



ST. ANNE'S COLLEGE OF ENGINEERING AND TECHNOLOGY

(Approved by AICTE, New Delhi. Affiliated to Anna University, Chennai)
ANGUCHETTYPALAYAM, PANRUTI -607 106.

EE8511 POWER ELECTRONICS LABORATORY MANUAL

For
III YEAR / V SEMESTER
B.E. ELECTRICAL AND ELECTRONICS ENGINEERING

Prepared by
V.BALAJI M.E.,
Assistant Professor

**DEPARTMENT OF
ELECTRICAL AND ELECTRONICS
ENGINEERING**

General Instructions to students for EEE Lab courses

- ❖ Be punctual to the lab class.
- ❖ Attend the laboratory classes wearing the prescribed uniform and shoes.
- ❖ Avoid wearing any metallic rings, straps or bangles as they are likely to prove dangerous at times.
- ❖ Girls should put their plait inside their overcoat
- ❖ Boy students should tuck in their uniform to avoid the loose cloth getting into contact with rotating machines.
- ❖ Acquire a good knowledge of the surrounding of your worktable. Know where the various live points are situated in your table.
- ❖ In case of any unwanted things happening, immediately switch off the mains in the worktable.
- ❖ This must be done when there is a power break during the experiment being carried out.
- ❖ Before entering into the lab class, **you must be well prepared for the experiment** that you are going to do on that day.
- ❖ You must bring the related text book which may deal with the relevant experiment.
- ❖ Get the circuit diagram approved.
- ❖ Prepare the list of equipments and components required for the experiment and get the indent approved.
- ❖ Plan well the disposition of the various equipments on the worktable so that the experiment can be carried out.
- ❖ **Make connections as per the approved circuit diagram and get the same verified. After getting the approval only supply must be switched on.**
- ❖ For the purpose of speed measurement in rotating machines, keep the tachometer in the extended shaft. Avoid using the brake drum side.
- ❖ Get the reading verified. Then inform the technician so that supply to the worktable can be switched off.
- ❖ You must get the observation note corrected within two days from the date of completion of experiment. Write the answer for all the discussion questions in the observation note. If not, marks for concerned observation will be proportionately reduced.
- ❖ Submit the record note book for the experiment completed in the next class.
- ❖ If you miss any practical class due to unavoidable reasons, intimate the staff in charge and do the missed experiment in the repetition class.
- ❖ Such of those students who fail to put in a minimum of 75% attendance in the laboratory class will run the risk of not being allowed for the University Practical Examination. They will have to repeat the lab course in subsequent semester after paying prescribed fee.
- ❖ **Use isolated supply for the measuring instruments like CRO in Power Electronics Laboratory experiments.**

INDEX

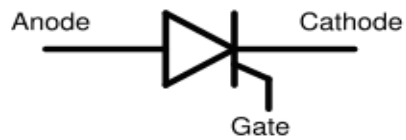
Ex.NO	DATE	NAME OF THE EXPERIMENTS	SIGNATURE
1.		CHARACTERISTICS OF SCR	
2.		CHARACTERISTICS OF TRIAC	
3.		CHARACTERISTICS OF MOSFET & IGBT	
4.		CHARACTERISTICS OF IGCT & GTO	
5.		TRANSIENT CHARACTERISTICS OF SCR AND MOSFET	
6.		TRIAC PHASE CONTROL CIRCUIT	
7.		SCR PHASE CONTROL CIRCUIT	
8.		AC TO DC HALF CONTROLLED CONVERTER	
9.		AC TO DC FULLY CONTROLLED CONVERTER	
10.		THREE PHASE HALF CONTROLLED CONVERTER	
11.		THREE PHASE FULLY CONTROLLED CONVERTER	
12.		STEP UP AND STEP DOWN MOSFET BASED CHOPPERS	
13.		IGBT BASED SINGLE PHASE PWM INVERTER	
14.		IGBT BASED THREE PHASE PWM INVERTER	
15.		RESONANT DC TO DC CONVERTER	
16.		CYCLOCONVERTER	
17.		SINGLE PHASE HALF CONTROLLED CONVERTER	
18.		SINGLE PHASE FULLY CONTROLLED CONVERTER	
19.		THREE PHASE HALF CONTROLLED CONVERTER	
20.		THREE PHASE FULLY CONTROLLED CONVERTER	
21.		SINGLE PHASE AC VOLTAGE CONVERTER	
22.		DC-DC CONVERTER	

STAFF IN CHARGE

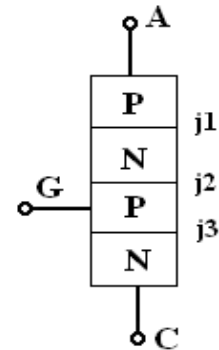
HOD

SILICON CONTROLLED RECTIFIER (SCR):

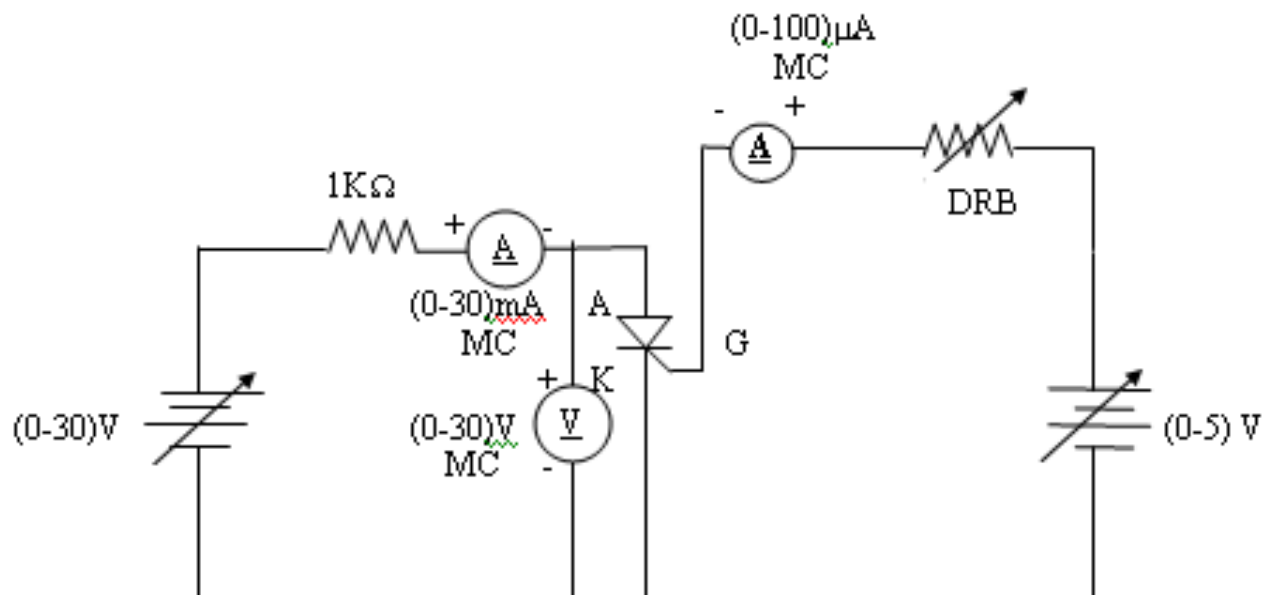
Symbol:



Block Construction:



CIRCUIT DIAGRAM:



Ex. No. 1

Date:

CHARACTERISTICS OF SCR

AIM :

To determine the characteristics of SCR with different gate current(I_g) and to draw the characteristics graphically.

APPARATUS REQUIRED:

S.No.	APPARATUS	RANGE	TYPE	QUANTITY
1	SCR Module kit	220 V / 5 A		1
2	Thyristor Trainer Module Kit Firing Circuit Module inbuilt			1
3	Voltmeter	(0-30) V	MC	1
4	Ammeter	(0-30)mA	MC	1
5	Ammeter	(0-100) μ A	MC	1
6	Resistor	1 k Ω		1
7	Patch Chords			10

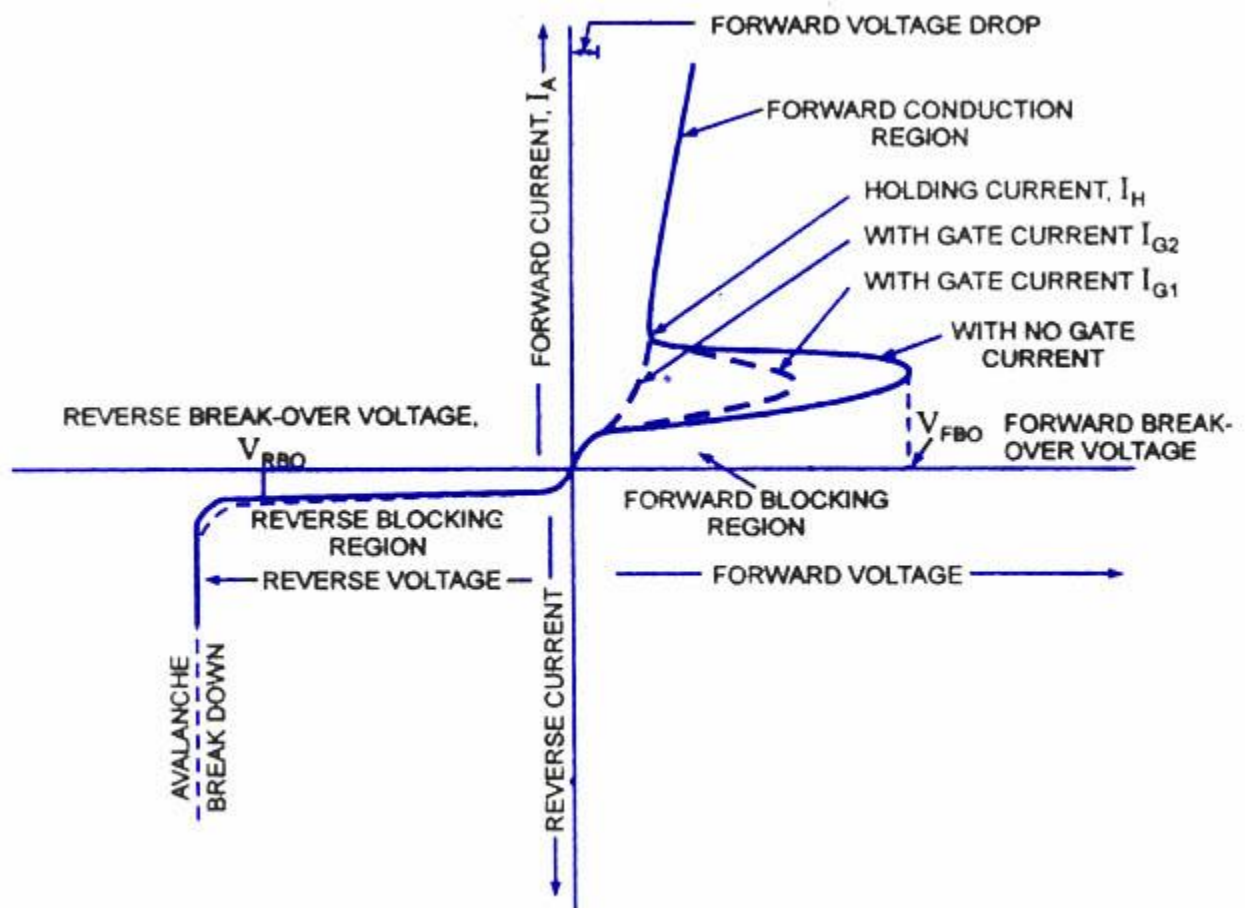
THEORY:

An SCR is a three terminal, four layer latching device. The three terminals are anode, cathode and gate. When the anode is more positive w.r.t the cathode, junctions j_1 , j_3 are forward biased and the junction j_2 is reverse biased. Only a small leakage current flows through the device. The device is said to be in the forward blocking state or OFF state. When the anode to cathode voltage is increased to break-over value, the junction j_2 breaks down and device starts conducting. The anode current must be more than a value known as latching current in order to maintain the device in the ON state. Once SCR starts conducting, it behaves like a conducting diode and gate has no control over the device

TABULAR COLUMN:

S.No.	$I_G = \dots(\mu A)$		$I_G = \dots(\mu A)$	
	$V_{AK} (V)$	$I_A (mA)$	$V_{AK} (V)$	$I_A (mA)$

MODEL GRAPH:



V-I Characteristics of SCR

PROCEDURE:

- 1) Make the connections as per the circuit diagram.
- 2) Switch on the supply
- 3) Set the gate current at a fixed value by varying RPS on the gate-cathode side.
- 4) Increase the voltage applied to anode-cathode side from zero until breakdown occurs.
- 5) Note down the breakdown voltage.
- 6) Draw the graph between anode to cathode voltage (V_{ak}) and anode current (i_a)
- 7) Repeat the experiment for different values of gate current.
- 8) Note the holding current (I_h) and latching current (I_l) from the graph.

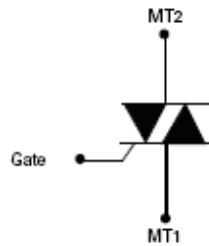
RESULT:

Thus the Characteristics of SCR is drawn as per the tabulation and the following values are noted:

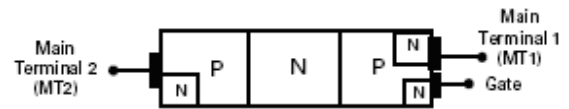
1. Gate Current (I_g) :
2. Break Down voltage (V_{ak}):
3. Holding Current (I_h) :
4. Latching Current (I_l) :

TRIODE AC SWITCH (TRIAC):

Symbol:

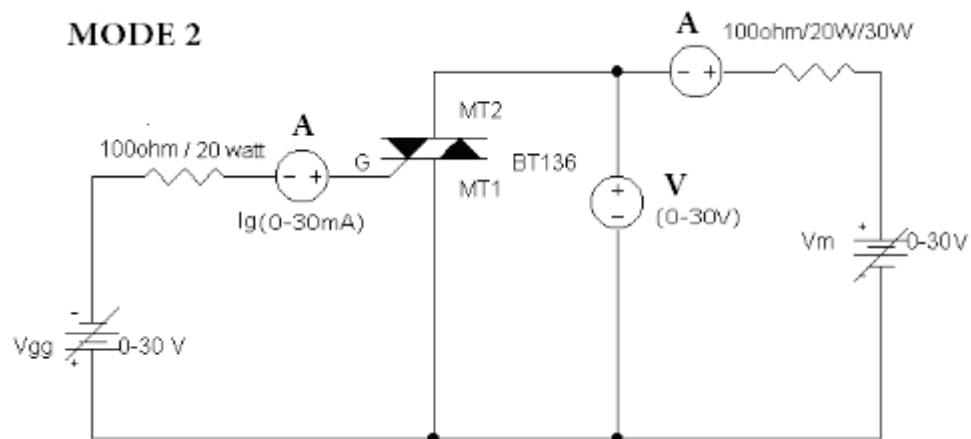
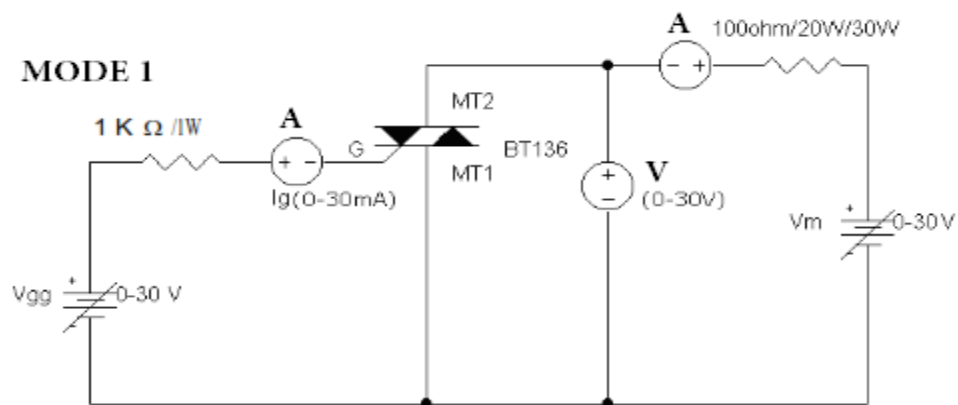


Block Construction:



CIRCUIT DIAGRAM:

MT2 positive with respect to MT1 negative



Ex. No. 2

Date:

CHARACTERISTICS OF TRIAC

AIM:

To determine the characteristics of TRIAC and draw the characteristics graphically.

APPARATUS REQUIRED:

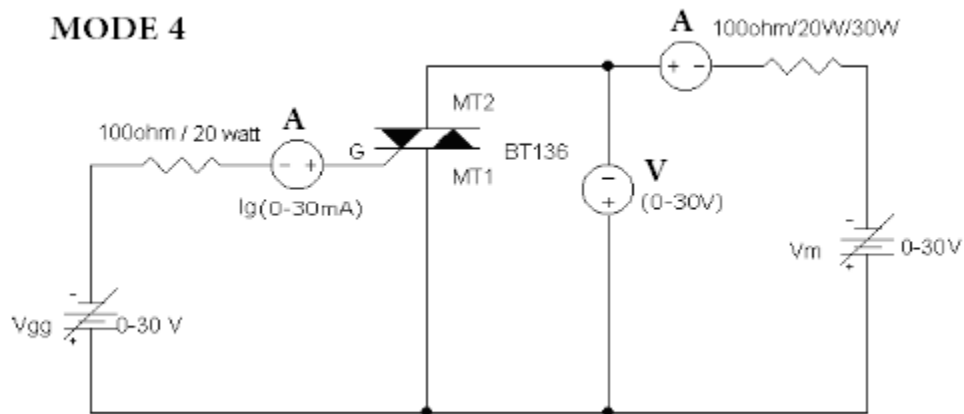
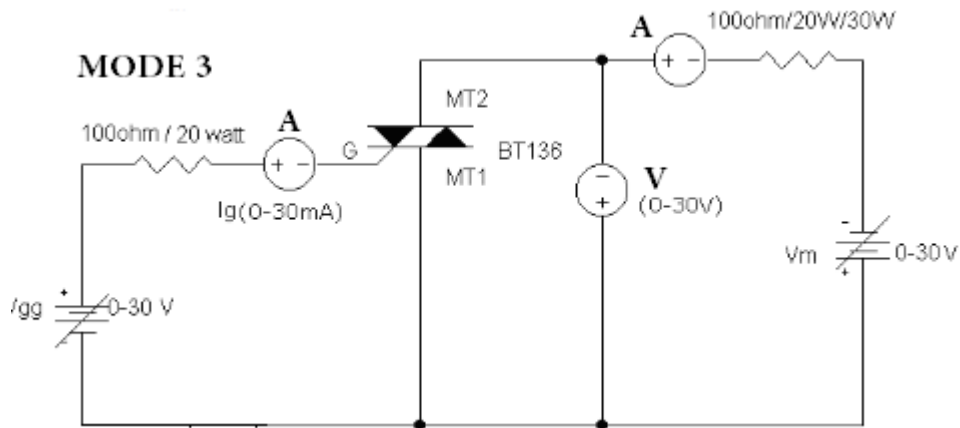
S.No.	APPARATUS	RANGE	TYPE	QUANTITY
1	TRIAC Module kit	220 V / 5 A		1
2	Voltmeter	(0-30) V	MC	1
3	Ammeter	(0-30)mA	MC	1
4	Ammeter	(0-50)mA	MC	1
5	Resistor	1 k Ω		1
6	Patch Chords			10

THEORY:

An SCR is a three terminal, four layer latching device. The three terminals are anode, cathode and gate. When the anode is more positive w.r.t the cathode, junctions j₁, j₃ are forward biased and the junction j₂ is reverse biased. Only a small leakage current flows through the device. The device is said to be in the forward blocking state or OFF state. When the anode to cathode voltage is increased to break-over value, the junction j₂ breaks down and device starts conducting. The anode current must be more than a value known as latching current in order to maintain the device in the ON state. Once SCR starts conducting, it behaves like a conducting diode and gate has no control over the device.

CIRCUIT DIAGRAM:

MT2 negative with respect to MT1 positive:



PROCEDURE:

1. Make the connections as per the circuit diagram.
2. Switch on the supply.
3. Set the gate current at a fixed value by varying RPS on the gate- cathode side.
4. Increase the voltage applied across anode and corresponding current is noted.
5. The TRIAC terminal (MT2) is made positive with respect to (MT1) and the gate current is positive
6. The graph lies in the I quadrant.
7. The gate current is made negative and the graph lies in II quadrant

8. The TRIAC terminal (MT1) is made positive with respect to (MT2) and the gate current is positive the graph lies in the III quadrant
9. The gate current is made negative and the graph lies in IV quadrant
10. The above steps are repeated for different values of I_G .
11. Draw the graph between anode to cathode voltage (V_{AK}) and anode current (I_A)

TABULAR COLUMN:

MT2 positive with respect to MT1 negative

MODE-1:

$I_G = \dots(\text{mA})$		$I_G = \dots(\text{mA})$	
V_{AK} (V)	I_A (mA)	V_{AK} (V)	I_A (mA)

MODE-2:

$I_G = \dots(\text{mA})$		$I_G = \dots(\text{mA})$	
V_{AK} (V)	I_A (mA)	V_{AK} (V)	I_A (mA)

MT2 negative with respect to MT1 positive:

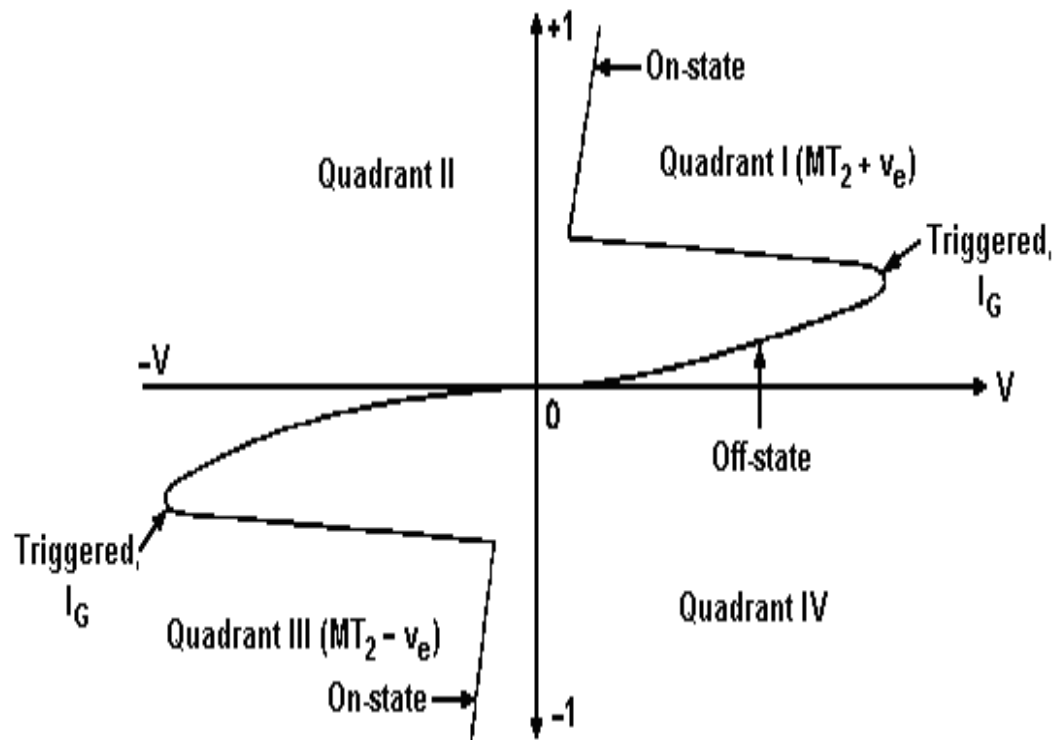
MODE-3:

$I_G = \dots(\text{mA})$		$I_G = \dots(\text{mA})$	
V_{AK} (V)	I_A (mA)	V_{AK} (V)	I_A (mA)

MODE-4:

$I_G = \dots(\text{mA})$		$I_G = \dots(\text{mA})$	
V_{AK} (V)	I_A (mA)	V_{AK} (V)	I_A (mA)

MODEL GRAPH:



RESULT :

Thus the Characteristics of TRIAC was obtained and drawn graphically.

Ex. No. 3

Date:

CHARACTERISTICS OF MOSFET & IGBT

AIM :

To determine the characteristics of MOSFET & IGBT and draw the characteristics graphically.

APPARATUS REQUIRED:

S.No.	APPARATUS	RANGE	TYPE	QUANTITY
1	MOSFET & IGBT Module kit	220 V / 5 A		1
2	Voltmeter	(0-5) V	MC	1
3	Voltmeter	(0-30) V	MC	1
4	Ammeter	(0-5)mA	MC	1
5	Resistor	5 K Ω , 1 k Ω		1
6	Patch Chords			10

THEORY

MOSFET

a) Output characteristics:

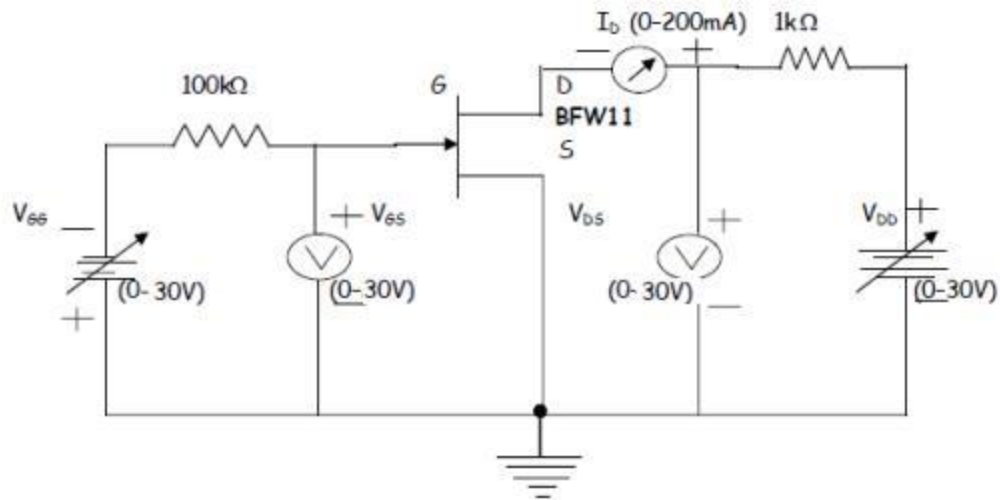
It indicates the variation of drain current I_D as a function of drain-source voltage V_{GS} as a parameter. For low values of V_{DS} , the graph between I_D - V_{DS} is almost linear; this indicates a constant value of on resistance $R_{DS} = V_{DS} / I_D$. For given V_{GS} , if V_{DS} is increased, output characteristic is relatively flat indicating that drain current is nearly constant.

b) Transfer characteristics:

This characteristic shows the variation of drain current I_D as a function of gate-source voltage V_{GS} . Threshold voltage V_{GST} is an important parameter of MOSFET. V_{GST} is the minimum positive voltage between gate and source to induce n-channel. Thus, for threshold voltage below V_{GST} , device is in the OFF-state. Magnitude of V_{GST} is of the order of 2 to 3V.

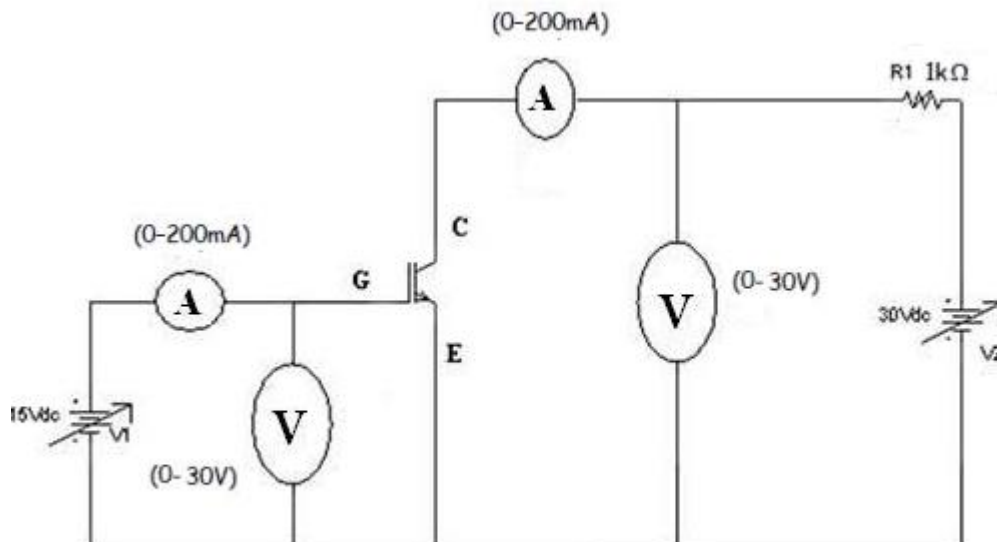
CIRCUIT DIAGRAM:

METAL OXIDE FIELD EFFECT TRANSISTOR (MOSFET):



CIRCUIT DIAGRAM:

INSULATED GATE BASE TRANSISTOR (IGBT):



TABULAR COLUMN (MOSFET):

TRANSFER CHARACTERISTICS:

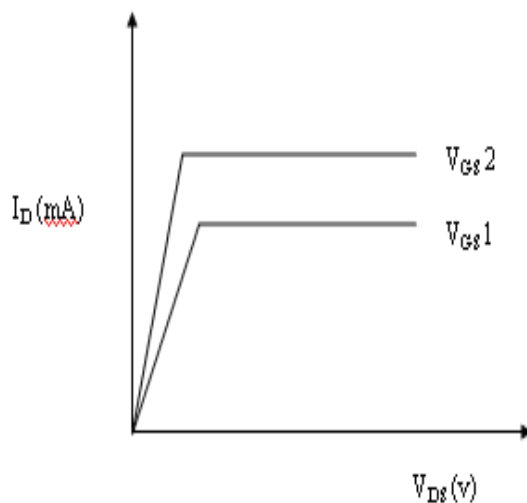
$V_{GS} = \dots\dots\dots(V)$		$V_{GS} = \dots\dots\dots(V)$	
V_{DS} (mV)	I_D (mA)	V_{DS} (mV)	I_D (mA)

DRAIN CHARACTERISTICS:

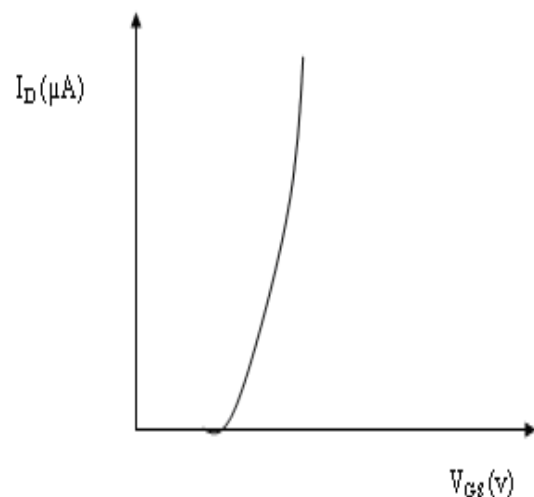
V_{GS} (mV)	I_D (mA)

MODEL GRAPH (MOSFET):

TRANSFER CHARACTERISTICS



DRAIN CHARACTERISTICS



IGBT:

a) Output characteristics:

Output characteristics of an IGBT show the plot of collector current I_C versus Collector- Emitter voltage V_{CE} for various values of Gate-Emitter voltages. In the forward direction, the shape of the output characteristics is similar to that of BJT. But here the controlling parameter is Gate-Emitter voltage V_{GE} because IGBT is a voltage controlled device. When the device is OFF, junction j_2 blocks forward voltage and in case reverse

voltage appears across collector and emitter, junction j₁ blocks it. V_{RM} is the maximum reverse breakdown voltage.

b) Transfer characteristics:

The transfer characteristics of an IGBT is a plot of collector current I_C versus Gate- Emitter voltage V_{GE}. This characteristic is identical to that of power MOSFET. When V_{GE} is less than the threshold voltage V_{GET}, IGBT is in the OFF-state

PROCEDURE:

- 1) Make the connections as per the circuit diagram.
- 2) Switch on the supply.
- 3) Set the gate current at a fixed value by varying RPS on the gate-cathode side.
- 4) Vary the voltage applied across Gate and corresponding V_{DS} (V_{CE}) and I_D (I_C) is noted .
- 5) The above steps are repeated for different values of I_G .
- 6) Vary the voltage across Collector and Emitter and noted down V_{GE} and I_C.
- 7) Draw the graph between V_{GS} (V_{CE}) and I_D (I_C) and V_{GS} (V_{GE}) and I_D (I_C).

