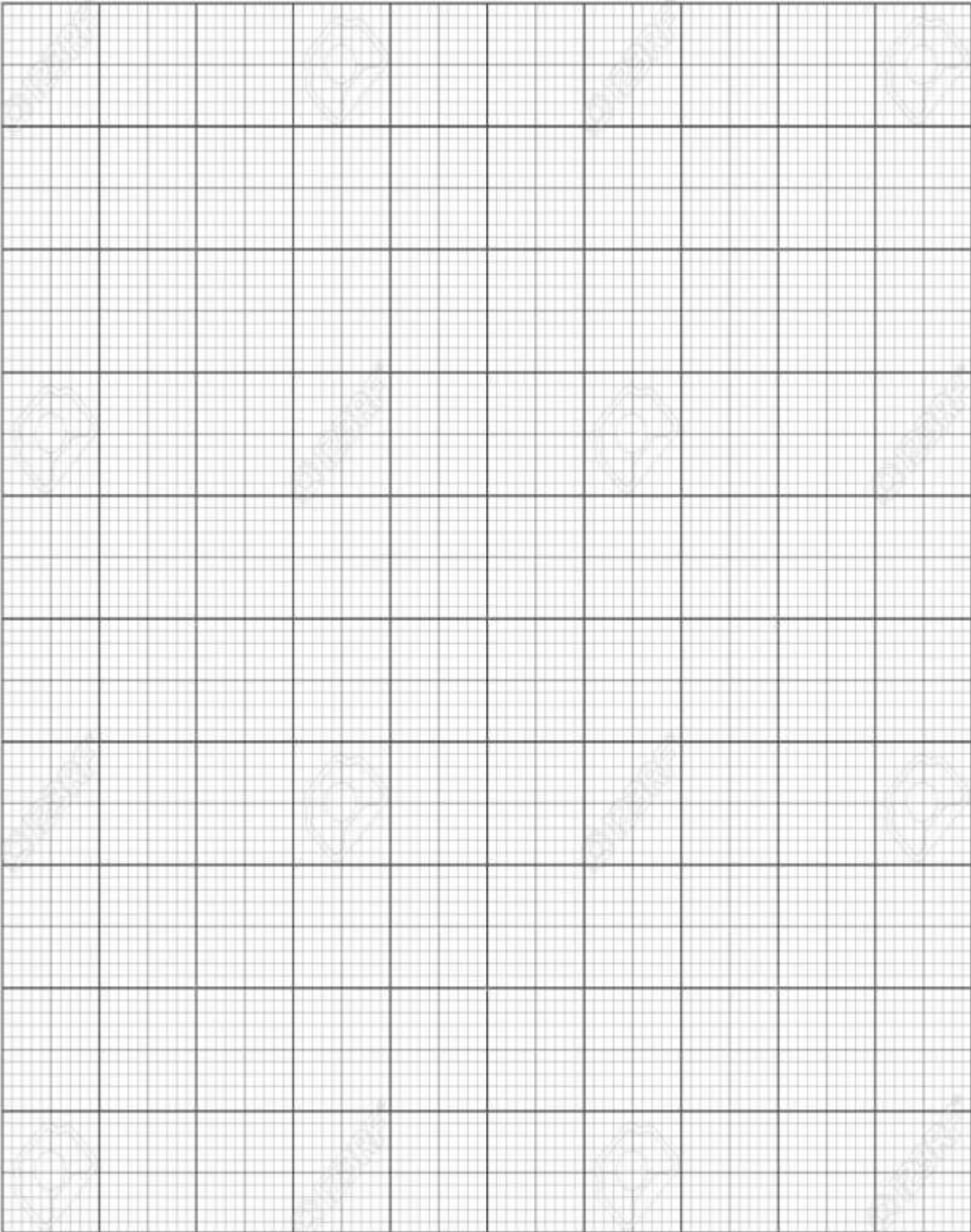


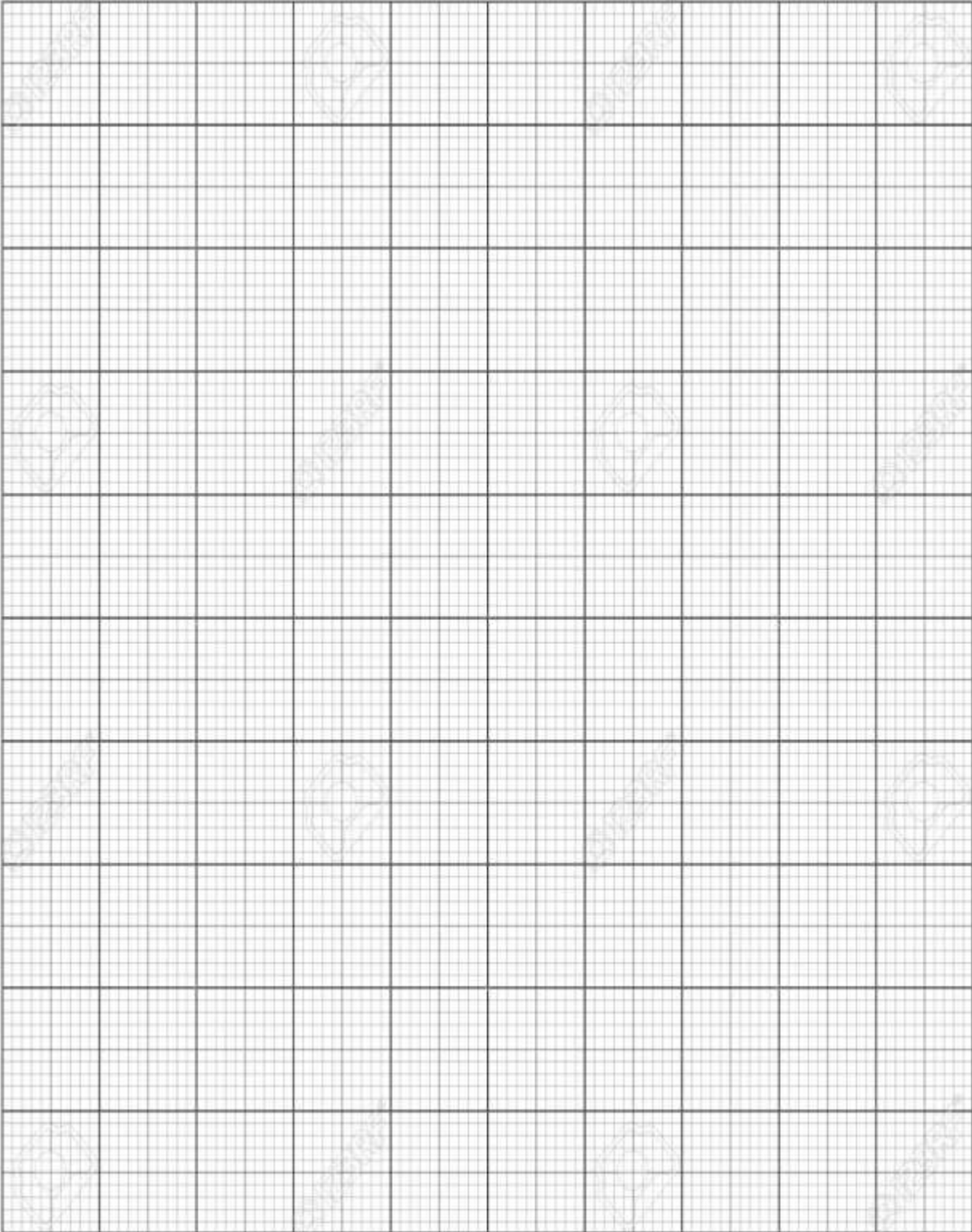
Fig - 1.3.3 Output Characteristics of IGBT

RESULT:

POST LAB VIVAQUESTIONS:

1. What are merits of IGBT?
2. What are demerits of IGBT?
3. What are the applications of IGBT's?
4. Why silicon used in all power semiconductor devices and why not? Germanium?
5. What is pinch off voltage?
6. What is threshold voltage?





EXPERIMENT NO: 02

DATE:

GATE FIRING CIRCUITS FOR SCR'S

Aim:

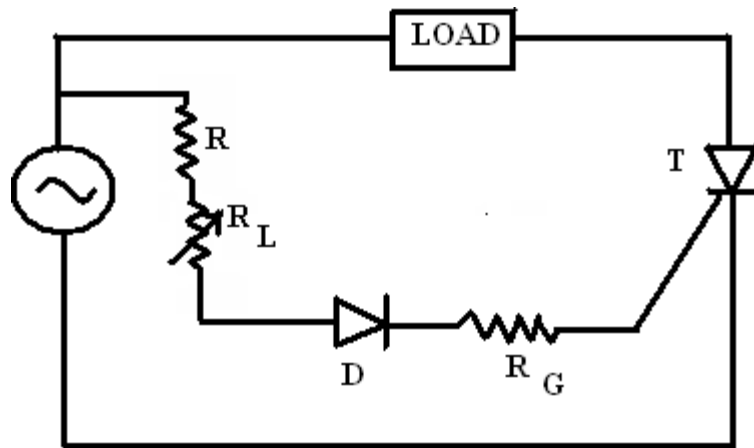
To study the operation of SCR using R & RC gate firing circuits.

Apparatus:

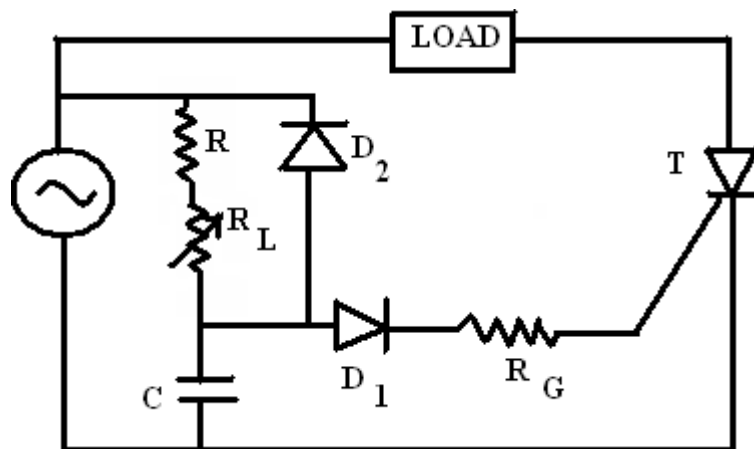
S.No.	Apparatus	Range	Quantity
1	R & RC Gate Firing Circuits Kit	---	1
2	CRO	(0-30)MHz	1
3	BNC Adaptors	---	1
4	R Load	47Ω/25W	1
5	Patch Cords	---	Some

Circuit Diagrams:

R Firing Circuit:



RC Firing Circuit:



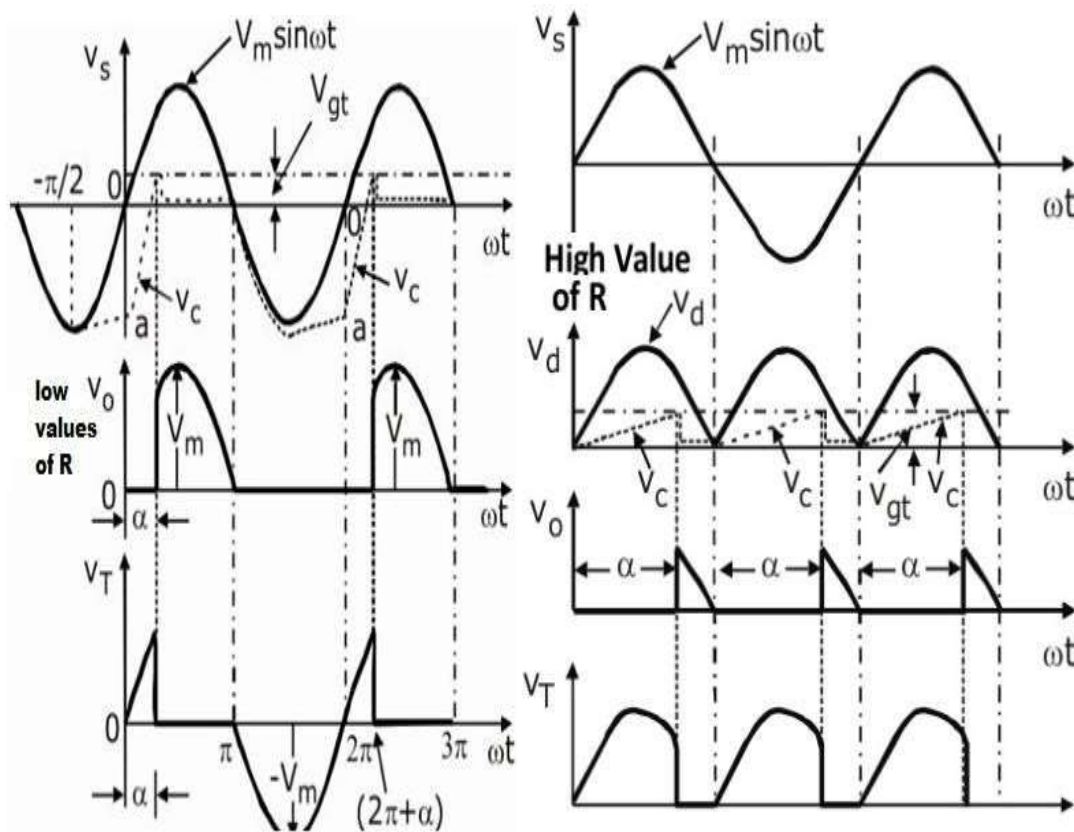
Precautions:

- 1) Connections should be right and tight.
- 2) Keep the potentiometers in the minimum firing angle position at the beginning of the experiment.
- 3) To reset the kit switch OFF all the main supply switches.

Procedure:

- 1) Connect the circuit as shown in circuit diagram.
- 2) Follow the precautions and switch ON the supply.
- 3) By varying the firing angle from 0° to 90° , observe the input voltage, output voltage, gate pulses and voltage across the thyristors.
- 4) Compare the practical values of output voltage with theoretical values and tabulate them.
- 5) Plot the graphs for input voltage, output voltage, gate pulses and voltage across the thyristors for different values of firing angle.
- 6) Repeat the same procedure for RC firing by varying the firing angle from 0° to 180° approximately.
- 7) Follow the precautions and switch OFF the supply.

Model Waveforms:



R Firing Circuits Waveforms

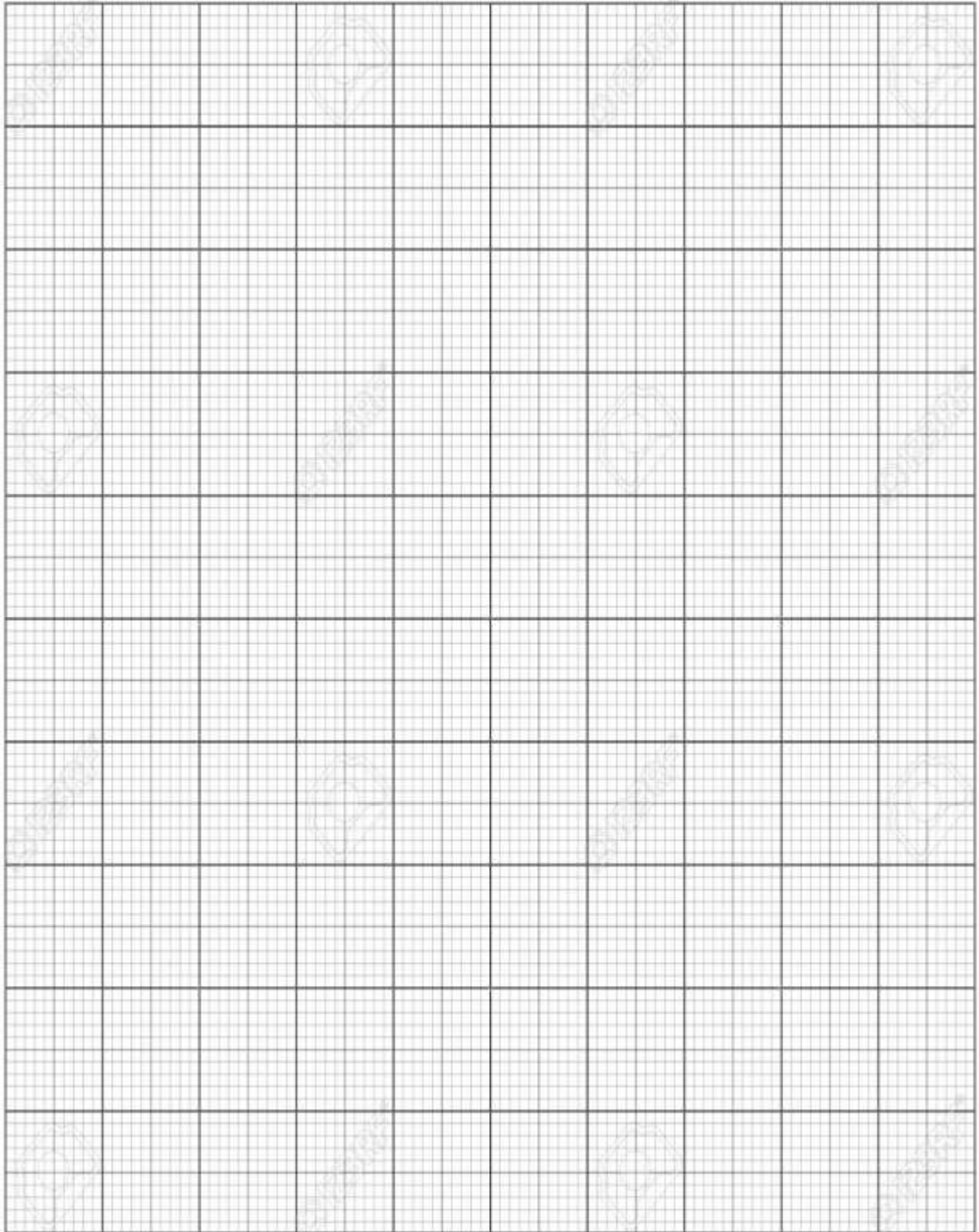
RC Firing Circuits

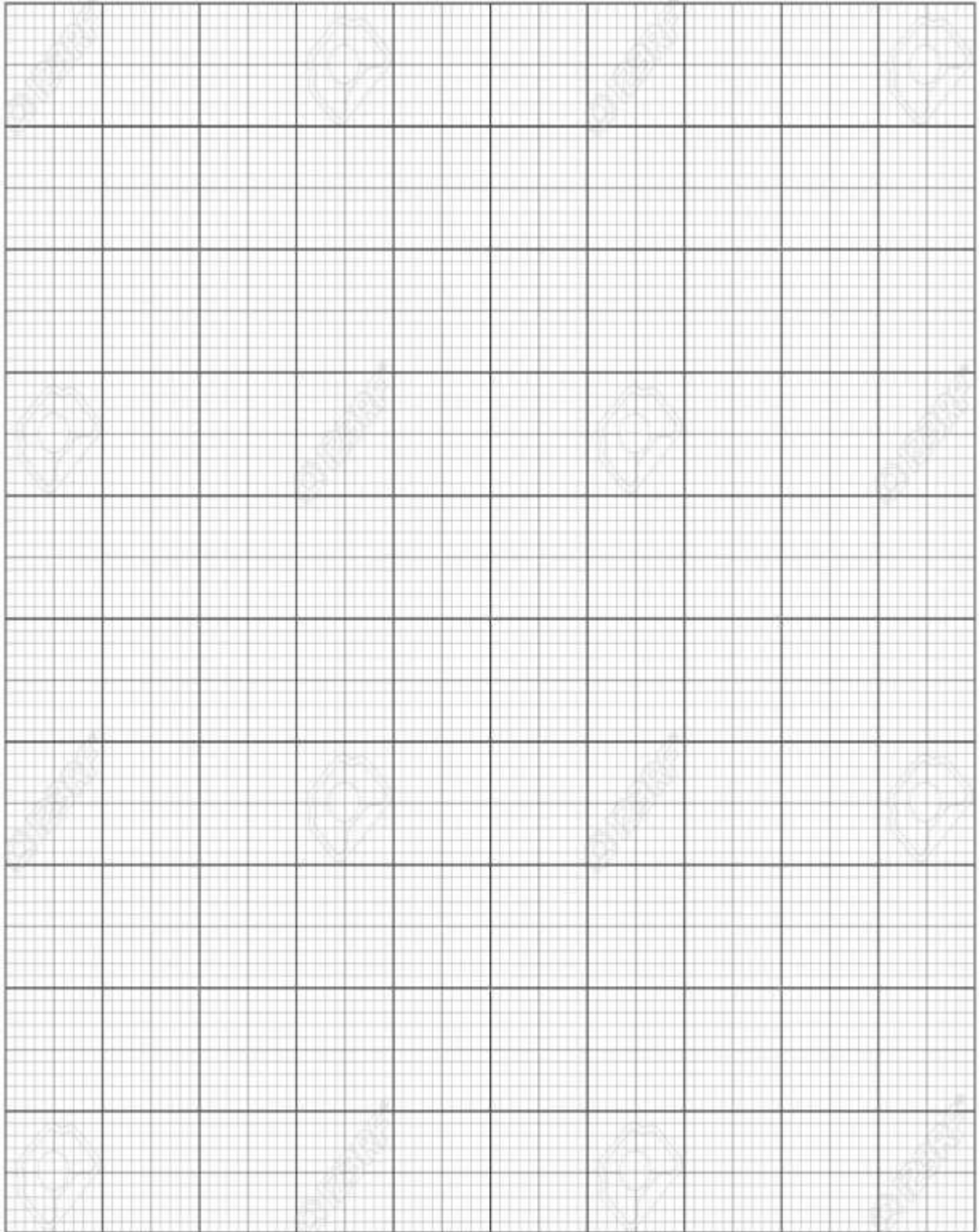
Calculations:

Result:

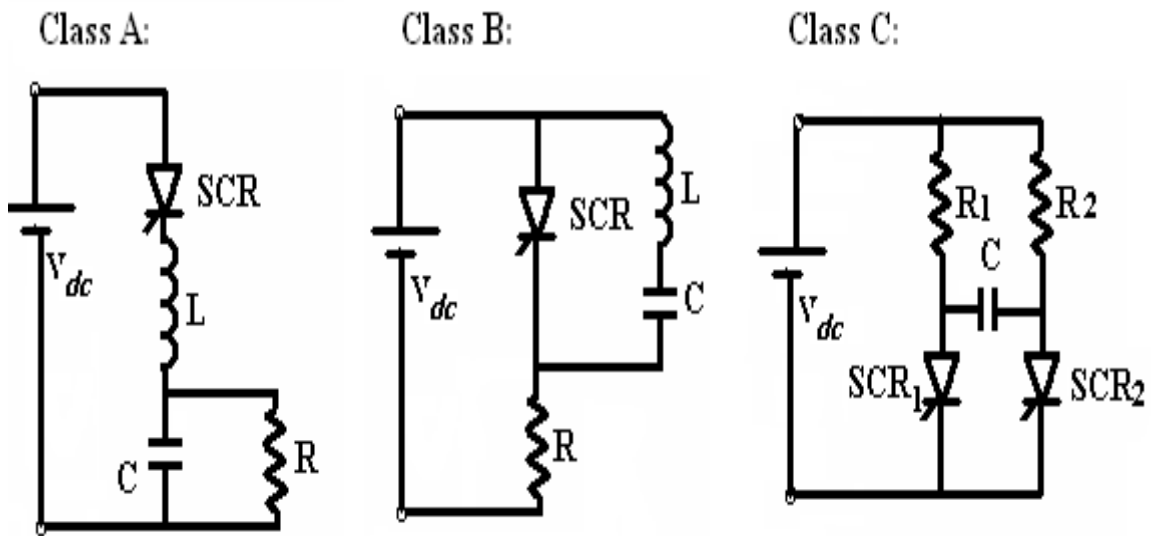
Viva Questions:

- 1.How many methods of gate controls are available for triggering SCRs?
- 2.What are methods of gate controls are available for triggering SCRs?
3. Why UJT is useful for pulse generation? What are the other devices available for the same purpose? Name them?
4. What is the purpose of using pulse transformer? What is its turns ratio? Will it influence the quality of gate signal?
5. Which is the best firing circuit suitable one for 1-phase fully control bridge converter? Suggest the circuit with reasons?
6. What are the different Pulse triggering methods?
7. What are the turn-on methods of SCR ?
8. What are the turn-off methods of SCR ?
9. What are the different modes of operation of SCR ?
10. Name the different types Power Electronic Devices?





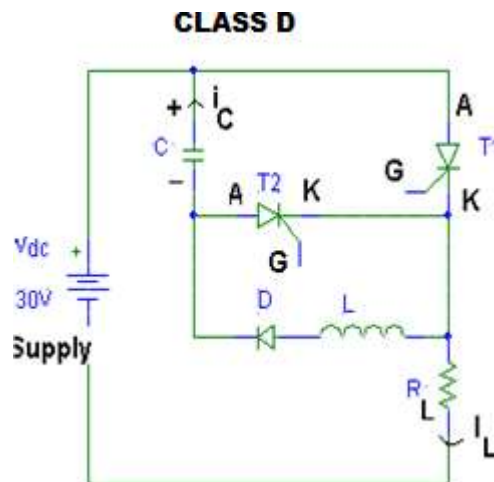
Circuit Diagrams:



Specifications:

Commutation Inductance: $L_1 = 250\text{mH}$, $L_2 = 500\text{mH}$, $L_3 = 1\text{H}$

Commutation Capacitance: $C_1 = 4.7\text{mF}$, $C_2 = 10\text{mF}$



EXPERIMENT NO: 03

DATE:

FORCED COMMUTATION CIRCUITS (CLASS A, B, C & D)

Aim:

To study the different commutation circuits (Class A, Class B, Class C, Class D).

Apparatus:

S.No.	Apparatus	Range	Quantity
1	Forced Commutation Study Kit	---	1
2	Rheostat	50Ω/2A(WW)	2
3	CRO	(0-30)MHz	1
4	BNC Adaptors	---	1
5	Patch Cords	---	Required some

Precautions:

- 1) Connections should be right and tight.
- 2) Keep the rheostats in maximum resistance position.
- 3) Keep the potentiometers in the minimum frequency & minimum duty cycle position at the beginning of the experiment.
- 4) For Class A and Class B commutation, duty cycle is of no use.
- 5) To reset the kit switch OFF all the main supply switches.

Procedure:

For Class A Commutation:

- 1) Connect the circuit as shown in circuit diagram.
- 2) Follow the precautions switch ON the DC supply.
- 3) By varying the frequency potentiometer observe the voltage waveforms across the load.
- 4) Repeat the same for L, C & R.
- 5) Follow the precautions and switch OFF the supply.

For Class B Commutation:

- 1) Connect the circuit as shown in circuit diagram.
- 2) Follow the precautions switch ON the DC supply.
- 3) By varying the frequency potentiometer observe the voltage waveforms across the load.
- 4) Repeat the same for L, C & R.
- 5) Follow the precautions and switch OFF the supply.

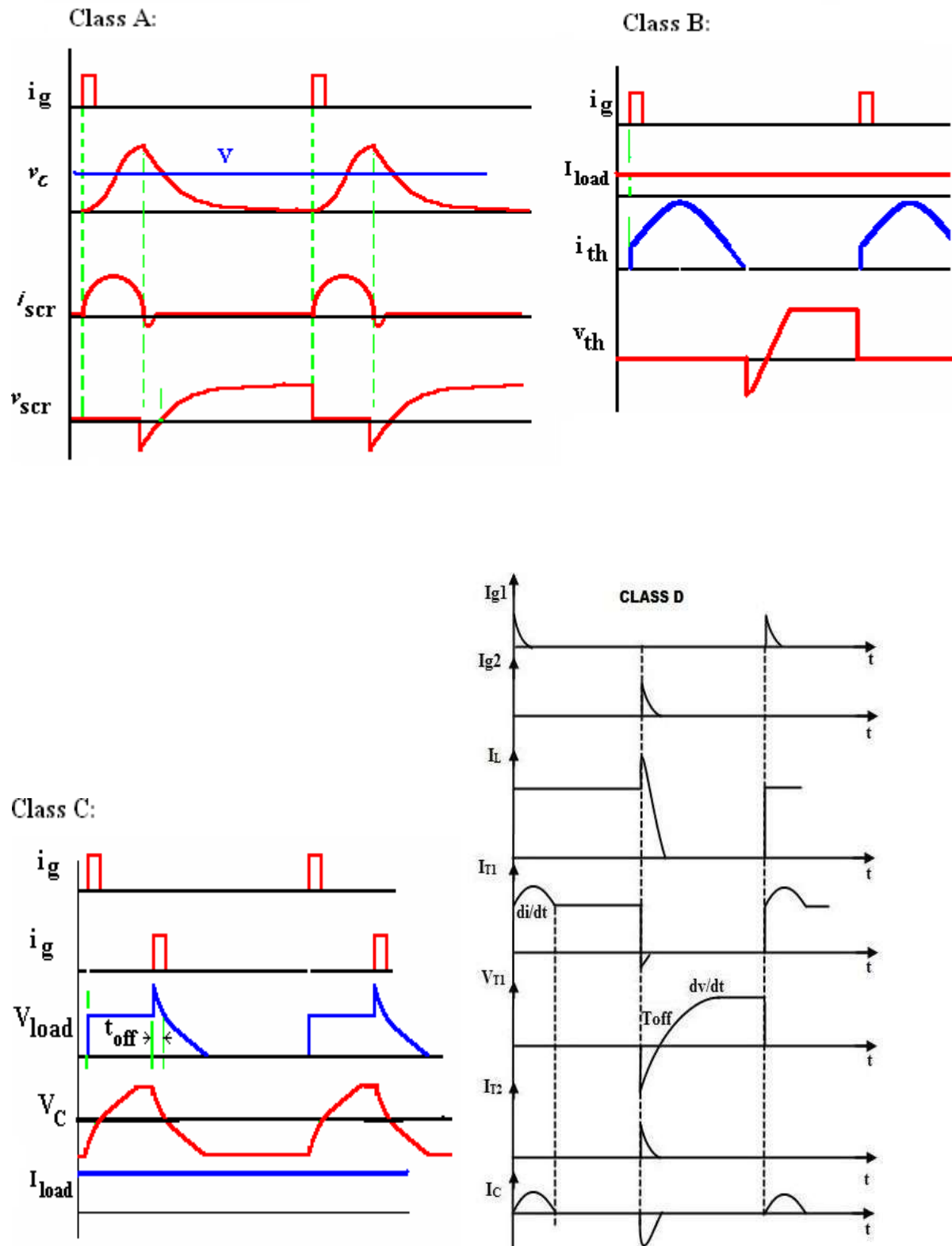
For Class C Commutation:

- 1) Connect the circuit as shown in circuit diagram.
- 2) Follow the precautions switch ON the DC supply.
- 3) By varying the frequency and duty cycle potentiometers observe the voltage waveforms across the R₁, R₂, C.
- 4) Repeat the same for different values of C.
- 5) Follow the precautions and switch OFF the supply.

For Class D Commutation:

- 1) This is again the same set-up as expt.4
- 2) The circuit then has chopper configuration and has been used to control DC motor of a battery powered vehicle.

Model Waveforms:

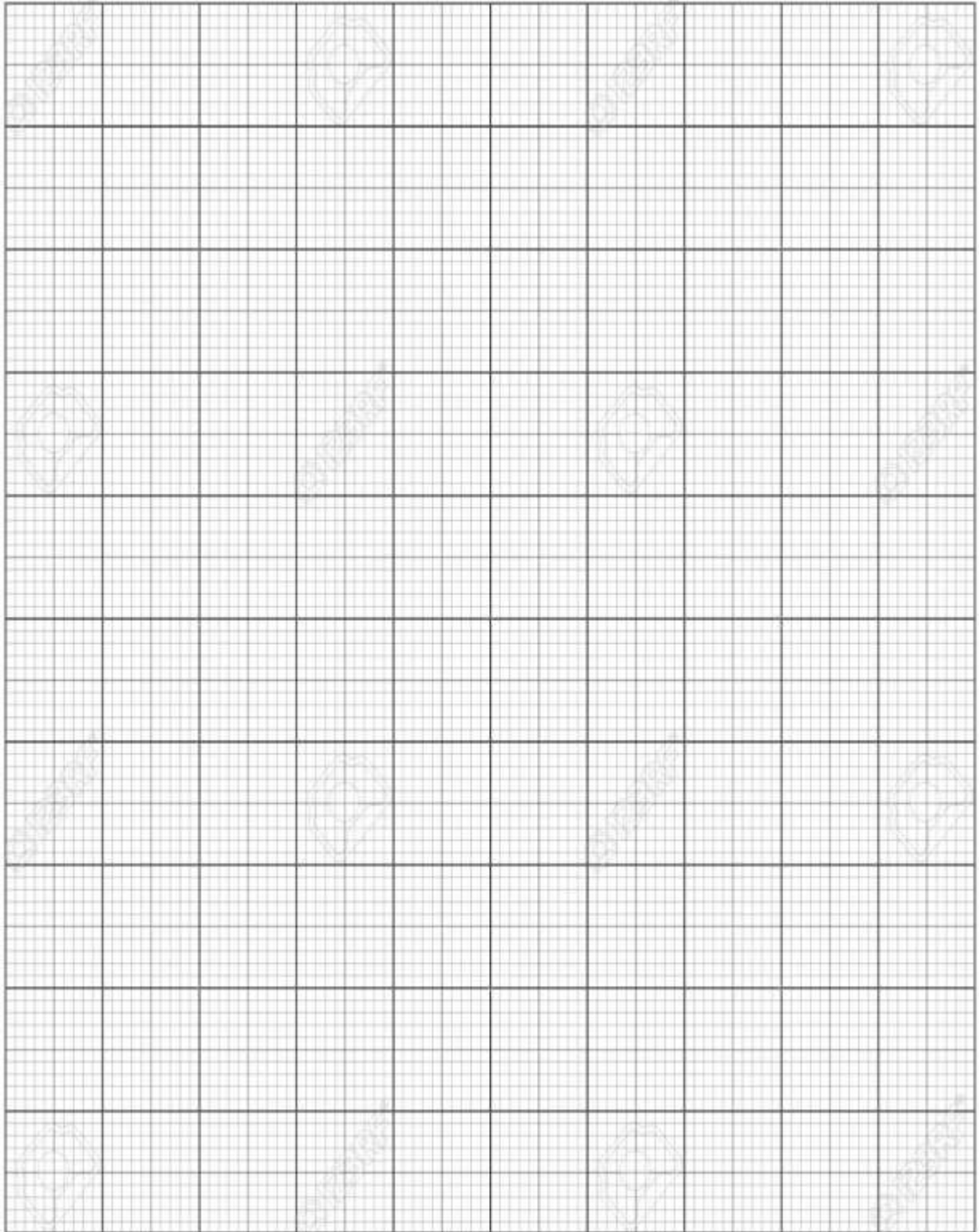


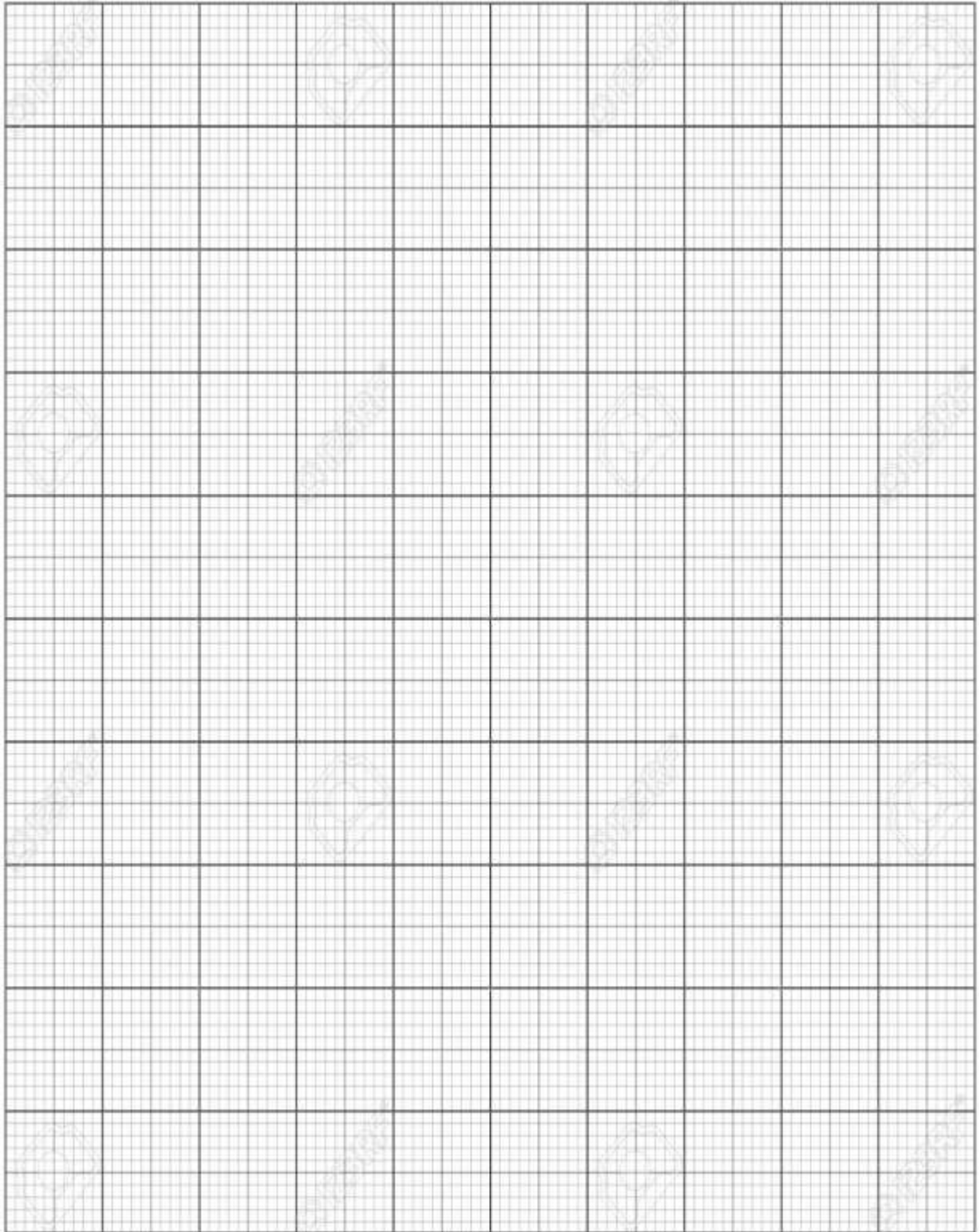
Calculations:

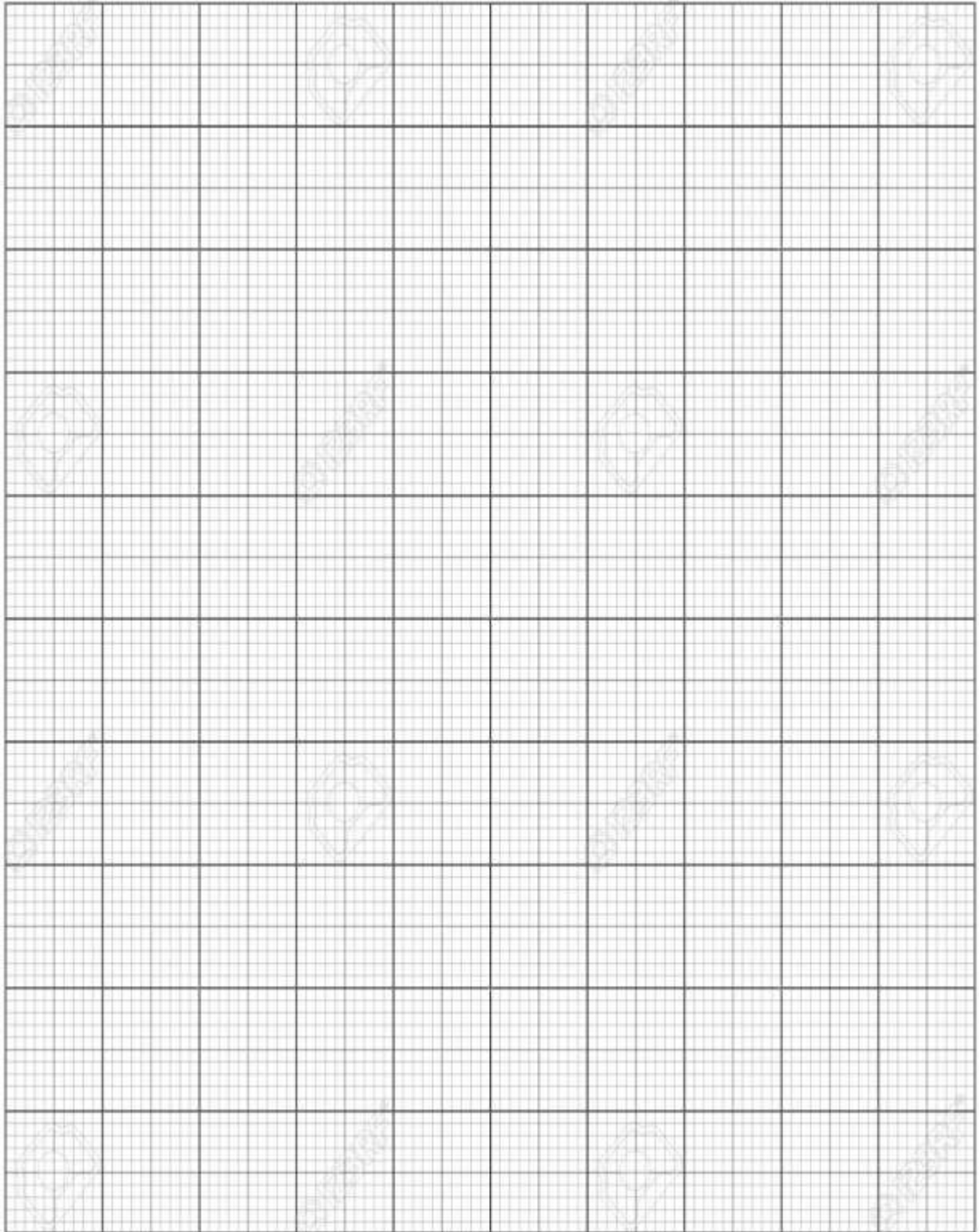
Result:

Viva Questions:

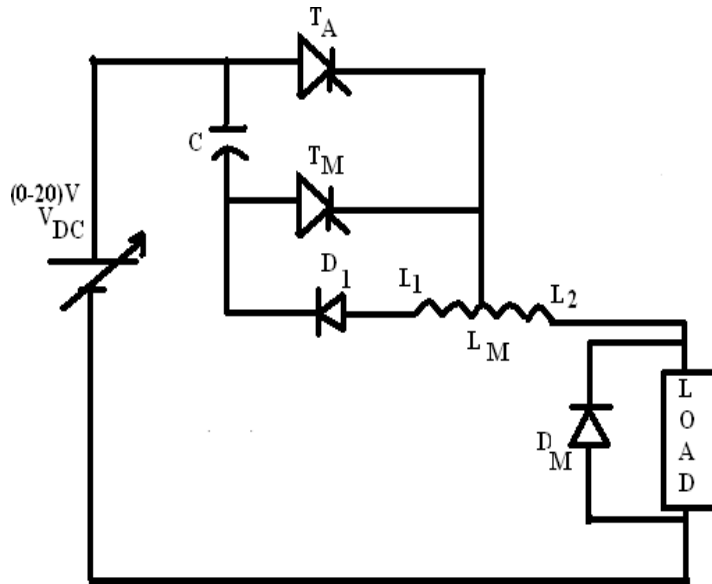
- 1 .What is the difference between Forced and natural commutation?
2. What is principle of self commutation?
3. What are the differences between Voltage and current commutation?
4. What is the purpose of commutation circuit?
5. Why should be the reverse bias time greater than the turn-off time of an SCR?
6. How the voltage of commutation capacitor is reversed in a commutation circuit ?
7. What is load commutation?
8. What is voltage commutation?
9. What is current commutation?
10. What is complementary commutation?