TABULAR COLUMN (IGBT):

TRANSFER CHARACTERISTICS

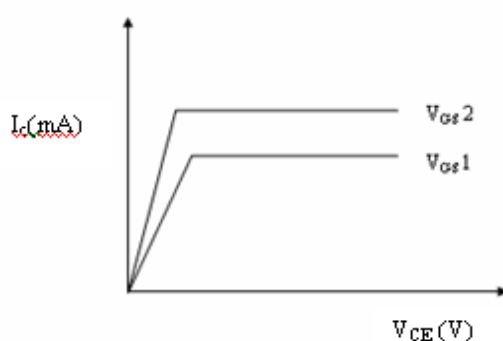
V _{GE} =....(V)		V _{GE} =....(V)	
V _{CE} (mV)	I _C (mA)	V _{CE} (mV)	I _C (mA)

DRAIN CHARACTERISTICS

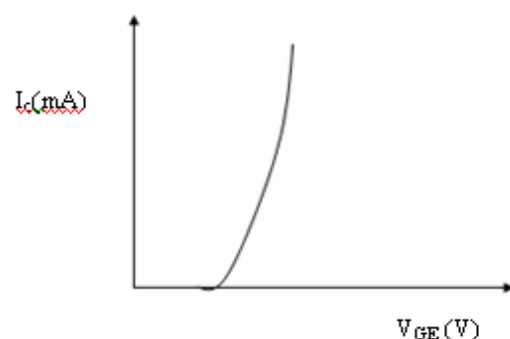
V _{CE} =....(V)	
V _{GE} (mV)	I _C (mA)

MODEL GRAPH (IGBT):

TRANSFER CHARACTERISTICS



DRAIN CHARACTERISTICS



PROCEDURE:

- 1) Make the connections as per the circuit diagram.
- 2) Switch on the supply.
- 3) Set the gate current at a fixed value by varying RPS on the gate-cathode side.
- 4) Vary the voltage applied across Gate and corresponding V_{DS} (V_{CE}) and I_D (I_C) is noted .
- 5) The above steps are repeated for different values of I_G .
- 6) Vary the voltage across Collector and Emitter and noted down V_{GE} and I_C .
- 7) Draw the graph between V_{GS} (V_{CE}) and I_D (I_C) and V_{GS} (V_{GE}) and I_D (I_C).

RESULT:

Thus the Characteristics of MOSFET & IGBT were obtained and drawn graphically.

MOSFET:

Threshold Voltage =

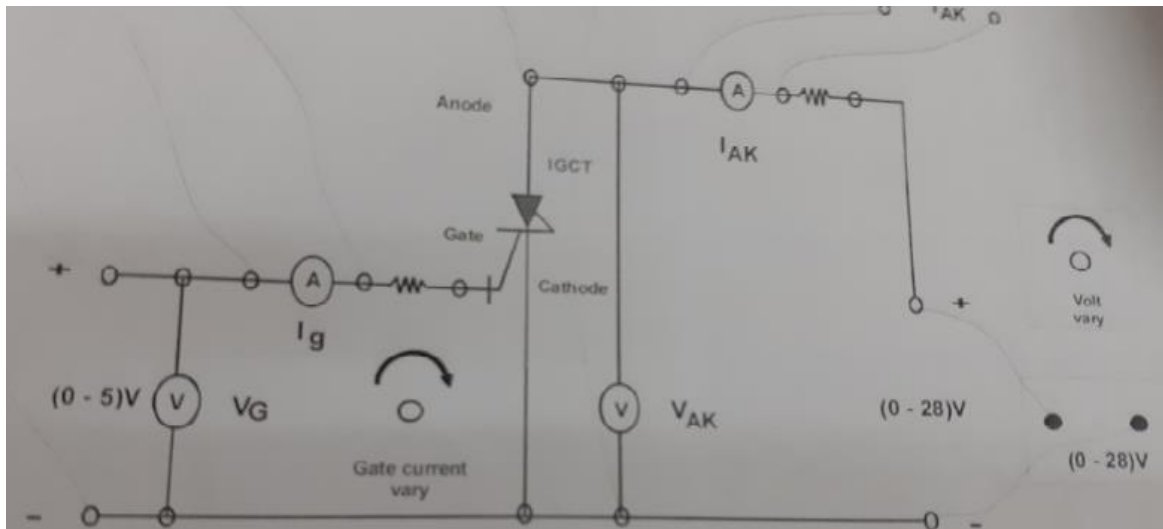
Saturated Current =

IGBT:

Threshold Voltage =

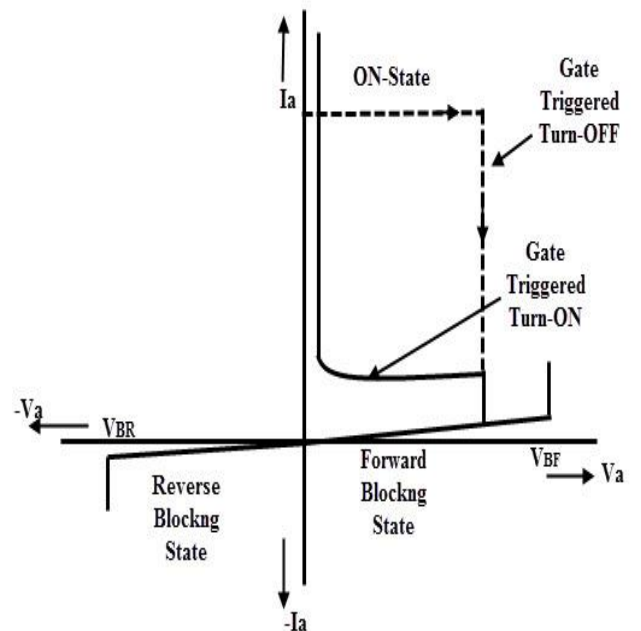
Saturated Current =

CIRCUIT DIAGRAM (IGCT) :



Tabular column:

$V_{Gk} = \dots (V)$		$V_{Gk} = \dots (V)$	
V_{ak} (mV)	I_{ak} (mA)	V_{ak} (mV)	I_{ak} (mA)



Ex. No. 4

Date:

CHARACTERISTICS OF IGCT & GTO

AIM :

To determine the characteristics of IGCT & GTO and draw the characteristics graphically.

APPARATUS REQUIRED:

S.No.	APPARATUS	RANGE	TYPE	QUANTITY
1	IGCT & GTO Module kit	220 V / 5 A		1
2	Voltmeter	(0-5) V	MC	1
3	Voltmeter	(0-30) V	MC	1
4	Ammeter	(0-5)mA	MC	1
5	Resistor	5 K Ω , 1 k Ω		1
6	Patch Chords			10

THEORY

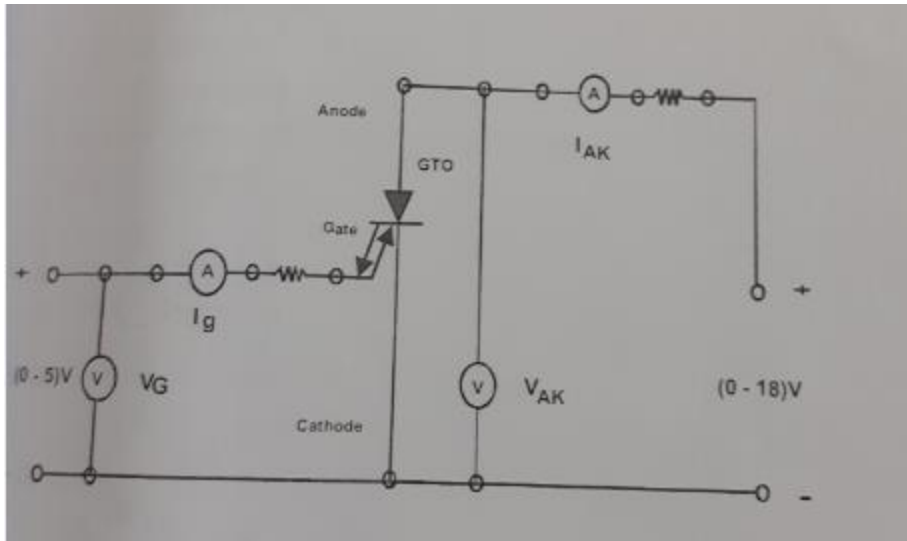
IGCT:

The integrated gate-commutated thyristor (IGCT) is a power semiconductor electronic device, used for switching electric current in industrial equipment. An IGCT is a special type of thyristor. It is made of the integration of the gate unit with the Gate Commutated Thyristor (GCT) wafer device. The close integration of the gate unit with the wafer device ensures fast commutation of the conduction current from the cathode to the gate. The wafer device is similar to a gate turn-off thyristor (GTO). They can be turned on and off by a gate signal, and withstand higher rates of voltage rise (dv/dt), such that no snubber is required for most applications.

GTO:

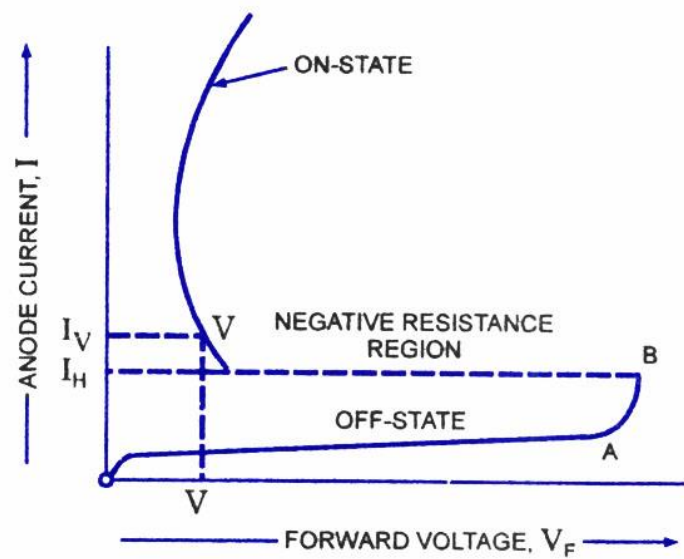
The GTO stands for gate turn OFF thyristor. The GTO is a three terminal device. The terminals names are anode, cathode and gate. It can be turned ON by applying a pulse of positive gate current and it can be turned OFF by applying a pulse of negative gate of proper amplitude. The GTO can resist higher voltage and current than the power transistor or MOSFET (metal oxide semiconductor FET)

CIRCUIT DIAGRAM (GTO) :



Tabular column:

$V_{Gk} = \dots (V)$		$V_{Gk} = \dots (V)$	
V_{ak} (mV)	I_{ak} (mA)	V_{ak} (mV)	I_{ak} (mA)



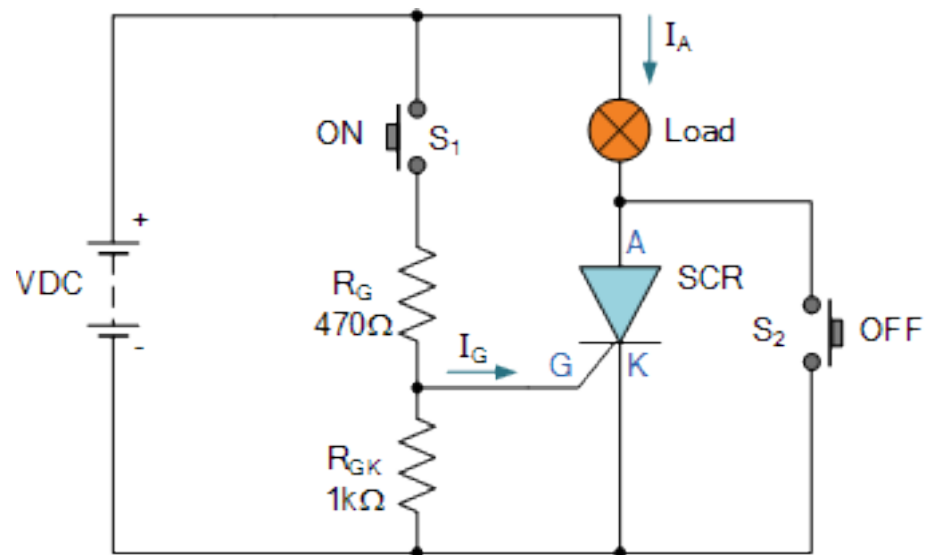
PROCEDURE:

1. Connections are made as per the circuit diagram
2. set the voltage between anode and cathode as anode is made positive with respect to the cathode.
3. set the voltage between gate and cathode as gate is made positive with respect to cathode.
4. vary the gate current by varying the potentiometer.
5. note the different values of V_{AK} , I_g , I_{AK} and make the gate current constant.
6. draw the graph between V_{AK} and I_g .
7. make the gate voltage zero and gate current pot to minimum. still the device will be in ON state.
8. set the voltage between gate and cathode as gate is made negative with respect to cathode for turn off the device.
9. vary the gate current (negative). now the device will come to OFF state.
10. note the different values for V_{AK} and I_g .
11. draw the graph between V_{AK} and I_g .

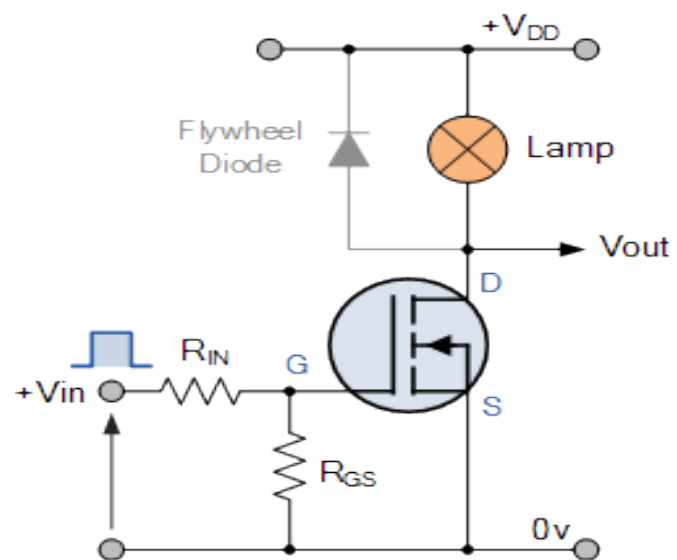
RESULT:

Thus the Characteristics of IGCT & GTO were obtained and drawn graphically.

CIRCUIT DIAGRAM (SCR) :



CIRCUIT DIAGRAM (MOSFET):



TRANSIENT CHARACTERISTICS OF SCR AND MOSFET

AIM:

To determine the transient characteristics of MOSFET & IGBT.

APPARATUS REQUIRED:

S.No.	APPARATUS	RANGE	TYPE	QUANTITY
1	MOSFET & IGBT Module kit	220 V / 5 A		1
2	Regulated Power Supply	(0-30) V		1
3	Voltmeter	(0-5) V	MC	1
4	Voltmeter	(0-30) V	MC	1
5	Ammeter	(0-5)mA	MC	1
6	Ammeter	(0-10)mA	MC	1
7	Ammeter	(0-100) μ A	MC	1
8	DRB			1
19	Resistor	1 K Ω , 1k Ω , 100 k Ω , 5 k Ω ,		1
10	Patch Chords			10

THEORY:

SCR

a) Turn-ON characteristics:

A forward biased thyristor is usually turned ON by applying a positive gate voltage between gate and cathode. Thyristor turn ON time is defined as the time during which it changes from forward blocking state to final ON state. Total turn ON time can be divided into three intervals; 1. Delay time (t_d) 2. Rise time (t_r) 3. Spread time (t_p).

- Delay time is measured from the instant at which gate current reaches $0.9 I_g$ to the instant at which anode current reaches $0.1 I_a$.
- Rise time is the time taken by the anode current to rise from $0.1 I_a$ to $0.9 I_a$
- Spread time is the time taken by the anode current to rise from $0.9 I_a$ to I_a .

TABULAR COLUMN (SCR) :

T _{ON}		T _{OFF}	
T(μs)	v	T _{rr} (μs)	T _q (μs)
		t(μs)	v

b) Turn-OFF characteristics:

Thyristor turn-OFF means that it has changed from ON to OFF state and is capable of blocking the forward voltage. This dynamic process of the SCR from conduction state to forward blocking state is called turn OFF process. The turn OFF time is divided into two intervals; Reverse Recovery time t_{rr} and Gate Recovery time t_{gr} . The time required for the reversed anode current to recover to zero or nearly zero value. The charges around junction j_2 of SCR are trapped and these only by recombination. This recombination of charges between t_3 and t_4 is called gate recovery time.

MOSFET

a) Turn-ON characteristics:

- The turn ON time is defined as the sum of turn ON delay time and rise time of the device.

During turn ON delay time t_{dn} , the input capacitance charges to gate threshold voltage V_{GST} and the drain current is zero.

- During rise time period, gate voltage rises to V_{GSP} which is the gate source peak voltage. This voltage is sufficient to drive the MOSFET into ON state. Then drain current increases from zero to full value of current I_D .
- Thus the total turn ON time is $t_{on} = t_{dn} + t_r$. The turn ON time can be reduced by using low impedance gate drive circuit.

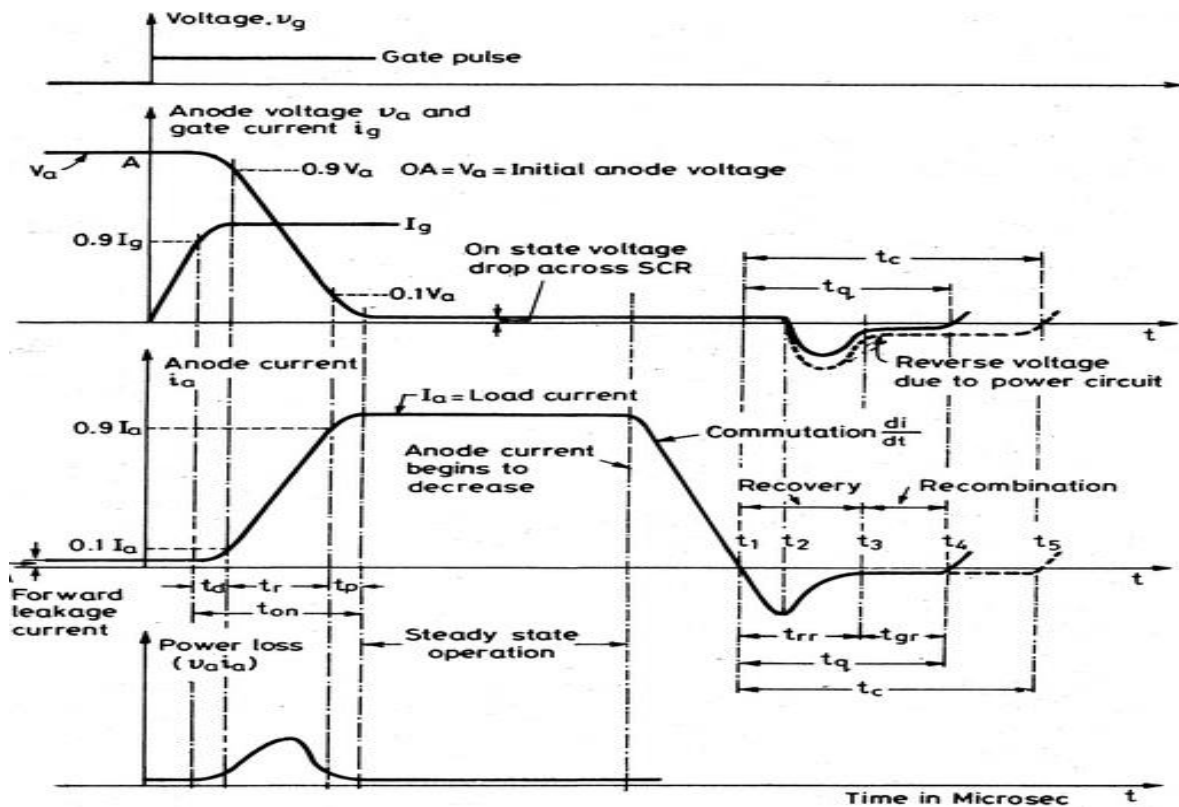
b) Turn-OFF characteristics:

The turn OFF process is initiated by the removal of gate source voltage V_{GS} at time t_1 , because MOSFET is a majority carrier device. The turn OFF time is the sum of turn OFF delay time t_{df} and fall time t_f .

- During this period t_{df} , the input capacitance discharges from over drive gate voltage V_1 to V_{GSP} but the drain current I_D does not change.
- During fall time t_f , the input capacitance again discharges from V_{GSP} to threshold

voltage V_{GST} . Then drain current fall from I_D to zero. So when $V_{GS} \leq V_{GST}$, power MOSFET turn OFF is completed. $T_{off} = t_{df} + t_f$.

MODEL GRAPH (SCR):



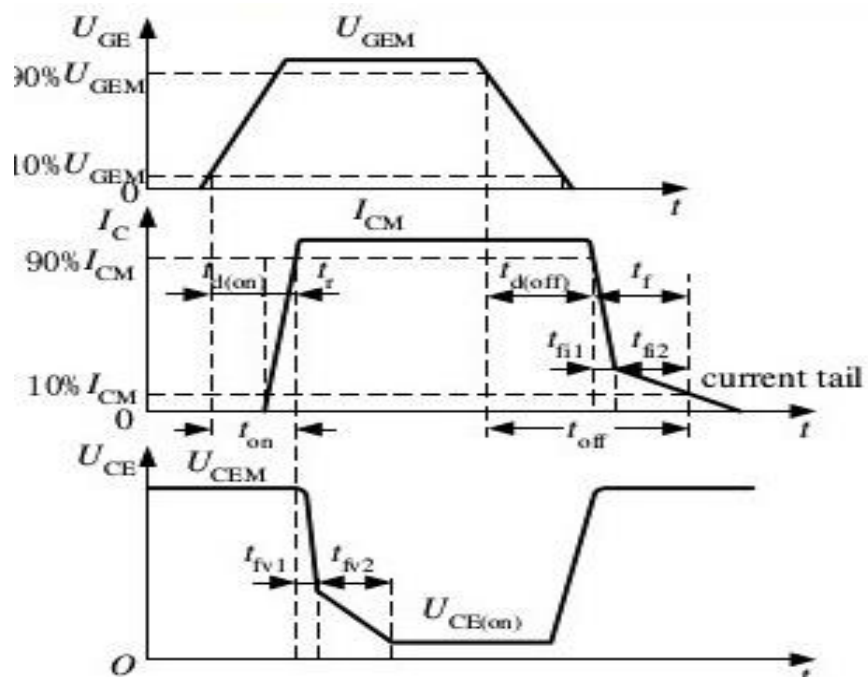
PROCEDURE:

- 1) Make the connections as per the circuit diagram.
- 2) Switch on the supply.
- 3) Set the 30 V in the voltmeter using variable DC supply provided in the trainer.
gate current at a fixed value by varying RPS on the gate-cathode side.
- 4) Vary the gate current and note down the values of I_A & V_A .
- 5) The above steps are repeated for different values of I_G .
- 6) Draw the graph between V_{AK} (V_{DS}) and I_G (V_{GS}) and I_A (I_D) and I_G (V_{GS}).

TABULAR COLUMN (MOSFET):

TON (μs)	TOFF (μs)	CONDUCTING TIME(μs)	VOLTAGE(V)

MODEL GRAPH (MOSFET):



RESULT:

Thus the Transient Characteristics of SCR & MOSFET were obtained and drawn graphically.

Ex. No. 6

Date:

TRIAC PHASE CONTROL CIRCUIT

AIM:

To construct the Single Phase AC Control Circuit using TRIAC for various values of firing angle.

APPARATUS REQUIRED:

S.No.	APPARATUS	RANGE	TYPE	QUANTITY
1	TRIAC Power Circuit kit	Single Phase 230 V / 10 A		1
2	TRIAC Firing Circuit kit	Single Phase 230 V / 5 A		1
3	Isolation Transformer	230 / 115 V		1
4	Auto Transformer	230 / 230 V		1
5	CRO	20 MHZ		1
6	Loading Rheostat	100 Ω , 2 A		1
7	Patch Chords			10

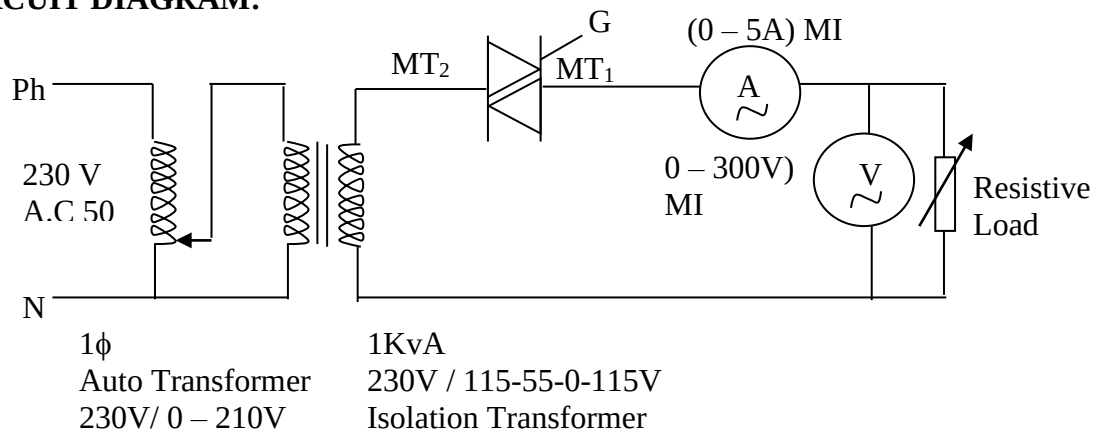
PROCEDURE:

- 1) Make the connections as per the circuit diagram.
- 2) Keep the multiplication factor of CRO's probe at maximum position.
- 3) Switch on the TRIAC knob and firing circuit kit.
- 4) Vary the firing angle in steps and note the readings of waveforms in CRO.
- 5) Switch of the supply.
- 6) Draw the output waveforms.

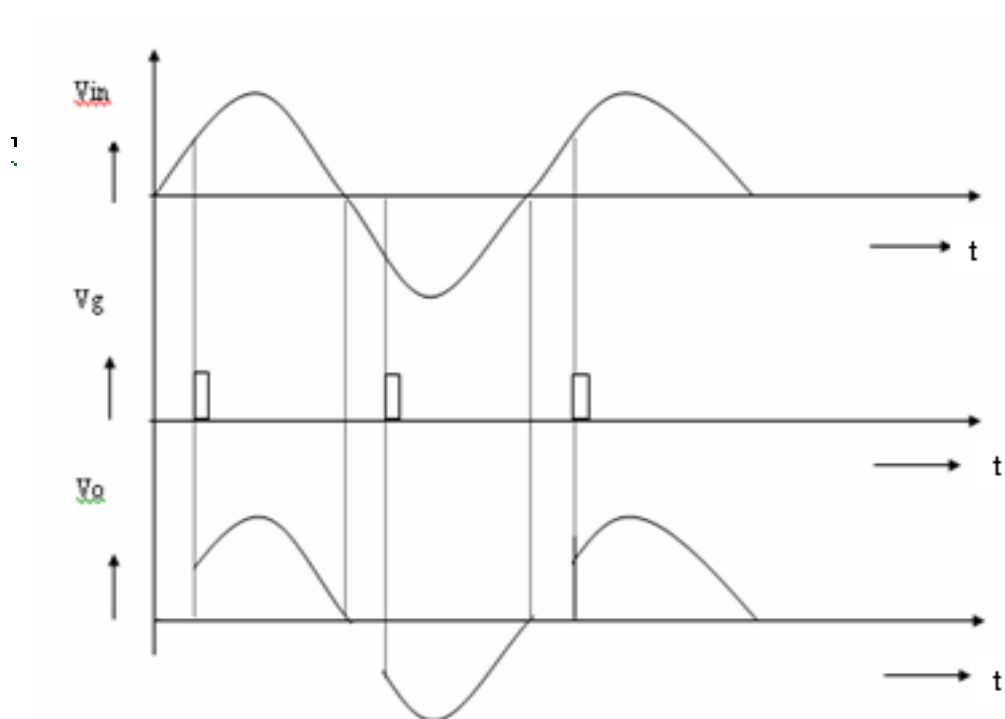
TABULAR COLUMN:

Voltage (v)	Total Time Period T(ms)	Non-Conductingtime 't ' (ms)	Firing angle α	Measured Voltage(V)	Calculatedvoltage (V)

CIRCUIT DIAGRAM:



MODEL GRAPH:



RESULT:

Thus Single Phase AC Phase Control Circuit using TRIAC was constructed and output waveforms was drawn.

Ex. No. 7

Date:

SCR PHASE CONTROL CIRCUIT

AIM:

To construct the Single Phase AC Control Circuit using SCR for various values of firing angle.

APPARATUS REQUIRED:

S.No.	APPARATUS	RANGE	TYPE	QUANTITY
1	SCR Power Circuit kit	Single Phase 230 V / 10 A		1
2	Isolation Transformer	230 / 115 V		1
3	Auto Transformer	230 / 230 V		1
4	CRO	20 MHZ		1
5	Loading Rheostat	100 Ω , 2 A		1
6	Patch Chords			10

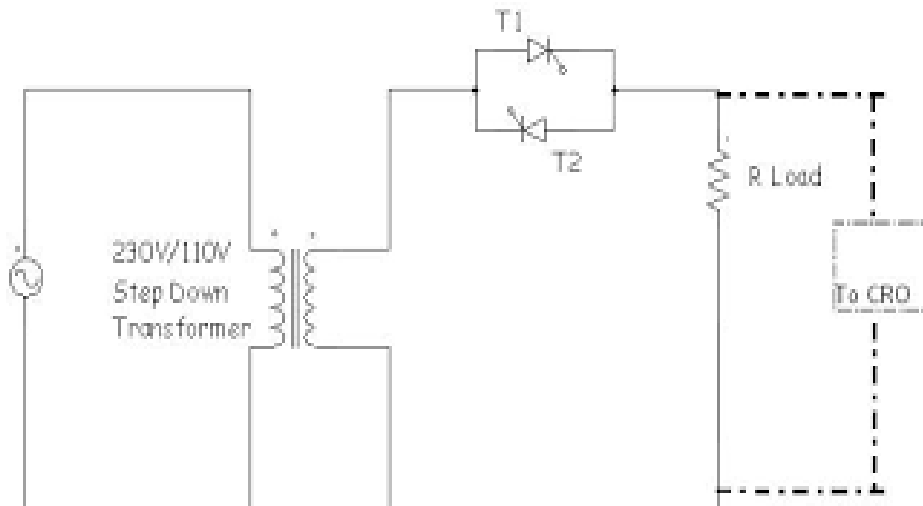
PROCEDURE:

- 1) Make the connections as per the circuit diagram.
- 2) Keep the multiplication factor of CRO's probe at maximum position.
- 3) Switch on the SCR knob and firing circuit kit.
- 4) Vary the firing angle in steps and note the readings of waveforms in CRO.
- 5) Switch of the supply.
- 6) Draw the output waveforms.

RESULT:

Thus Single Phase AC Phase Control Circuit using TRIAC was constructed and output waveforms was drawn.

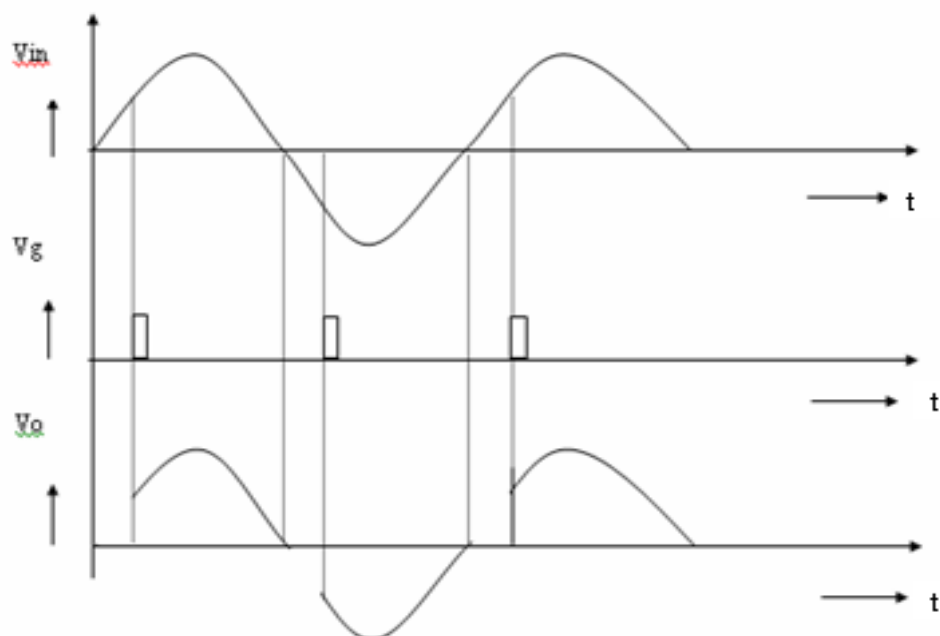
CIRCUIT DIAGRAM:



TABULAR COLUMN:

Voltage (v)	Total Time Period T(ms)	Non-Conductingtime 't ' (ms)	Firing angle α	Measured Voltage(V)	Calculated voltage (V)

MODEL GRAPH:



Ex. No. 8

Date:

AC TO DC HALF CONTROLLED CONVERTER

AIM:

To construct a single phase half controlled Converter and plot its output response.

APPARATUS REQUIRED:

S.NO.	APPARATUS	RANGE	TYPE	QUANTITY
1	Half controlled Converter Power circuit kit	1 ϕ , 230V,10A	-	1
3	SCR firing circuit kit	1 Φ ,230V,5A	-	1
4	Isolation Transformer	230V/115-55-0- 55-115	-	1
5	Auto-transformer	230V/0-270V, 4A	-	1
6	Loading Rheostat	100 Ω / 2A	-	1
7	CRO	20MHz	-	1
8	Patch chords	-	-	15

FORMULA:

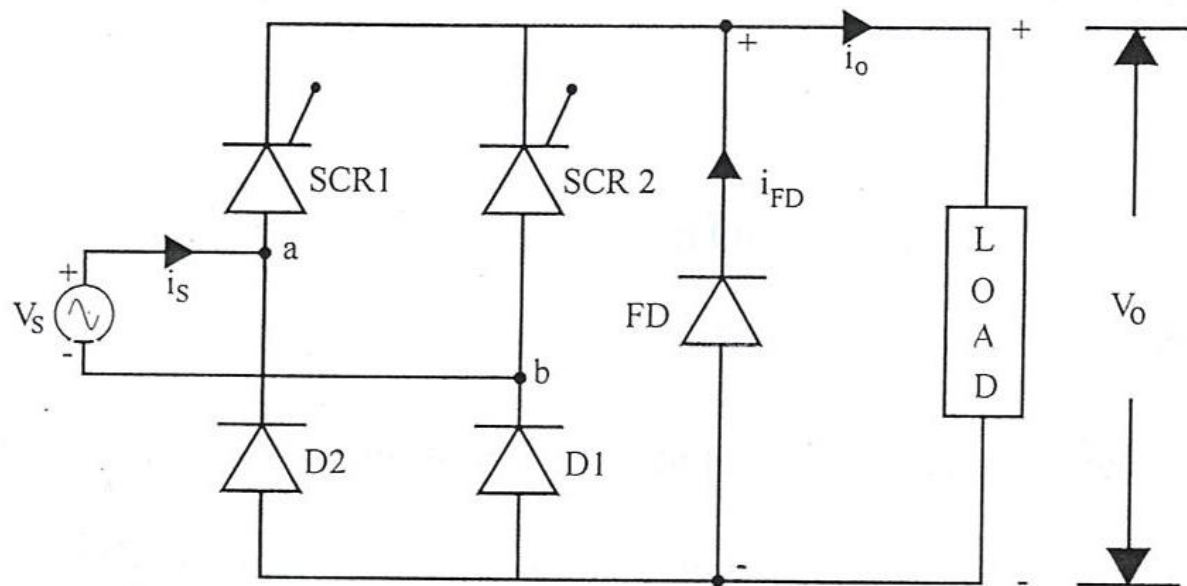
$$\text{Firing angle } \alpha_0 = \frac{t}{10\text{ms}} \times 180$$

t= non-conducting period of thyristor.

PROCEDURE:

1. Make the connections as per the circuit diagram.
2. Keep the multiplication factor of the CRO's probe at the maximum position.
3. Switch on the thyristor kit and firing circuit kit.
4. Keep the firing circuit knob at the 180 °position.
5. Vary the firing angle in steps.
6. Note down the voltmeter reading and waveform from the CRO.
7. Switch off the power supply and disconnect.

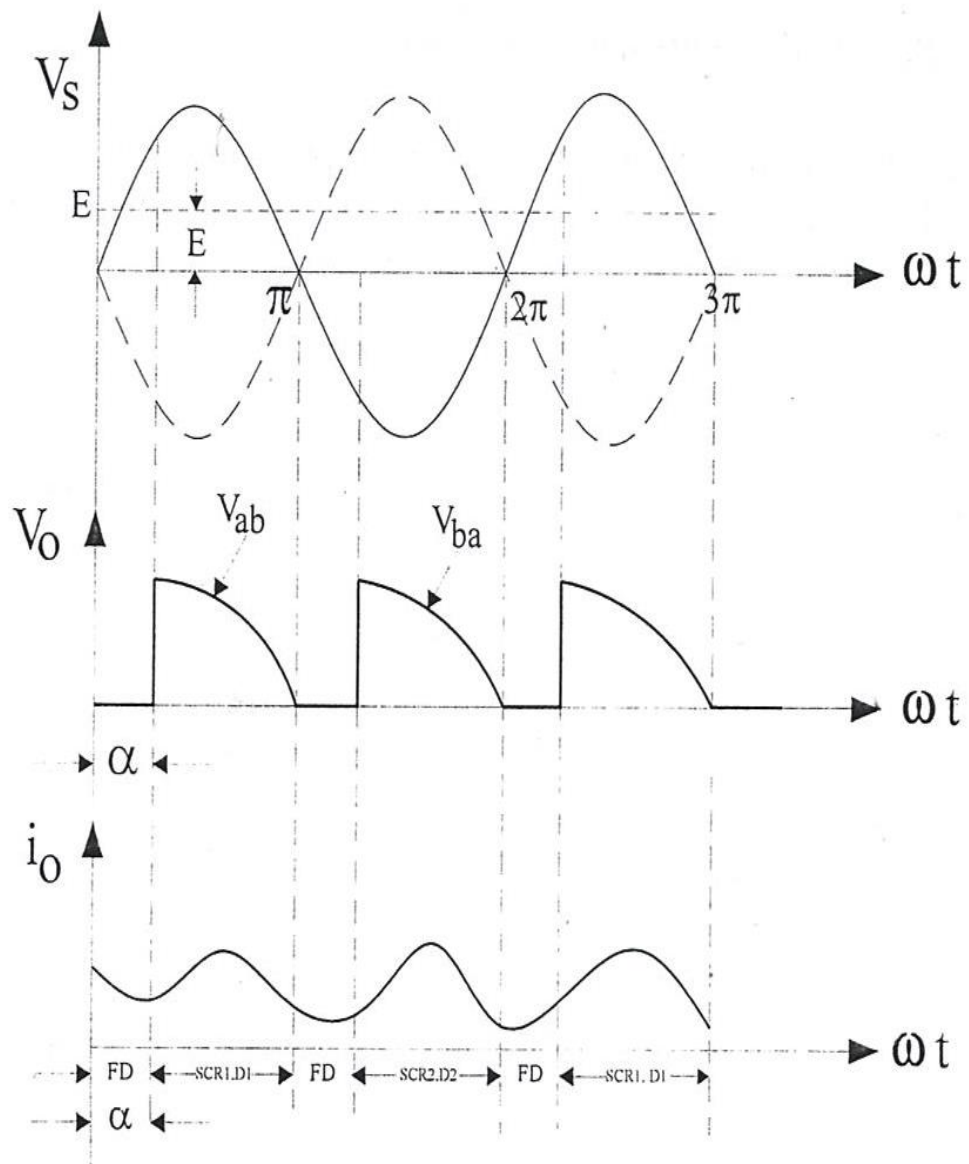
CIRCUIT DIAGRAM :



TABULATOR COLUMN:

LOAD	Voltage (v)	Total Time Period T(ms)	Non- Conductingtime 't ' (ms)	Firing angle α	Measured Voltage(V)	Calculated voltage (V)
R Load						
RL load						
RL with FD						

MODEL GRAPH:

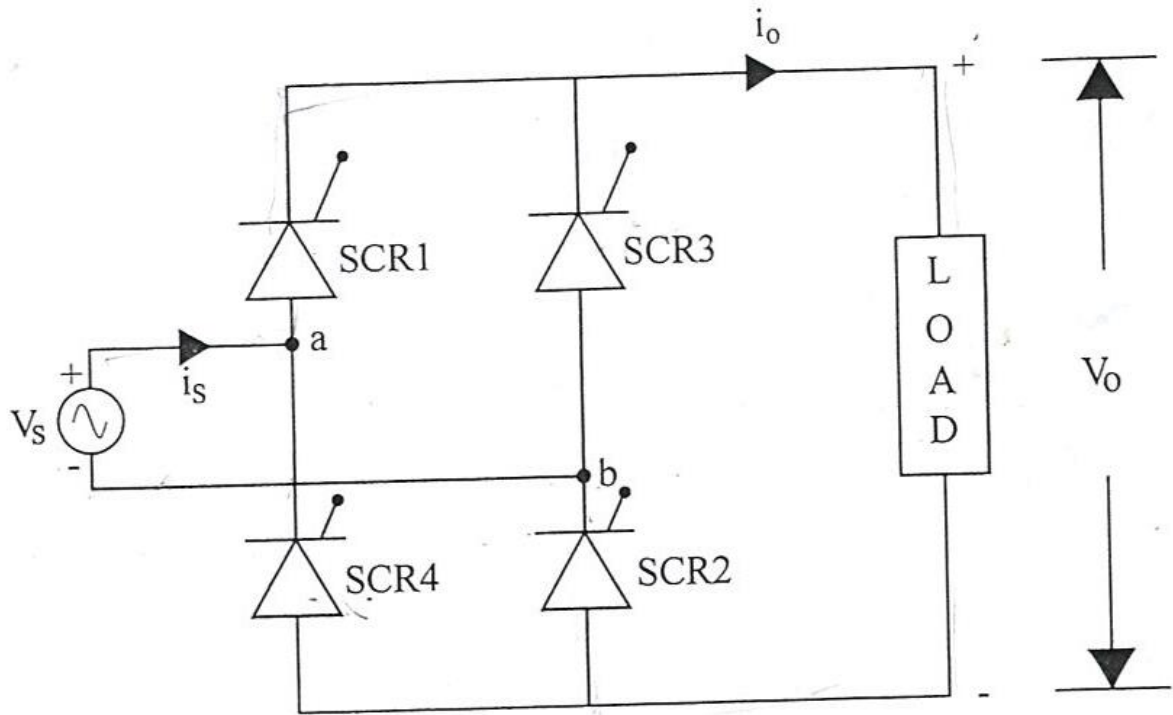


RESULT :

Thus the single phase half controlled converter is done and the output waveform is observed in CRO and verified using the calculated values and waveforms were plotted.

Load	Firing Angle (α)	Output Voltage	
		Calculated	Observed

CIRCUIT DIAGRAM :



TABULAR COLUMN:

LOAD	Voltage (v)	Total Time Period T(ms)	Non- Conductingtime 't ' (ms)	Firing angle α	Measured Voltage(V)	Calculated voltage (V)
R Load						
RL load						
RL with FD						

Ex. No. 9

Date:

AC TO DC FULLY CONTROLLED CONVERTER

AIM:

To construct a single phase fully controlled Converter and plot its response.

APPARATUS REQUIRED:

S.NO	ITEM	RANGE	TYPE	QUANTIT Y
1	Fully controlled Converter Power circuit kit	1 ϕ , 230V,10A	-	1
3	SCR firing circuit kit	1 Φ ,230V,5A	-	1
4	Isolation Transformer	230V/115-55-0-55-115	-	1
5	Auto-transformer	230V/0-270V, 4A	-	1
6	Loading Rheostat	100 Ω / 2A	-	1
7	CRO	20MHz	-	1
8	Patch chords	-	-	15

FORMULA :

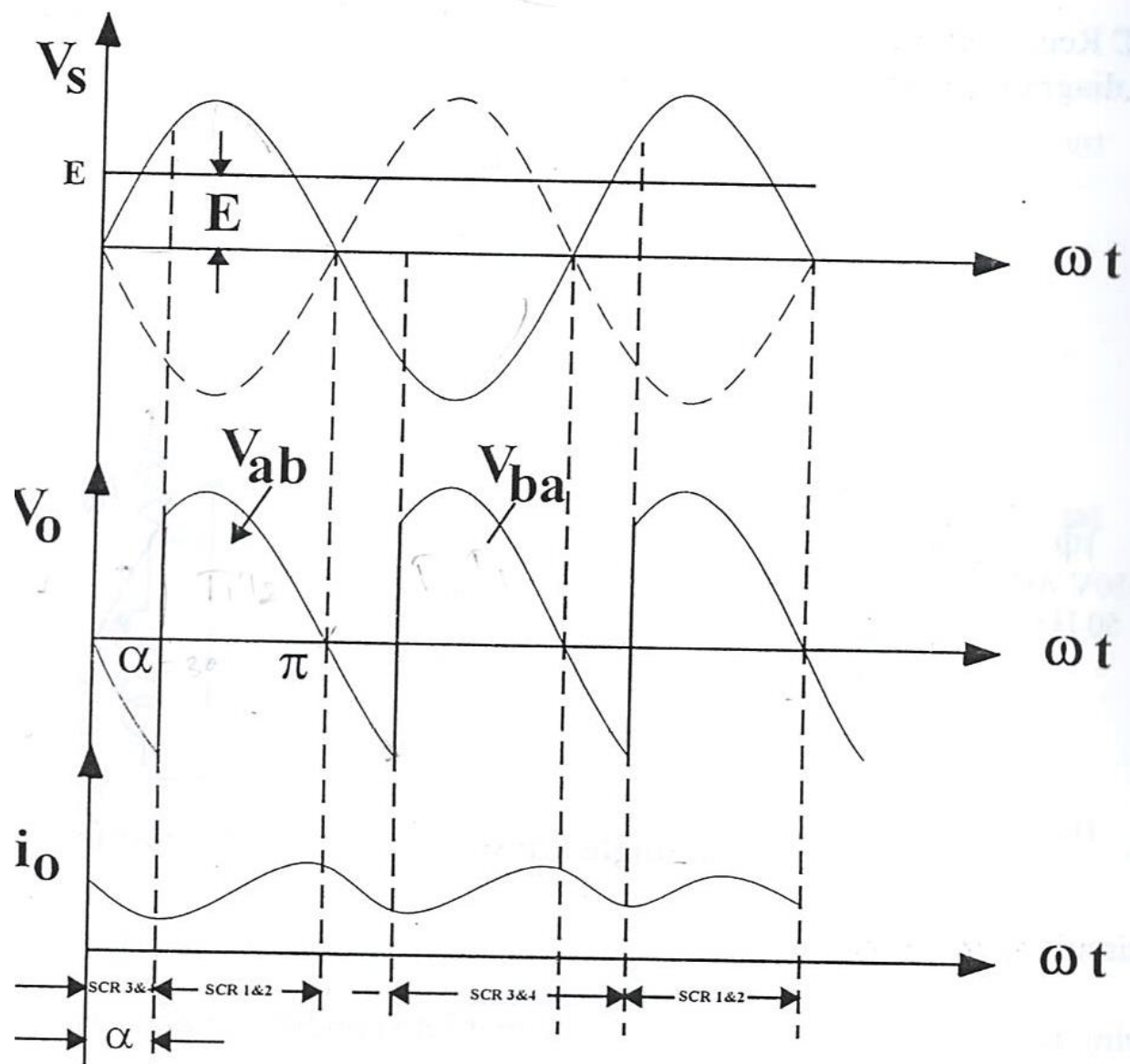
$$\text{Firing angle } \alpha_0 = \frac{t}{10\text{ms}} \times 180$$

t = non-conducting period of thyristor.

PROCEDURE:

1. Make the connections as per the circuit diagram..
2. Keep the multiplication factor of the CRO's probe at the maximum position.
3. Switch on the thyristor kit and firing circuit kit.
4. Keep the firing circuit knob at the 180 ° position.
5. Vary the firing angle in steps.
6. Note down the voltmeter reading and waveform from the CRO.
7. Switch off the power supply and disconnect.

MODEL GRAPH :



RESULT:

Thus the single phase full controlled converter is done and the output waveform is observed in CRO and verified using the calculated values and the graph is plotted.

Load	Firing Angle (α)	Output Voltage	
		Calculated	Observed

Ex:No: 10

Date:

THREE PHASE HALF CONTROLLED CONVERTER

AIM:

To study the operation of three phase SCR half controlled converter using VPET-218 module.

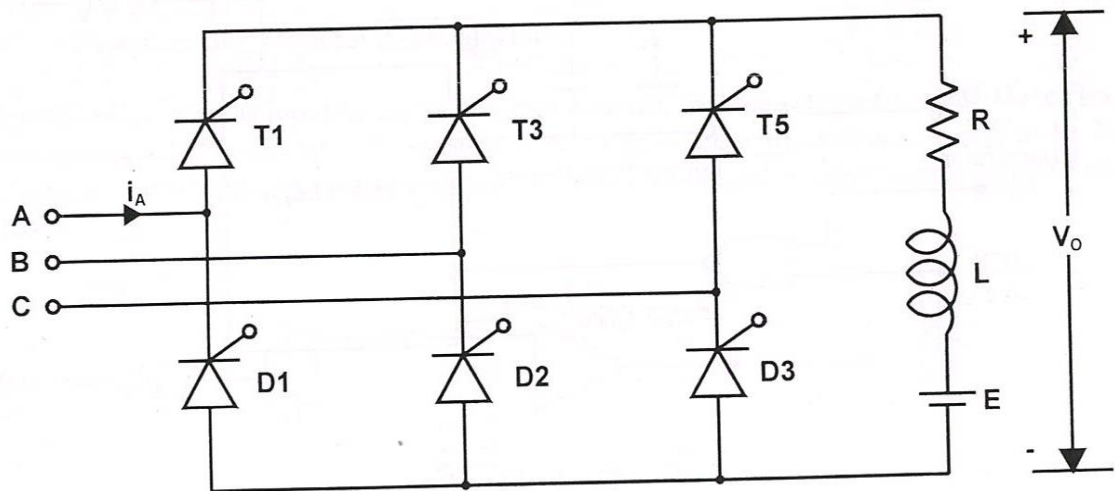
APPARATUS REQUIRED:

S.No	Apparatus	Range	Quantity
1	Kit module	VPET-218	1
2	Multimeter	-	1
3	Pulse chords	-	-
4	Power chord	-	1
5	CRO	-	1
6	Connecting wires	-	Few

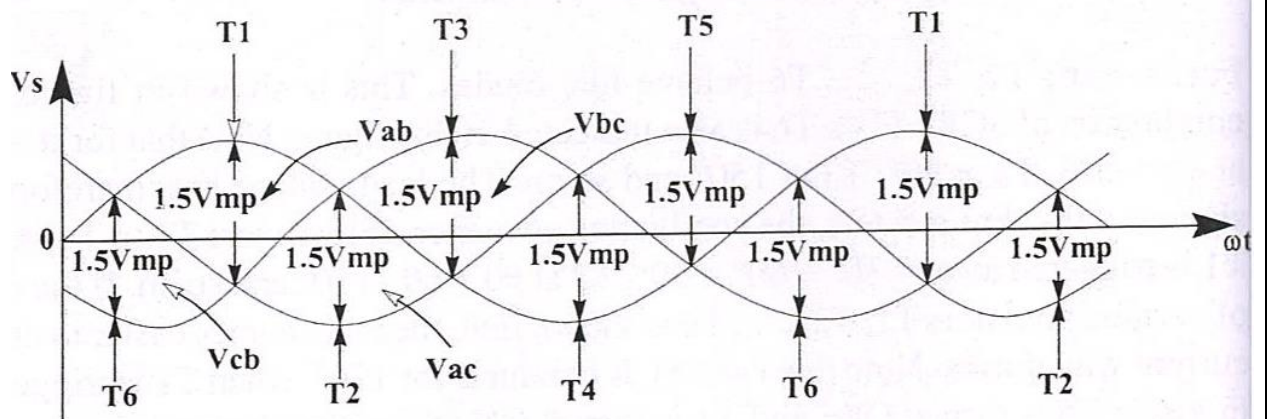
THEORY:

Three phase half controlled bridge rectifier circuit consists of three SCRs in three arms and three diodes in the other three arms. The output voltage V_0 across the load terminals is controlled by varying the firing angles of SCRs T_1 , T_2 , T_3 . The diodes D_1 , D_2 and D_3 provide merely a return path for the current to the most negative line terminal. For firing angle less than 30° , the output terminal voltage of the converter is always positive, and the freewheeling diode does not come into operation. As the firing angle is retarded beyond this point, so the load current starts to freewheel through the diode for certain periods, thus cutting off the input line current, and preventing the output terminal load voltage from swinging into the negative direction. Hence the effect of the freewheeling diode is to cause a reduction of ripple voltage of the output terminals and at the same time to divert the load current away from the input lines

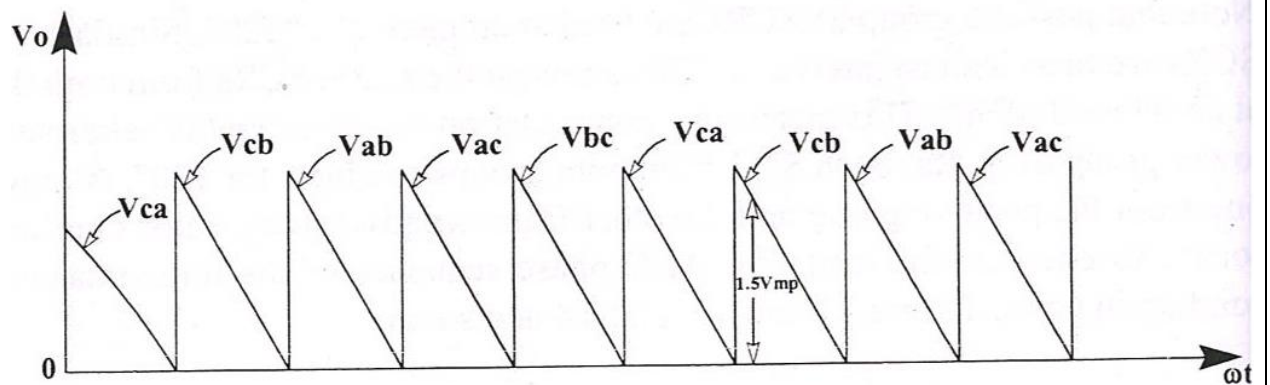
CIRCUIT DIAGRAM:



MODEL GRAPH:



T5	T1	T3	T5	T1	+ ve Group
T2	T6	T2	T4	T6	- ve Group



TABULAR COLUMN:

Voltage (v)	Total Time Period T(ms)	Non-Conducting time 't ' (ms)	Firing angle α	Measured Voltage (V)	Calculated voltage (V)

MODEL CALCULATION:**FORMULA USED:**

Average output voltage,

$$V_o = (3\sqrt{3} V_m / 2\pi) * (1 + \cos\alpha)$$

Where,

V_m = Peak phase voltage, Volts

α = Firing angle, degrees

PROCEDURE:

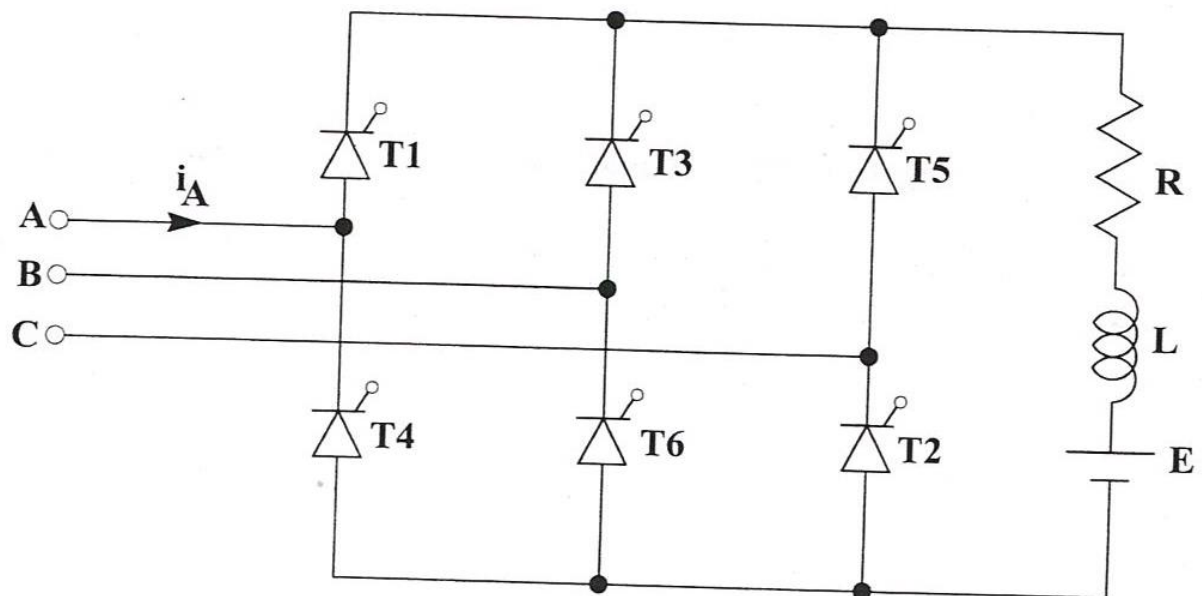
1. Switch ON the power supply ON/OFF switch.
2. Switch ON the pulse ON/OFF switch.
3. Vary the firing angle step by step in the range $180^\circ - 0^\circ$.
4. For each firing angle observe the output waveform through CRO.
5. Tabulate the readings.

RESULT:

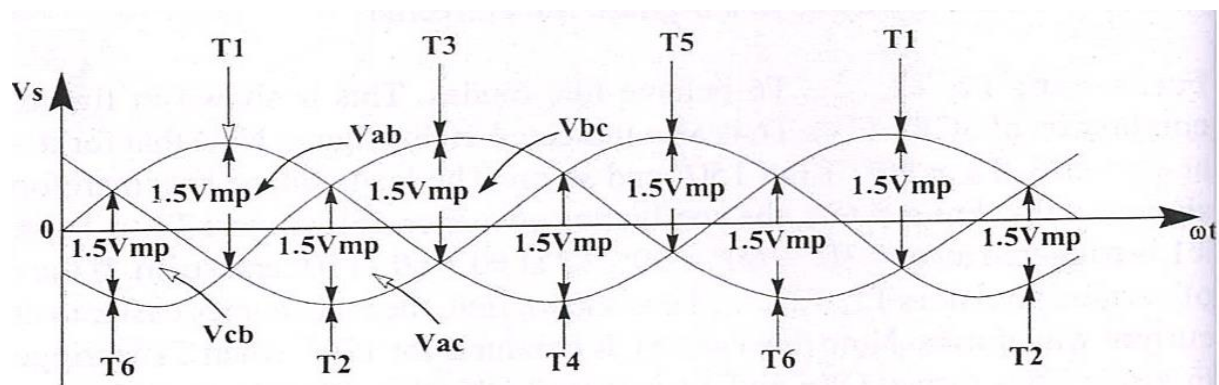
Thus the three phase half controlled converter is done and the output waveform is observed in CRO and verified using the calculated values.

Load	Firing Angle (α)	Output Voltage	
		Calculated	Observed

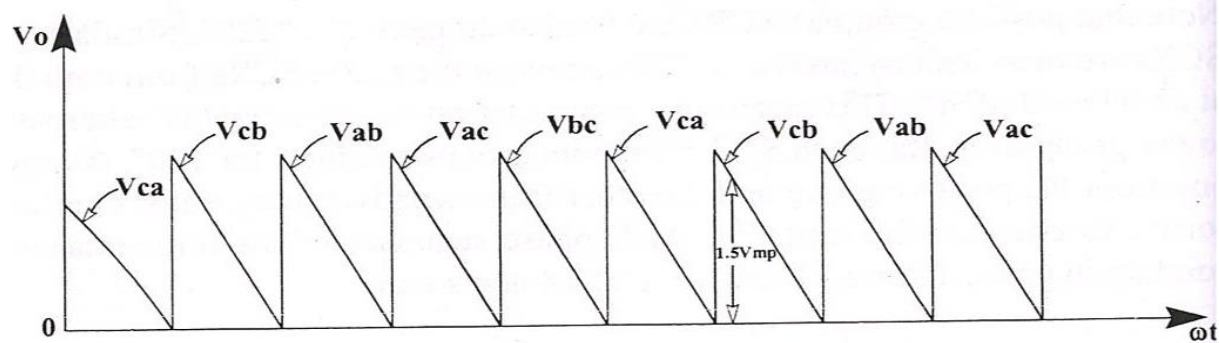
CIRCUIT DIAGRAM:



MODEL GRAPH:



T5	T1	T3	T5	T1	+ ve Group
T2	T6	T2	T4	T6	- ve Group



Ex.No:11

Date:

THREE PHASE FULLY CONTROLLED CONVERTER

AIM:

To study the operation of three phase SCR fully controlled converter using VPET-215 module

APPARATUS REQUIRED:

S.No	Apparatus	Range	Quantity
1	Kit module	VPET-215	1
2	Multimeter	-	1
3	Pulse chords	-	Few
4	Power chord	-	1
5	CRO	-	1
6	Connecting wires	-	Few

THEORY:

A three-phase fully-controlled bridge rectifier can be constructed using six SCRs. The bridge circuit has two halves, the positive half consisting of the SCRs T1, T3 and T5 and the negative half consisting of the SCRs T2, T4 and T6. At any time, one SCR from each half conducts when there is current flow. The SCRs are triggered in the sequence T1, T2, T3, T4, T5, T6 and T1 and so on. When the SCRs are fired at 0° firing angle, the output of the bridge rectifier would be the same as that of the circuit with diodes. For instance, it is seen that D1 starts conducting only after $\theta = 30^\circ$. In fact, it can start conducting only after $\theta = 30^\circ$, since it is reverse-biased before $\theta = 30^\circ$. The bias across D1 becomes zero when $\theta = 30^\circ$ and diode D1 starts getting forward biased only after $\theta = 30^\circ$.