Circuit Diagram:



EXPERIMENT NO: 04

DATE:

D.C JONES CHOPPER WITH R & RL LOADS

Aim:

To study SCR based D.C Jones chopper Circuit.

Apparatus:

S.No.	Apparatus	Range	Quantity
1	D.C Jones Chopper Firing Unit	---	1
2	D.C Jones Chopper Kit	---	1
3	Rheostat	50Ω/2A(WW)	1
4	Inductor	(0-150)mH/2A	1
5	CRO	(0-30)MHz	1
6	Regulated Power Supply	(0-30)V/2A	1
7	BNC Adaptors	---	1
8	Patch Cords	---	Some

Precautions:

- 1) Connections should be right and tight.
- 2) Keep the rheostat in maximum resistance position.
- 3) Keep the potentiometers in the minimum position at the beginning of the experiment.
- 4) To reset the kit switch OFF all the main supply switches.

Procedure:

For R Load:

- 1) Connect the circuit as shown in circuit diagram.
- 2) By observing the precautions switch ON the supply.
- 3) By varying the duty cycle and frequency (by keeping any one of them as constant), observe the input voltage, output voltage and voltage across the thyristors.
- 4) Compare the practical values of output voltage with theoretical values and tabulate them.
- 5) Plot the graphs for input voltage, output voltage and voltage across the thyristors.
- 6) Observing the precautions switch OFF the supply.

For RL Load:

- 1) Connect a 150mH/2A inductor in series with R Load and repeat the steps from 2 to 6.

TABULAR COLUMN:

Constant Duty Cycle

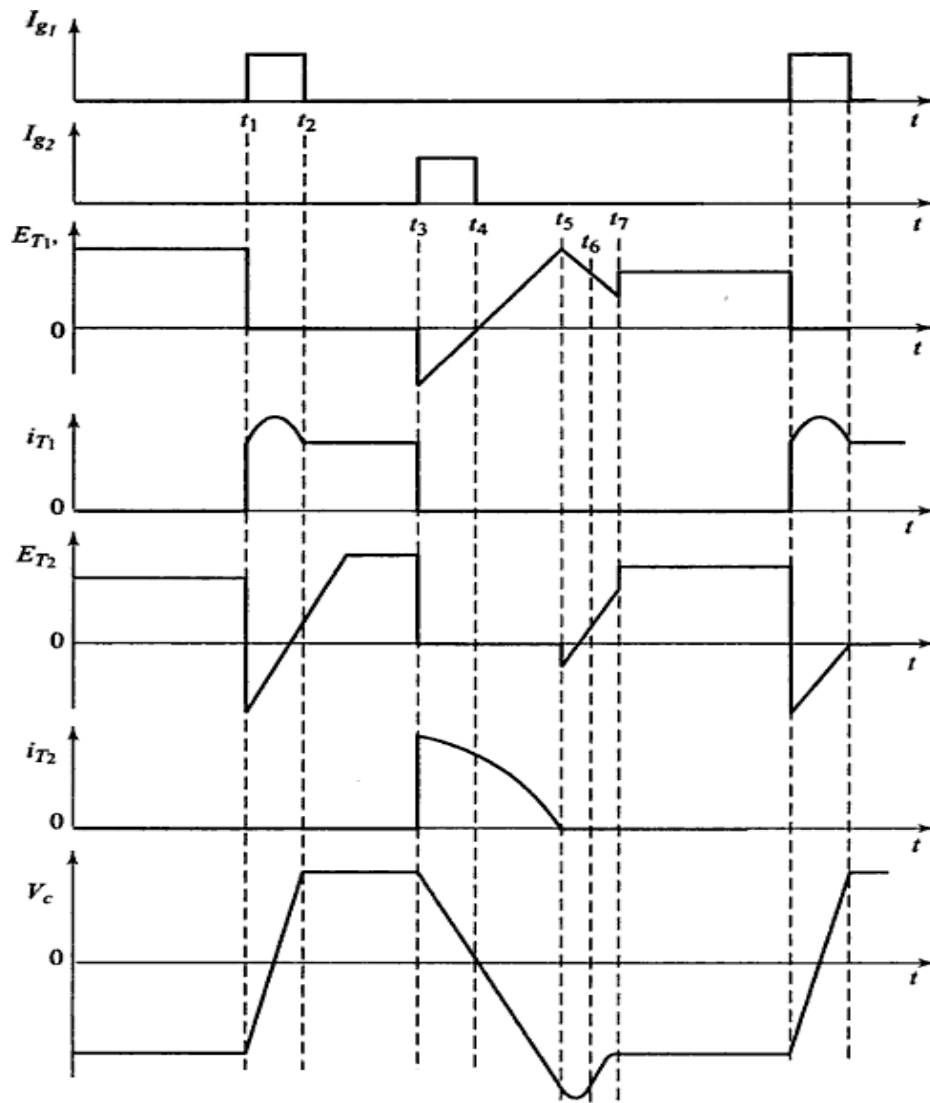
Duty Cycle: 50%, $V_{IN}=10$ to 15 V

S. No	Frequency(Hz)	V0(Volts)
1		
2		
3		
4		
5		
6		
7		
8		
9		
10		

Constant Frequency, Frequency Control

S. No	$T_{ON}(\text{sec})$	$T_{OFF}(\text{sec})$	Duty Cycle (%)	V_o (Volts)
1				
2				
3				
4				
5				
6				
7				
8				
9				

Model Waveforms:

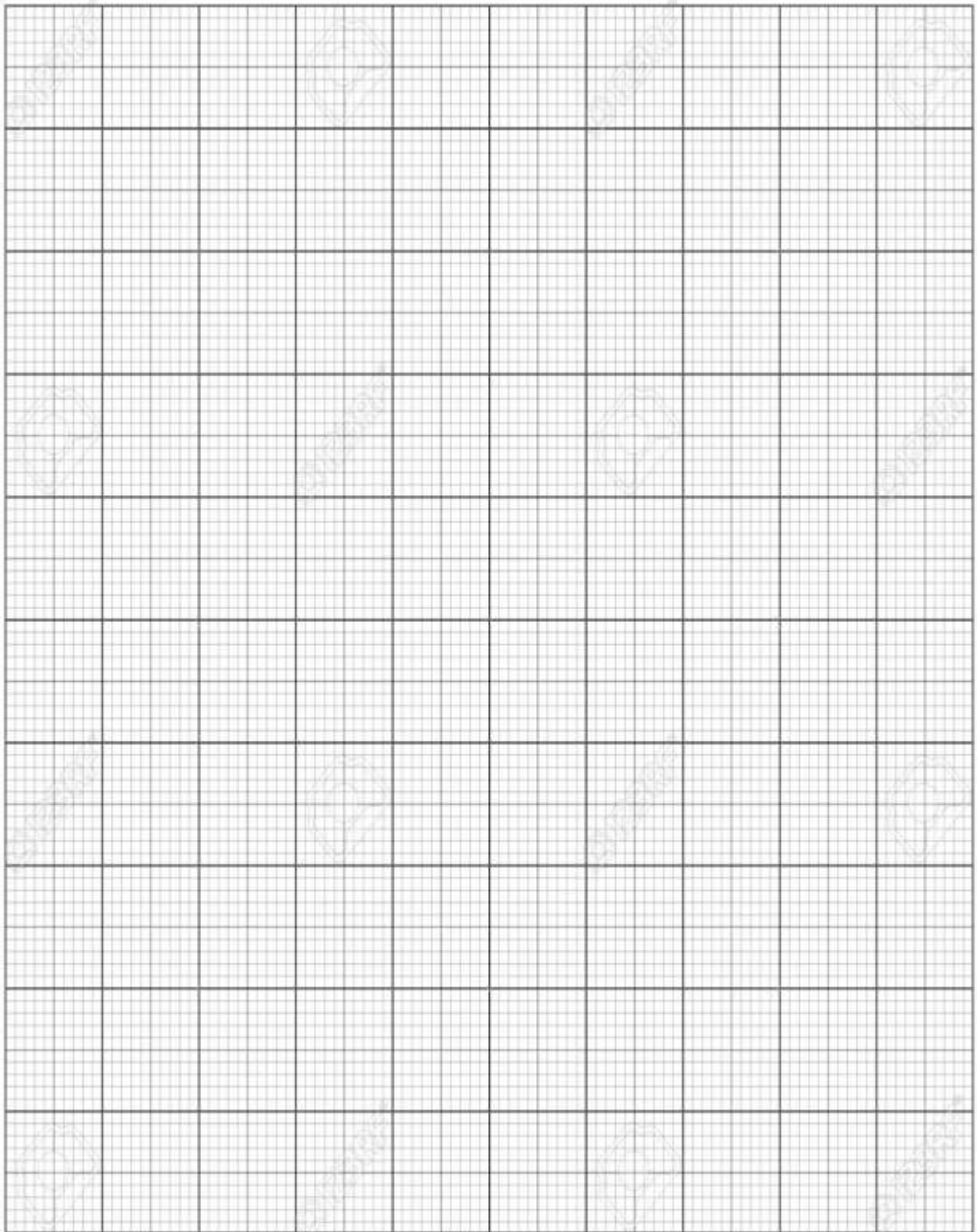


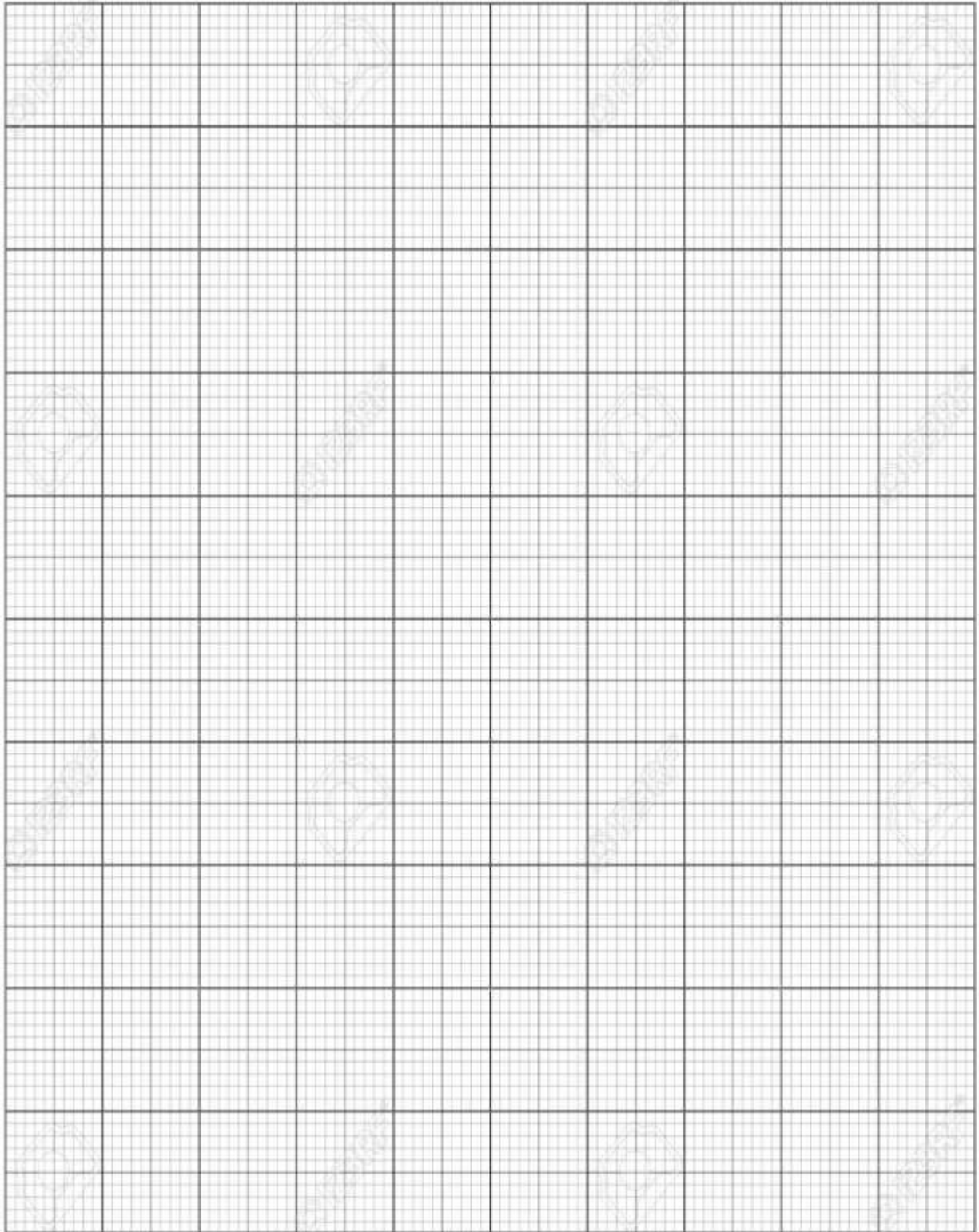
Calculations:

Result:

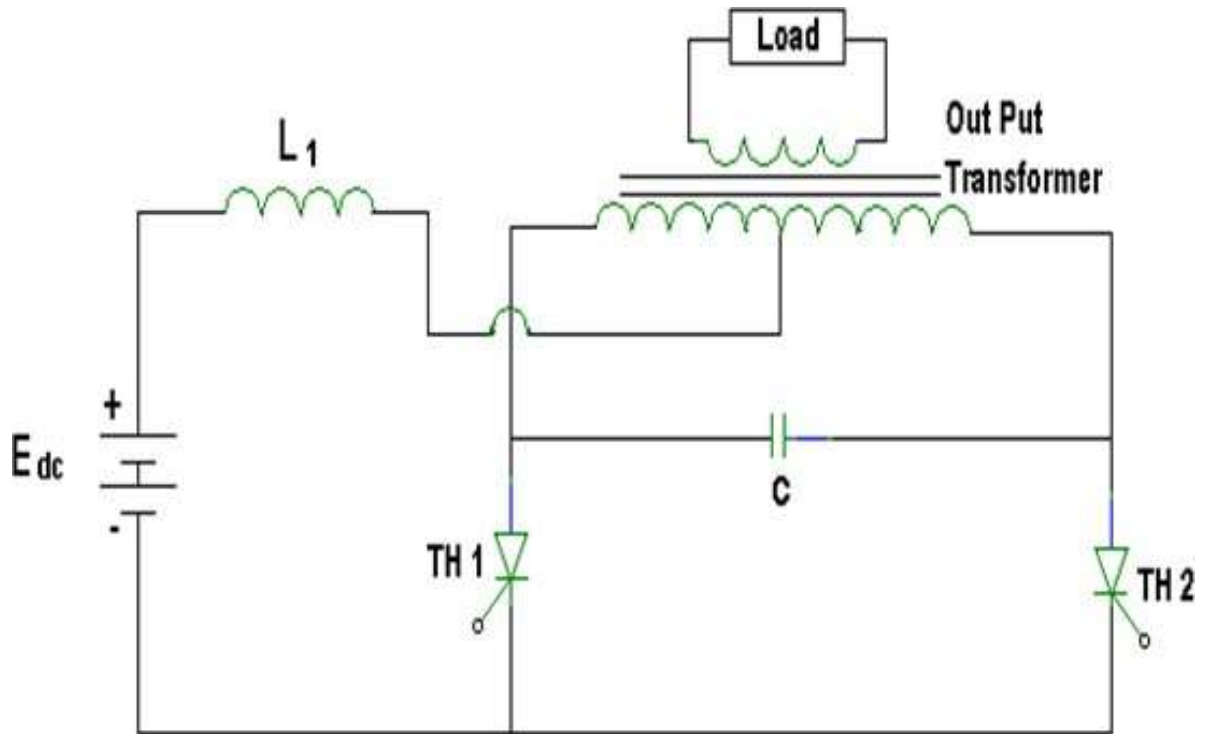
Viva Questions:

1. What is a DC Chopper?
2. What is frequency modulation control of a chopper?
3. What is pulse-width modulation control of a chopper?
4. What is the purpose of diode in the commutation circuit of Jones Chopper?
5. What are the applications of a chopper?
6. What is the purpose of a commutation circuit in a chopper?
7. What is difference between the circuit turn off and turn-off time of a thyristor?
8. Why does a commutation capacitor gets over charged?
9. What are the advantages and disadvantages of variable frequency chopper?
10. What is the type of commutation used in Jones Chopper?
11. What is the principle of operation of step down and step up chopper?





CIRCUIT DIAGRAM:



EXPERIMENT NO: 05 DATE:

SINGLE PHASE PARALLEL INVERTER

AIM: To study module and waveforms of a 1- ϕ parallel inverter power circuit with R and RL loads.

APPARATUS REQUIRED:

1. 1- ϕ parallel inverter firing circuit and power circuit unit
2. Inductor : 300mH, 2A.
3. Capacitor : 6.8 μ f, 100V.
4. Output transformer : Primary- 30V-25V-0-25V-30V Secondary- 0-30V2A
5. Loading Rheostat : 50 Ohms 2 A.
6. Loading Inductor : 50 mH, 2A
7. 20 MHz dual trace oscilloscope with 1:10 BNC probes

SPECIFICATIONS:

1. Input : 230V, 50Hz, 1- ϕ AC supply
2. Load : Rand RL
3. Thyristors : 10 A, 600V.
4. Diodes : 10A, 600V.
5. Capacitor : 6.8 μ f, 100V.
6. Inductor : 300 μ H, 2A.
7. Fuses : 2A Glass fuse

THEORY:

A Parallel inverter is used to produce a square wave from a DC source is connected in alternative sense to the two halves of the transformer primary, thereby inducing a square wave voltage across the load in the transformer secondary.

In this inverter, the commutating capacitor comes in parallel with the load during the operation of the inverter and hence this inverter is called as a parallel inverter. The capacitor in the given figure is required for commutation, but as the capacitor is effectively in parallel with the load via the transformer, an inductor L is required in series with the DC source to prevent the instant discharge of capacitor C via the source when thyristor switching occurs.

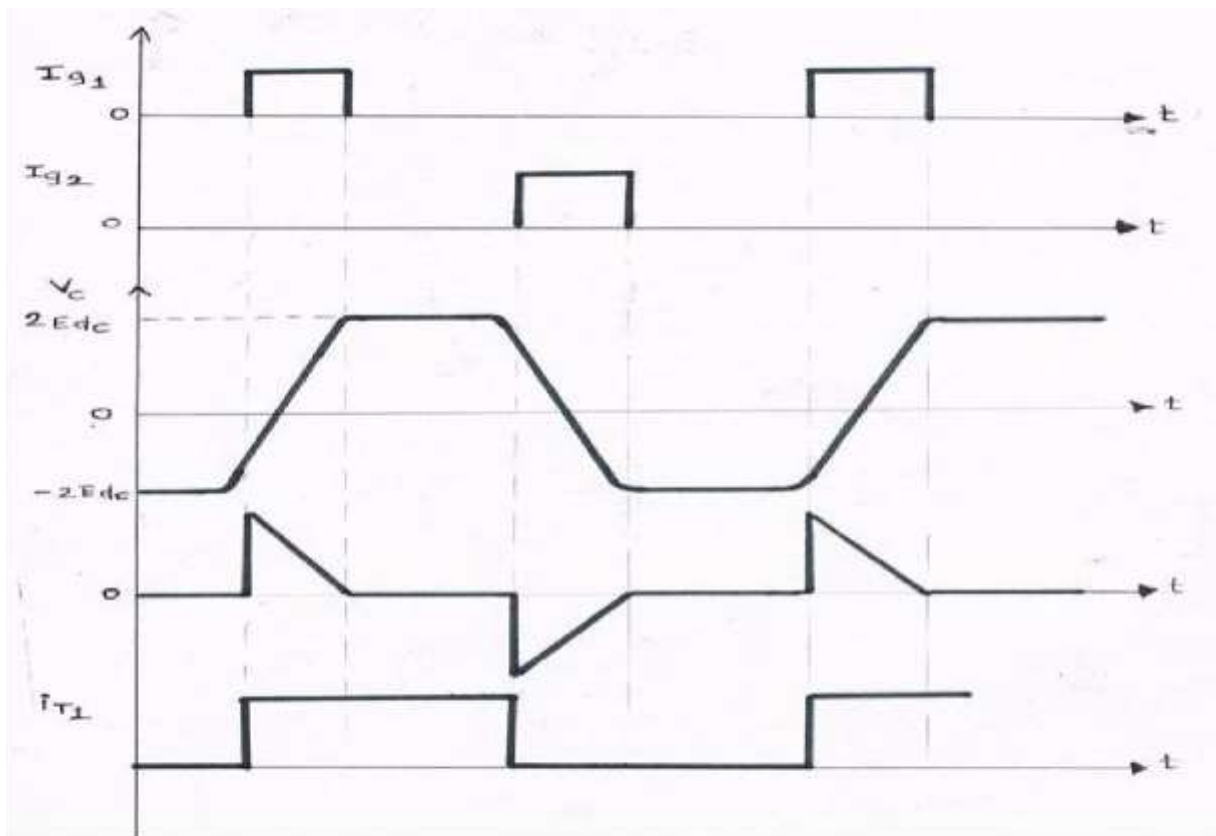
PRECAUTIONS:

1. Make sure all the connecting links are tightly fixed.
2. Ensure all the controlling knobs in fully counter clock wise position before starting experiment.
3. Handle everything with care.
4. Make sure the firing pulses are proper before connecting to the power circuit.
5. Make sure to connect firing pulses from the firing circuit to their respective SCRs in the power circuit.
6. Ensure switch OFF the input supply first and then trigger pulses to avoid short circuit.

PROCEDURE:

1. Switch ON the main supply to the firing circuit. Observe the trigger outputs in the firing circuit by varying frequency potentiometer and by operating ON/OFF switch. Make sure the firing pulses are proper before connecting to the power circuit.
2. Make the connections as per the circuit diagram.
3. Connect the firing pulses from the firing circuit to the respective SCRs in the power circuit.
4. Connect the DC input from 30V, 2A regulated power supply.
5. Switch ON the DC supply, set input voltage to 15 V and switch ON the trigger pulses by Operating ON/OFF/ switch in the firing circuit.
6. Observe the voltage waveform across load using oscilloscope.
7. Vary the frequency, load and observe the voltage waveform across load with and without freewheeling diode.
8. Draw the waveforms in the graph at different frequencies.
9. To switch OFF the inverter, switch OFF the input supply first and then trigger pulses.
10. Since the parallel inverter works on forced commutation, there is a chance of failure. If the commutation fails, switch OFF the DC supply and then trigger outputs. Check the connections and try again.

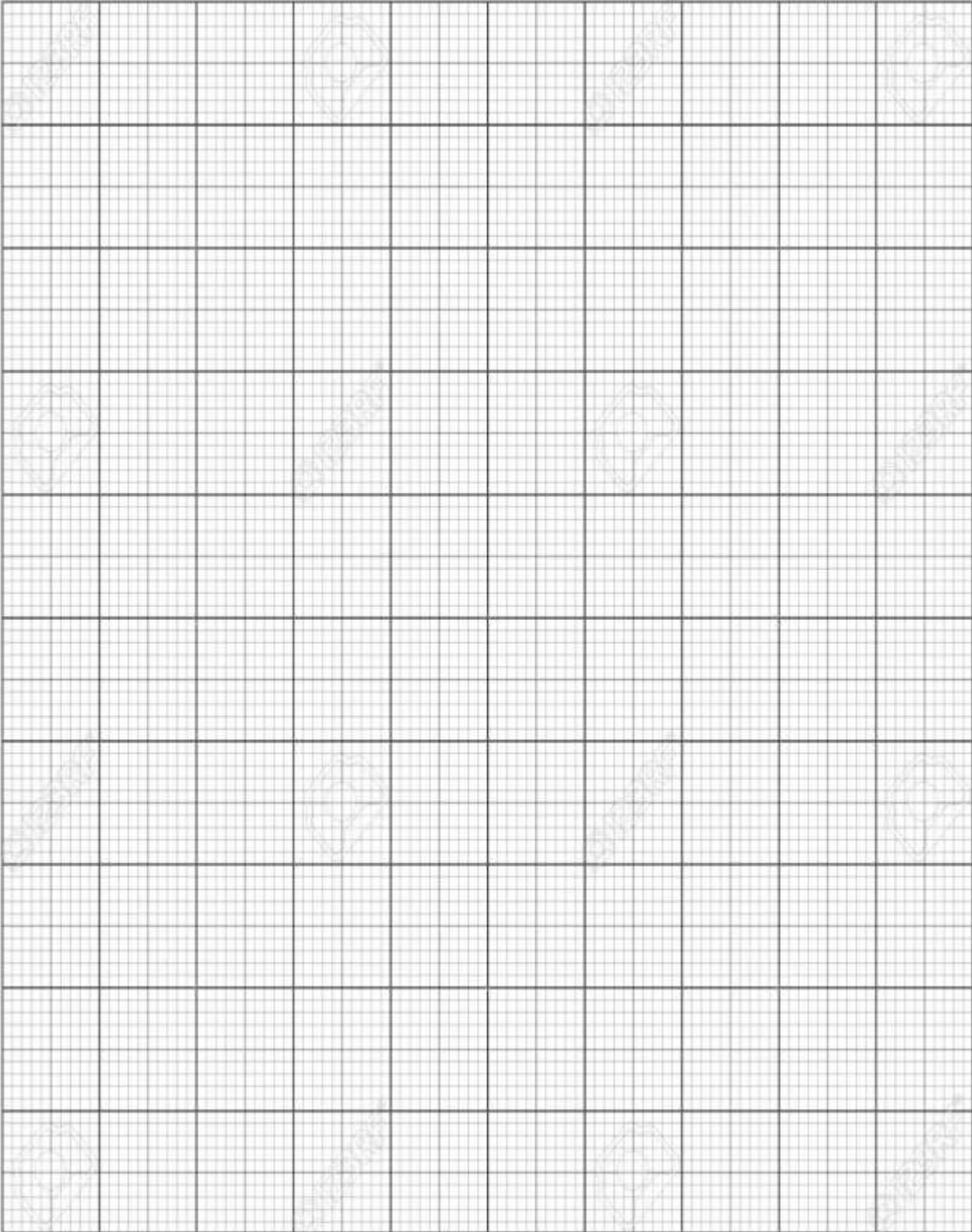
MODEL WAVE FORMS:



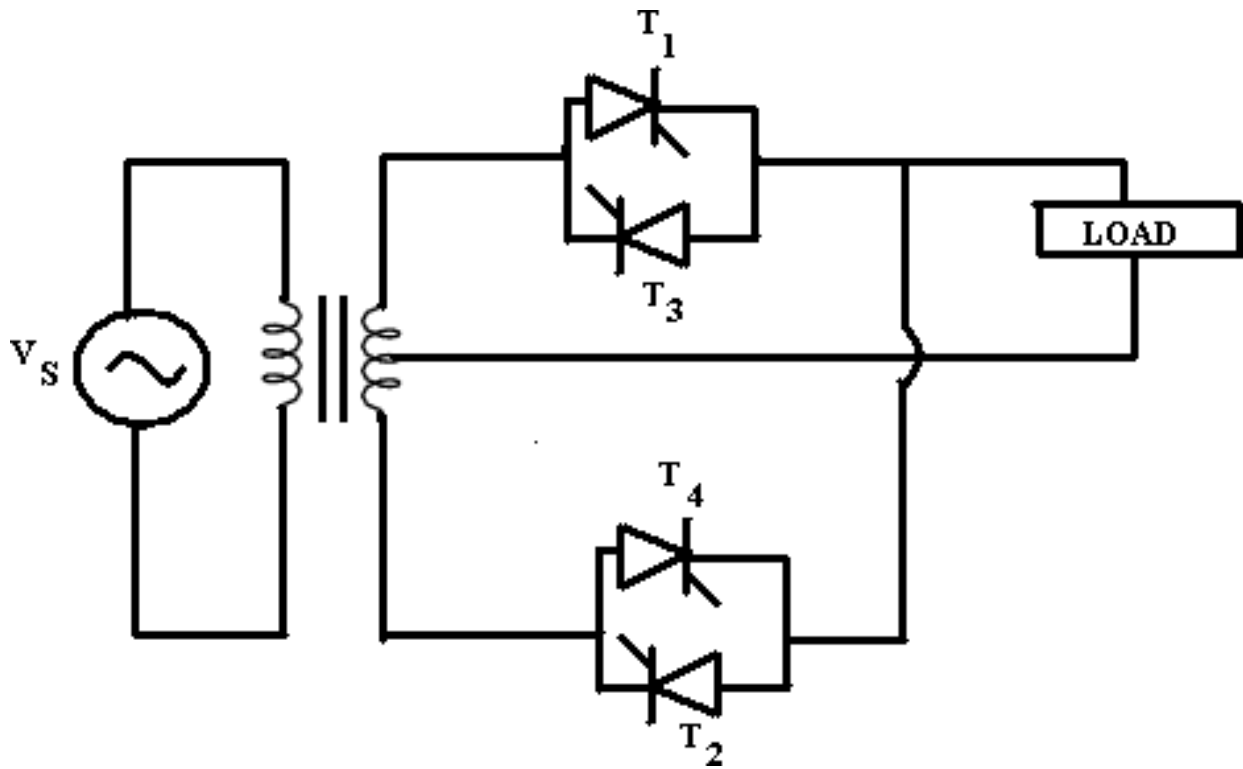
RESULT:

VIVA QUESTIONS:

1. What is parallel inverter? Why is it called so?
2. What is the purpose of capacitor in the parallel inverter?
3. What is the purpose of transformer in the parallel inverter?
4. Is the parallel inverter naturally commutated or force commutated?
5. What are the advantages of parallel resonant inverters?



Circuit Diagram:



EXPERIMENT NO: 06

DATE:

SINGLE PHASE CYCLOCONVERTER WITH R & RL LOADS

Aim:

To study the operation of Single Phase Cycloconverter using R Load for different firing angles and frequency divisions.

Apparatus:

S.No.	Apparatus	Range	Quantity
1	Single Phase Cycloconverter Kit	---	1
2	Single Phase Cycloconverter Firing Unit	---	1
3	Single Phase Isolation Transformer (Center Tap Type)	Primary: 230V, Secondary: (0-30-60-115-230)V	1
4	Rheostat	150Ω/5A(WW)	1
5	CRO	(0-30)MHz	1
6	BNC Adaptors	---	1
7	Patch Cords	---	Required some

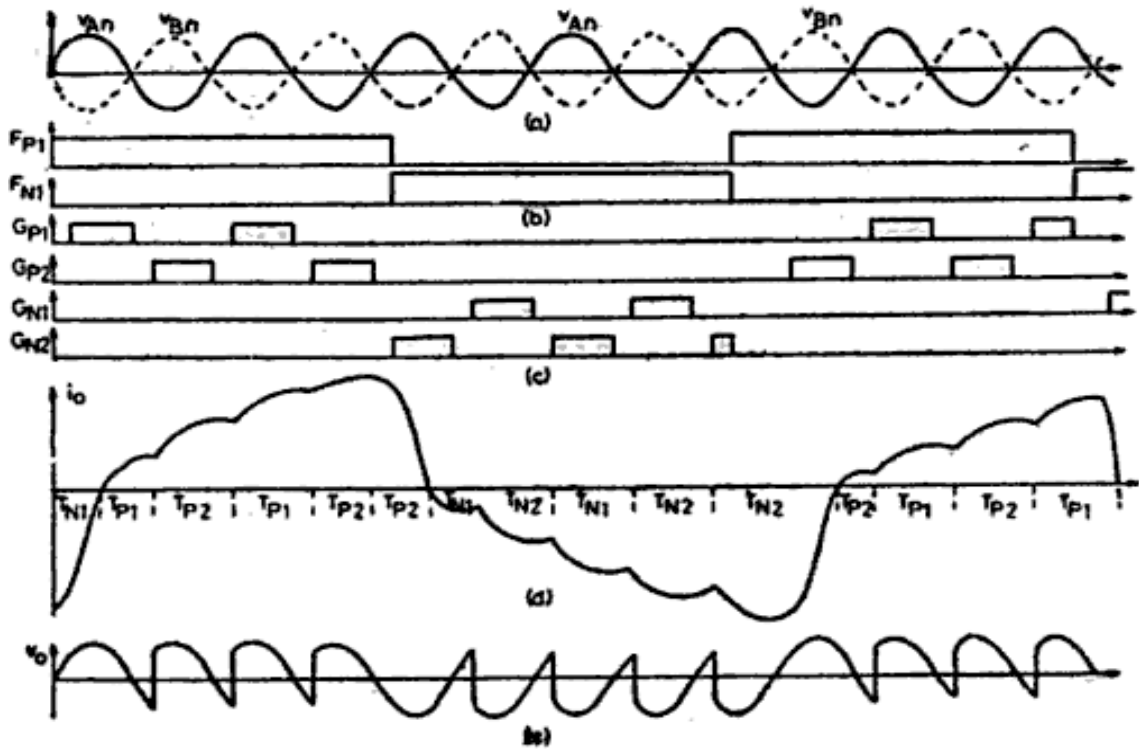
Precautions:

- 1) Connections should be right and tight.
- 2) Keep the potentiometers in the minimum firing angle position at the beginning of the experiment.
- 3) Keep the rheostat in maximum resistance position.
- 4) To reset the kit switch OFF all the main supply switches.

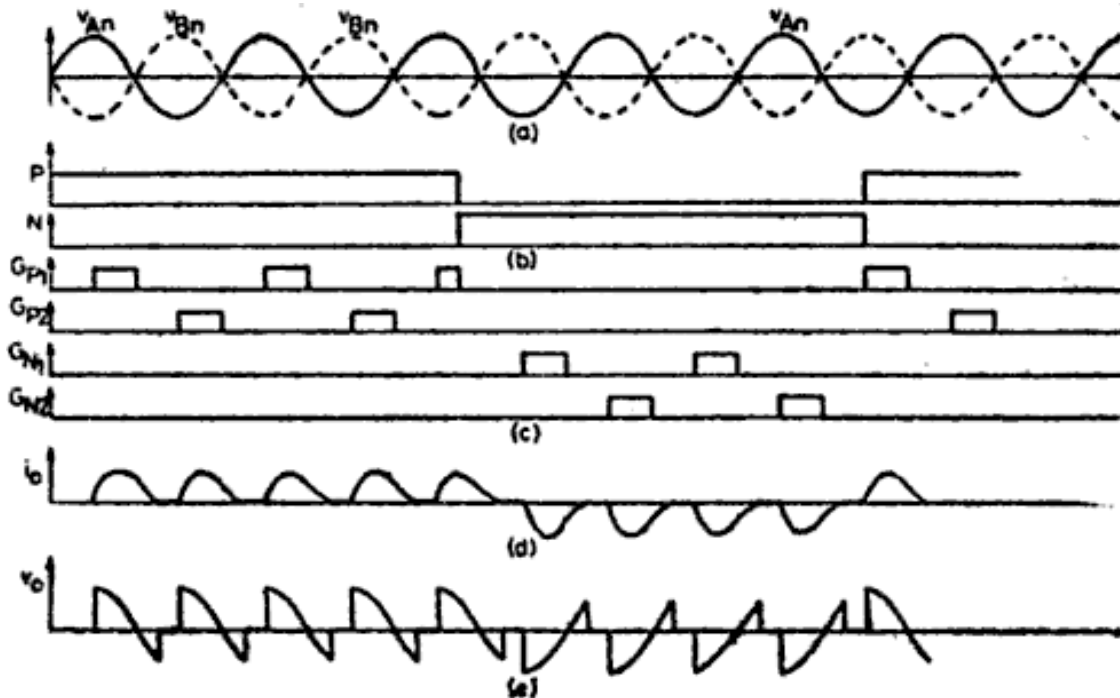
Procedure:

- 1) Connect the circuit as shown in circuit diagram.
- 2) Connect 30V -0 – 30V tapping from the center tapped transformer.
- 3) Follow the precautions and switch ON the supply to the firing unit and kit.
- 4) Switch ON the MCBswitches ON the trigger outputs. Set the frequency division to 2 and vary the firing angles and observe the waveforms.
- 5) Compare the practical values of output voltage with theoretical values and tabulate them.
- 6) Plot the graphs for input voltage, output voltage, gate pulses and voltage across the thyristors for different values of firing angle.
- 7) Repeat the same procedure for RL load.
- 8) Follow the precautions and switch OFF the supply.

Model Waveforms:



Waveforms of 1- Φ cycloconverter with continuous load current; (a) Supply Voltage, (b) Blanking Signals, (c) Trigger pulses to thyristors, (d) Load Current, (e) Load Voltage.



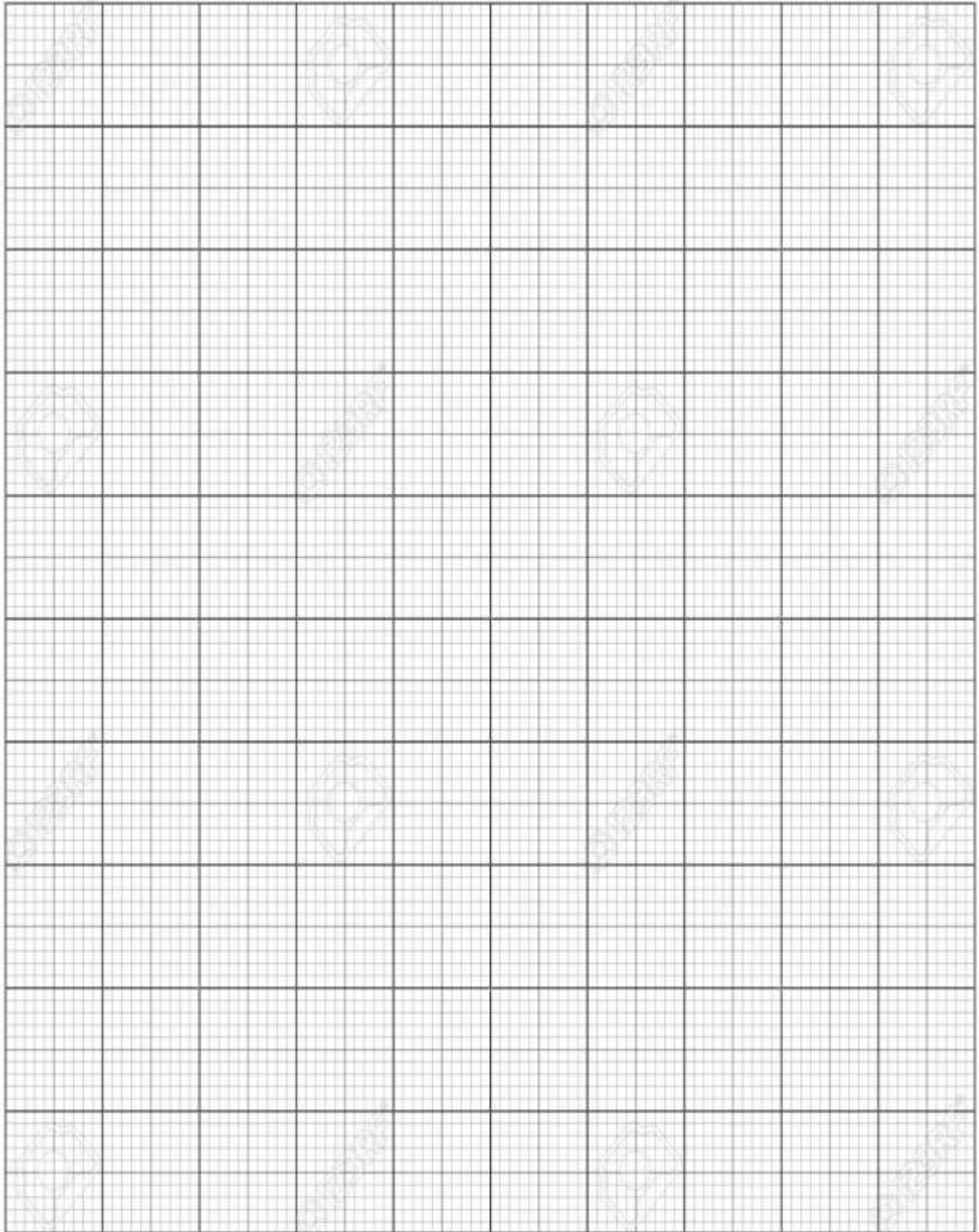
Waveforms of 1- Φ cycloconverter with discontinuous load current; (a) Supply Voltage, (b) Blanking Signals, (c) Trigger pulses to thyristors, (d) Load Current, (e) Load Voltage.