TABULAR COLUMN:

Voltage (v)	Total Time Period	Non-Conducting time	Firing angle α	Measured Voltage	Calculated voltage

MODEL CALCULATION:**FORMULA USED:**

Average output voltage,

$$V_o = (3\sqrt{3} V_m / \pi) \cos \alpha$$

Where,
 V_m = Peak phase voltage, Volts
 α = Firing angle, degrees

PROCEDURE:

1. Switch ON the power supply ON/OFF switch.
2. Switch ON the pulse ON/OFF switch.
3. Vary the firing angle step by step in the range $180^\circ - 0^\circ$.
4. For each firing angle observe the output waveform through CRO.
5. Tabulate the readings.

RESULT:

Thus the three phase half controlled converter is done and the output waveform is observed in CRO and verified using the calculated values

Load	Firing Angle (α)	Output Voltage	
		Calculated	Observed

Ex. No. 12

Date:

STEP UP AND STEP DOWN MOSFET BASED CHOPPERS

AIM:

To construct Step down & Step up MOSFET based choppers and to draw its output response.

APPARATUS REQUIRED:

S.NO	ITEM	RANGE	QUANTIT Y
1	Step up & Step down MOSFET based chopper kit		1
2	CRO	20 MHZ	1
3	Patch chords		15

THEORY:

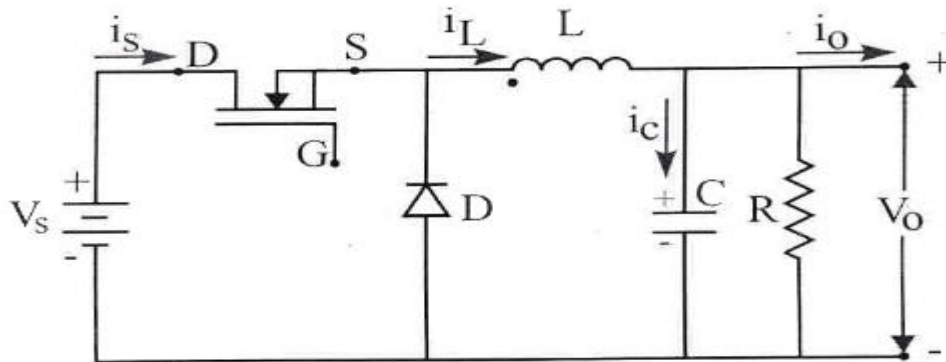
This experiment is intended to study the closed loop operation of Buck-Boost converter. The set voltage to the PWM generator is set at 1V. Feedback voltage from Buck-Boost converter power circuit is connected to the PWM generator circuit. On varying the DC input voltage slowly from 0 to 15V, the output voltage is measured as constant

PROCEDURE (STEP UP CHOPPER & STEP DOWN CHOPPER) :

1. Initially keep all the switches in the OFF position
2. Initially keep duty cycle POT in minimum position
3. Connect banana connector 24V DC source to 24V DC input.
4. Connect the driver pulse [output to MOSFET input
5. Switch on the main supply
6. Check the test point waveforms with respect to ground.
7. Vary the duty cycle POT and tabulate the T_{on} , T_{off} & output voltage
8. Trace the waveforms of V_o Vs & I_o
9. Draw the graph for V_o Vs Duty cycle, K

CIRCUIT DIAGRAM

STEP DOWN CHOPPER :

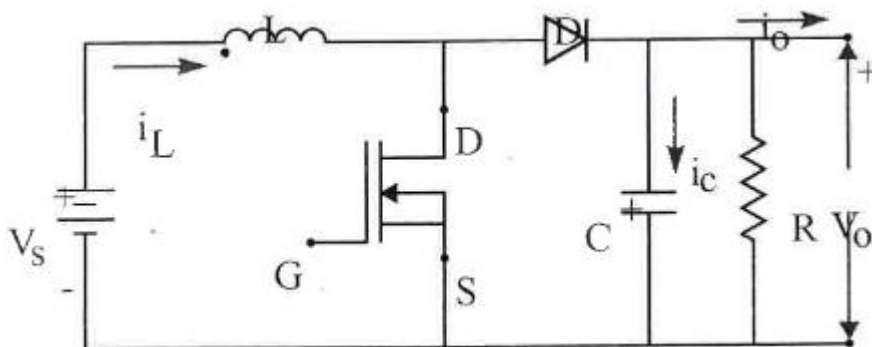


TABULAR COLUMN (STEP UP CHOPPER):

$V_s = \underline{\hspace{2cm}}$ V

S.NO	T _{ON} (sec)	T _{OFF} (sec)	T (sec)	Duty Ratio, $k = T_{ON} / T$	Measured Voltage (V)	Calculated V_o $V_o = k / (1 - k) V_s (V)$

STEP UP CHOPPER:

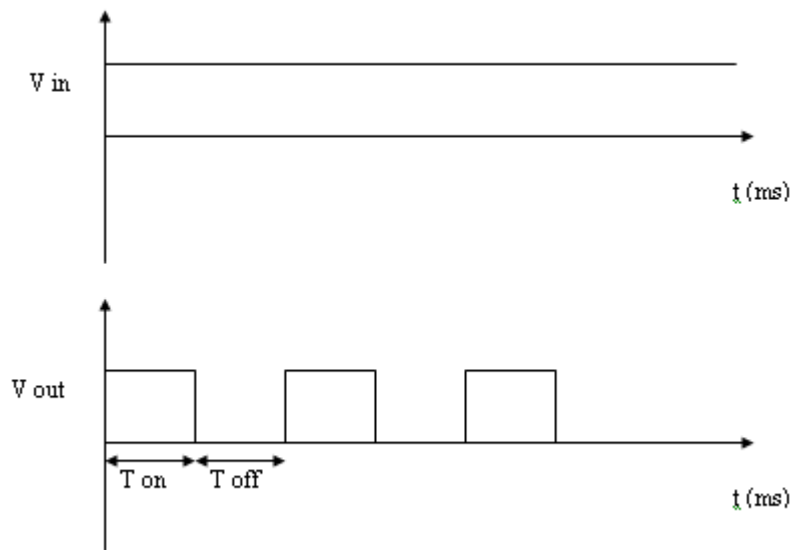


TABULAR COLUMN (STEP DOWN CHOPPER):

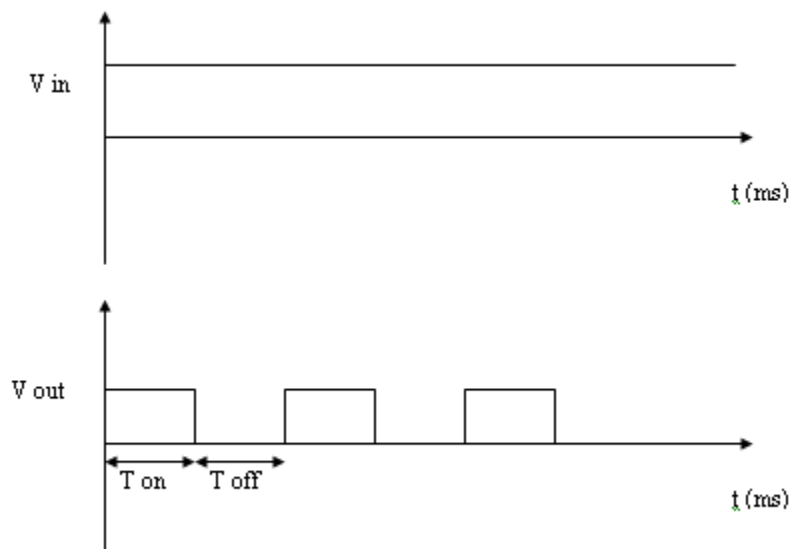
$V_s = \underline{\hspace{2cm}}$ V

V_s	T _{ON} (sec)	T _{OFF} (sec)	T (sec)	Duty Ratio, $k = T_{ON} / T$	Measured Voltage (V)	Calculated V_o $V_o = k V_s (V)$

MODEL GRAPH (STEP UP CHOPPER) :



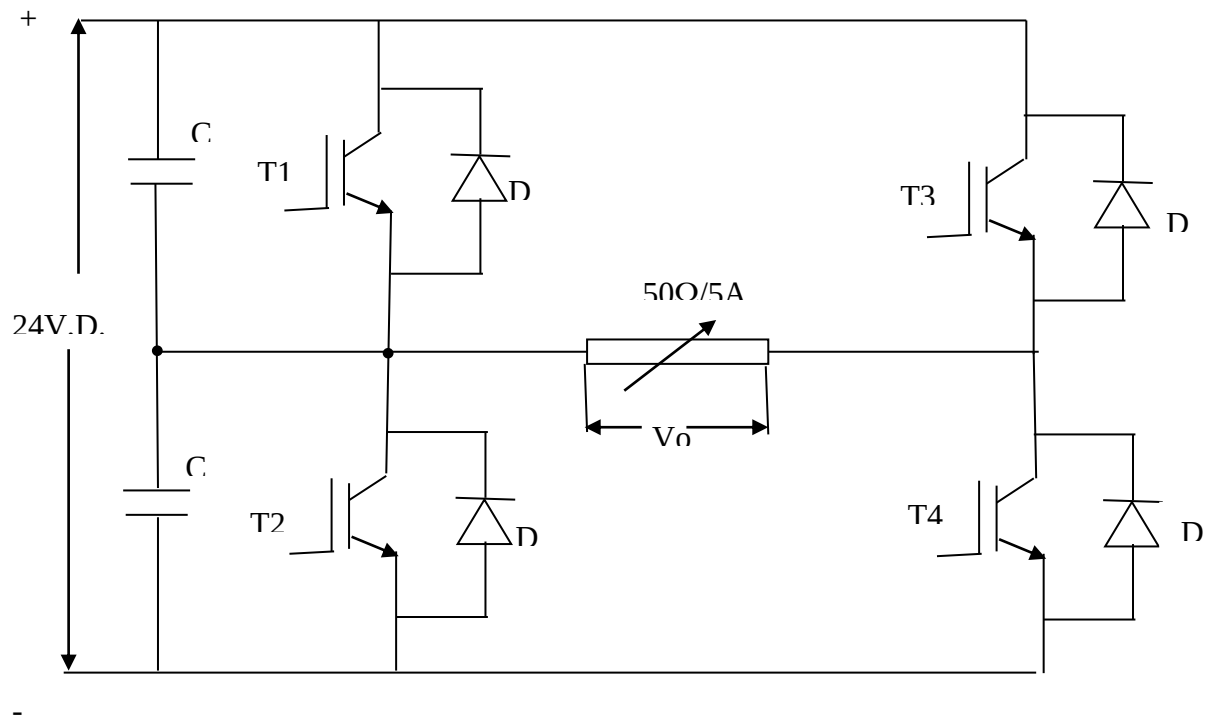
MODEL GRAPH (STEP DOWN CHOPPER) :



RESULT:

Thus the output response of Step down & Step up MOSFET based choppers were drawn.

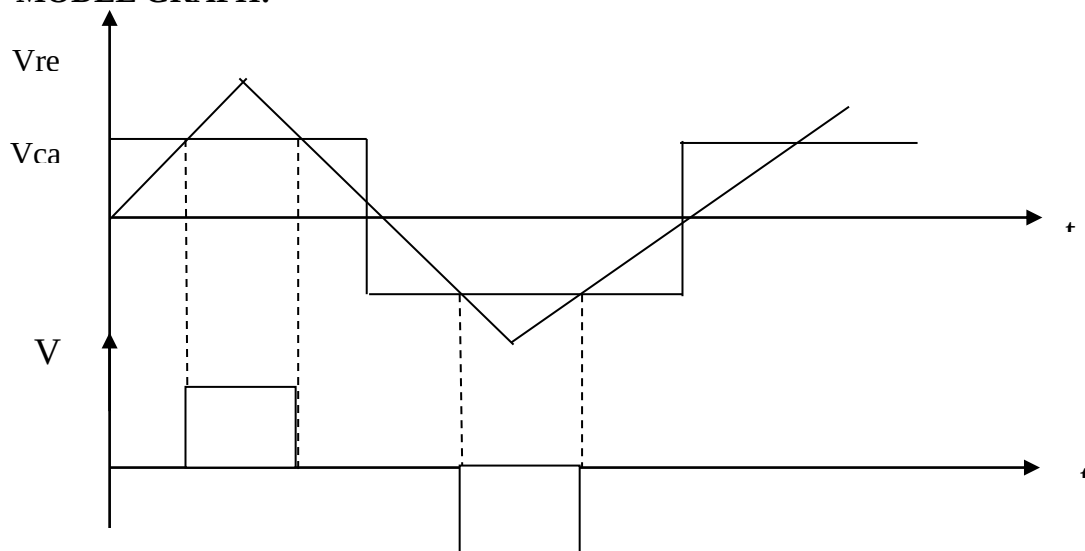
CIRCUIT DIAGRAM:



TABULAR COLUMN:

S.no	Output voltage (v)	Time (ms)

MODEL GRAPH:



Ex. No. 13

Date:

IGBT BASED SINGLE PHASE PWM INVERTER

AIM :

To obtain Single phase output wave forms for IGBT based PWM inverter

APPARATUS REQUIRED:

S.No.	ITEM	RANGE	TYPE	QUANTITY
1	IGBT Based PWM inverter Kit	220/10A		1
2	CRO	20MHZ		1
3	Patch Chord			10
4	Load rheostat	50 Ω /5A		1

THEORY:

It consists of four IGBTs S₁, S₂, S₃, S₄ and four inverse parallel diodes D₁, D₂, D₃, D₄. The diodes are essential to conduct the reactive current, and thereby to feed back the stored energy in the inductor to the DC source. These diodes are known as feed back diodes. For many industrial applications the output AC voltage of the inverter must be sinusoidal in shape and the amplitudes and frequency must be controllable. This is achieved by PWM of the inverter switches.

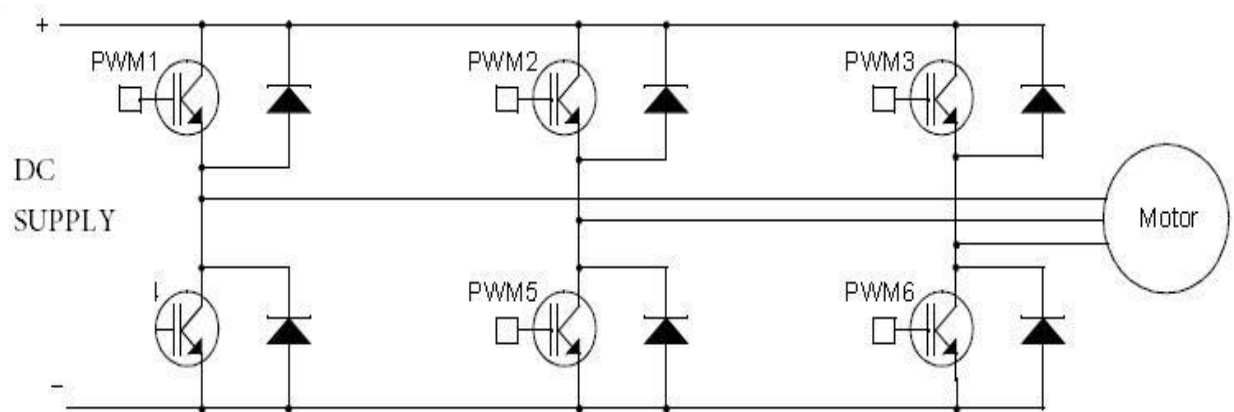
PROCEDURE :

1. Make the connection as per the circuit diagram.
2. Connect the gating signal from the inverter module.
3. Switch ON D.C 24 V.
4. Keep the frequency knob to particulars frequency.
5. Observe the rectangular and triangular carrier waveforms on the CRO.
6. Obtain the output waveform across the load Rheostat.

RESULT :

Thus the output waveform for IGBT inverter (PWM) was obtained.

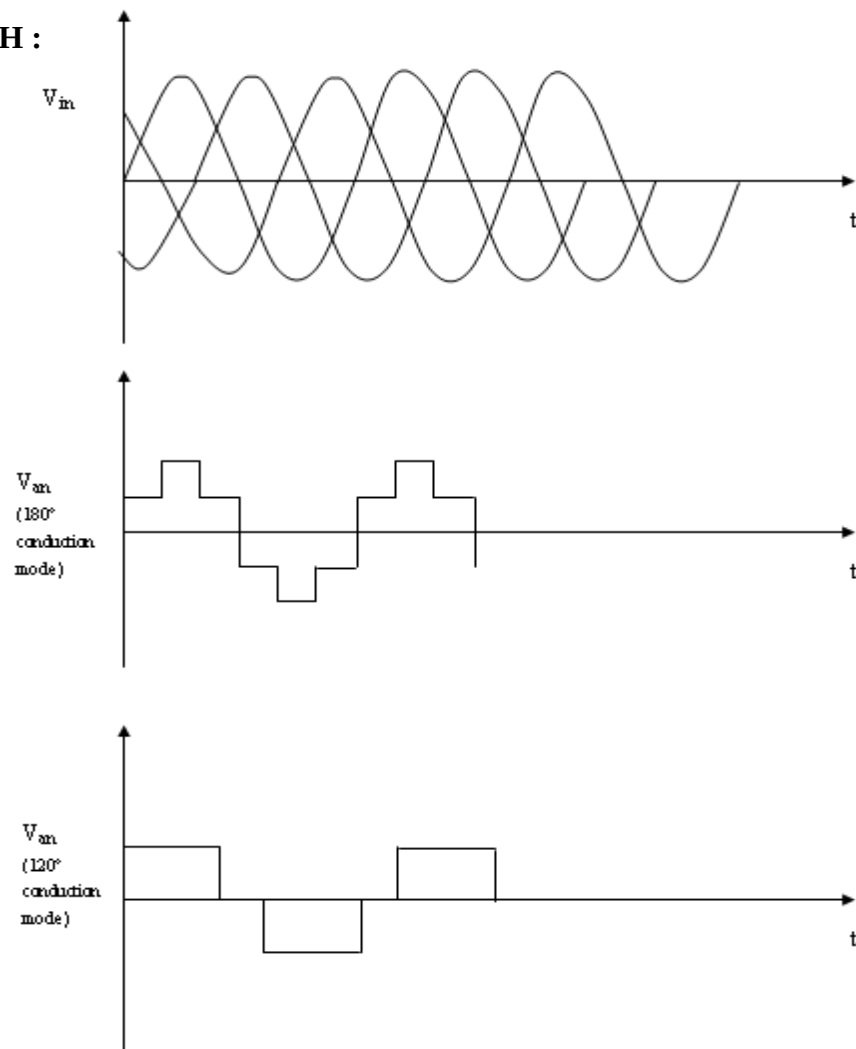
CIRCUIT DIAGRAM:



TABULAR COLUMN:

S.No.	Output Voltage (V)	Time (ms)

MODEL GRAPH :



Ex. No. 14

Date:

IGBT BASED THREE PHASE PWM INVERTER

AIM:

To obtain Three phase output wave forms for IGBT based PWM inverter

APPARATUS REQUIRED:

S.No.	ITEM	RANGE	TYPE	QUANTITY
1	IGBT Based PWM inverter Kit	220/10A		1
2	CRO	20MHZ		1
3	Patch Chord			10
4	Load rheostat	50 Ω /5A		1

THEORY:

The most frequently used three phase inverter circuit of three legs, one for each phase. For this configuration, output transformer is not required. Also, this circuit uses six IGBTs. The inverter configuration is also termed as six step bridge inverter. In inverter terminology, a step is defined as a change in the firing from one IGBT to the next IGBT in proper sequence. For one cycle of 360°, each step would be of 60° for a six step inverter. This means that IGBT's would be gated at regular intervals of 60°.

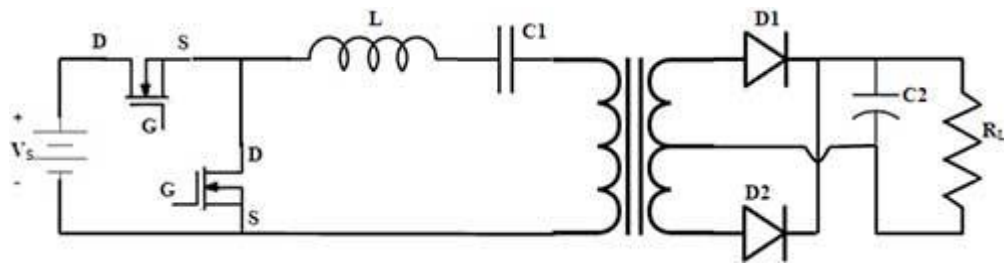
PROCEDURE:

1. Make the connection as per the circuit diagram.
2. Connect the gating signal from the inverter module.
3. Switch ON D.C 24 V.
4. Keep the frequency knob to particulars frequency.
5. Observe the input and output waveforms for 180° conduction mode and 120° conduction mode in the CRO.
6. Obtain the output waveform across the load Rheostat.

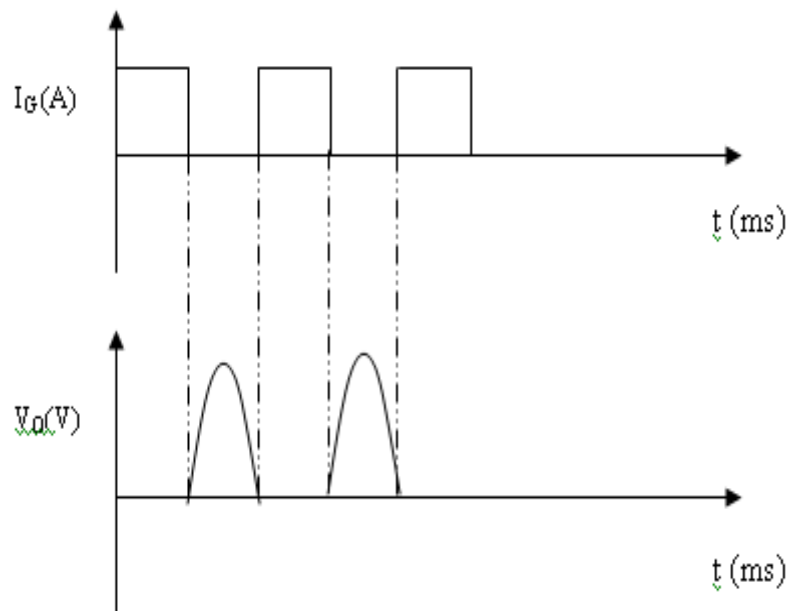
RESULT :

Thus the output waveform for IGBT inverter (PWM) was obtained.

CIRCUIT DIAGRAM (ZVS RESONANT CONVERTER) :



MODEL GRAPH (ZVS RESONANT CONVERTER) :



TABULAR COLUMN (ZVS RESONANT CONVERTER) :

RESONANT FREQUENCY =

S.No.	Input voltage (V_i) V	Switching frequency (hz)	Output voltage V_o (V)	Output current i_o (A)

Ex. No. 15

Date:

RESONANT DC TO DC CONVERTER

AIM:

To study Zero Voltage Switching Resonant converter and Zero Current Switching Resonant Converter and plot its output waveforms.

APPARATUS REQUIRED:

S.No.	ITEM	RANGE	TYPE	QUANTITY
1	Resonant Converter Module			1
2	Loading rheostat	100 Ω / 2A		1
3	CRO	20 MHZ		1
4	Patch chords			15

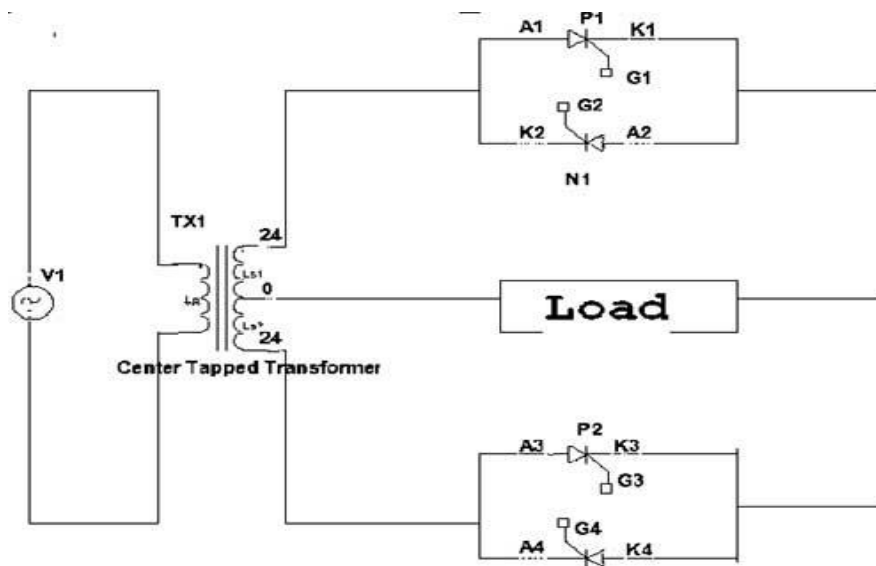
PROCEDURE (ZVS & ZCS RESONANT CONVERTER):

1. Make the connections as per the circuit diagram.
2. Switch on the resonant converter module.
3. Keep the frequency knob of the firing circuit kit below the resonance Frequency of power circuit kit
4. Switch on the DC power supply connected to the power circuit kit and Switch on the firing circuit kit
5. Vary the frequency knob of the firing circuit kit
6. Observe the waveform from the CRO.
7. Repeat the same procedure for different values of switching frequency.
8. Switch of the power supply and disconnect the connection

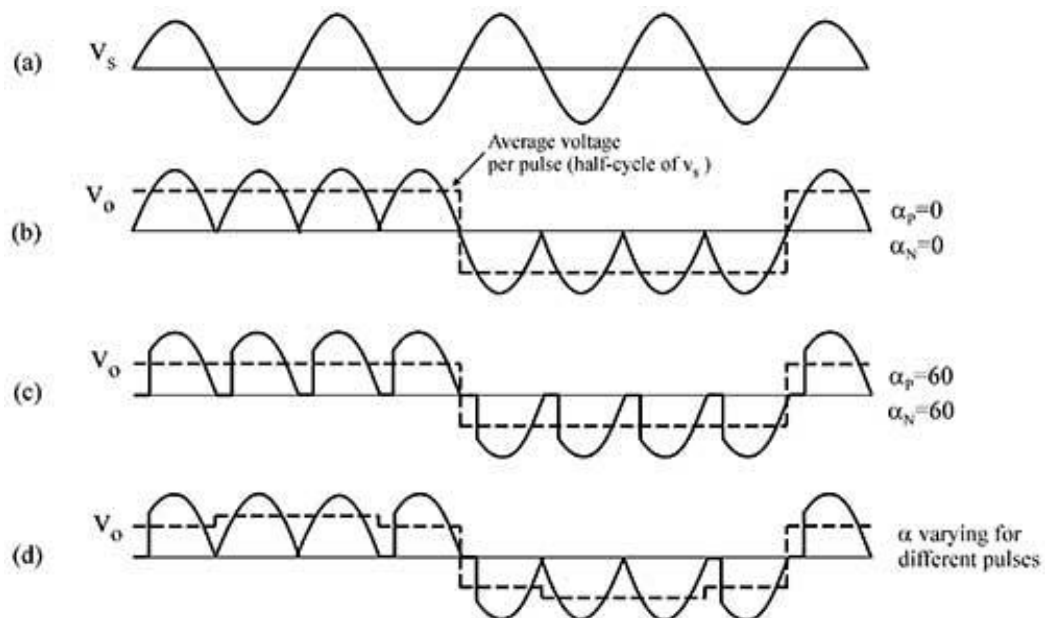
RESULT :

Thus ZVS and ZCS Resonant Converter operation were studied and its output waveform were plotted.

CIRCUIT DIAGRAM:



Voltage & Current waveform for a cycloconverter with discontinuous load current



Ex. No. 16

Date:

CYCLOCONVERTER

AIM :

To determine the working of cycloconverter and plot its output waveform with different frequency.

APPARATUS REQUIRED :

S.No.	ITEM	RANGE	TYPE	QUANTITY
1	cycloconverter power circuit kits	1 Φ ,230 V , 2 A		1
2	Loading rheostat	100 Ω / 2A		1
3	Regulated power supply	(0 – 30 V) / 2 A		1
4	CRO	20 MHZ		1
5	Patch chords			as req.

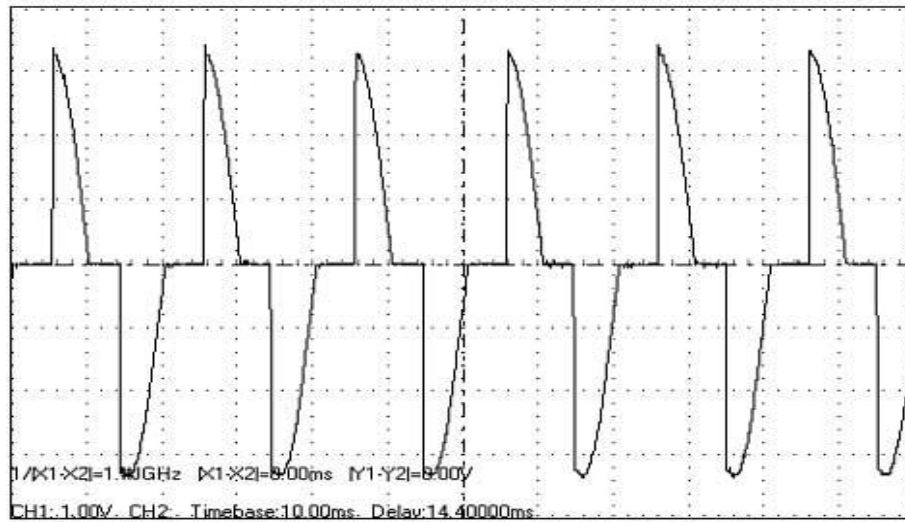
THEORY:

The basic control principle of an ideal cycloconverter is to continuously modulate the firing angles of individual converter, so that each produces the same sinusoidal ac voltage at its output terminal. Thus the voltages of the two generators in fig. 1 have the same amplitude, frequency and phase and the voltage at the output terminals of the cycloconverter is equal to the voltage of either of these generators. It is possible for the mean power to flow either “to” or “from” the output terminals and the cycloconverter is inherently capable of operation with loads of any phase angle within a complete spectrum of 360.