Calculations:

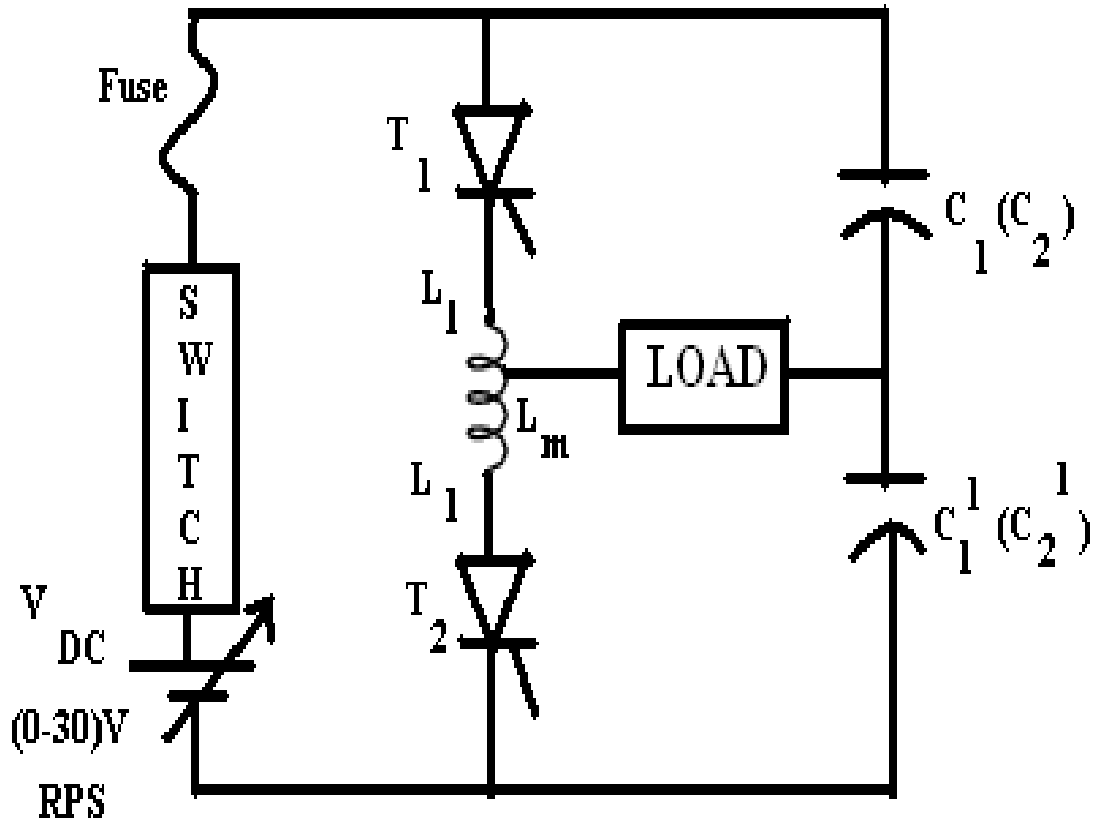
Tabular Column:

S.No	V _{in} (Volts)	Frequency Division	Firing Angle	V _o volts	I _o Amps.
1					
2					
3					
4					

Result:



Circuit Diagram:



Specifications:

$L_2 - L_1 - L_m - L_1 - L_2 : 10\text{mH} - 5\text{mH} - 0 - 5\text{mH} - 10\text{mH} / 2\text{A}$.
 $C_1 \& C_1^1 : 6.8 \mu\text{F} / 100 \text{V}$
 $C_2 \& C_2^1 : 10 \mu\text{F} / 100 \text{V}$

EXPERIMENT NO:07DATE:

SINGLE PHASE SERIES INVERTER WITH R & RL LOADS

Aim:

To study the operation of Single Phase Series Inverter using R Load.

Apparatus:

S.No.	Apparatus	Range	Quantity
1	Single Phase Series Inverter Kit	---	1
2	Single Phase Series Inverter Firing Unit	---	1
3	R.P.S	0-30 V / 2A	1
4	Rheostat	50Ω/2A(WW)	1
5	CRO	(0-30)MHz	1
6	BNC Adaptors	---	1
7	Patch Cords	---	Required some

Precautions:

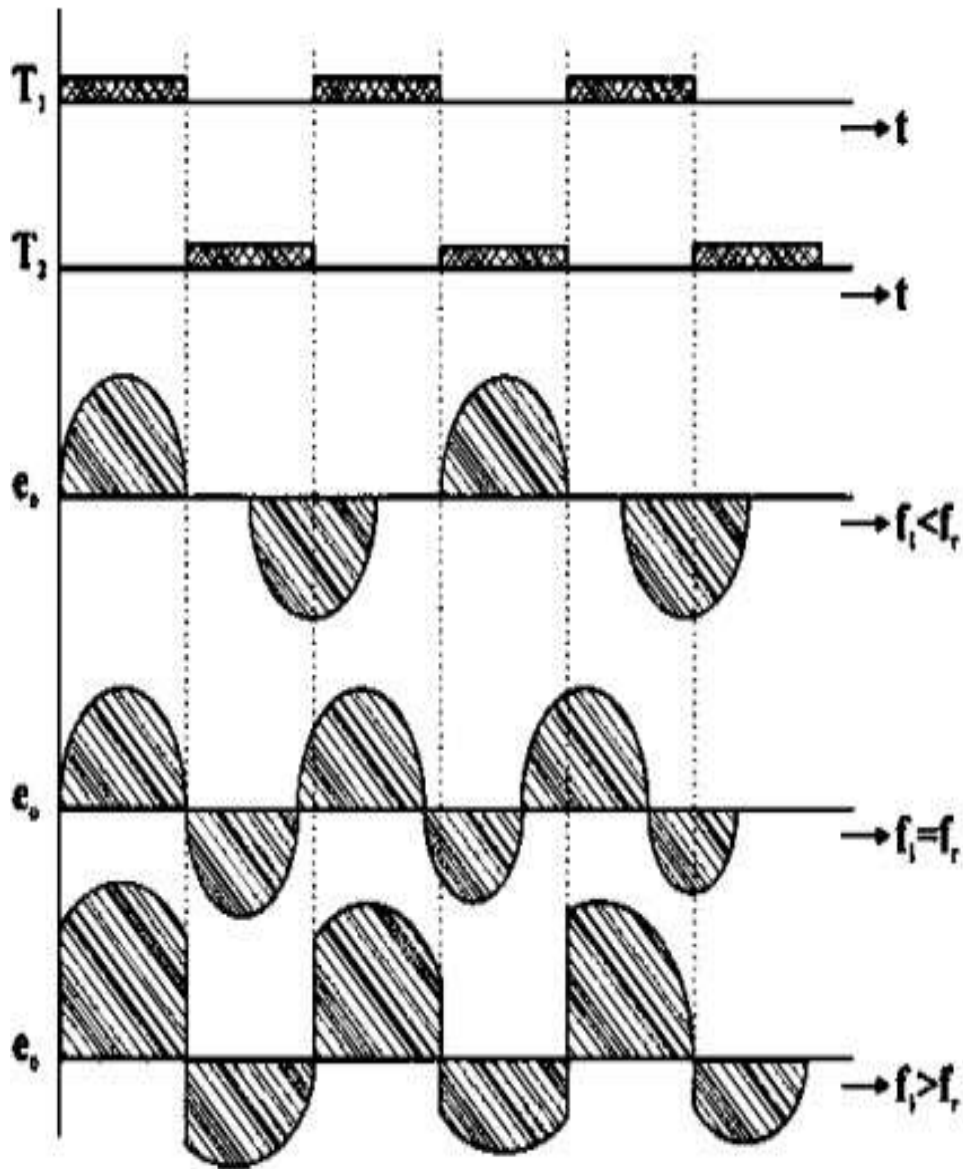
- 1) Connections should be right and tight.
- 2) To reset the kit switch OFF all the main supply switches.
- 3) To switch OFF the inverter, switch OFF supply first and then trigger pulses.

Procedure:

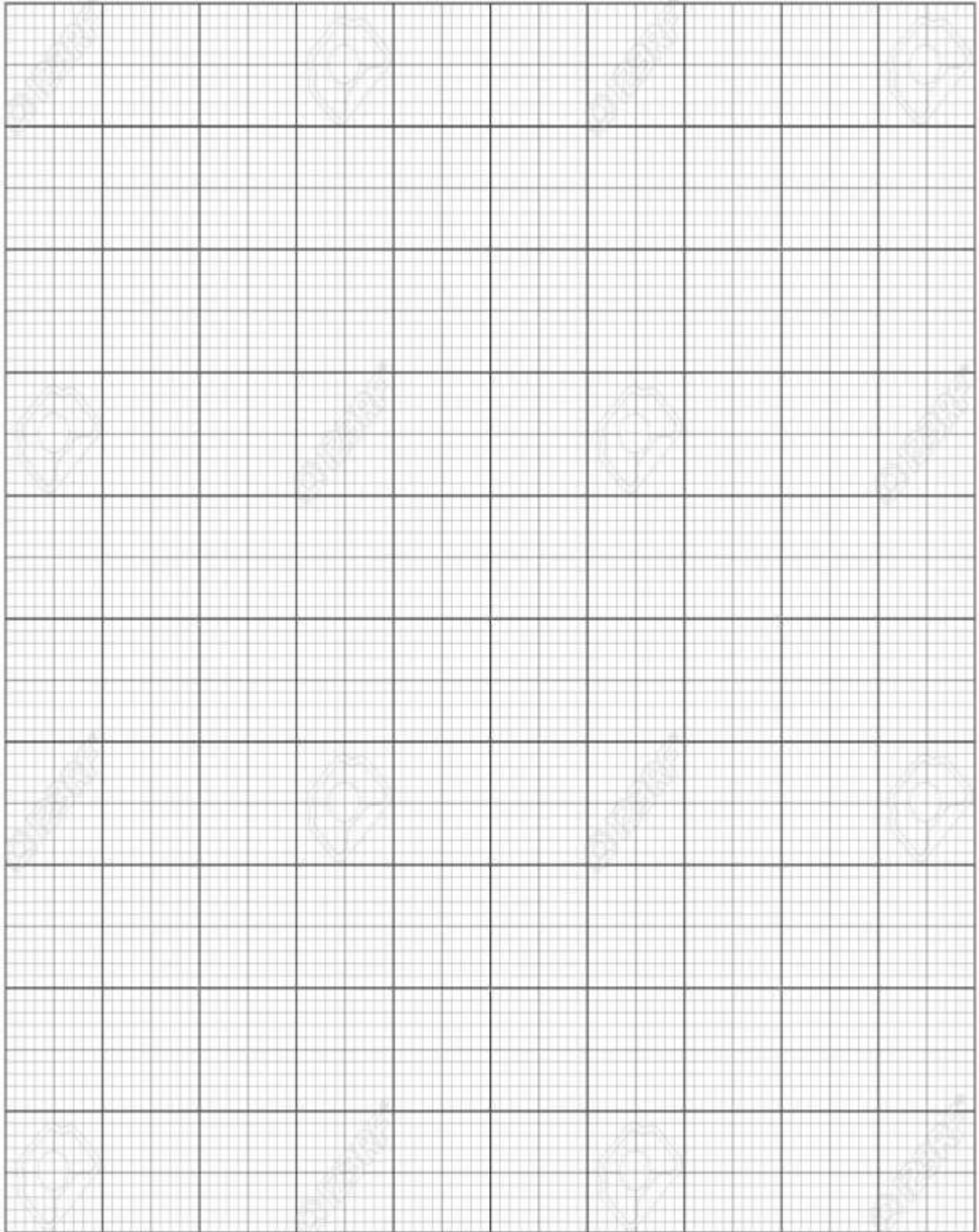
- 1) Connect the circuit as shown in circuit diagram.
- 2) Switch ON the DC power supply And apply trigger pulses to the SCR's.
- 3) Vary the frequency and observe the waveforms.
- 4) If the inverter frequency is above the resonant frequency of the power circuit commutation will fail.
- 5) Repeat the same with different values of L, C and load.
- 6) Observe the waveforms with and without fly wheel diodes.
- 7) Follow the precautions and switch OFF the supply.

Calculations:

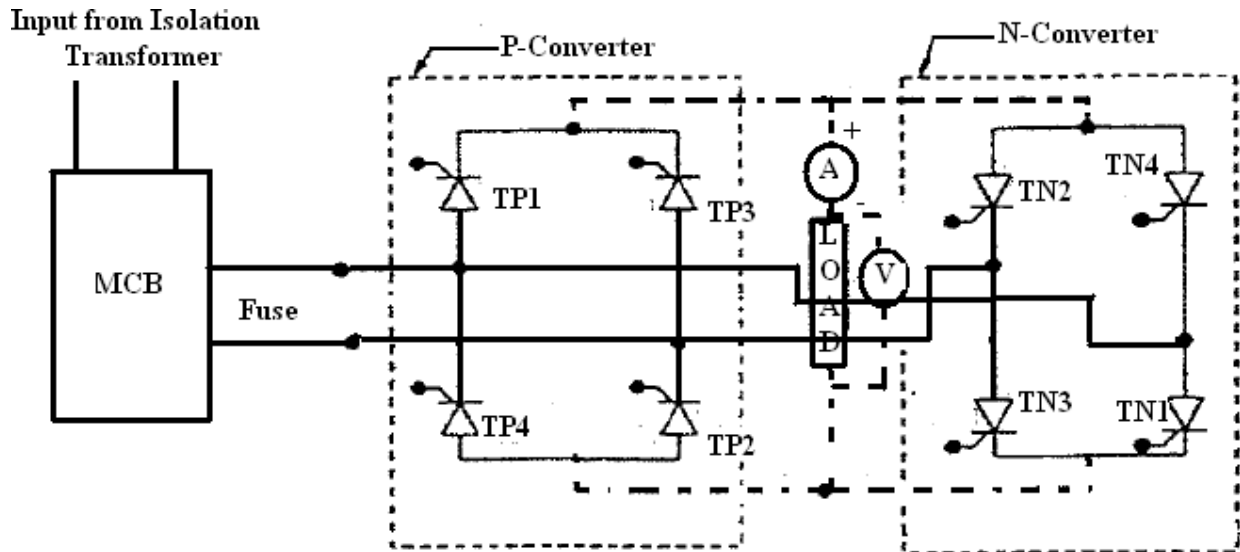
Model Graph:



Result:



Circuit Diagram:



EXPERIMENT NO: 08 DATE:

SINGLE PHASE DUAL CONVERTER WITH R & RL LOADS

Aim:

To study Single Phase Dual Converter.

Apparatus:

S.No.	Apparatus	Range	Quantity
1	Single Phase Dual Converter Firing Unit (Microcontroller Based)	---	1
2	Single Phase Dual Converter Kit	---	1
3	Single Phase Isolation Transformer	Primary: 230V, Secondary: (0-30-60-115-230)V	1
4	Rheostat	150Ω/5A(WW)	1
5	CRO	(0-30)MHz	1
6	BNC Adaptors	---	1
7	Patch Cords	---	Some

Precautions:

- 1) Make fresh connections before you make a new experiment.
- 2) Preferably work at low voltages (20-30V) for every new connection after careful verification it can be raised to the maximum ratings. (This is to reduce damages due to wrong connections and high starting current problems)

Procedure:

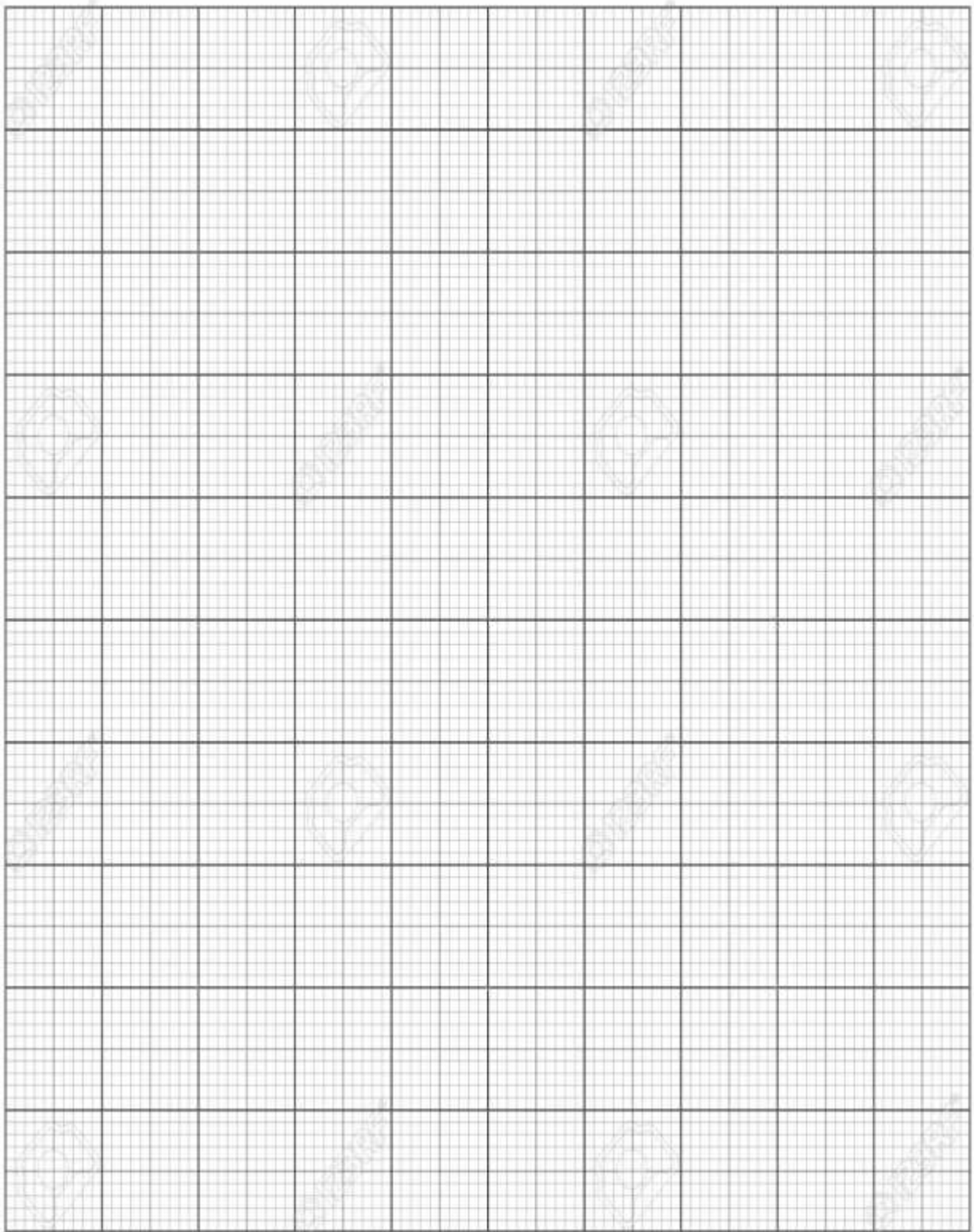
- 1) Switch ON the single phase dual converter firing converter circuit. Observe the test points.
- 2) Observe trigger outputs both in P and N-converter by varying firing angle in non-circulatory mode.
- 3) Make sure that all the pulses are proper before connecting to power circuit to conduct the experiment.
- 4) Check all the SCRs in the power circuit. This can be done by checking the resistance between gate and cathode and also between anode and cathode using a multi meter putting it in the diode range.
- 5) If the SCRs are good, it shows some resistance value of the order of 20Ω to 200Ω between gate and cathode. And very high resistance value of the few mega ohms between anode and cathode. If the resistance between gate / cathode is zero or very high means the device is faulty. If the resistance between anode / cathode is zero means the device is faulty.
- 6) Make sure that all the SCRs are good before connecting the firing pulses from the firing circuit.
- 7) Make the connections in the power circuit as shown in the circuit diagram for non circulatory current mode.
- 8) Connect firing pulses from the firing circuit to the respective SCR's gate / cathode terminals in the power circuit.
- 9) Connect input AC supply to the power circuit through an isolation transformer for safety for measurement. Initially adjust the input voltage for 30V.
- 10) Connect a rheostat of suitable value (150Ω/5A) between output terminals. Connect the ammeter and voltmeter as shown.
- 11) Switch ON the firing circuit. Select NCC mode. Switch ON the MCB.
- 12) Vary the firing angle by dec key and press ON / OFF key to ON and observe the voltage waveforms across load and the devices.
- 13) Note down the voltmeter and ammeter readings for different values of firing angle. Note down the reading in the tabular column.

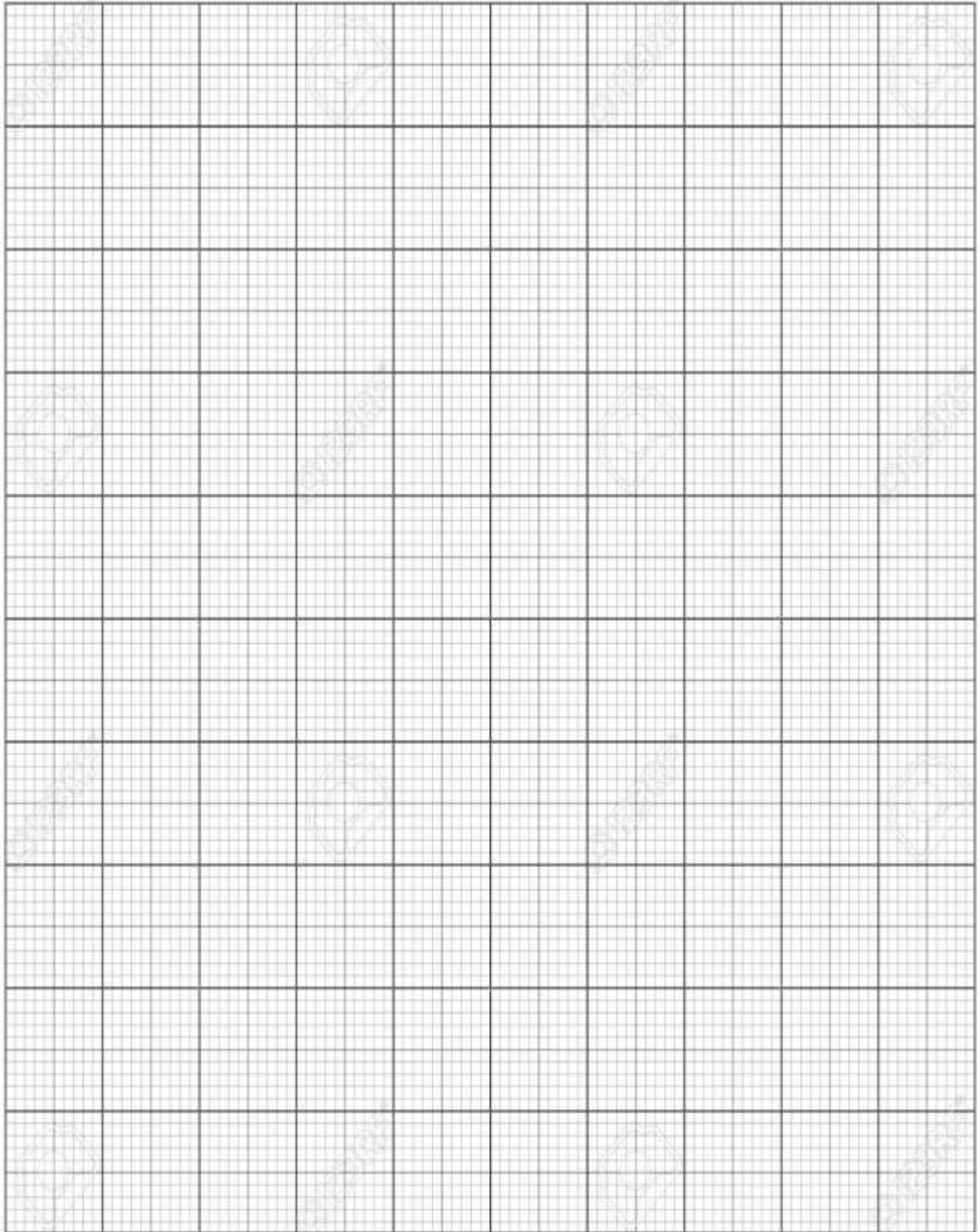
Tabular Columns:

S. No.	Input Voltage V_{in}	Firing Angle	Output Voltage V_o	Output Current I_o
1				
2				
3				
4				

Calculations:

Result:





EXPERIMENT NO: 09(A) DATE:

PSPICE SIMULATION OF SINGLE PHASE FULL WAVE RECTIFIER USING RLE LOADS

AIM:

To obtain the performance characteristics of Single Phase full converter for R, RL, RLE Loads Using MATLAB / Simulink

APPARATUS:

S. No.	Name of the Equipment
1.	PC With Desktop
2.	Matlab / Simulink

CIRCUIT DIAGRAM:

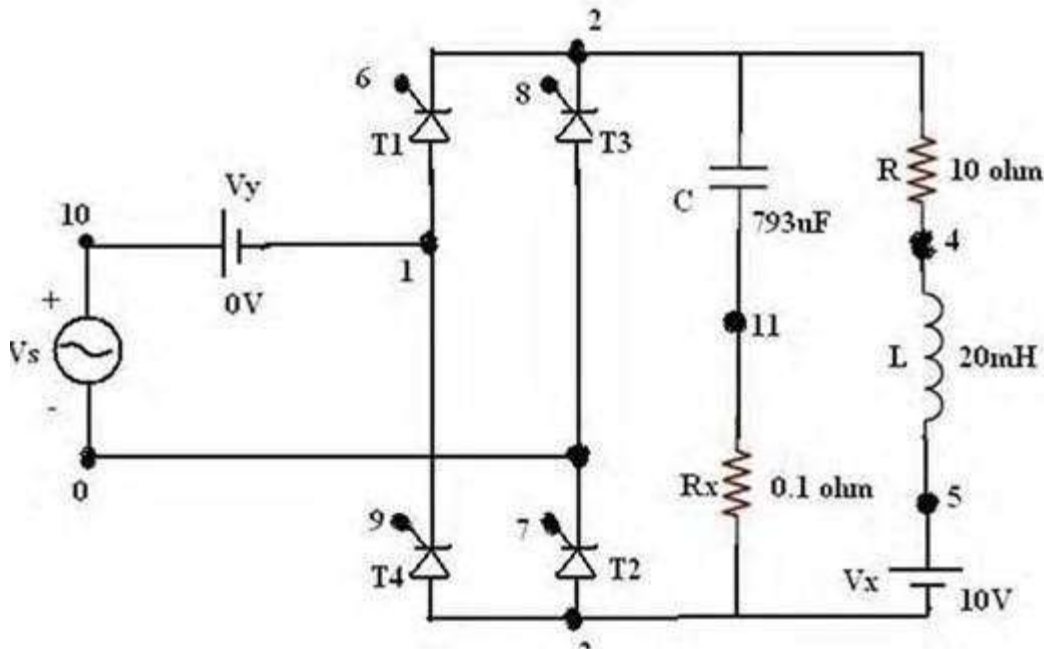


Fig - Circuit Diagram of PSPICE Simulation of Single Phase Full Wave Rectifier

PROCEDURE:

1. Represent the nodes for a given circuit.
2. Write spice program by initializing all the circuit parameter as per given flowchart.
3. From desktop of your computer click on "START" menu followed by "programs" and then clicking appropriate program group as "DESIGN LAB EVAL8 followed by "DESIGN MANAGER".
4. Open the run text editor from microsim window & start writing pspice program.
5. Save the program with .cir extension.
6. Open the run spice A / D window from microsim window.
7. Open file menu from run spice A / D window then open saved circuit file.
8. If there are any errors, simulation will be displayed with statement as "simulation error occurred".
9. To see the errors click on o/p file icon and open examine o / p.
10. To make changes in the program open the circuit file, modify, save & Run the program.
11. If there are no errors, simulation will be completed & it will be displayed with a statement as "simulation completed".
12. To see the o / p click on o / p file icon & open examine o / p then note down the values.
13. If probe command is used in the program, click on o / p file icon & open run probe. Select variables to plot on graphical window and observe the o / p plots then take print outs of that.

PROGRAM CODE:

CLC

VS 10 0 SIN (0 325V 50HZ)

VG1 6 2 PULSE (0V 10V 2500US 1NS 1NS 100US

20000US) VG2 7 0 PULSE (0V 10V 2500US 1NS 1NS

100US 20000US) VG3 8 2 PULSE (0V 10V 12500US

1NS 1NS 100US 20000US) VG4 9 1 PULSE (0V 10V

12500US 1NS 1NS 100US 20000US) R 2 4 10

L 4 5 20MH

VX 5 3 DC 10V

VY 10 1 DC 10V

C 2 11 793UF

```
RX 11 3 0.1
XT1 1 2 6 2SCR
XT2 3 0 7 0SCR
XT3 0 2 8 2SCR
XT4 3 1 9 1SCR
.SUBCKT SCR 1 2 3 2
S1 1 5 6 2 SMOD
RG 3 4 50
VX 4 2 DC0V
VY 5 7 DC0V
DT 7 2DMOD
RT 6 2 1
```

```
CT 6 2 10UF
F1 2 6 POLY (2) VX VY 0 50 11
.MODEL SMOD VSWITCH (RON=0.0105 ROFF=10E+5 VON=0.5V VOFF=0V)
.MODEL DMOD D (IS=2.2E-15 BV=1200V TT=0 CJO=0)
.ENDS SCR
.TRAN 50US 100MS 50MS 50US
.PROBE
.OPTIONS ABSTOL=1.00N RELTOL=1.0M VNTOL=0.1 ITL5=20000
.FOUR 50HZ I(VY)
.END
Plot v (2)
```

MODEL WAVEFORMS:

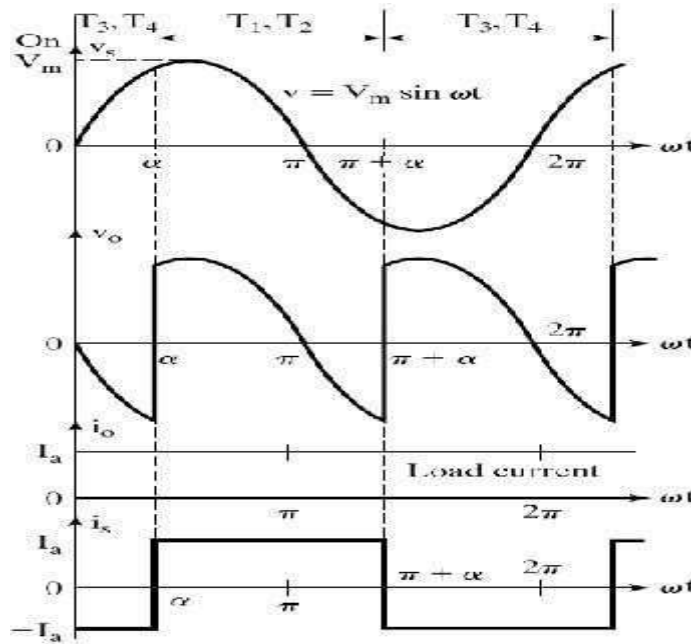


Fig - Output Wave Forms of PSPICE Simulation of Single Phase Full Wave Rectifier

RESULT:

PRE LAB VIVA QUESTIONS:

1. If we place freewheeling diode what happens in full controlled converters?
2. What is the output voltage equation of a 3-phase fully controlled bridge converter?
3. What are the parameters that affect the frequency of the ripple in the output voltage of 3-phase semiconverter?

POST LAB VIVA QUESTIONS:

1. Explain operation of rectifier with RL load.
2. How to plot rectifier output waveforms?

Expt.No: 9 (B)

Date:

PSPICE SIMULATION OF SINGLE PHASE AC VOLTAGE CONTROLLER USING RLE LOADS

AIM:

To obtain the performance characteristics of Single Phase for R, RL, RLE Loads Using MATLAB / Simulink

APPARATUS:

S. No.	Name of the Equipment
1.	PC With Desktop
2.	MATLAB / Simulink

CIRCUITDIAGRAM:

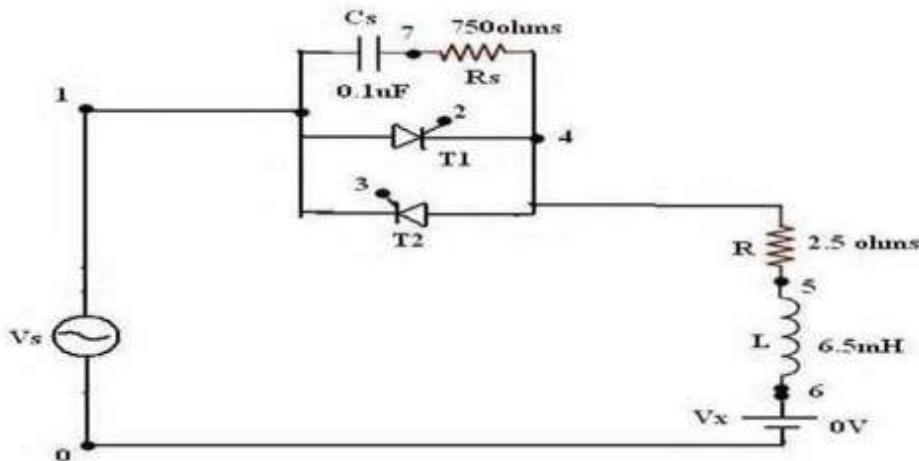


Fig - Circuit Diagram of PSPICE Simulation of Single Phase Ac Voltage Controller

PROCEDURE:

1. Represent the nodes for again circuit.
2. Write PSPICE program by initializing all the circuit parameters as per given flow chart 3.From desktop of your computer click “start” menu followed “PROGRAMS” & then clicking appropriate program group as “DESIGN LAB tv 218” followed by designmanager.
3. Open the Run text editor from microsim window & start writing PSPICEprogram.
4. Save the program with .cir extension. (Ex:DA.cir).
5. Open the RUN SPICE A / D window from microsimwindow.
6. Open file menu from RUN SPICE A / D window then open saved circuitfile.

7. If there are any errors, simulation will be displayed with statement as “simulation error occurred.” To see the errors click on output file icon & open examine output.
8. To make changes in the program open the circuit file modifies & run the program.
9. If there are no errors simulation modifies be displayed with a statement as “simulation completed”.
10. To see the output click on the output file icon & open examine output then note down the values.
11. If probe command is used in the program click on output file icon & open Run probe select variable to plot on graphical window & observe the plots then the printouts of that.

PROGRAM CODE:

CLC

VS 1 0 SIN (0 325V 50HZ)

VG1 2 4 PULSE (0V 10V 2500US 1NS 1NS 100US

20000US) VG2 3 1 PULSE (0V 10V 2500US 1NS 1NS

100US 20000US) R 4 5 2.5

L 5 6 6.5MH

VX 6 0 DC 10V

XT1 1 4 2 4SCR

XT2 4 1 3 1SCR

.SUBCKT SCR 1 2 3 2

S1 1 5 6 2 SMOD

RG 3 4 50

VX 4 2 DC 0V

VY 5 7 DC 0V

DT 7 2 DMOD

RT 6 2 1

CT 6 2 100F

F1 2 6 POLY (2) VX VY 0 50 11

.MODEL SMOD VSWITCH (RON=0.0105 ROFF=10E+5 VON=0.5V VOFF=0V)

.MODEL DMOD D (IS=2.2E-15 BV=1200V TT=0 CJO=0)

.ENDS SCR

.TRAN 50US 100MS 50MS 50US

.PROBE

.FOUR 50HZ I(VX)

.END

PLOT V(2)

MODELWAVEFORMS:

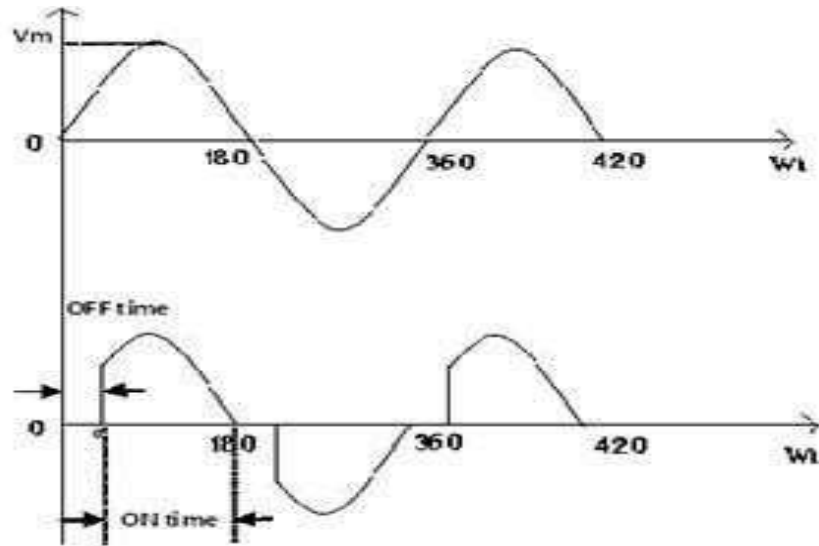


Fig –Output Wave Forms of PSPICE Simulation of Single Phase Ac Voltage Controller

POST LAB VIVAQUESTIONS:

1. What are the applications of fully controlled rectifiers?
2. What are the advantages of fully controlled over half controlled rectifiers?

RESULT:

Expt.No: 10

Date:

PSPICE SIMULATION OF RESONANT PULSE COMMUTATION CIRCUIT

AIM:

To obtain the performance characteristics of a Resonant Pulse Commutation Circuit

APPARATUS:

S. No.	Name of the Equipment
1.	PC With Desktop
2.	MATLAB / Simulink

CIRCUIT DIAGRAM:

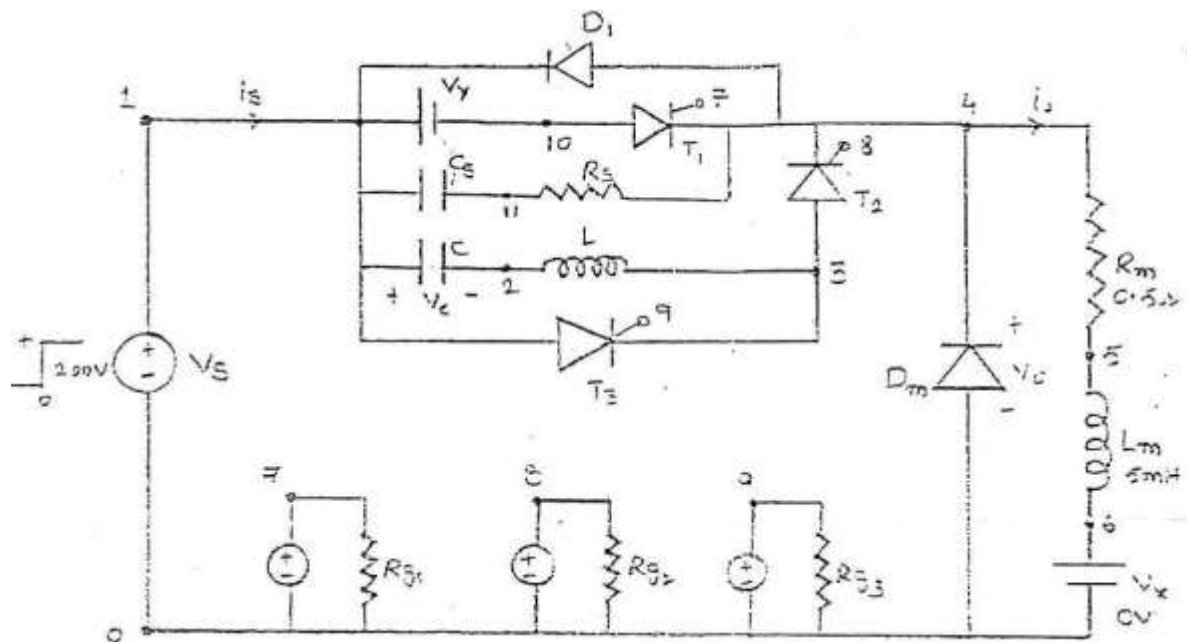


Fig - Circuit Diagram of PSPICE Simulation of Resonant Pulse Commutation Circuit

PROCEDURE:

1. Represent the nodes for a given circuit.
2. Write spice program by initializing all the circuit parameter as per given flowchart.
3. From desktop of your computer click on "START" menu followed by "programs" and then clicking appropriate program group as "DESIGN LAB EVAL8 followed by

“DESIGN MANAGER.”

4. Open the run text editor from microsim window & start writing pspiceprogram.
5. Save the program with .circuitextension.
6. Open the run spice A / D window from microsimwindow.
7. Open file menu from run spice A / D window then open saved circuitfile.
8. If there are any errors, simulates will be displayed with statement as “simulation error occurred”.
9. To see the errors click on o/p file icon and open examine o /p.
10. To make changes in the program open the circuit file, modify, save & Run theprogram.
11. If there are no errors, simulation will be completed & it will be displayed with a statement as “simulationcompleted”.
12. To see the o / p click on o / p file icon & open examine o / p then note down thevalues.
13. If .probe command is used in the program, click on o / p file icon &open run probe. Select variables to plot on graphical window and observe the o / p plots then take print outs ofthat.