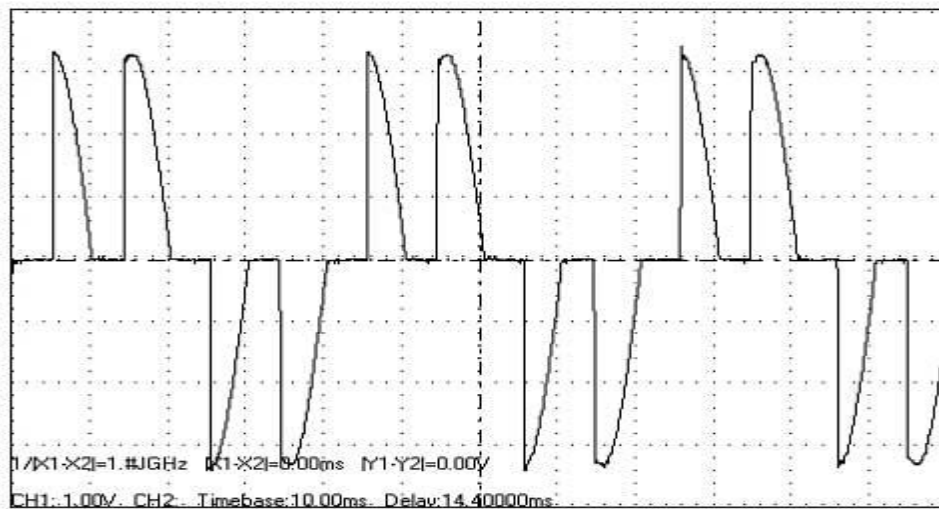
Bridge Configuration:

Two Single phase fully controlled bridges are connected in opposite direction. Bridge 1 supplies load current in the positive half of the output cycle and bridge 2 supplies load current in the negative half of the output cycle. The two bridges should not conduct together as this will produce a short circuit at the input. Instead of one thyristor in the centre tap transformer configuration, two thyristor come in series with the each voltage source in bridge configuration. For resistive loads the SCR undergo natural commutation and produce discontinuous current operation..For inductive load ,the load current may be continuous or discontinuous depending upon the firing angle and load power factor. The load voltage and current waveforms for continuous and discontinuous load current are drawn.

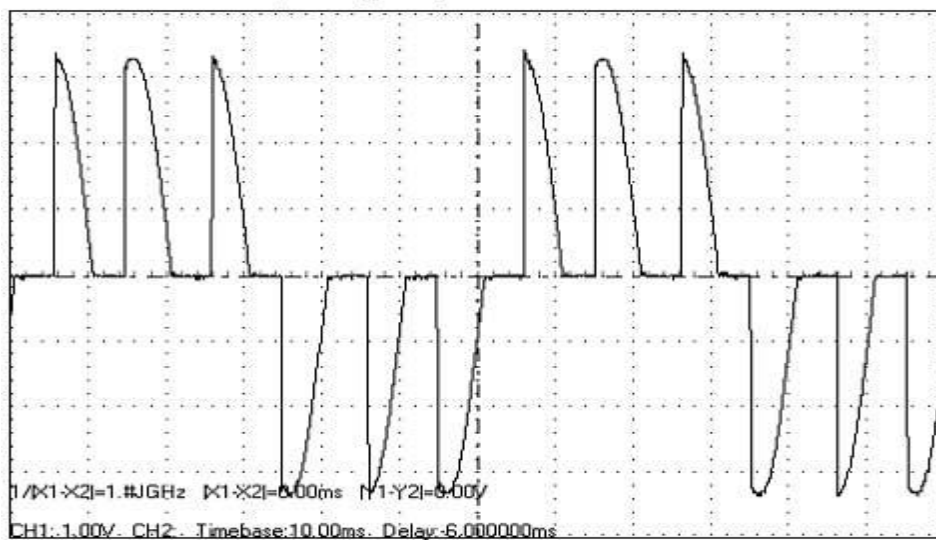
a) Frequency Select =F



a) Frequency Select =F/2



a) Frequency Select =F/3



Voltage & Current waveform for a 1- ϕ to 1- ϕ cycloconverter with continuous load current

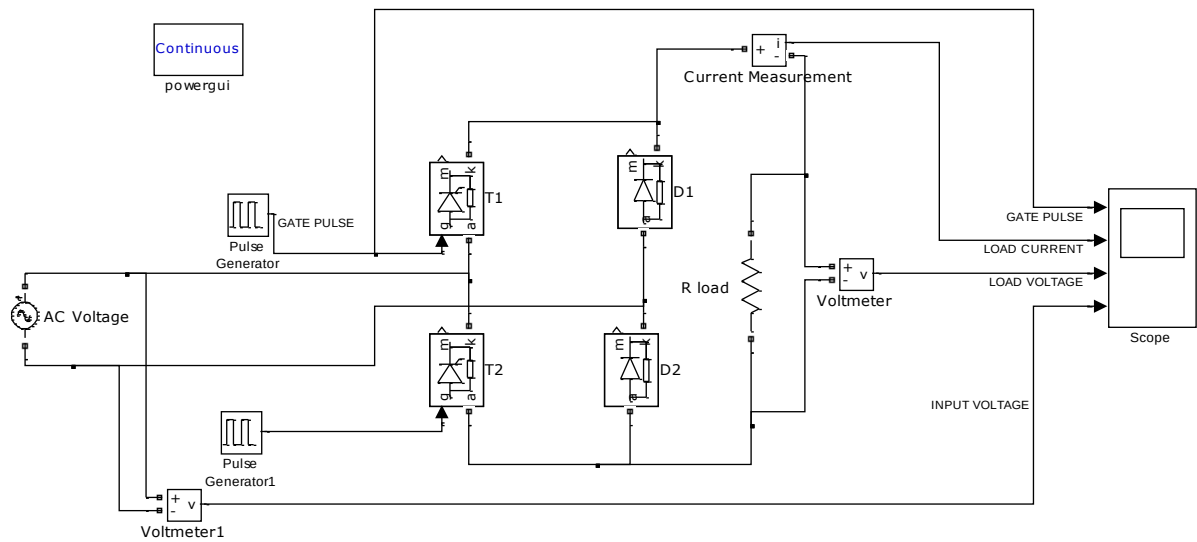
PROCEDURE:

1. 1.Make the connections as per the circuit diagram.
2. Switch on the resonant converter module.
3. Keep the frequency knob of the firing circuit kit below the resonance Frequency of power circuit kit
4. Switch on the DC power supply connected to the power circuit kit and Switch on the firing circuit kit
5. Vary the frequency knob of the firing circuit kit
6. Observe the waveform from the CRO.
7. Repeat the same procedure for different values of switching frequency.
8. Switch of the power supply and disconnect the connection

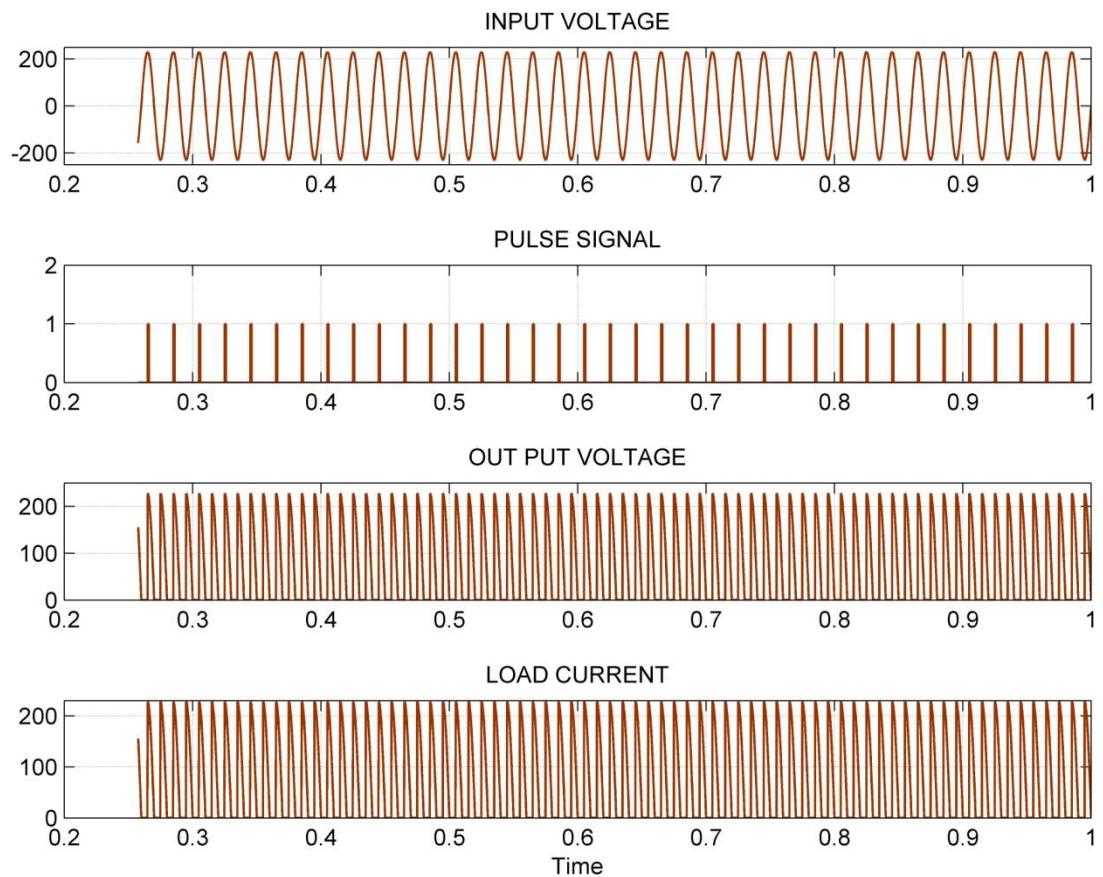
RESULT:

Thus a cycloconverter is done experimentally and its output waveform were plotted.

SINGLE PHASE HALF CONTROLLED CONVERTER WITH R LOAD **SIMULATION DIAGRAM FOR R LOAD: -**



MODEL WAVEFORMS: -



SINGLE PHASE HALF CONTROLLED CONVERTER WITH R AND RL LOAD

AIM: -

To simulate and study the Single Phase Half Controlled Converter with R and RL Load in different firing angles.

APPARATUS REQUIRED: -

Sl. No.	Description	Quantity
01	Computer	01
02	Printer	01

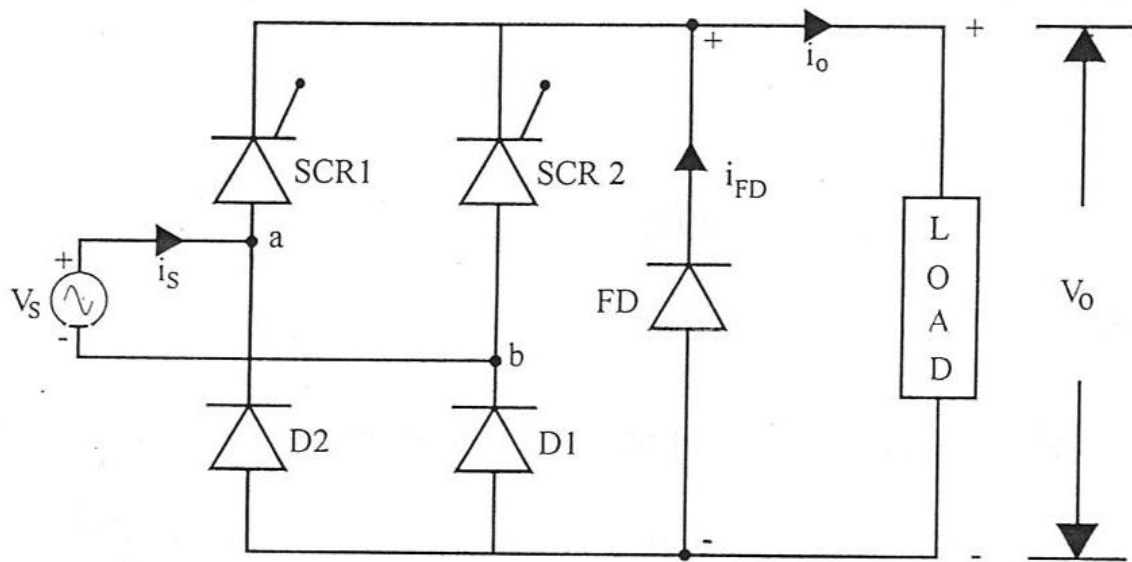
SOFTWARE REQUIRED: -

MATLAB 6.1 or above.

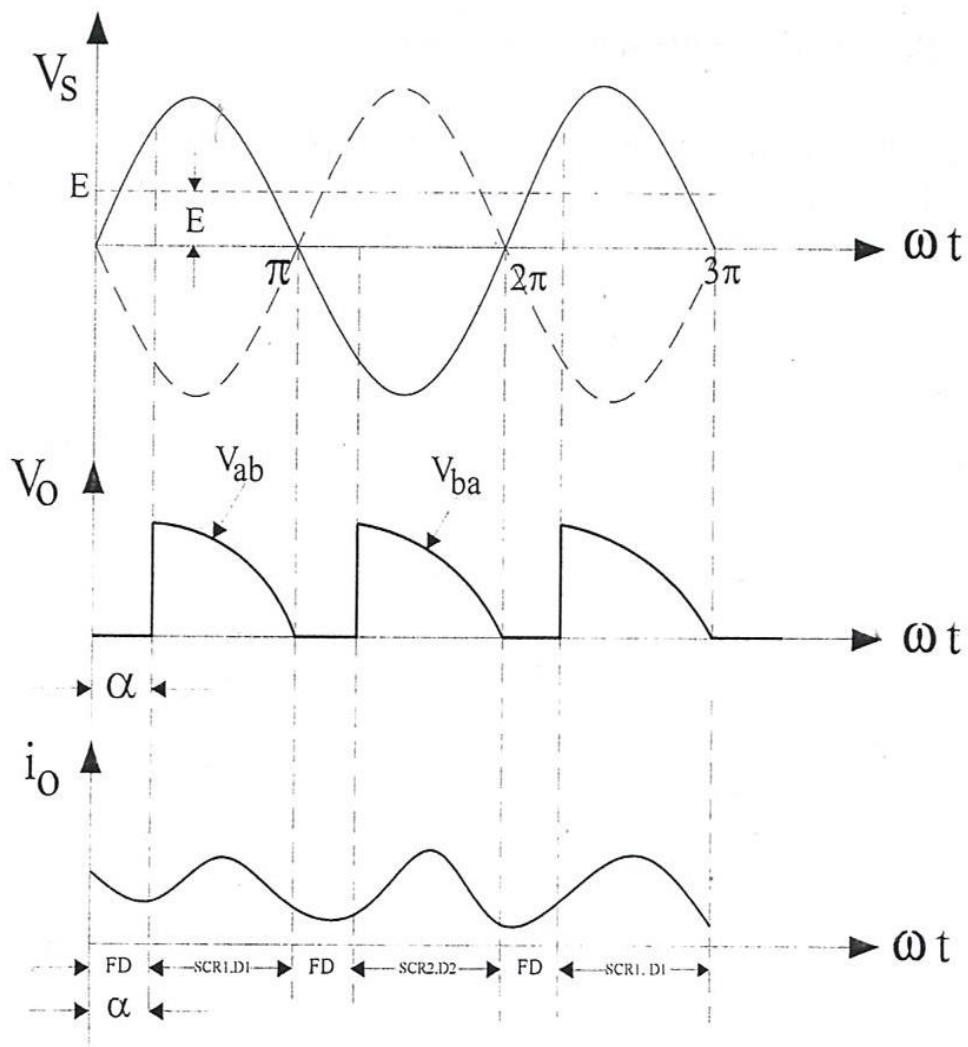
PROCEDURE FOR R LOAD: -

1. Study the circuit diagram, operation and model waveforms of a single phase half controlled converter.
2. Open the MATLAB software by double-clicking the MATLAB icon on the desktop and click the shortcut "Simulink" to open the "Simulink Library Browser".
3. Create a new model and click on "SimPowerSystems" block in "Simulink Library Browser".
4. Drag the "powergui" block in SimPowerSystems submenu to the new model.
5. Double click on the "Electrical Source" block in SimPowerSystems submenu to select the single phase supply source "AC Voltage Source" and drag it to the new model. Set the peak amplitude of voltage as 110 Volts.
6. In the SimPowerSystems submenu, select "Power Electronics" block to pick the "Thyristor" & "Diode" and drag them to the new model.
7. In the SimPowerSystems submenu, select "Elements" block to pick the load "Series RLC branch" and drag it to the new model. Set the values of L & C as zero by double-clicking on the load.
8. In the SimPowerSystems submenu, select "Measurements" block to pick the "Voltage Measurement" & "Current Measurement" and drag them to the new model.
9. In the Simulink submenu, select "Commonly Used Blocks" to pick the "SCOPE" and drag it to the new model.
10. In the Simulink submenu, select "Sources" to pick the "Pulse Generator" for T_1 & T_2 and drag it to the new model.
11. Arrange the components as per the circuit diagram and connect them.
12. Set the values of input source voltage parameters such as amplitude, phase angle and frequency.
13. Set the pulse generator parameters such as pulse type, amplitude, period, pulse width, phase delay, etc. as per circuit design formula.
14. Set the scope axis as per the circuit requirements like input voltage, output voltage, output current, etc.
15. Set the load resistance value as 10 ohms.
16. Simulate the circuit and take the output waveforms in the scope.

CIRCUIT DIAGRAM: -



MODEL WAVEFORMS: -



FORMULA USED FOR DESIGN OF PHASE DELAY: -

1. Firing angle delay for pulse generator 1 = $\frac{(0 + \alpha) * 0.02}{360}$ sec
2. Firing angle delay for pulse generator 2 = $\frac{(180 + \alpha) * 0.02}{360}$ sec

Where, α = firing angle delay in degrees.

Frequency = 50 Hz, i.e. time period = 0.02 sec.

PHASE DELAY CALCULATIONS: -

Let the firing angle be 30° .

i.e., $\alpha = 30^\circ$

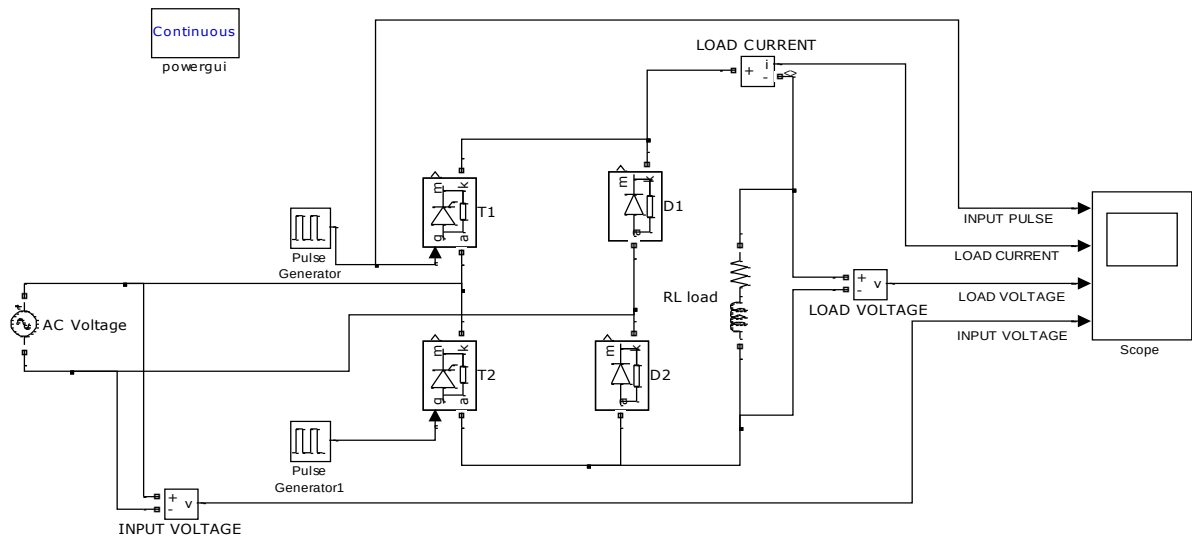
$$\begin{aligned}\text{Firing angle delay for pulse generator 1} &= \frac{(0 + 30) * 0.02}{360} \text{ sec} \\ &= 0.001667 \text{ sec.}\end{aligned}$$

$$\begin{aligned}\text{Firing angle delay for pulse generator 2} &= \frac{(180 + 30) * 0.02}{360} \text{ sec} \\ &= 0.011667 \text{ sec.}\end{aligned}$$

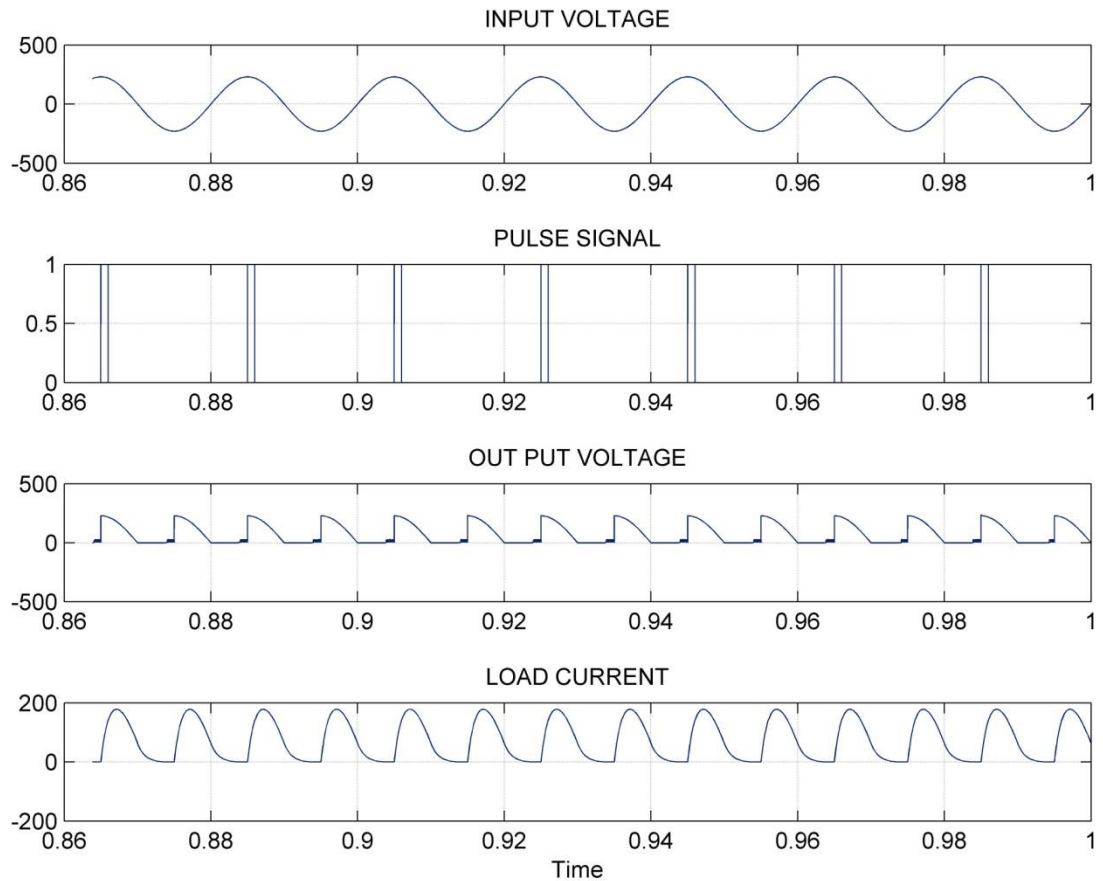
Firing angle delays in sec for pulse generators 1 & 2 for various firing angles can be calculated and tabulated as follows.

FIRING ANGLE (α) in deg	Phase angle delay for (in sec.)	
	Pulse Generator 1	Pulse Generator 2
0	0	0.01
30	0.001667	0.011667
60	0.00333	0.01333
90	0.005	0.015
120	0.00667	0.01667
150	0.00833	0.01833
180	0.01	0.02

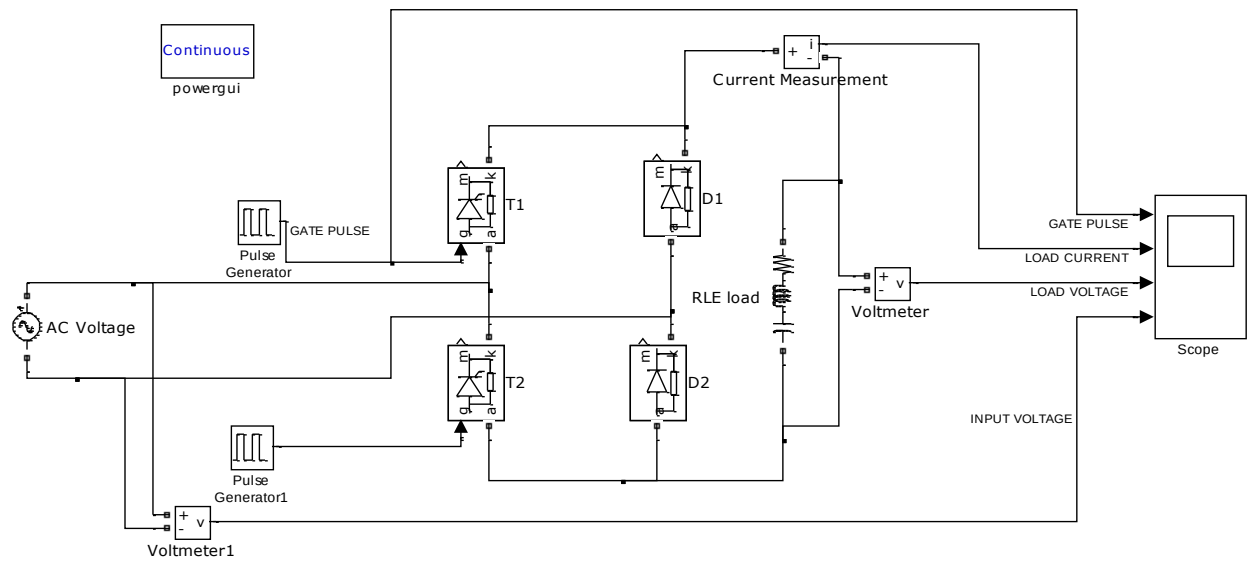
SIMULATION DIAGRAM FOR RL LOAD: -



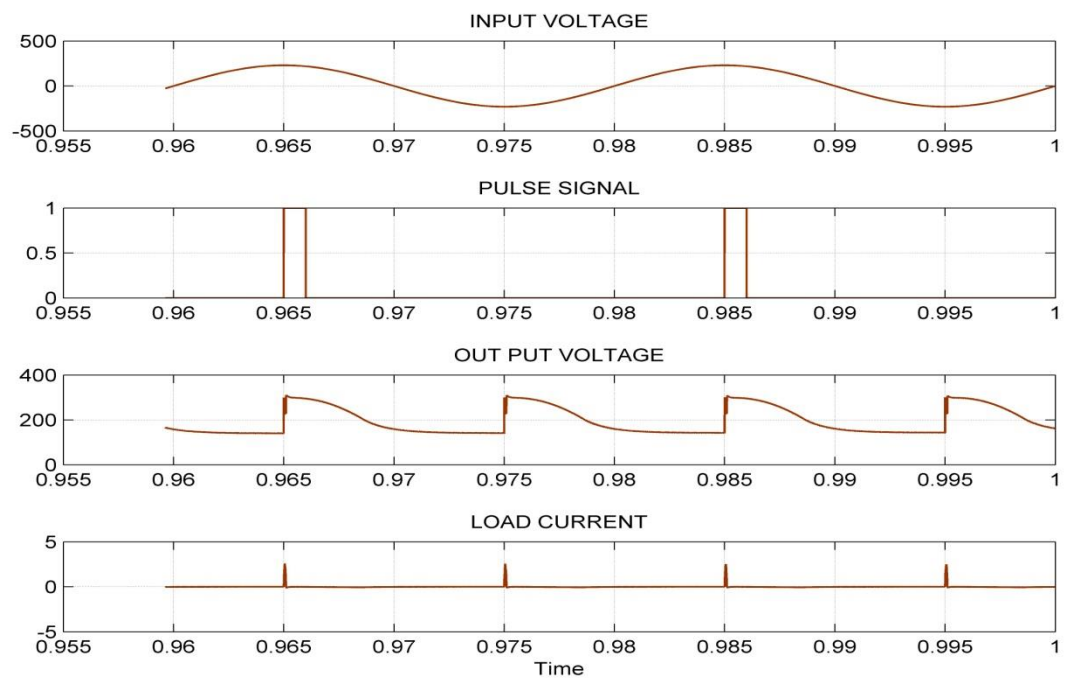
MODEL WAVEFORMS: -



SIMULATION DIAGRAM FOR RLE LOAD:



WAVEFORMS OF RLE LOAD



PROCEDURE FOR RL LOAD: -

1. Study the circuit diagram, operation and model waveforms of a single phase half controlled converter.
2. Open the MATLAB software by double-clicking the MATLAB icon on the desktop and click the shortcut “Simulink” to open the “Simulink Library Browser”.
3. Create a new model and click on “SimPowerSystems” block in “Simulink Library Browser”.
4. Drag the “powergui” block in SimPowerSystems submenu to the new model.
5. Double click on the “Electrical Source” block in SimPowerSystems submenu to select the single phase supply source “AC Voltage Source” and drag it to the new model. Set the peak amplitude of voltage as 110 Volts.
6. In the SimPowerSystems submenu, select “Power Electronics” block to pick the “Thyristor” & “Diode” and drag them to the new model.
7. In the SimPowerSystems submenu, select “Elements” block to pick the load “Series RLC branch” and drag it to the new model. Set the value of C as zero by double-clicking on the load.
8. In the SimPowerSystems submenu, select “Measurements” block to pick the “Voltage Measurement” & “Current Measurement” and drag them to the new model.
9. In the Simulink submenu, select “Commonly Used Blocks” to pick the “SCOPE” and drag it to the new model.
10. In the Simulink submenu, select “Sources” to pick the “Pulse Generator” for T_1 & T_2 and drag it to the new model.
11. Arrange the components as per the circuit diagram and connect them.
12. Set the values of input source voltage parameters such as amplitude, phase angle and frequency.
13. Set the pulse generator parameters such as pulse type, amplitude, period, pulse width, phase delay, etc. as per circuit design formula.
14. Set the scope axis as per the circuit requirements like input voltage, output voltage, output current, etc.
15. Set the load resistance and inductance values as 10 ohms and 1 mH respectively.
16. Simulate the circuit and take the output waveforms in the scope.

RESULT: -

Thus the single phase half controlled converter was simulated with R and RL load of resistance of 10 ohms and inductance of 1 mH for the firing angles of 45° and 135° and the related waveforms were obtained.

Expt. No.: 18

Date:

SINGLE PHASE FULLY CONTROLLED CONVERTER WITH R AND RL LOAD

AIM: -

To simulate and study the Single Phase Fully Controlled Converter with R and RL Load for different firing angles.

APPARATUS REQUIRED: -

Sl. No.	Description	Quantity
01	Computer	01
02	Printer	01

SOFTWARE REQUIRED: -

MATLAB 6.1 or above.