PROGRAM CODE:

```
CLC
VS 1 0 DC 200V
VG1 7 0 PULSE (0V 100V 0 1US 1US 0.4MS 1MS)
VG2 8 0 PULSE (0V 100V 0.4MS 1US 1US 0.6MS
1MS) VG3 9 0 PULSE (0V 100V 0.1US 1US 1US
0.2MS 1MS) RG1 7 0 10MEG
RG2 8 0 10MEG
RG3 9 0 10MEG
CS 10 11 0.1UF
RS 11 4 750
C 1 2 31.2UF IC=200V
L 2 3 6.4UH
D1 4 1 DMOD
DM 0 4 DMOD
.MODEL DMOD D(IS=1E-25
BV=1000V) RM 4 5 0.5
LM 5 6 5MH
```

```
VX 6 0 DC 0V
VY 1 10 DC 0V
XT1 10 4 7 0 DCSCR
XT2 3 4 8 0DCSCR
XT3 1 3 9 0DCSCR

SUBCKT DCSCR 1 2 3 4
DT 5 2 DMOD
ST 1 5 3 4 SMOD
.MODEL DMOD D (IS=1E-25 BV=1000V)
.MODEL SMOD VSWITCH (RON=0.1 ROFF=10E+6 VON=10 VOFF=5V)
.ENDS DCSCR
.TRAN 0.5US 3MS 1.5MS 0.5US
.PROBE
.END

PLOT I (C ) AND V(2)
```

RESULT:

PRE LAB VIVAQUESTIONS:

1. What is mean byCommutation?
2. What is mean by ResonantCommutation?
3. Explain Resonant Pulse Commutation.

POST LAB VIVAQUESTIONS:

1. What are the applications of Resonant pulseCommutation?

Expt.No: 11

Date:

PSPICE SIMULATION OF SINGLE PHASE INVERTER WITH PWM CONTROL

AIM:

To obtain the performance characteristics of single phase inverter with PWM control.

APPARATUS:

S. No	Name of the Equipment
1.	PC With Desktop
2.	PSPICE

CIRCUIT DIAGRAM:

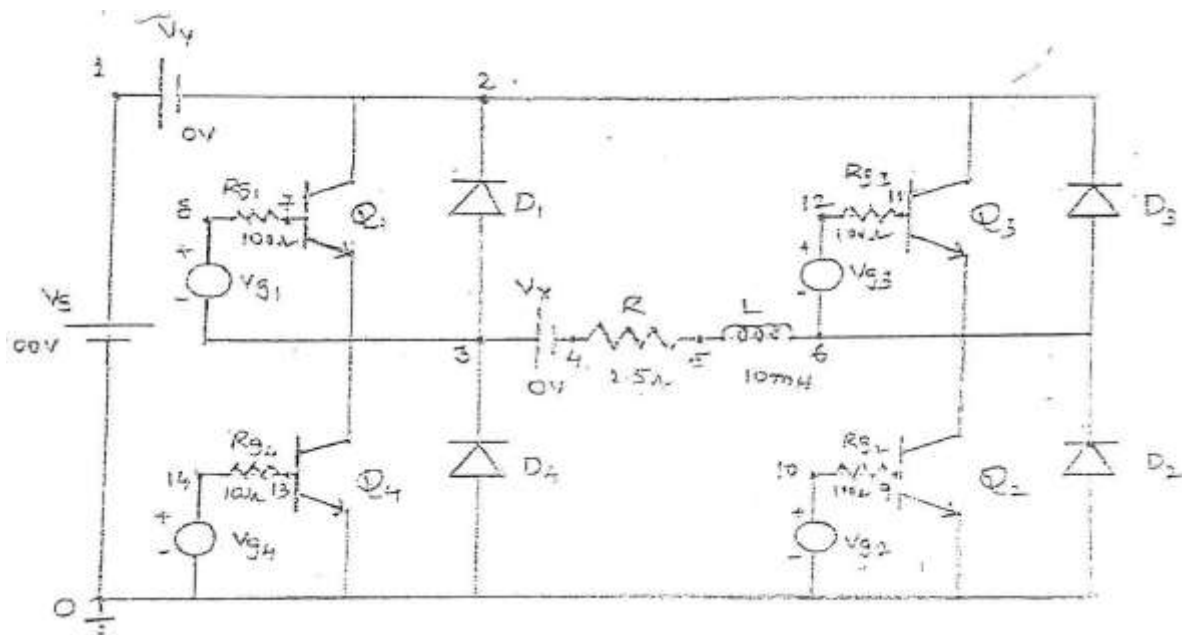


Fig - Circuit Diagram of PSPICE Simulation of Single Phase Inverter

PROCEDURE:

1. Represent the nodes for a given circuit.
2. Write spice program by initializing all the circuit parameter as per given flowchart.
3. From desktop of your computer click on "START" menu followed by "programs" and then clicking appropriate program group as "DESIGN LAB EVAL8 followed by "DESIGN MANAGER."
4. Open the run text editor from microsim window & start writing PSPICE program.

5. Save the program with .cir extension.
6. Open the run spice A / D window from microsimwindow.
7. Open file menu from run spice A / D window then open saved circuitfile.
8. If there are any errors, simulates will be displayed with statement as “simulation error occurred”.
9. To see the errors click on o / p file icon and open examineo/p.
10. To make changes in the program open the circuit file, modify, save & Run theprogram.
11. If there are no errors, simulation will be completed & it will be displayed with a statement as “simulationcompleted”.
12. To see the o / p click on o / p file icon & open examine o / p then note down thevalues.
13. If .probe command is used in the program, click on o / p file icon &open run probe. Select variables to plot on graphical window and observe the o / p plots then take print outs ofthat.

PROGRAMCODE:

```
VS 1 0 DC 100V
VT 17 0 PULSE (50V 0V 0 833.33US 833.33US 1NS
1666.67US) RT 17 0 2MEG
VC1 15 0 PULSE (0 -30V 1NS 1NS 8333.33US
1666.67US) RC1 15 0 2MEG
VC3 16 0 PULSE (0 -30V 8333.33US 1NS 1NS 8333.33US
16666.67US) RC3 16 0 2MEG
R 4 5 2.5
L 5 6 10MH
VX 3 4 DC0V
VY 1 2 DC0V
D1 3 2 DMOD
D2 0 6 DMOD
D3 6 2 DMOD
D4 0 3 DMOD
.MODEL DMOD D(IS=2.2E-15 BV=1800V
TT=0) Q1 2 7 3 QMOD
Q2 6 9 0 QMOD
```

```
Q3 2 11 6 QMOD
Q4 3 13 0 QMOD
.MODEL QMOD NPN(IS=6.74F BF=416.5 CJC=3.638P
CJE=4.451P) RG1 8 7 100
RG2 10 9 100
RG3 12 11100
RG4 14 13100
*SUBCKT CALL FOR PWM
CONTROL XPW1 17 15 8 3 PWM
XPW2 17 15 10 0PWM
XPW3 17 16 12 6PWM
XPW4 17 16 14 0PWM
.SUBCKT PWM 1 2 34
*model ref carrier +control -
control R1 1 51K
R2 2 51K
RIN 5 0 2MEG
RF 5 3 100K
R0 6 3 75
C0 3 4 10PF
E1 6 4 0 5 2E+5
.ENDS PWM
.TRAN 10US 16.67MS 0 10US
.PROBE
.options abstol=1.00n reltol=0.01 vntol=0.1 itl5=20000
.FOUR 60HZ V(3,6)
.END
PLOT V (14) I(VX) I (vy)V(10)
```

RESULT:

PRE LAB VIVAQUESTIONS:

1. What is the function of inverter?
2. What is meant by carrier wave form in PWM control?
3. What are the control strategies for inverter?

POST LAB VIVAQUESTIONS:

1. What are the advantages of PWM inverter?
2. What are the applications of PWM inverter?
3. What are the advantages of simulation using PSPICE?

WEBENCH® Design Center

WEBENCH Design Environments are unique and powerful software tools that deliver customized power, lighting, filtering, clocking and sensing designs in seconds. These easy-to-use tools help you generate, optimize and simulate designs that conform to your unique specifications. They allow you to make value-based tradeoffs at a design, system and supply chain level before your design is committed to production.

Overview

The WEBENCH® Power Designer is an end to end system that produces power supply solutions for a wide range of power supply needs. The flow for the tool is broken down into 4 steps.

SELECT: Enter you simple specifications. WEBENCH® Power Designer will generate many designs that meet these requirements. You refine the choices presented using the Advanced Options, Feature filters, Visualizer, and Optimizer Knob. You select one device that you would like to explore further.

DESIGN: Once a device is selected, the Dash board opens and allows you to look at the design a little more closely using the Schematic window, BOM, OpVals, Charts, and Optimizer Knob. You can further customize the design to meet your specific requirements.

ANALYZE: Now you are ready to fully exercise your solution to verify the performance using Electrical Simulation, Thermal simulation, Re-Compensation, and the Schematic Editor. You are able to verify the performance and continue to update the design to meet all of your requirements.

EXPORT: Now that you have a robust solution, you will want to build a prototype and test it in the lab. WEBENCH® Power Designer provides Schematic Export, PCB export, Simulation export, complete documentation and Build-It for you to get the prototype ready for production.

EXPERIMENT NO: 12 DATE:

STUDY OF DC TO DC BUCK CONVERTER TPS54160**Aim:**

- To investigate the efficiency of a TPS54160 buck regulator depends on line and Load conditions and on the switching frequency.
- To Analyze the influence of switching frequency f_s and capacitance and resistance of the input and out capacitors on steady state waveforms of TPS54160 buck regulator.

Apparatus:

Webench software Design Tool. Source:

Specifications:

		VALUE		UNIT
		MIN	MAX	
Input voltage	VIN	-0.3	65	V
	EN ⁽²⁾	-0.3	5	
	BOOT		73	
	VSENSE	-0.3	3	
	COMP	-0.3	3	
	PWRGD	-0.3	6	
	SS/TR	-0.3	3	
	RT/CLK	-0.3	3.6	
Output voltage	BOOT-PH		8	V
	PH	-0.6	65	
	PH, 10-ns Transient	-2	65	
Voltage Difference	PAD to GND		±200	mV
Source current	EN		100	μA
	BOOT		100	mA
	VSENSE		10	μA
	PH	Current Limit		A
	RT/CLK		100	μA
Sink current	VIN	Current Limit		A
	COMP		100	μA
	PWRGD		10	mA
	SS/TR		200	μA
Operating junction temperature		-40	150	°C

Theory:**Overview**

The TPS54160A device is a 60-V, 1.5-A, step-down (buck) regulator with an integrated high-side n-channel MOSFET. To improve performance during line and load transients the device implements a constant frequency, current mode control which reduces output capacitance and simplifies external frequency compensation design. The wide switching frequency of 100kHz to 2500kHz allows for efficiency and size optimization when selecting the output filter components. The switching frequency is adjusted using a resistor to ground on the RT/CLK pin.

The device has an internal phase lock loop (PLL) on the RT/CLK pin that is used to synchronize the power switch turn on to a falling edge of an external system clock.

The TPS54160A has a default start up voltage of approximately 2.5V. The EN pin has an internal pull-up current source that can be used to adjust the input voltage under voltage lockout (UVLO) threshold with two external resistors. In addition, the pull up current provides a default condition. When the EN pin is floating the

device operates. The operating current is 116 μA when not switching and under no load. When the device is disabled, the supply current is 1.3 μA .

The integrated 200m Ω high-side MOSFET allows for high efficiency power supply designs capable of delivering 1.5 A of continuous current to a load. The TPS54160A reduces the external component count by integrating the boot recharge diode. The bias voltage for the integrated high-side MOSFET is supplied by a capacitor on the BOOT to PH pin. The boot capacitor voltage is monitored by an UVLO circuit and turns on the high-side MOSFET off when the boot voltage falls below a preset threshold. The TPS54160A can operate at high duty cycles because of the boot UVLO. The output voltage can be stepped down to as low as the 0.8V reference.

The TPS54160A has a power good comparator (PWRGD) which asserts when the regulated output voltage is less than 92% or greater than 109% of the nominal output voltage. The PWRGD pin is an open drain output which deasserts when the VSENSE pin voltage is between 94% and 107% of the nominal output voltage allowing the pin to transition high when a pull-up resistor is used. The TPS54160A minimizes excessive output overvoltage (OV) transients by taking advantage of the OV power good comparator. When the OV comparator is activated, the high-side MOSFET is turned off and masked from turning on until the output voltage is lower than 107%. The SS/TR (slow start/tracking) pin is used to minimize inrush currents or provide power supply sequencing during power up. A small value capacitor should be coupled to the pin to adjust the slow start time. A resistor divider can be coupled to the pin for critical power supply sequencing requirements. The SS/TR pin is discharged before the output powers up. This discharging ensures a repeatable restart after an over-temperature fault, UVLO fault or a disabled condition. The TPS54160A, also, discharges the slow start capacitor during overload conditions with an overload recovery circuit. The overload recovery circuit will slow start the output from the fault voltage to the nominal regulation voltage once a fault condition is removed. A frequency fold back circuit reduces the switching frequency during start-up and over current fault conditions to help control the inductor current.

Specifications:

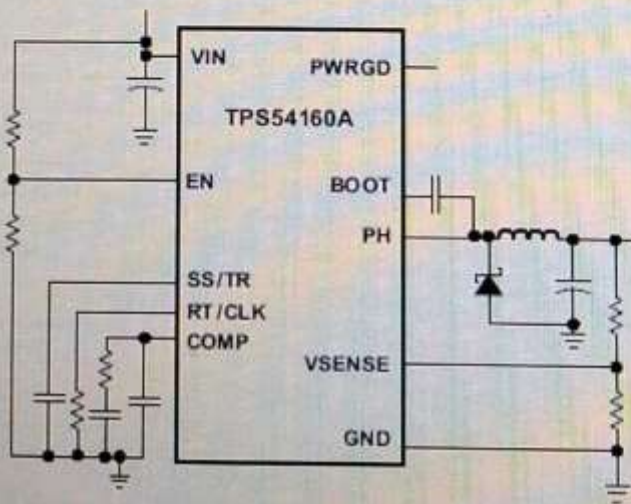
1 Features

- 3.5 V to 60 V Input Voltage Range
- 200-mΩ High-Side MOSFET
- High Efficiency at Light Loads with a Pulse Skipping Eco-mode™
- TPS54160A has Tighter Enable Threshold Than TPS54160 for More Accurate UVLO voltage
- Adjustable UVLO Voltage and Hysteresis
- 116 μA Operating Quiescent Current
- 1.3 μA Shutdown Current
- 100 kHz to 2.5 MHz Switching Frequency
- Synchronizes to External Clock
- Adjustable Slow Start/Sequencing
- UV and OV Power Good Output
- 0.8-V Internal Voltage Reference
- MSOP10 and 3mm x 3mm VSON Package with PowerPAD™
- Supported by WEBBENCH® and SwitcherPro™ Software Tool

2 Applications

- 12-V, 24-V and 48-V Industrial and Commercial Low Power Systems
- Aftermarket Auto Accessories: Video, GPS, Entertainment

4 Simplified Schematic



3 Description

The TPS54160A device is a 60-V, 1.5-A, step down regulator with an integrated high-side MOSFET. Current mode control provides simple external compensation and flexible component selection. A low ripple pulse skip mode reduces the no load, regulated output supply current to 116 μA. Using the enable pin, shutdown supply current is reduced to 1.3 μA.

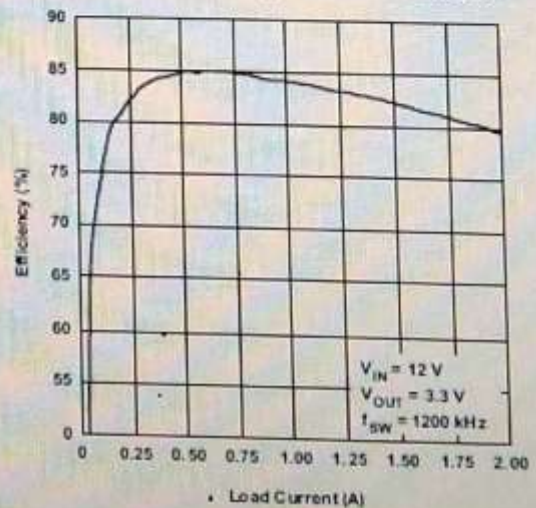
Under voltage lockout is internally set at 2.5 V, but can be increased using the enable pin. The output voltage startup ramp is controlled by the slow start pin that can also be configured for sequencing/tracking. An open drain power good signal indicates the output is within 94% to 107% of its nominal voltage.

A wide switching frequency range allows efficiency and external component size to be optimized. Frequency fold back and thermal shutdown protects the part during an overload condition.

Device Information

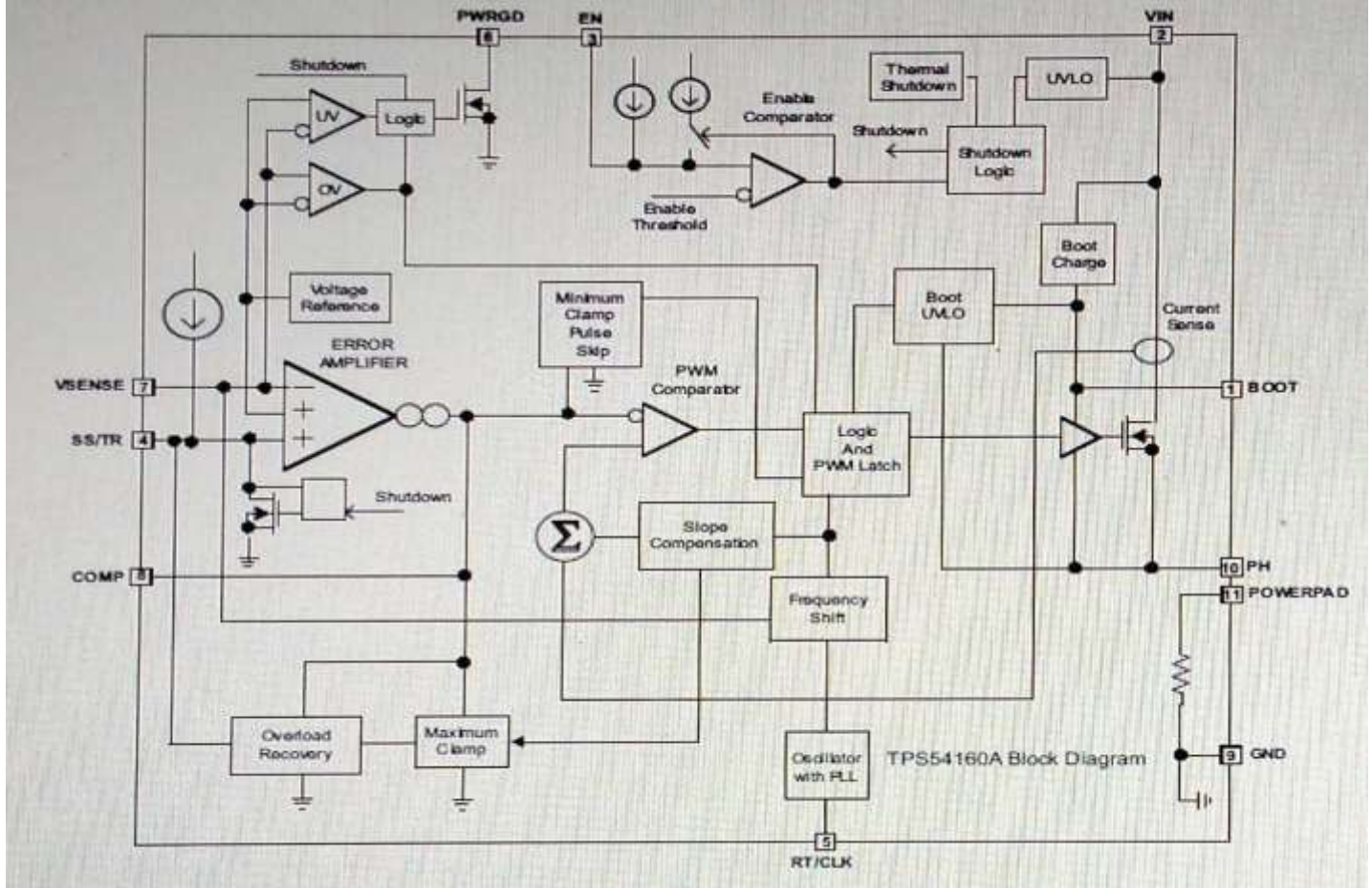
ORDER NUMBER	PACKAGE (PIN)	BODY SIZE
TPS54160DGQ	MSOP (10)	3mm x 3mm
TPS54160ADGQ		
TPS54160ADRC	VSON (10)	3mm x 3mm

Figure 1. Efficiency vs Load Current

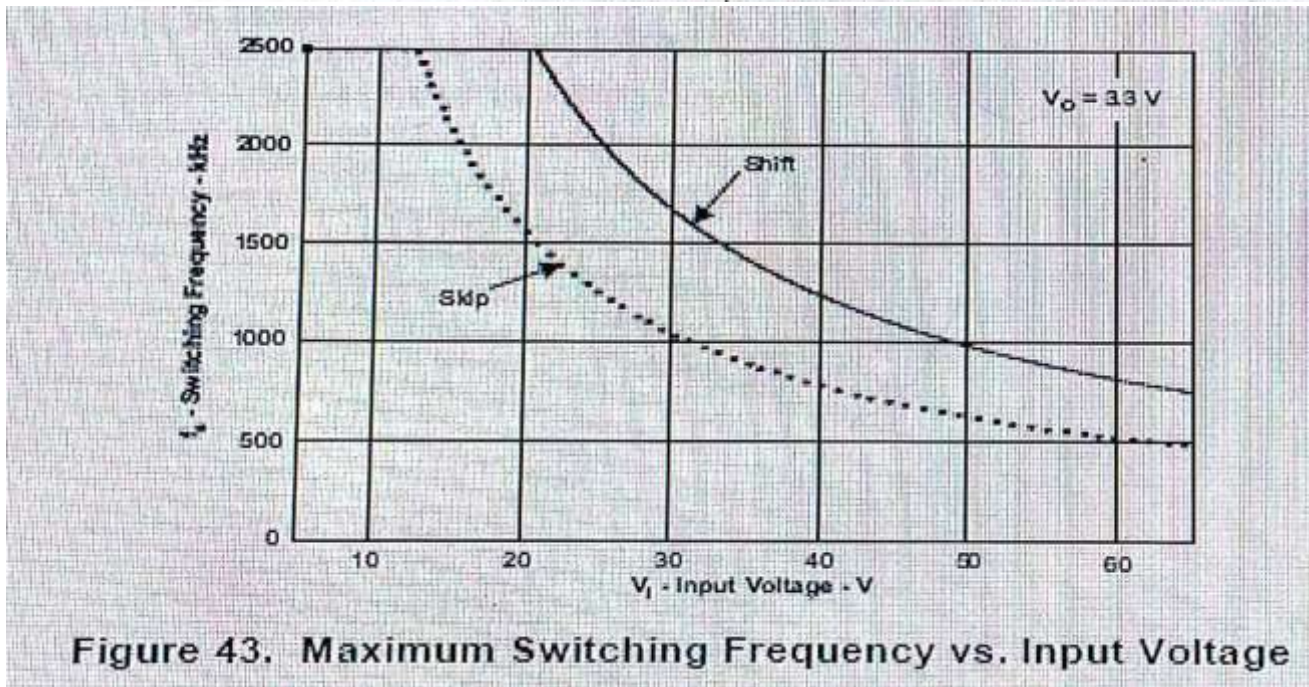
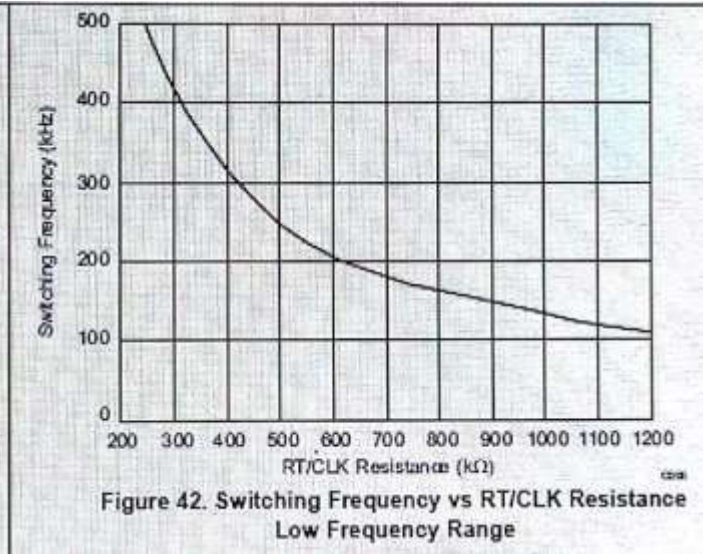
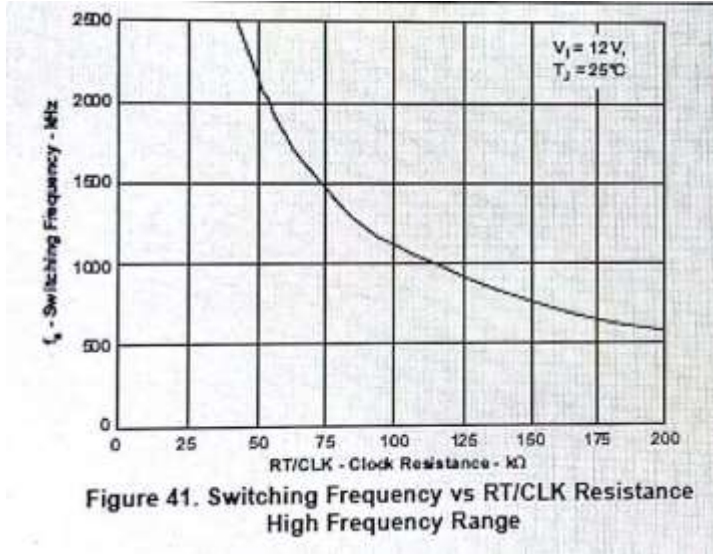


Functional Block Diagram:

8.2 Functional Block Diagram



Waveforms:



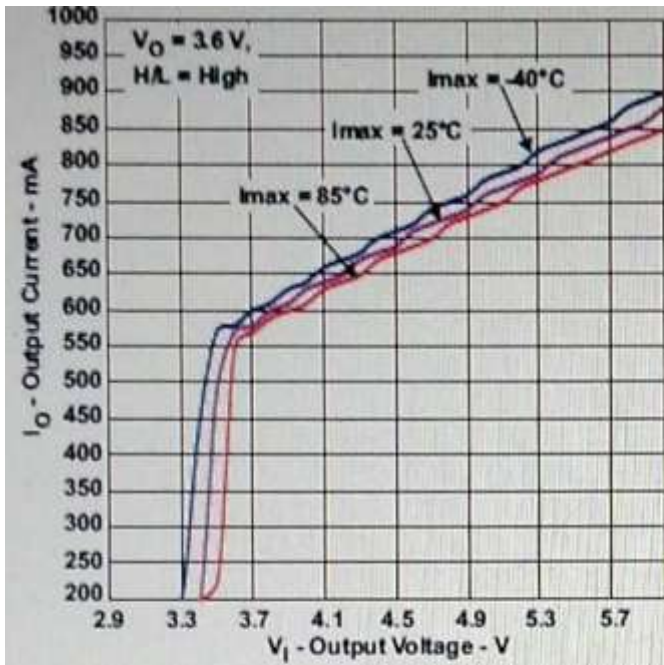


Figure 1. Maximum Output Current

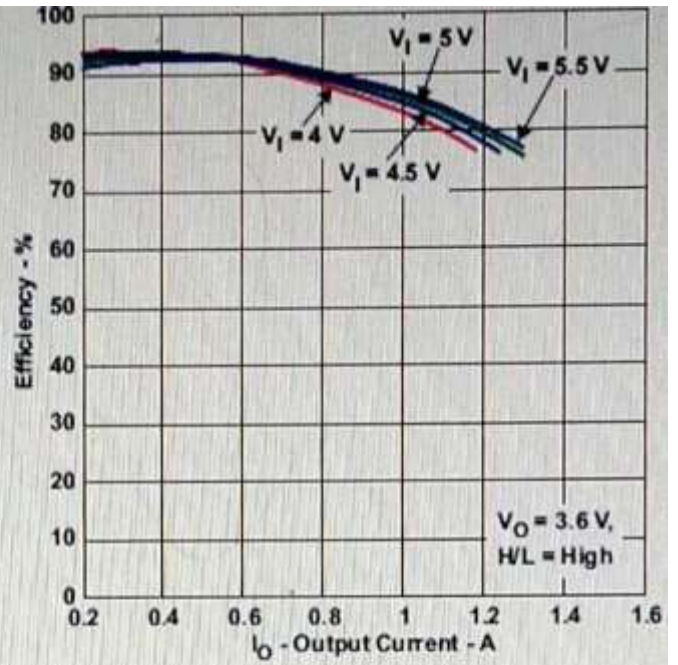


Figure 2. Efficiency vs Output Current

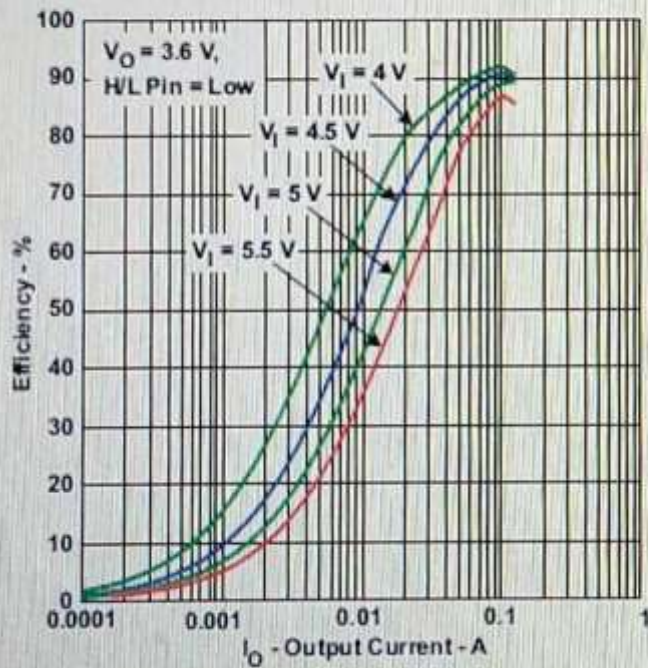


Figure 3. Efficiency vs Output Current

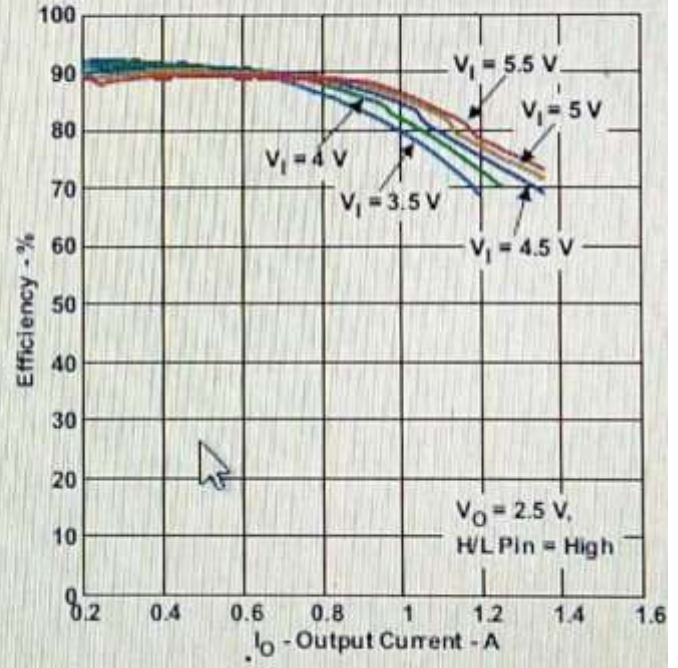


Figure 4. Efficiency vs Output Current

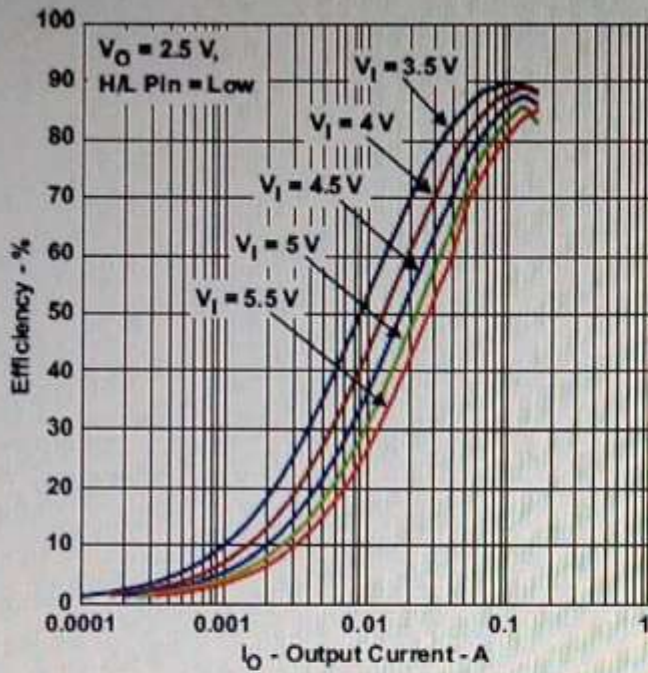


Figure 5. Efficiency vs Output Current

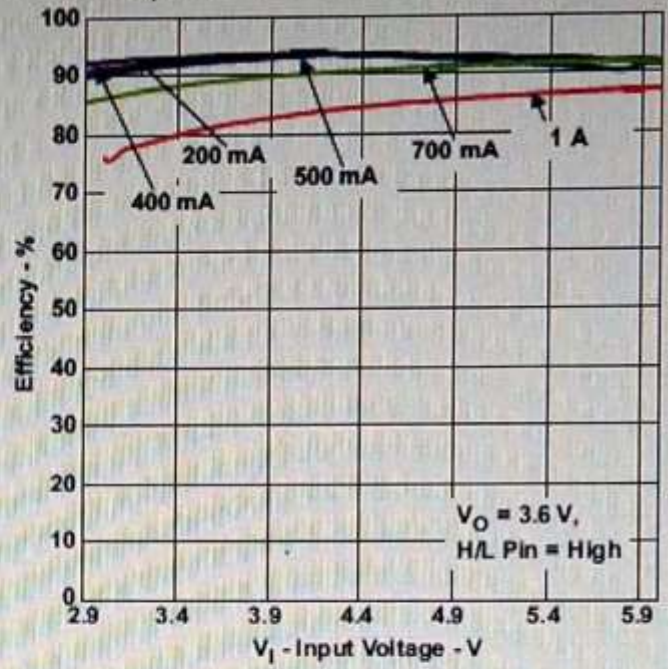


Figure 6. Efficiency vs Input Voltage

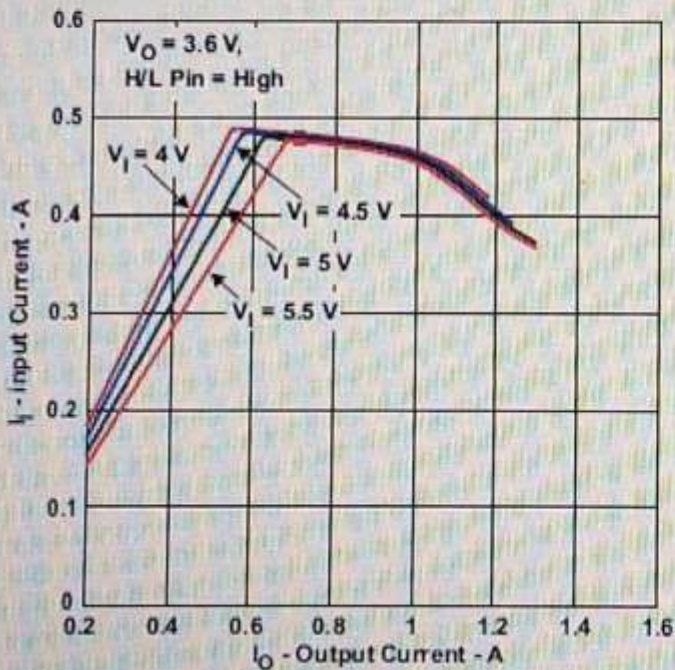


Figure 7. Input Current vs Output Current

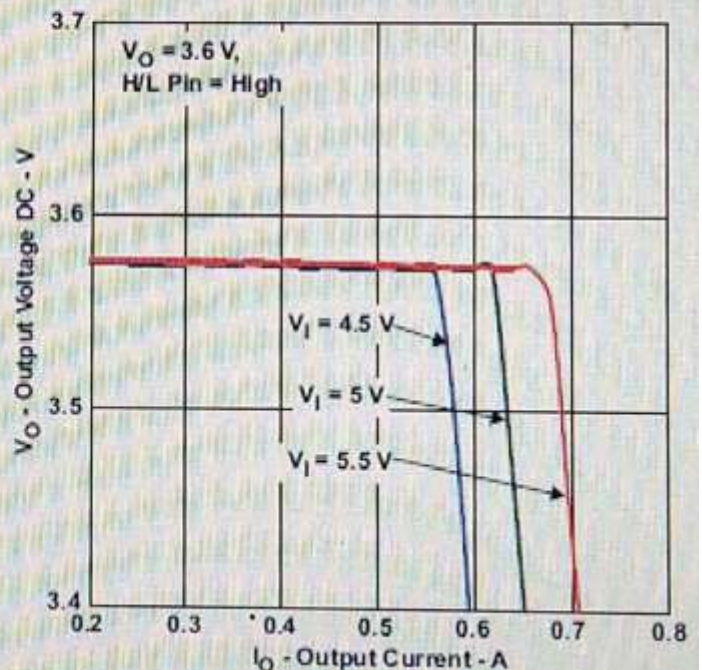


Figure 8. Output Voltage vs Output Current

Result:

Expt.No: 13

Date:

PSPICE SIMULATION OF BUCK CHOPPER**AIM:**

To obtain the performance characteristics of BUCK CHOPPER

APPARTUS:

S. No.	Name of theEquipment
1.	PC With Desktop
2.	PSPICE

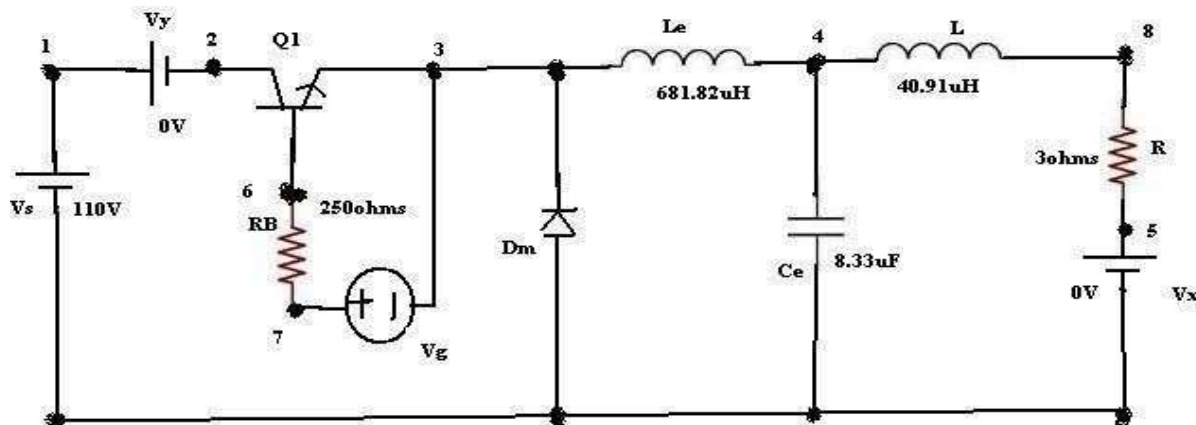
CIRCUIT DIAGRAM:

Fig –Circuit Diagram of PSPICE Simulation of Buck Chopper

PROCEDURE:

1. Represent the nodes for a givencircuit.
2. Write spice program by initializing all the circuit parameter as per given flowchart.
3. From desktop of your computer click on “ START ” menu followed by “ programs ” and then clicking appropriate program group as “ DESIGN LAB EVAL8 followed by “ DESIGN MANAGER.”
4. Open the run text editor from microsim window & start writing pspiceprogram.
5. Save the program with .cirextension.

6. Open the run spice A / D window from microsimwindow.
7. Open file menu from run spice A / D window then open saved circuitfile.
8. If there are any errors, simulates will be displayed with statement as “simulation error occurred”.
9. To see the errors click on o / p file icon and open examine o /p.
10. To make changes in the program open the circuit file, modify, save & Run theprogram.
11. If there are no errors, simulation will be completed & it will be displayed with a statement as “simulationcompleted”.
12. To see the o / p click on o / p file icon & open examine o / p then note down thevalues.
13. If .probe command is used in the program, click on o / p file icon &open run probe. Select variables to plot on graphical window and observe the o / p plots then take print outs ofthat.

PROGARM CODE:

```
CLC
VS 1 0 DC 110V
VY 1 2 DC 0V
VG 7 3 PULSE (0V 20V 0 0.1NS 0.1NS
27.28US 50US) RB 7 6 250
LE 3 4 681.82UH
CE 4 0 8.33UF IC=60V
L 4 8 40.91UH
R 8 5 3
VX 5 0 DC 0V
DM 0 3 DMOD
.MODEL DMOD D (IS=2.2E-15
BV=1800V TT=0) Q1 2 6 3 QMOD
.MODEL QMOD NPN (IS=6.734F BF=416.4 BR=0.7371
CJC=3.638P CJE=4.493P TR=239.5N TF=301.2P)
.TRAN 1US 1.6MS 1.5MS 1US UIC
.PROBE
.FOUR 20KHZ I (VY)
```

.END

PLOT I (LE) I(VX) V4

RESULT:

VIVAQUESTIONS:

1. What is the output voltage equation for step up chopper?
2. Define duty cycle.
3. What is the output voltage equation for step down chopper?
4. What is the output voltage equation for step up / down chopper?