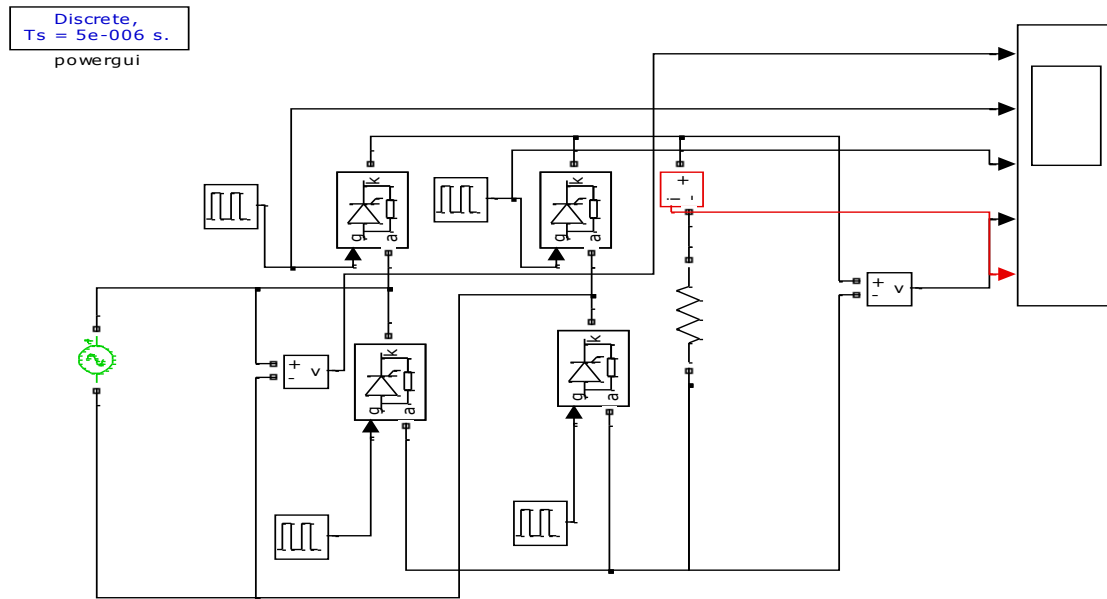
PROCEDURE FOR R LOAD: -

1. Study the circuit diagram, operation and model waveforms of a single phase fully controlled converter.
2. Open the MATLAB software by double-clicking the MATLAB icon on the desktop and click the shortcut "Simulink" to open the "Simulink Library Browser".
3. Create a new model and click on "SimPowerSystems" block in "Simulink Library Browser".
4. Drag the "powergui" block in SimPowerSystems submenu to the new model.
5. Double click on the "Electrical Source" block in SimPowerSystems submenu to select the single phase supply source "AC Voltage Source" and drag it to the new model. Set the peak amplitude of voltage as 110 Volts.
6. In the SimPowerSystems submenu, select "Power Electronics" block to pick the "Thyristor" and drag it to the new model.
7. In the SimPowerSystems submenu, select "Elements" block to pick the load "Series RLC branch" and drag it to the new model. Set the values of L & C as zero by double-clicking on the load.
8. In the SimPowerSystems submenu, select "Measurements" block to pick the "Voltage Measurement" & "Current Measurement" and drag them to the new model.
9. In the Simulink submenu, select "Commonly Used Blocks" to pick the "SCOPE" and drag it to the new model.
10. In the Simulink submenu, select "Sources" to pick the "Pulse Generator" for T_1 & T_2 and drag it to the new model.
11. Arrange the components as per the circuit diagram and connect them.
12. Set the values of input source voltage parameters such as amplitude, phase angle and frequency.
13. Set the pulse generator parameters such as pulse type, amplitude, period, pulse width, phase delay, etc. as per circuit design formula.
14. Set the scope axis as per the circuit requirements like input voltage, output voltage, output current, etc.
15. Set the load resistance value as 10 ohms.
16. Simulate the circuit and take the output waveforms in the scope.

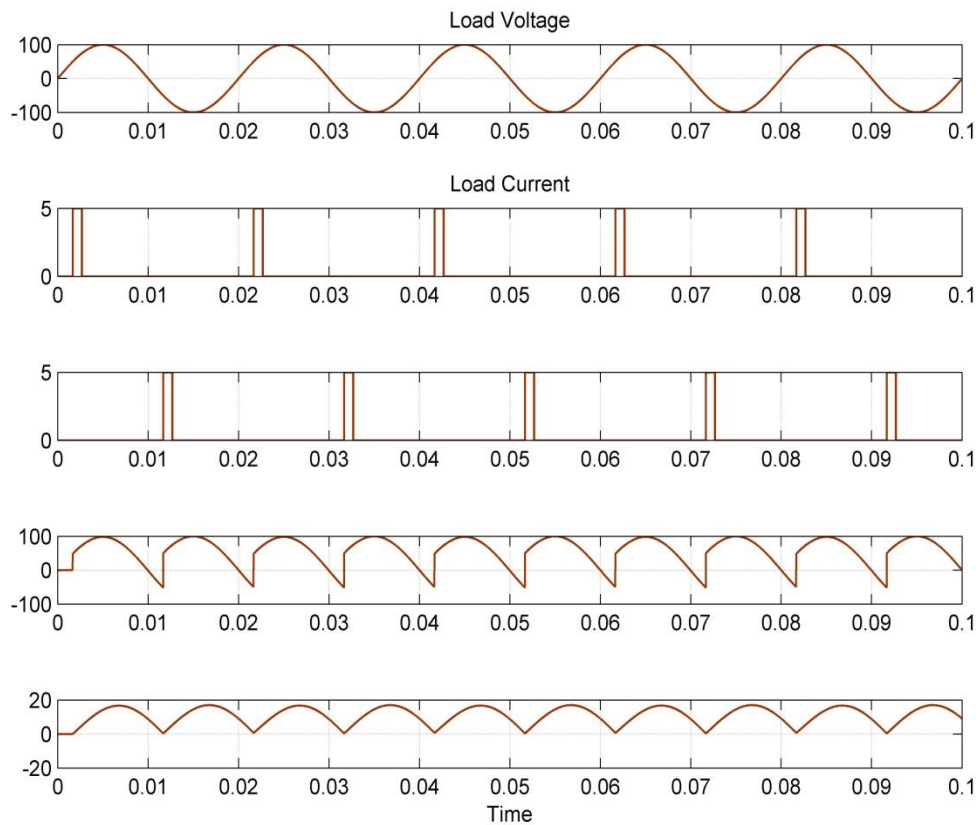
SINGLE PHASE FULLY CONTROLLED CONVERTER WITH R LOAD

SIMULATION DIAGRAM FOR R LOAD: -

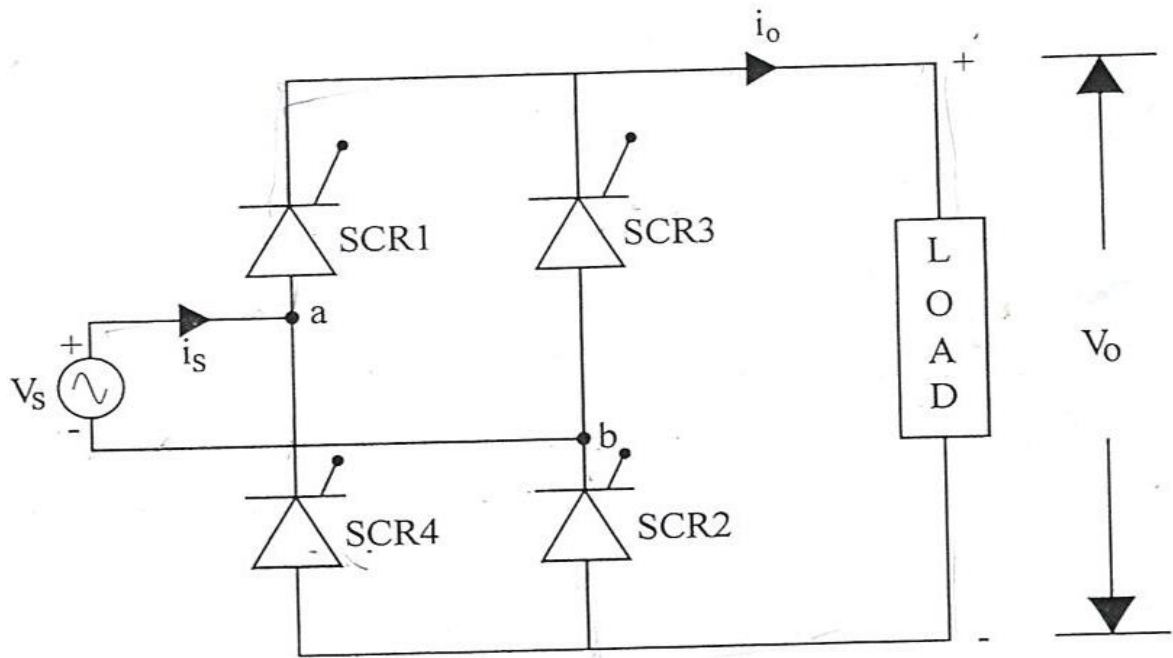
Single phase Bridge Contolled Rectifier



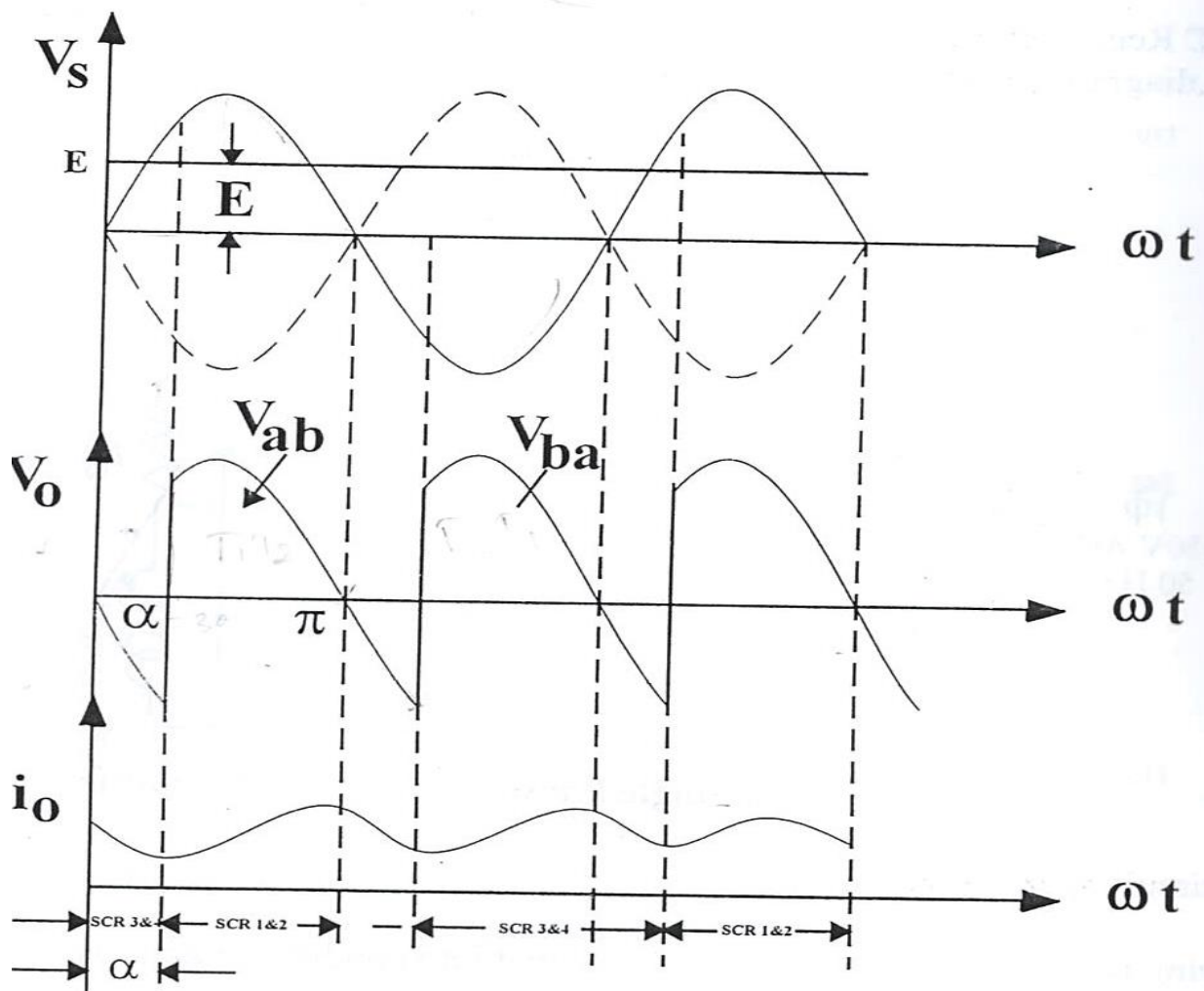
WAVEFORMS OF R LOAD



CIRCUIT DIAGRAM: -



MODEL WAVEFORMS: -



FORMULA USED FOR DESIGN OF PHASE DELAY: -

1. Firing angle delay for pulse generator 1 for T_1 & $T_2 = \frac{(0 + \alpha) * 0.02}{360}$ sec
2. Firing angle delay for pulse generator 2 for T_3 & $T_4 = \frac{(180 + \alpha) * 0.02}{360}$ sec

Where, α = firing angle delay in degrees.

Frequency = 50 Hz, i.e. time period = 0.02 sec.

PHASE DELAY MODEL CALCULATIONS: -

Let the firing angle be 30° .

i.e., $\alpha = 30^\circ$

$$\begin{aligned}\text{Firing angle delay for pulse generator 1 for } T_1 \text{ \& } T_2 &= \frac{(0 + 30) * 0.02}{360} \text{ sec} \\ &= 0.001667 \text{ sec.}\end{aligned}$$

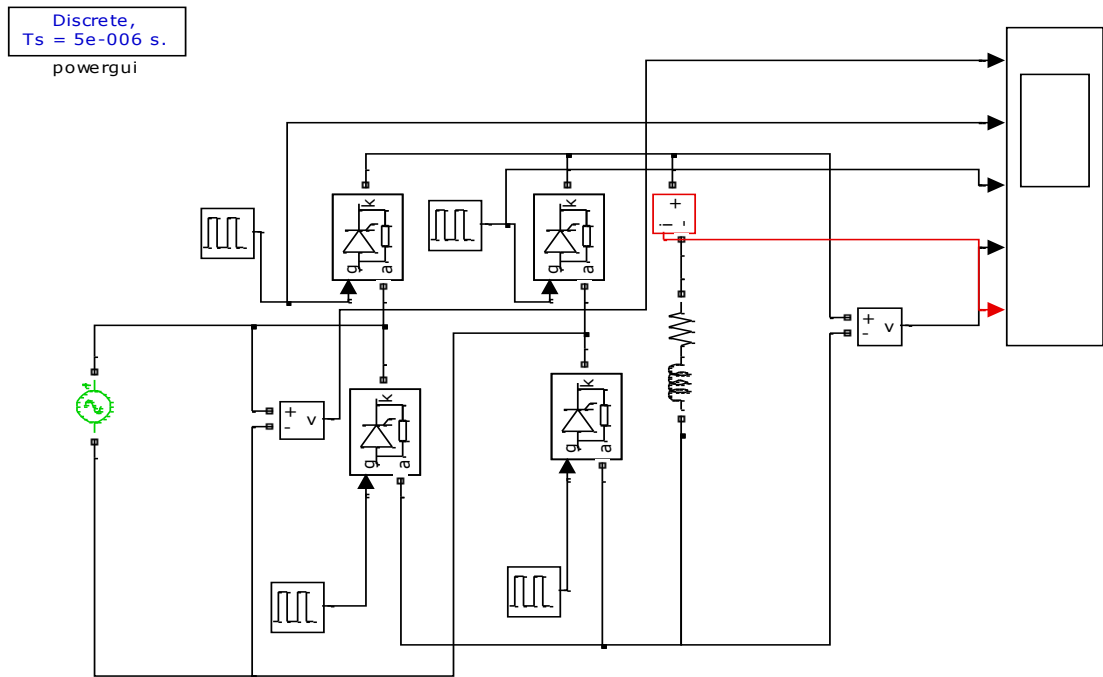
$$\begin{aligned}\text{Firing angle delay for pulse generator 2 for } T_3 \text{ \& } T_4 &= \frac{(180 + 30) * 0.02}{360} \text{ sec} \\ &= 0.011667 \text{ sec.}\end{aligned}$$

Firing angle delays in sec for pulse generators 1 & 2 for various firing angles can be calculated and tabulated as follows.

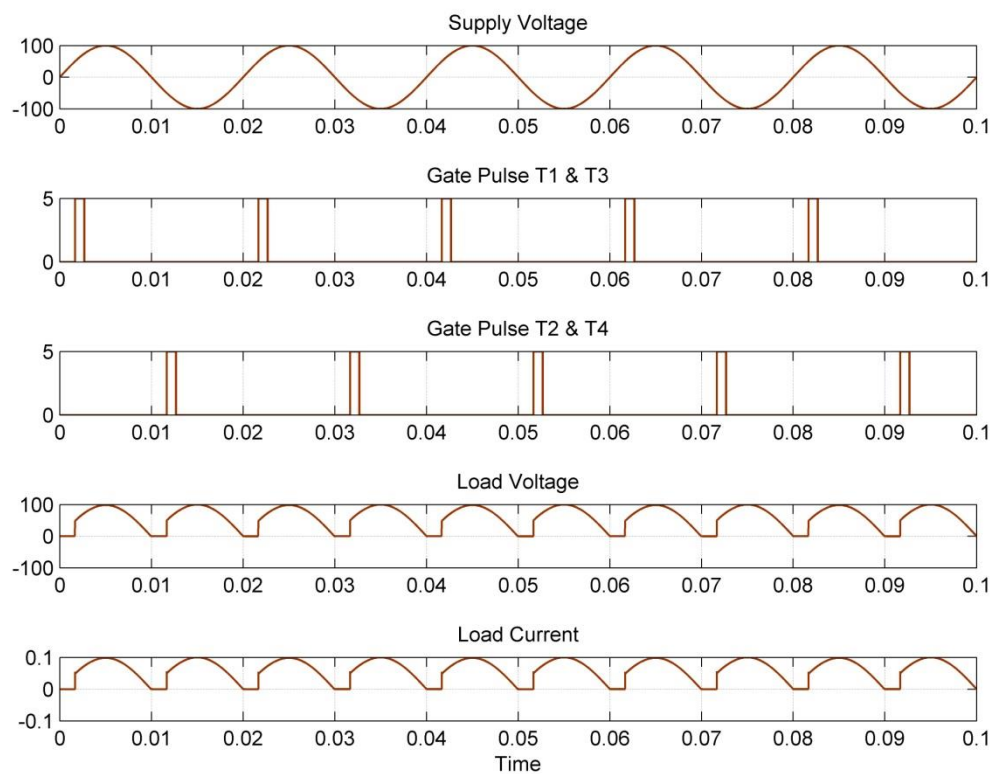
FIRING ANGLE (α) in deg	Phase angle delay for (in sec.)	
	Pulse Generator 1	Pulse Generator 2
0	0	0.01
30	0.001667	0.011667
60	0.00333	0.01333
90	0.005	0.015
120	0.00667	0.01667
150	0.00833	0.01833
180	0.01	0.02

SIMULATION DIAGRAM FOR RL LOAD: -

Single phase Bridge Contolled Rectifier with RL Load



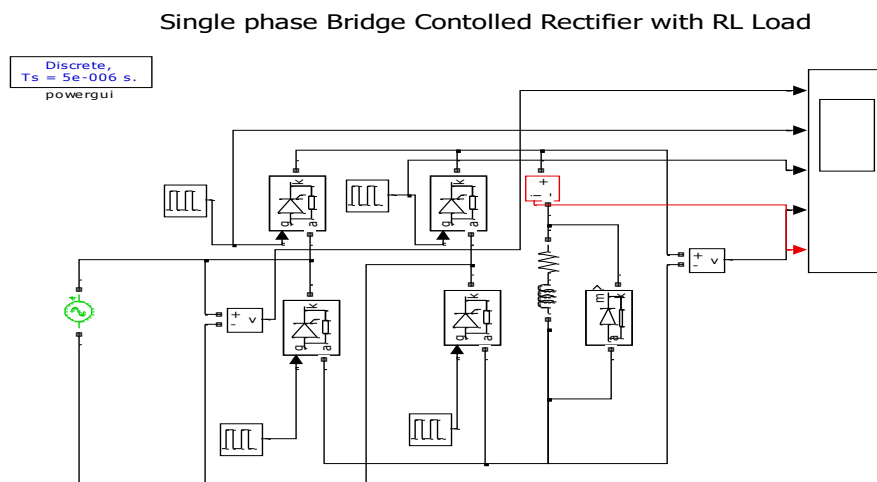
MODEL WAVEFORMS: -



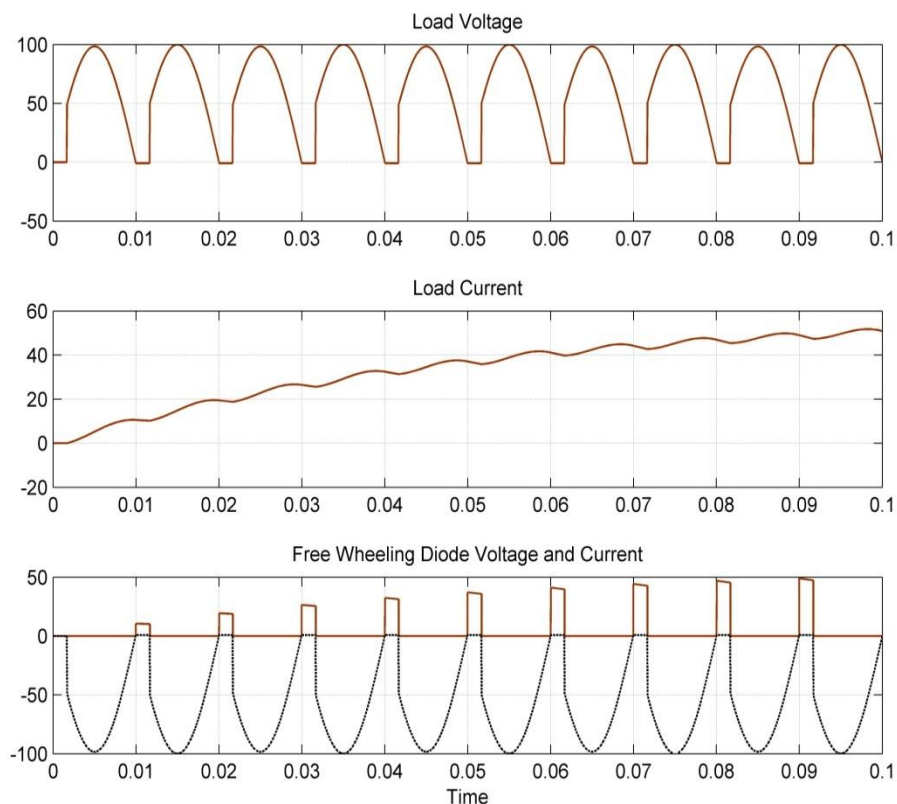
PROCEDURE FOR RL LOAD: -

1. Study the circuit diagram, operation and model waveforms of a single phase fully controlled converter.
2. Open the MATLAB software by double-clicking the MATLAB icon on the desktop and click the shortcut “Simulink” to open the “Simulink Library Browser”.
3. Create a new model and click on “SimPowerSystems” block in “Simulink Library Browser”.
4. Drag the “powergui” block in SimPowerSystems submenu to the new model.
5. Double click on the “Electrical Source” block in SimPowerSystems submenu to select the single phase supply source “AC Voltage Source” and drag it to the new model. Set the peak amplitude of voltage as 110 Volts.
6. In the SimPowerSystems submenu, select “Power Electronics” block to pick the “Thyristor” & “Diode” as Freewheeling Diode (FD) and drag it to the new model.
7. In the SimPowerSystems submenu, select “Elements” block to pick the load “Series RLC branch” and drag it to the new model. Set the value of C as zero by double-clicking on the load.
8. In the SimPowerSystems submenu, select “Measurements” block to pick the “Voltage Measurement” & “Current Measurement” and drag them to the new model.
9. In the Simulink submenu, select “Commonly Used Blocks” to pick the “SCOPE” and drag it to the new model.
10. In the Simulink submenu, select “Sources” to pick the “Pulse Generator” for T_1 – T_2 & T_3 – T_4 and drag it to the new model.
11. Arrange the components as per the circuit diagram and connect them.
12. Set the values of input source voltage parameters such as amplitude, phase angle and frequency.
13. Set the pulse generator parameters such as pulse type, amplitude, period, pulse width, phase delay, etc. as per circuit design formula.
14. Set the scope axis as per the circuit requirements like input voltage, output voltage, output current, etc.
15. Set the load resistance and inductance values as 10 ohms and 1 H respectively.
16. Simulate the circuit and take the output waveforms in the scope.

WAVEFORMS OF RL LOAD WITH FREEWHEELING DIODE :-



WAVEFORMS OF RL LOAD FOR FIRING ANGLE with FD: -

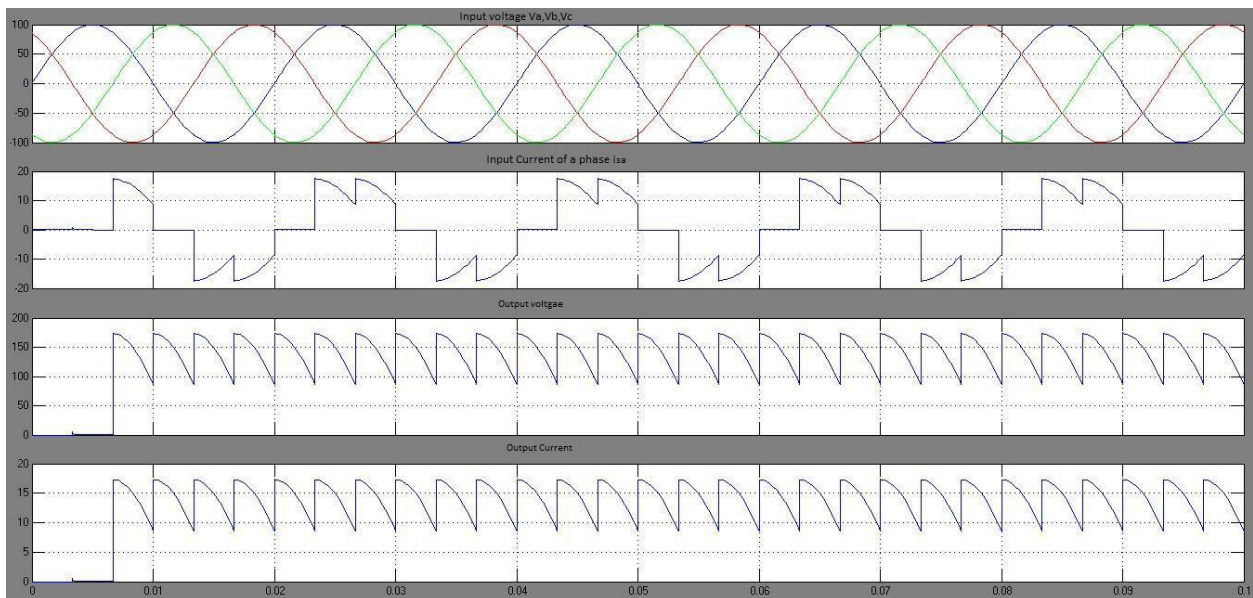
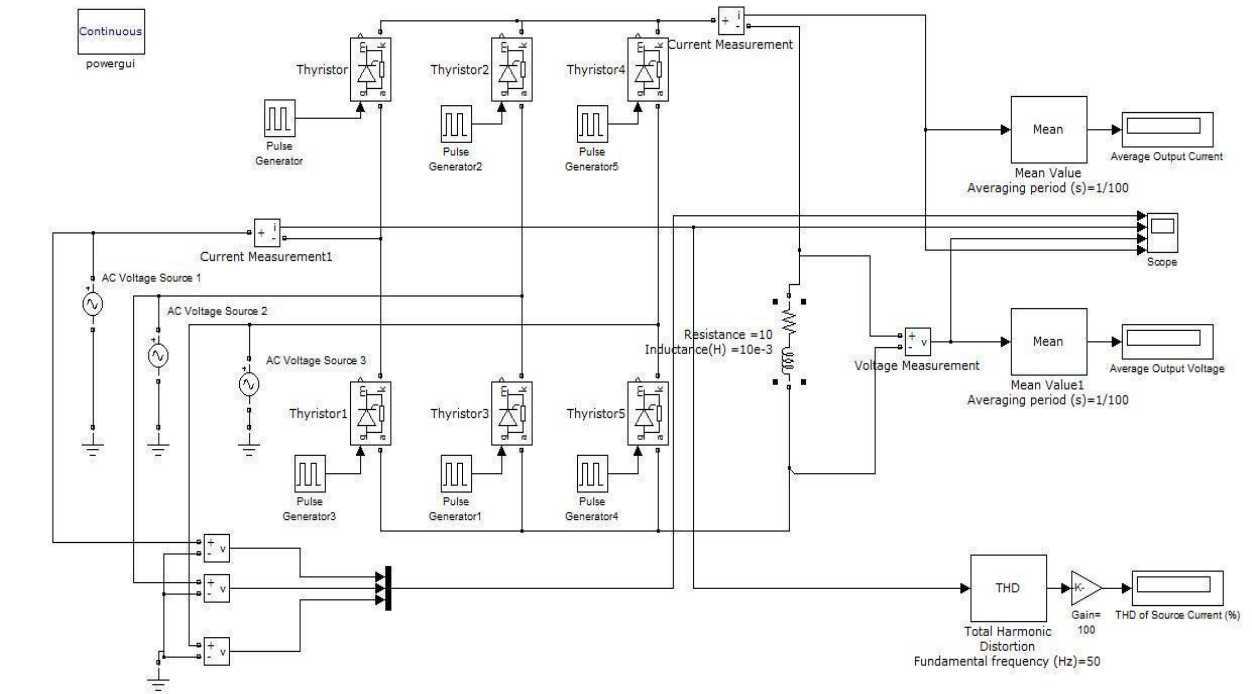


RESULT: -

Thus the single phase fully controlled converter was simulated with R and RL load of resistance of 10 ohms and inductance of 1 H for the firing angles of 30° and 120° and the related waveforms were obtained.

THREE PHASE HALF CONTROLLED CONVERTER WITH R AND RL LOAD

SIMULATION DIAGRAM FOR R LOAD: -



THREE PHASE HALF CONTROLLED CONVERTER WITH R AND RL LOAD

AIM: -

To simulate and study the Three Phase Half Controlled Converter with R and RL Load in different firing angles.

APPARATUS REQUIRED: -

Sl. No.	Description	Quantity
01	Computer	01
02	Printer	01

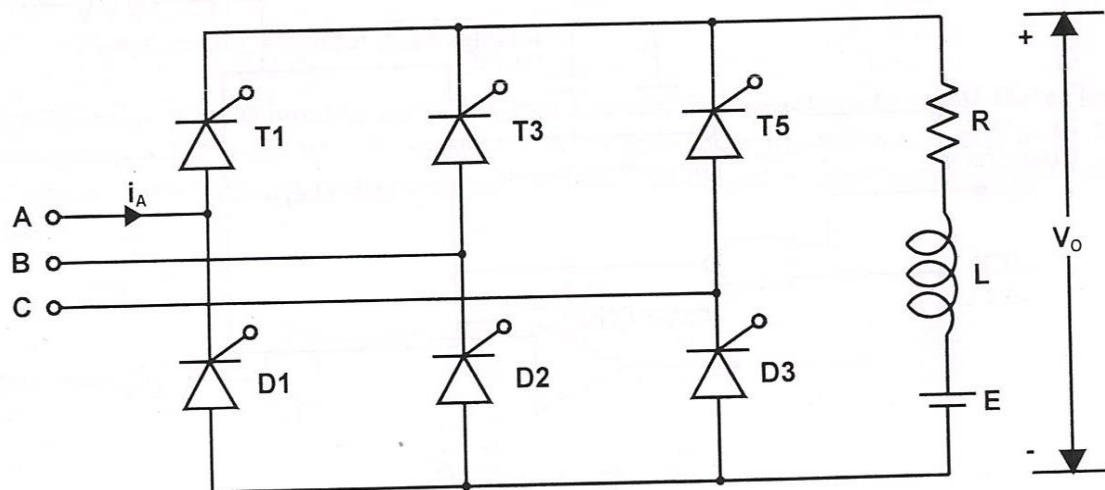
SOFTWARE REQUIRED: -

MATLAB 6.1 or above.

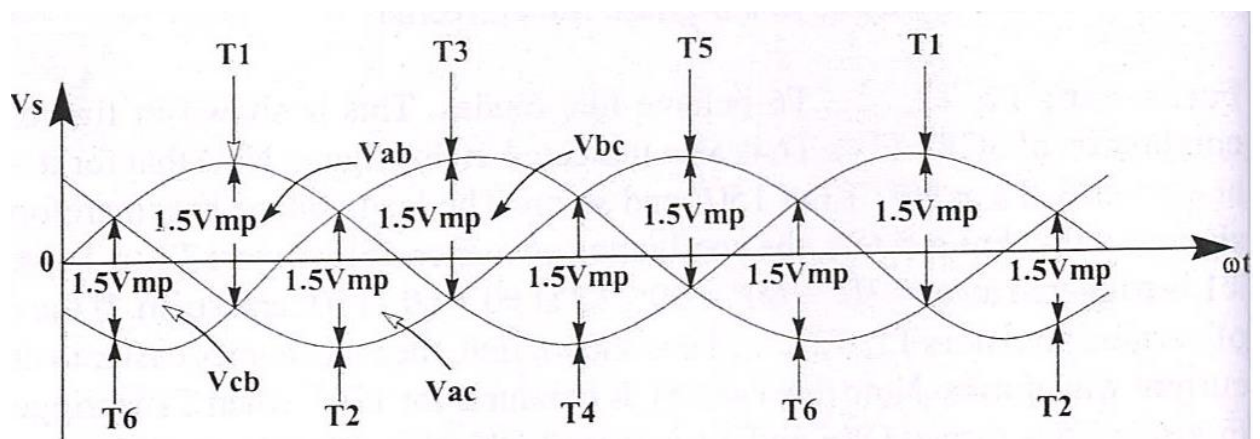
PROCEDURE FOR R LOAD: -

1. Study the circuit diagram, operation and model waveforms of a three phase half controlled converter.
2. Open the MATLAB software by double-clicking the MATLAB icon on the desktop and click the shortcut "Simulink" to open the "Simulink Library Browser".
3. Create a new model and click on "SimPowerSystems" block in "Simulink Library Browser".
4. Drag the "powergui" block in SimPowerSystems submenu to the new model.
5. Double click on the "Electrical Source" block in SimPowerSystems submenu to select the three phase supply source i.e. "Three Phase Programmable Voltage Source" and drag it to the new model. Set the peak amplitude of voltage as 100 Volts.
6. In the SimPowerSystems submenu, select "Power Electronics" block to pick the "Thyristor" & "Diode" and drag them to the new model.
7. In the SimPowerSystems submenu, select "Elements" block to pick the load "Series RLC branch" and drag it to the new model. Set the values of L & C as zero by double-clicking on the load.
8. In the SimPowerSystems submenu, select "Measurements" block to pick the "Three-Phase V-I Measurement", "Voltage Measurement" & "Current Measurement" and drag them to the new model.
9. In the Simulink submenu, select "Commonly Used Blocks" to pick the "SCOPE" and drag it to the new model.
10. In the Simulink submenu, select "Sources" to pick the "Pulse Generator" from it for T_1 , T_2 and T_3 and drag it to the new model.
11. Arrange the components as per the circuit diagram and connect them.
12. Set the values of input source voltage parameters such as amplitude, phase angle and frequency.
13. Set the pulse generator parameters such as pulse type, amplitude, period, pulse width, phase delay, etc. as per circuit design formula.
14. Set the scope axis as per the circuit requirements like input voltage, output voltage, output current, etc.
15. Set the load resistance value as 10 ohms.
16. Simulate the circuit and take the output waveforms in the scope.

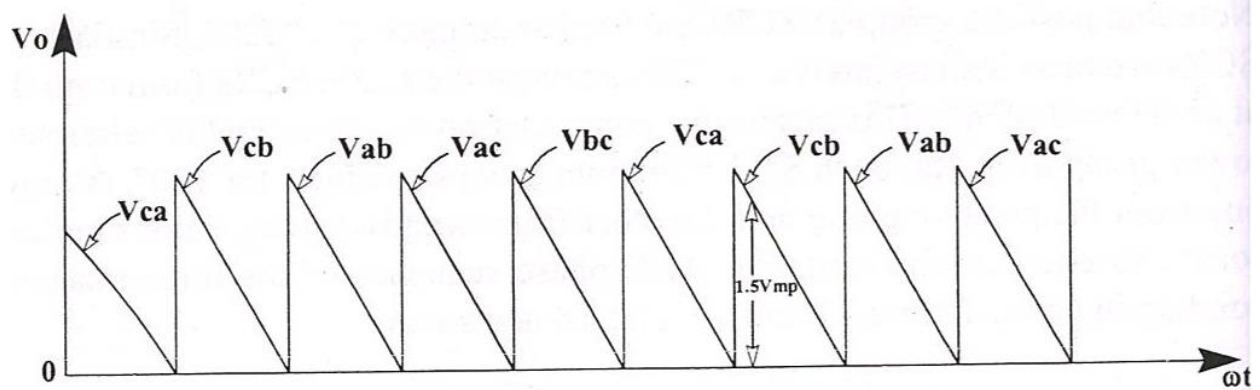
CIRCUIT DIAGRAM: -



MODEL WAVEFORMS: -



T5		T1		T3		T5		T1		+ ve Group
T2	T6	T2		T4		T6				- ve Group



FORMULA USED FOR DESIGN OF PHASE DELAY: -

1. Firing angle delay for pulse generator 1 for $T_1 = \frac{(30 + \alpha) * 0.02}{360}$ sec
2. Firing angle delay for pulse generator 2 for $T_2 = \frac{(30 + 120 + \alpha) * 0.02}{360}$ sec
3. Firing angle delay for pulse generator 2 for $T_3 = \frac{(30 + 240 + \alpha) * 0.02}{360}$ sec

Where, α = firing angle delay in degrees.

Frequency = 50 Hz, i.e. time period = 0.02 sec.

PHASE DELAY MODEL CALCULATIONS: -

Let the firing angle be 30° .

i.e., $\alpha = 30^\circ$

$$\text{Firing angle delay for pulse generator 1 for } T_1 = \frac{(30 + 30) * 0.02}{360} \text{ sec}$$

$$= 0.00333 \text{ sec.}$$

$$\text{Firing angle delay for pulse generator 2 for } T_2 = \frac{(30 + 120 + 30) * 0.02}{360} \text{ sec}$$

$$= 0.01 \text{ sec.}$$

$$\text{Firing angle delay for pulse generator 2 for } T_2 = \frac{(30 + 240 + 30) * 0.02}{360} \text{ sec}$$

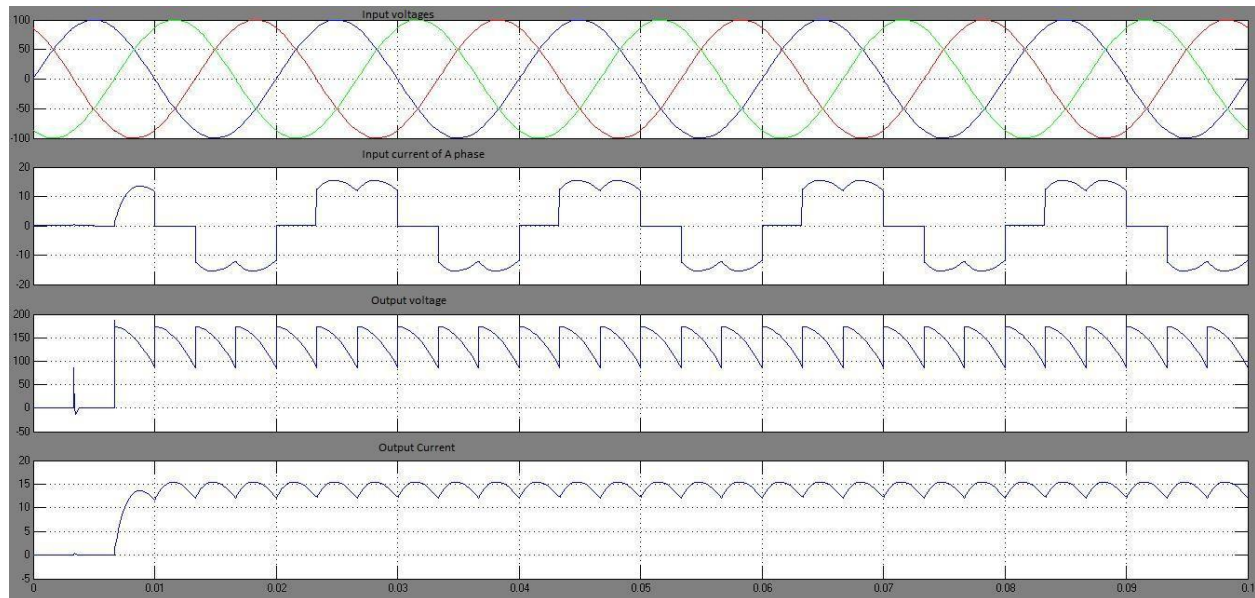
$$= 0.01667 \text{ sec.}$$

Firing angle delays in sec for pulse generators 1, 2 & 3 for various firing angles can be calculated and tabulated as follows.

FIRING ANGLE (α) in deg	Phase angle delay for (in sec.)		
	Pulse Generator 1	Pulse Generator 2	Pulse Generator 3
0	0.001667	0.00833	0.015
30	0.00333	0.01	0.01667
60	0.005	0.011667	0.01833
90	0.00667	0.01333	0.02
120	0.00833	0.015	0.021667
150	0.01	0.01667	0.02333
180	0.011667	0.018333	0.025

THREE PHASE HALF CONTROLLED CONVERTER WITH R AND RL LOAD

SIMULATION DIAGRAM FOR RL LOAD: -



PROCEDURE FOR RL LOAD: -

1. Study the circuit diagram, operation and model waveforms of a three phase half controlled converter.
2. Open the MATLAB software by double-clicking the MATLAB icon on the desktop and click the shortcut "Simulink" to open the "Simulink Library Browser".
3. Create a new model and click on "SimPowerSystems" block in "Simulink Library Browser".
4. Drag the "powergui" block in SimPowerSystems submenu to the new model.
5. Double click on the "Electrical Source" block in SimPowerSystems submenu to select the three phase supply source i.e. "Three Phase Programmable Voltage Source" and drag it to the new model. Set the peak amplitude of voltage as 110 Volts.
6. In the SimPowerSystems submenu, select "Power Electronics" block to pick the "Thyristor" & "Diode" and drag them to the new model.
7. In the SimPowerSystems submenu, select "Elements" block to pick the load "Series RLC branch" and drag it to the new model. Set the value of C as zero by double-clicking on the load.
8. In the SimPowerSystems submenu, select "Measurements" block to pick the "Three-Phase V-I Measurement", "Voltage Measurement" & "Current Measurement" and drag them to the new model.
9. In the Simulink submenu, select "Commonly Used Blocks" to pick the "SCOPE" and drag it to the new model.
10. In the Simulink submenu, select "Sources" to pick the "Pulse Generator" for T_1 , T_2 and T_3 and drag it to the new model.
11. Arrange the components as per the circuit diagram and connect them.
12. Set the values of input source voltage parameters such as amplitude, phase angle and frequency.

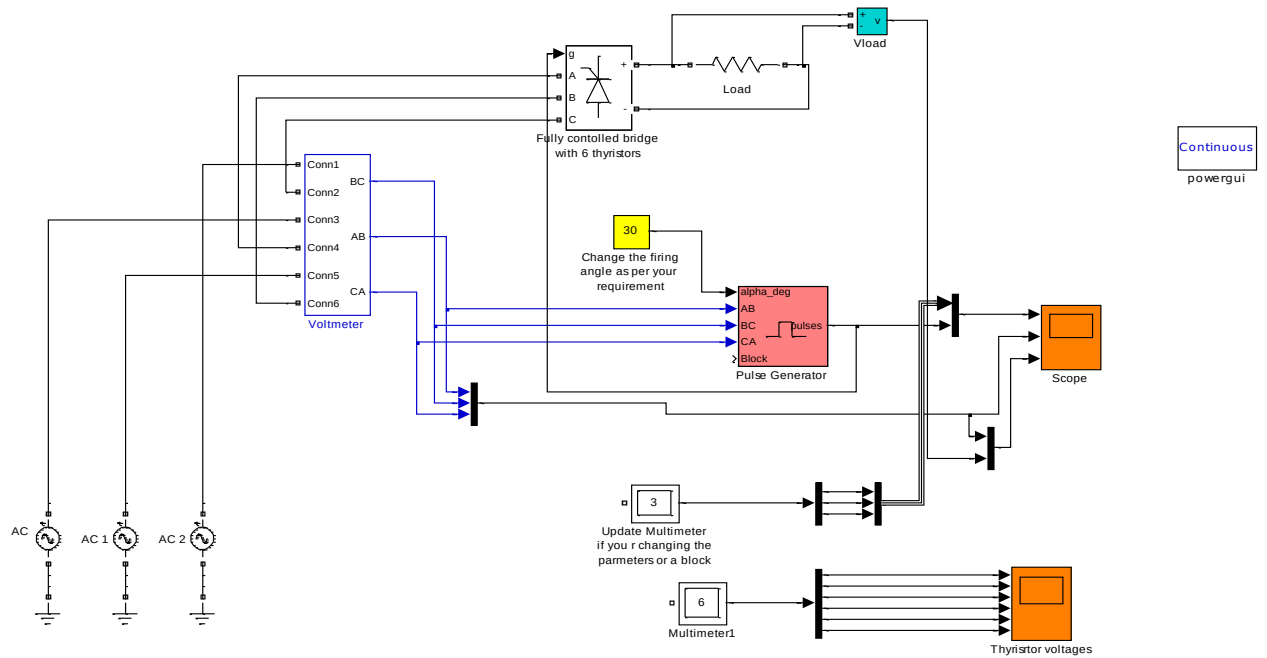
13. Set the pulse generator parameters such as pulse type, amplitude, period, pulse width, phase delay, etc. as per circuit design formula.
14. Set the scope axis as per the circuit requirements like input voltage, output voltage, output current, etc.
15. Set the load resistance and inductance values as 10 ohms and 1 mH respectively.
16. Simulate the circuit and take the output waveforms in the scope.

RESULT: -

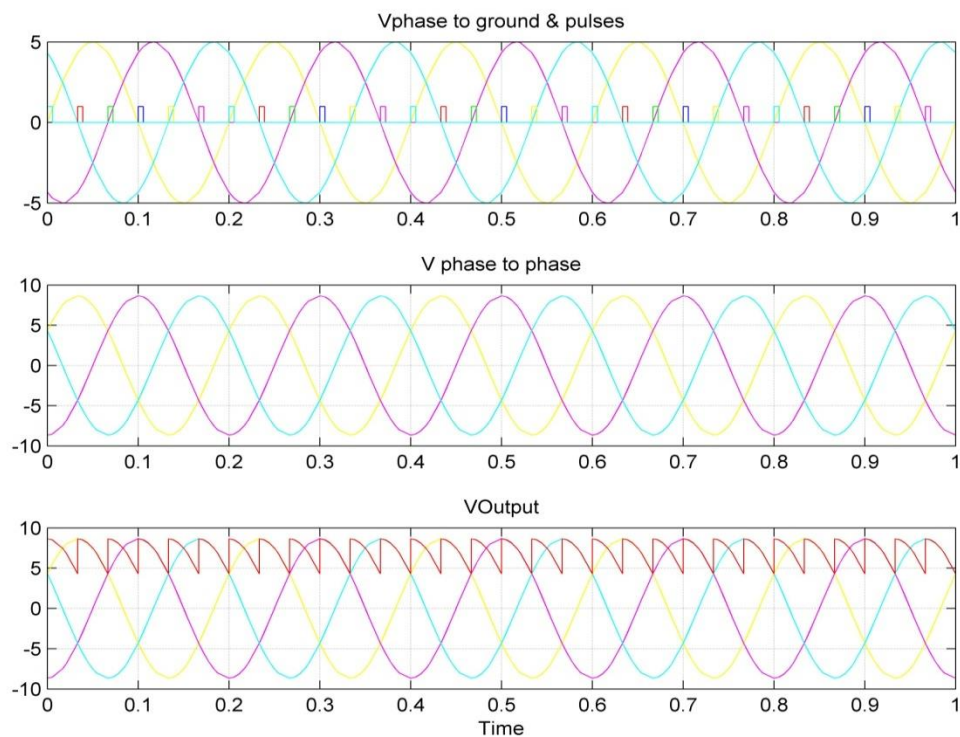
Thus the three phase half controlled converter was simulated with R and RL load of resistance of 10 ohms and inductance of 1 mH for the firing angles of 30° , 60° , 90° and 120° and the related waveforms were obtained.

THREE PHASE FULLY CONTROLLED CONVERTER WITH R AND RL LOAD

SIMULATION DIAGRAM FOR R LOAD: -



MODEL WAVEFORM:



Expt. No.: 20

Date:

THREE PHASE FULLY CONTROLLED CONVERTER WITH R AND RL LOAD

AIM: -

To simulate and study the Three Phase Fully Controlled Converter with R and RL Load in different firing angles.

APPARATUS REQUIRED: -

Sl. No.	Description	Quantity
01	Computer	01
02	Printer	01

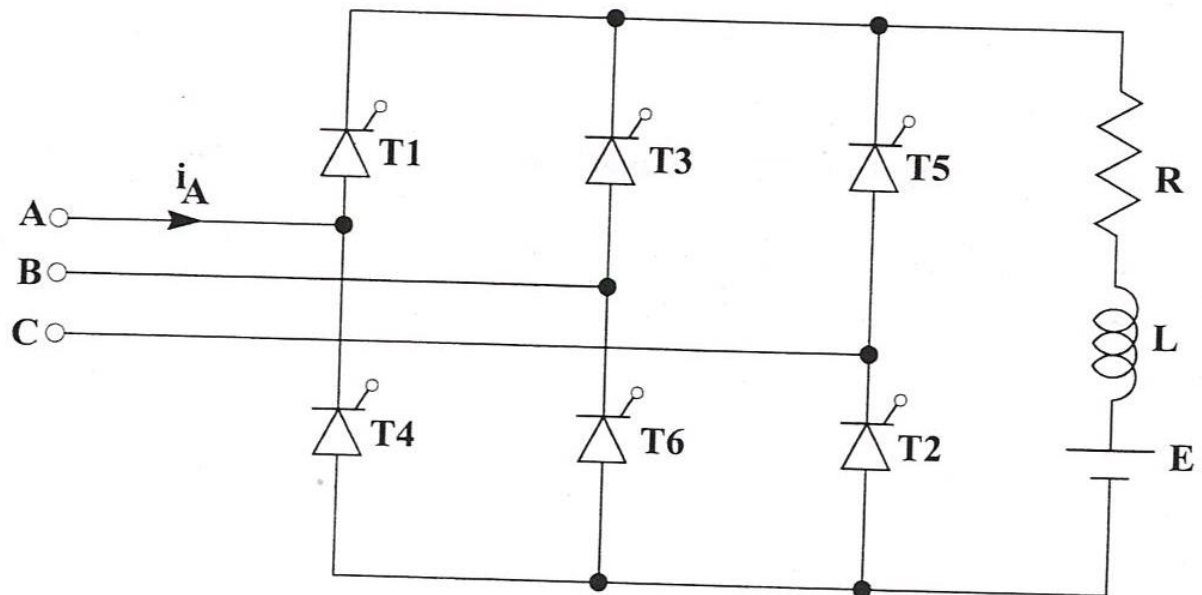
SOFTWARE REQUIRED: -

MATLAB 6.1 or above.

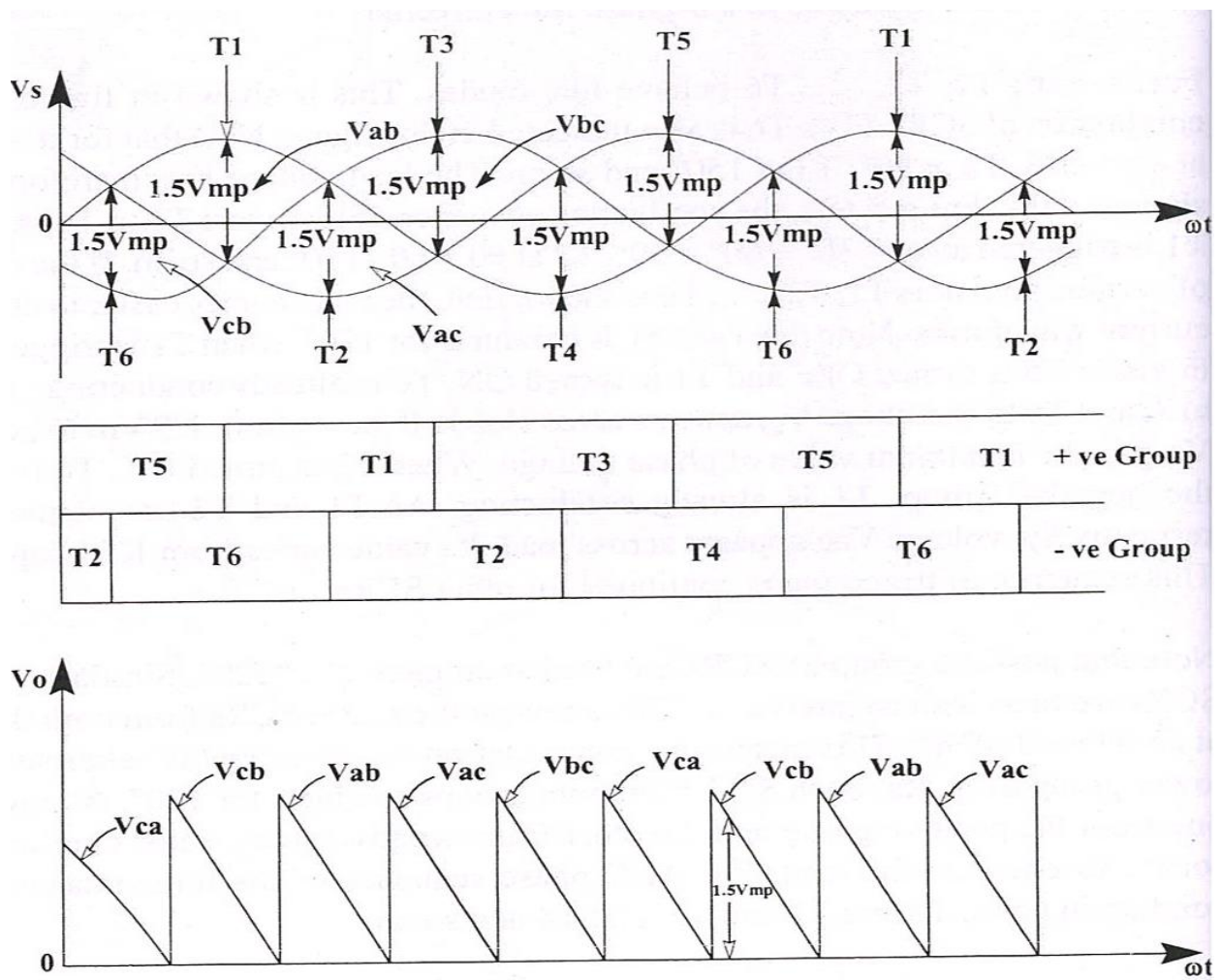
PROCEDURE FOR R LOAD: -

1. Study the circuit diagram, operation and model waveforms of a three phase fully controlled converter.
2. Open the MATLAB software by double-clicking the MATLAB icon on the desktop and click the shortcut "Simulink" to open the "Simulink Library Browser".
3. Create a new model and click on "SimPowerSystems" block in "Simulink Library Browser".
4. Drag the "powergui" block in SimPowerSystems submenu to the new model.
5. Double click on the "Electrical Source" block in SimPowerSystems submenu to select the three phase supply source i.e. "Three Phase Programmable Voltage Source" and drag it to the new model. Set the peak amplitude of voltage as 110 Volts.
6. In the SimPowerSystems submenu, select "Power Electronics" block to pick the "Thyristor" and drag it to the new model.
7. In the SimPowerSystems submenu, select "Elements" block to pick the load "Series RLC branch" and drag it to the new model. Set the values of L & C as zero by double-clicking on the load.
8. In the SimPowerSystems submenu, select "Measurements" block to pick the "Three-Phase V-I Measurement", "Voltage Measurement" & "Current Measurement" and drag them to the new model.
9. In the Simulink submenu, select "Commonly Used Blocks" to pick the "SCOPE" and drag it to the new model.
10. In the Simulink submenu, select "Sources" to pick the "Pulse Generator" for Thyristors T_1 , T_2 , T_3 , T_4 , T_5 & T_6 and drag it to the new model.
11. Arrange the components as per the circuit diagram and connect them.
12. Set the values of input source voltage parameters such as amplitude, phase angle and frequency.
13. Set the pulse generator parameters such as pulse type, amplitude, period, pulse width, phase delay, etc. as per circuit design formula.
14. Set the scope axis as per the circuit requirements like input voltage, output voltage, output current, etc.
15. Set the load resistance value as 10 ohms.
16. Simulate the circuit and take the output waveforms in the scope.

CIRCUIT DIAGRAM: -



MODEL WAVEFORMS: -



FORMULA USED FOR DESIGN OF PHASE DELAY: -

1. Firing angle delay for pulse generator 1 for $T_1 = \frac{(30 + \alpha) * 0.02}{360} \text{ sec}$
2. Firing angle delay for pulse generator 2 for $T_2 = \frac{(30 + 60 + \alpha) * 0.02}{360} \text{ sec}$
3. Firing angle delay for pulse generator 2 for $T_3 = \frac{(30 + 120 + \alpha) * 0.02}{360} \text{ sec}$
4. Firing angle delay for pulse generator 2 for $T_4 = \frac{(30 + 180 + \alpha) * 0.02}{360} \text{ sec}$
5. Firing angle delay for pulse generator 2 for $T_5 = \frac{(30 + 240 + \alpha) * 0.02}{360} \text{ sec}$
6. Firing angle delay for pulse generator 2 for $T_3 = \frac{(30 + 300 + \alpha) * 0.02}{360} \text{ sec}$

Where, α = firing angle delay in degrees.

Frequency = 50 Hz, i.e. time period = 0.02 sec.

PHASE DELAY MODEL CALCULATIONS: -

Let the firing angle be 30° . i.e., $\alpha = 30^\circ$

$$\begin{aligned} \text{Firing angle delay for pulse generator 1 for } T_1 &= \frac{(30 + 30) * 0.02}{360} \text{ sec} \\ &= 0.00333 \text{ sec.} \end{aligned}$$

$$\begin{aligned} \text{Firing angle delay for pulse generator 2 for } T_2 &= \frac{(30 + 60 + 30) * 0.02}{360} \text{ sec} \\ &= 0.00667 \text{ sec.} \end{aligned}$$

$$\begin{aligned} \text{Firing angle delay for pulse generator 3 for } T_3 &= \frac{(30 + 120 + 30) * 0.02}{360} \text{ sec} \\ &= 0.01 \text{ sec.} \end{aligned}$$

$$\begin{aligned} \text{Firing angle delay for pulse generator 4 for } T_4 &= \frac{(30 + 180 + 30) * 0.02}{360} \text{ sec} \\ &= 0.01333 \end{aligned}$$

$$\begin{aligned} \text{Firing angle delay for pulse generator 5 for } T_5 &= \frac{(30 + 240 + 30) * 0.02}{360} \text{ sec} \\ &= 0.01667 \end{aligned}$$

$$\begin{aligned} \text{Firing angle delay for pulse generator 6 for } T_6 &= \frac{(30 + 300 + 30) * 0.02}{360} \text{ sec} \\ &= 0.02 \end{aligned}$$

Firing angle delays in sec for pulse generators 1, 2 & 3 for various firing angles can be calculated and tabulated as follows.

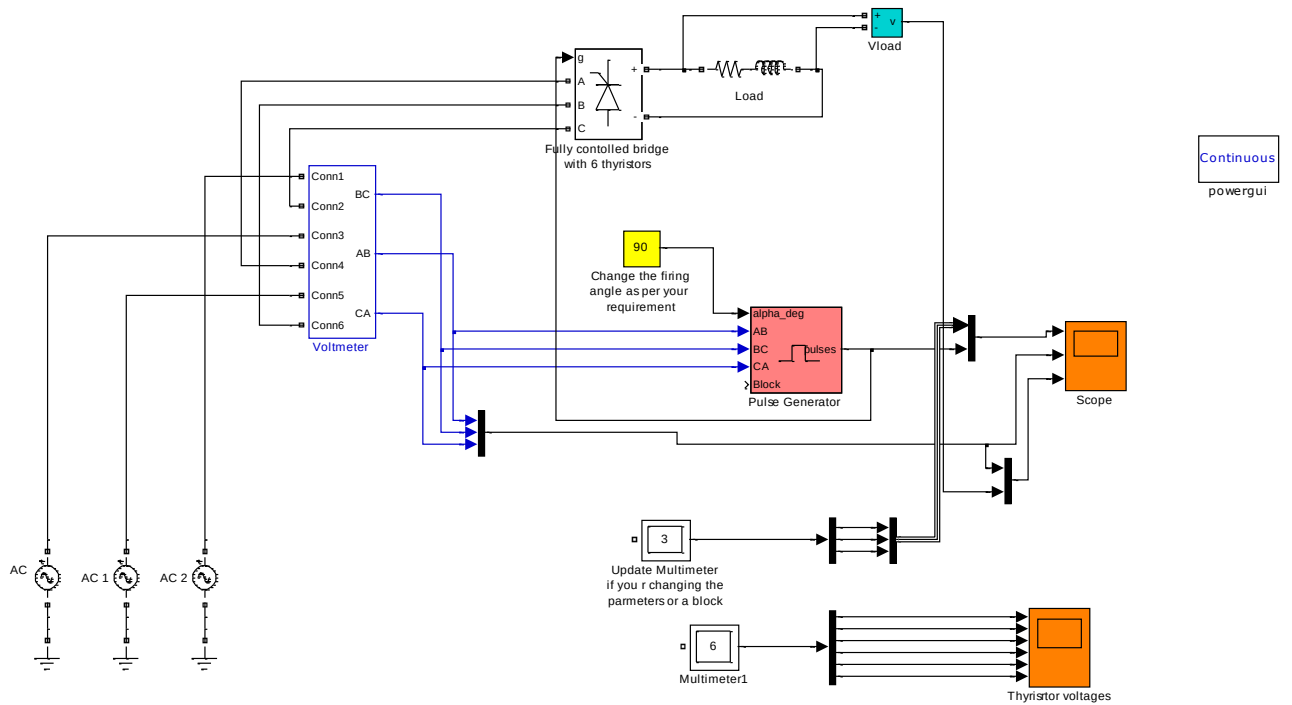
FIRING ANGLE (α) in deg	Phase angle delay for (in sec.)					
	PG 1	PG 2	PG 3	PG 4	PG 5	PG 6
0	0.00167	0.005	0.00833	0.01167	0.015	0.01833
30	0.00333	0.00667	0.01	0.01333	0.01667	0.02
60	0.005	0.00833	0.01167	0.015	0.01833	0.02167
90	0.00667	0.01	0.01333	0.01667	0.02	0.02333
120	0.00833	0.01167	0.015	0.01833	0.02167	0.025
150	0.01	0.01333	0.01667	0.02	0.02333	0.02667
180	0.01167	0.015	0.01833	0.02167	0.025	0.02833

RESULT: -

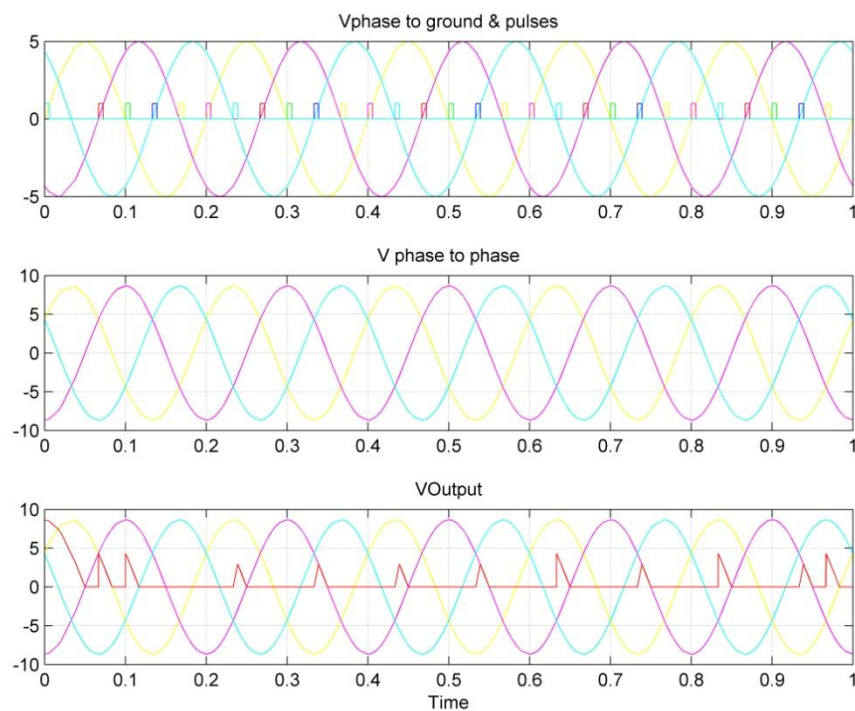
Thus the three phase half controlled converter was simulated with R and RL load of resistance of 10 ohms and inductance of 1 mH for the firing angles of 30°, 60°, 90° and 120° and the related waveforms were obtained.

THREE PHASE FULLY CONTROLLED CONVERTER WITH R AND RL LOAD

SIMULATION DIAGRAM FOR RL LOAD: -



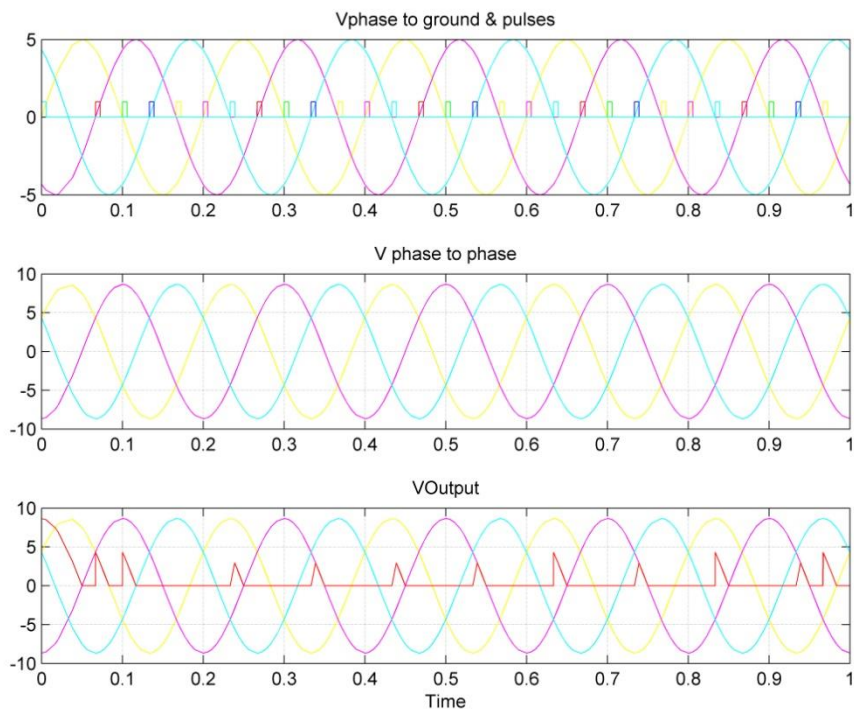
MODEL GRAPH:



PROCEDURE FOR RL LOAD: -

1. Study the circuit diagram, operation and model waveforms of a three phase fully controlled converter.
2. Open the MATLAB software by double-clicking the MATLAB icon on the desktop and click the shortcut “Simulink” to open the “Simulink Library Browser”.
3. Create a new model and click on “SimPowerSystems” block in “Simulink Library Browser”.
4. Drag the “powergui” block in SimPowerSystems submenu to the new model.
5. Double click on the “Electrical Source” block in SimPowerSystems submenu to select the three phase supply source i.e. “Three Phase Programmable Voltage Source” and drag it to the new model. Set the peak amplitude of voltage as 110 Volts.
6. In the SimPowerSystems submenu, select “Power Electronics” block to pick the “Thyristor” and drag them to the new model.
7. In the SimPowerSystems submenu, select “Elements” block to pick the load “Series RLC branch” and drag it to the new model. Set the value of C as zero by double-clicking on the load.
8. In the SimPowerSystems submenu, select “Measurements” block to pick the “Three-Phase V-I Measurement”, “Voltage Measurement” & “Current Measurement” and drag them to the new model.
9. In the Simulink submenu, select “Commonly Used Blocks” to pick the “SCOPE” and drag it to the new model.
10. In the Simulink submenu, select “Sources” to pick the “Pulse Generator” for T₁, T₂, T₃, T₄, T₅ & T₆ and drag them to the new model.
11. Arrange the components as per the circuit diagram and connect them.
12. Set the values of input source voltage parameters such as amplitude, phase angle and frequency.
13. Set the pulse generator parameters such as pulse type, amplitude, period, pulse width, phase delay, etc. as per circuit design formula.
14. Set the scope axis as per the circuit requirements like input voltage, output voltage, output current, etc.
15. Set the load resistance and inductance values as 10 ohms and 1 H respectively.
16. Simulate the circuit and take the output waveforms in the scope.

WAVEFORMS OF RL LOAD FOR FIRING ANGLE OF 90°: -



RESULT: -

Thus the three phase fully controlled converter was simulated with R and RL load of resistance of 10 ohms and inductance of 1 H for the firing angles of 30°, 60°, 90° and 120° and the related waveforms were obtained.

SINGLE PHASE AC VOLTAGE CONVERTER WITH R AND RL LOAD

AIM: -

To simulate and study the Single Phase AC Voltage Converter with R and RL Load in different firing angles.

APPARATUS REQUIRED: -

Sl. No.	Description	Quantity
01	Computer	01
02	Printer	01

SOFTWARE REQUIRED: -

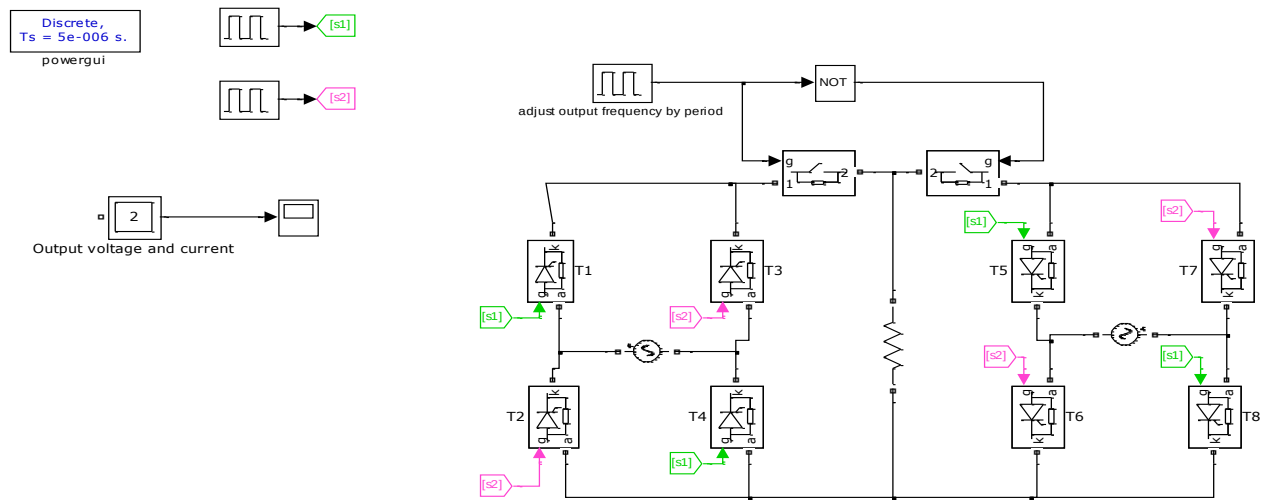
MATLAB 6.1 or above.

PROCEDURE FOR R LOAD: -

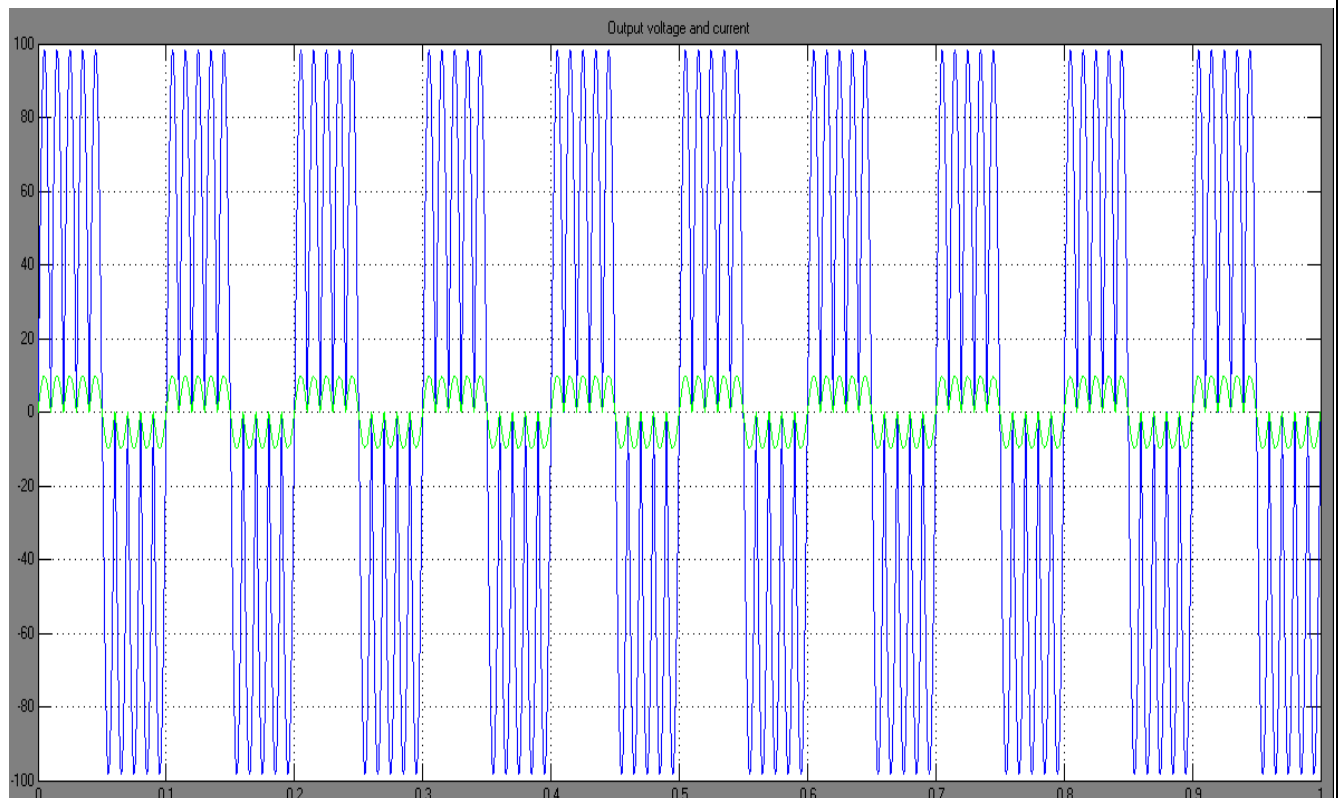
1. Study the circuit diagram, operation and model waveforms of a single phase AC Voltage controller.
2. Open the MATLAB software by double-clicking the MATLAB icon on the desktop and click the shortcut "Simulink" to open the "Simulink Library Browser".
3. Create a new model and click on "SimPowerSystems" block in "Simulink Library Browser".
4. Drag the "powergui" block in SimPowerSystems submenu to the new model.
5. Double click on the "Electrical Source" block in SimPowerSystems submenu to select the single phase supply source "AC Voltage Source" and drag it to the new model. Set the peak amplitude of voltage as 110 Volts.
6. In the SimPowerSystems submenu, select "Power Electronics" block to pick the "Thyristor" and drag them to the new model.
7. In the SimPowerSystems submenu, select "Elements" block to pick the load "Series RLC branch" and drag it to the new model. Set the values of L & C as zero by double-clicking on the load.
8. In the SimPowerSystems submenu, select "Measurements" block to pick the "Voltage Measurement" & "Current Measurement" and drag them to the new model.
9. Also find the "RMS" block for converting the output signal to rms value in the "measurements" submenu under Extra Library in SimPowerSystems menu and drag it to the new model.
10. Find the "Display" block for displaying the rms value of the output signal in the "Sinks" submenu under "Simulink" menu and drag it to the new model.
11. In the Simulink submenu, select "Commonly Used Blocks" to pick the "SCOPE" and drag it to the new model.
12. In the Simulink submenu, select "Sources" to pick the "Pulse Generator" for T_1 & T_2 and drag it to the new model.
13. Arrange the components as per the circuit diagram and connect them.
14. Set the values of input source voltage parameters such as amplitude, phase angle and frequency.
15. Set the pulse generator parameters such as pulse type, amplitude, period, pulse width, phase delay, etc. as per circuit design formula.
16. Set the scope axis as per the circuit requirements like input voltage, output voltage, output current, etc.
17. Set the load resistance value as 10 ohms.
18. Simulate the circuit and take the output waveforms in the scope.

SINGLE PHASE AC VOLTAGE CONVERTER WITH R AND RL LOAD

SIMULATION DIAGRAM FOR R LOAD: -



MODEL GRAPH:



FORMULA USED FOR DESIGN OF PHASE DELAY: -

1. Firing angle delay for pulse generator 1 = $\frac{(0 + \alpha) * 0.02}{360}$ sec
2. Firing angle delay for pulse generator 2 = $\frac{(180 + \alpha) * 0.02}{360}$ sec

Where, α = firing angle delay in degrees.

Frequency = 50 Hz, i.e. time period = 0.02 sec.

PHASE DELAY CALCULATIONS: -

Let the firing angle be 30° .

i.e., $\alpha = 30^\circ$

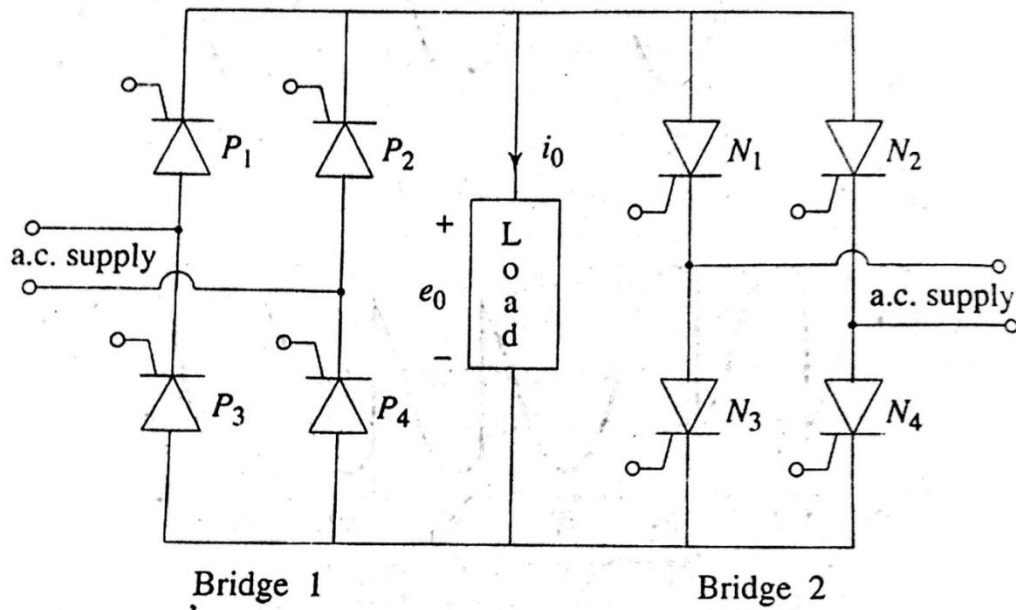
$$\begin{aligned}\text{Firing angle delay for pulse generator 1} &= \frac{(0 + 30) * 0.02}{360} \text{ sec} \\ &= 0.001667 \text{ sec.}\end{aligned}$$

$$\begin{aligned}\text{Firing angle delay for pulse generator 2} &= \frac{(180 + 30) * 0.02}{360} \text{ sec} \\ &= 0.011667 \text{ sec.}\end{aligned}$$

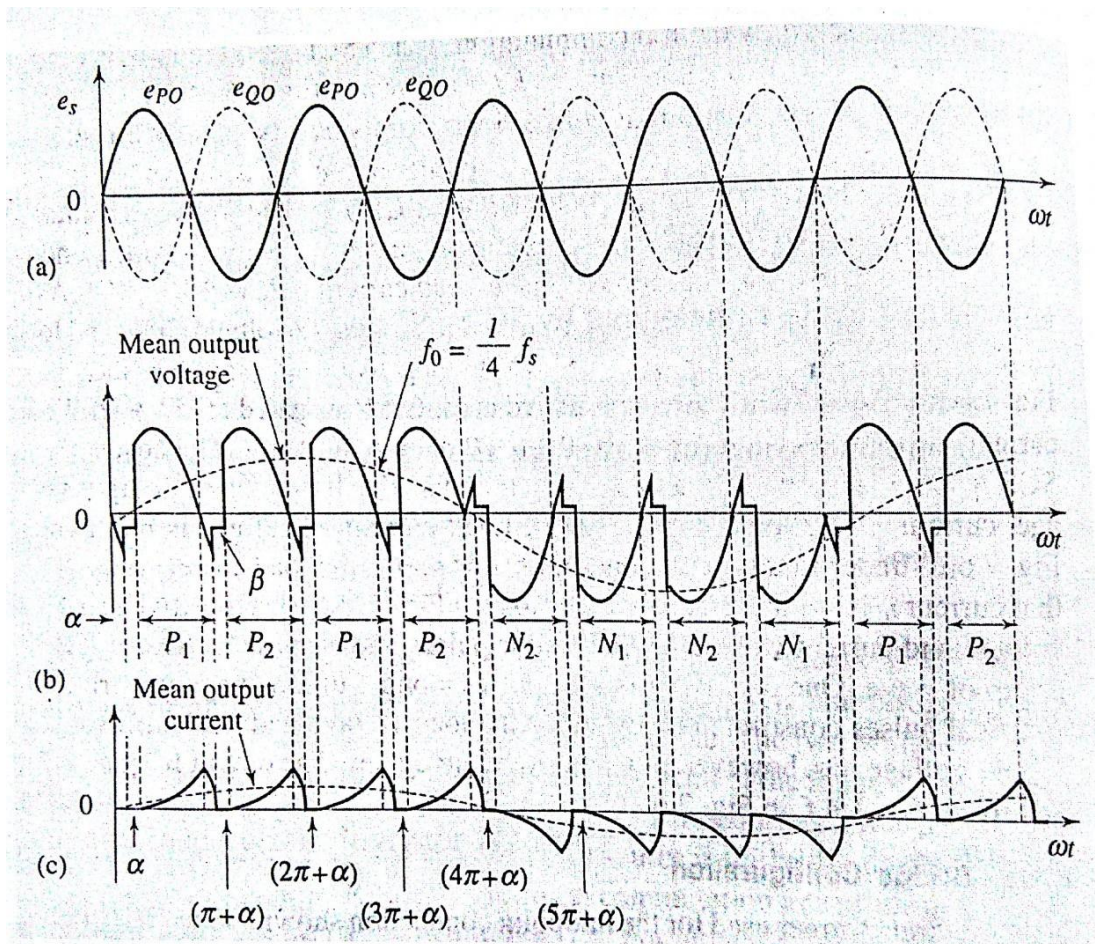
Firing angle delays in sec for pulse generators 1 & 2 for various firing angles can be calculated and tabulated as follows.

FIRING ANGLE (α) in deg	Phase angle delay for (in sec.)	
	Pulse Generator 1	Pulse Generator 2
0	0	0.01
30	0.001667	0.011667
60	0.00333	0.01333
90	0.005	0.015
120	0.00667	0.01667
150	0.00833	0.01833
180	0.01	0.02

CIRCUIT DIAGRAM: -

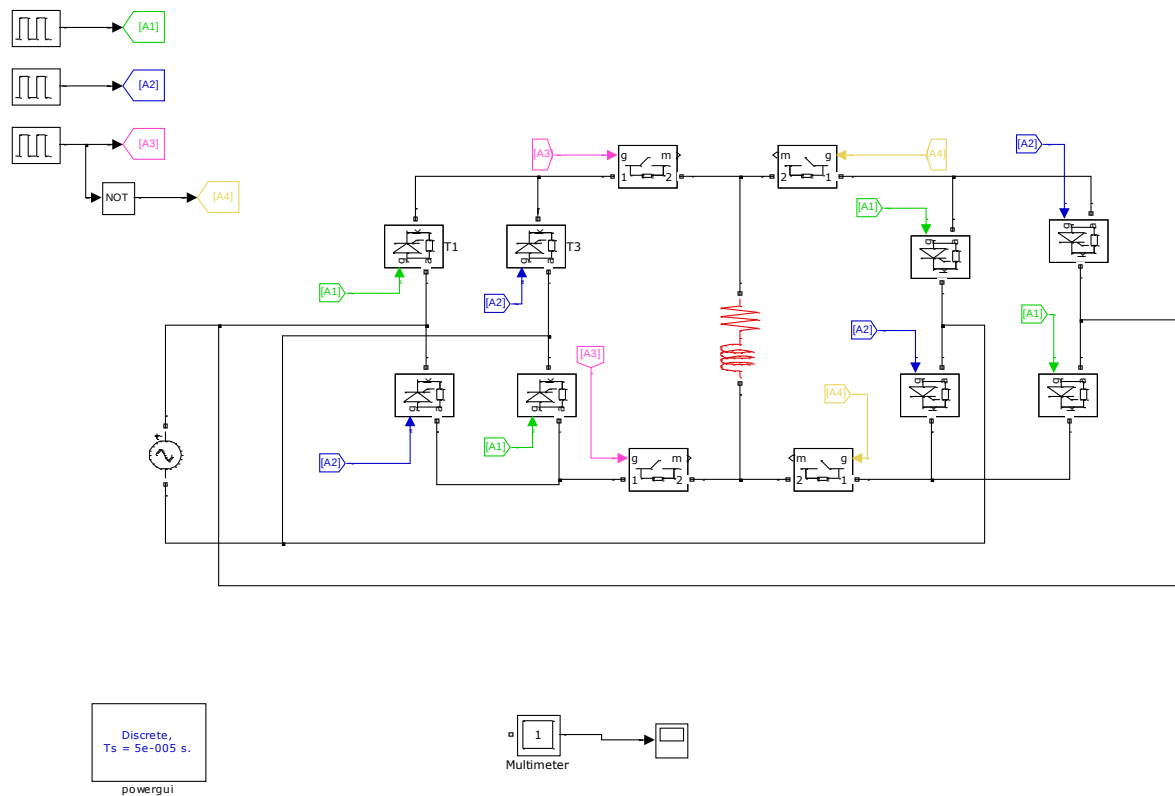


MODEL WAVEFORMS: -

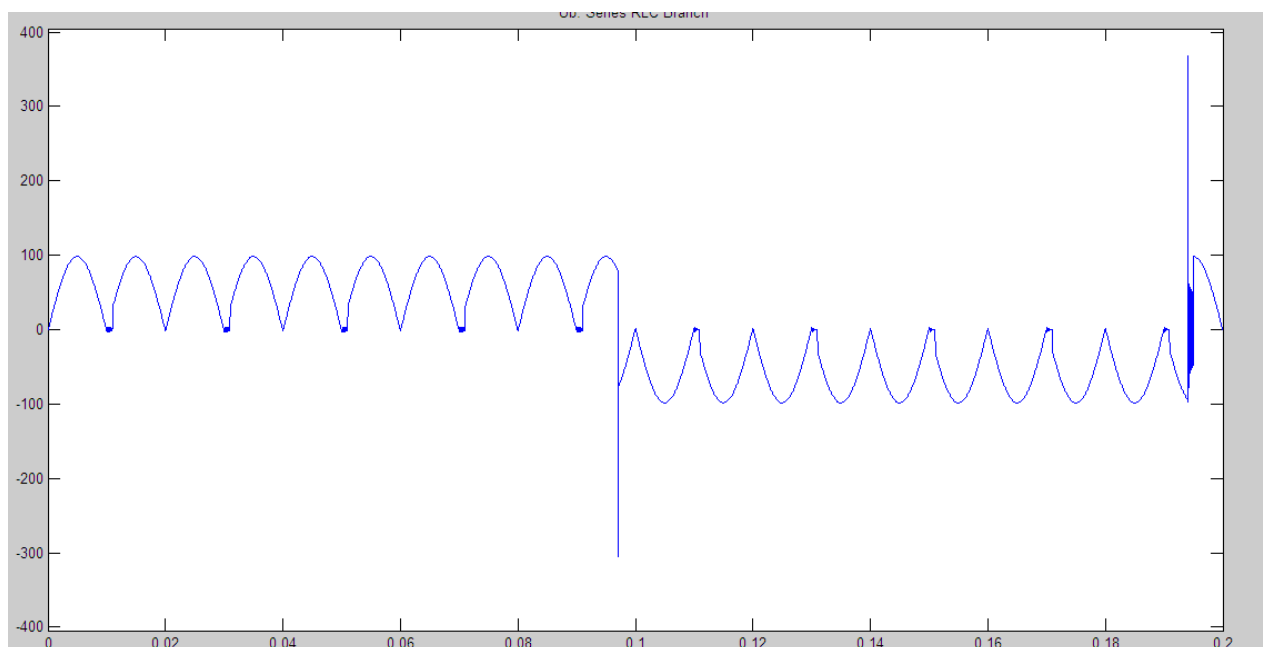


SINGLE PHASE AC VOLTAGE CONVERTER WITH R AND RL LOAD

SIMULATION DIAGRAM FOR RL LOAD: -



GRAPH:



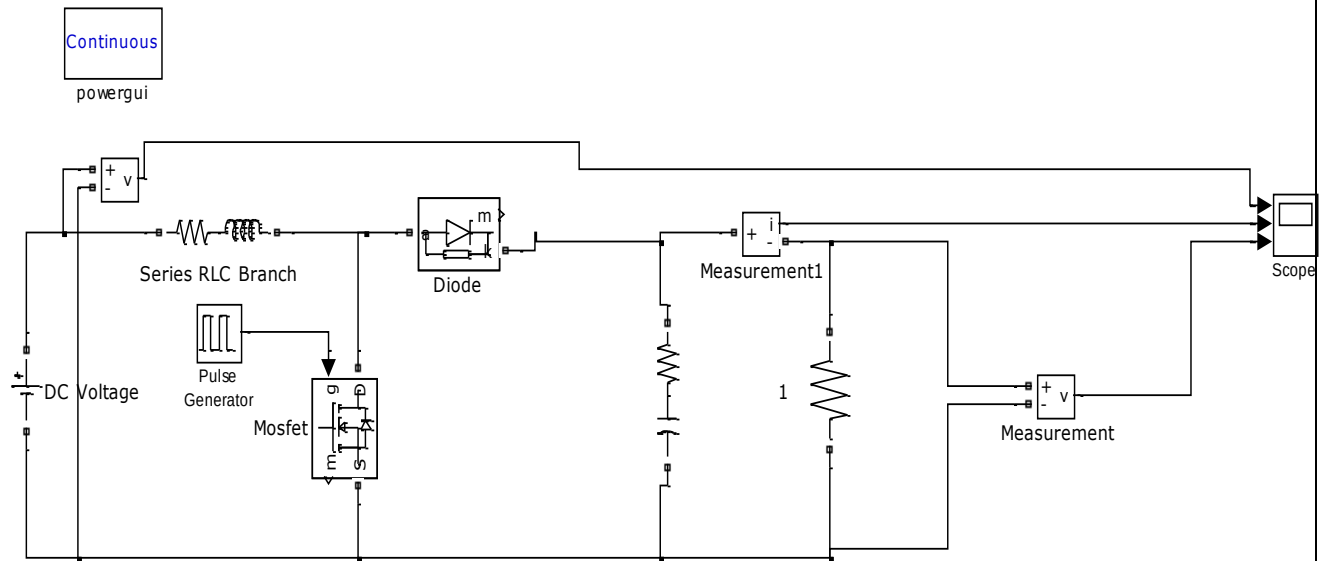
PROCEDURE FOR RL LOAD: -

1. Study the circuit diagram, operation and model waveforms of a single phase AC Voltage controller.
2. Open the MATLAB software by double-clicking the MATLAB icon on the desktop and click the shortcut "Simulink" to open the "Simulink Library Browser".
3. Create a new model and click on "SimPowerSystems" block in "Simulink Library Browser".
4. Drag the "powergui" block in SimPowerSystems submenu to the new model.
5. Double click on the "Electrical Source" block in SimPowerSystems submenu to select the single phase supply source "AC Voltage Source" and drag it to the new model. Set the peak amplitude of voltage as 110 Volts.
6. In the SimPowerSystems submenu, select "Power Electronics" block to pick the "Thyristor" and drag them to the new model.
7. In the SimPowerSystems submenu, select "Elements" block to pick the load "Series RLC branch" and drag it to the new model. Set the value of C as zero by double-clicking on the load.
8. In the SimPowerSystems submenu, select "Measurements" block to pick the "Voltage Measurement" & "Current Measurement" and drag them to the new model.
9. Also find the "RMS" block for converting the output signal to rms value in the "measurements" submenu under Extra Library in SimPowerSystems menu and drag it to the new model.
10. Find the "Display" block for displaying the rms value of the output signal in the "Sinks" submenu under "Simulink" menu and drag it to the new model.
11. In the Simulink submenu, select "Commonly Used Blocks" to pick the "SCOPE" and drag it to the new model.
12. In the Simulink submenu, select "Sources" to pick the "Pulse Generator" for T_1 & T_2 and drag it to the new model.
13. Arrange the components as per the circuit diagram and connect them.
14. Set the values of input source voltage parameters such as amplitude, phase angle and frequency.
15. Set the pulse generator parameters such as pulse type, amplitude, period, pulse width, phase delay, etc. as per circuit design formula.
16. Set the scope axis as per the circuit requirements like input voltage, output voltage, output current, etc.
17. Set the load resistance and inductance values as 10 ohms and 1 H respectively.
18. Simulate the circuit and take the output waveforms in the scope.

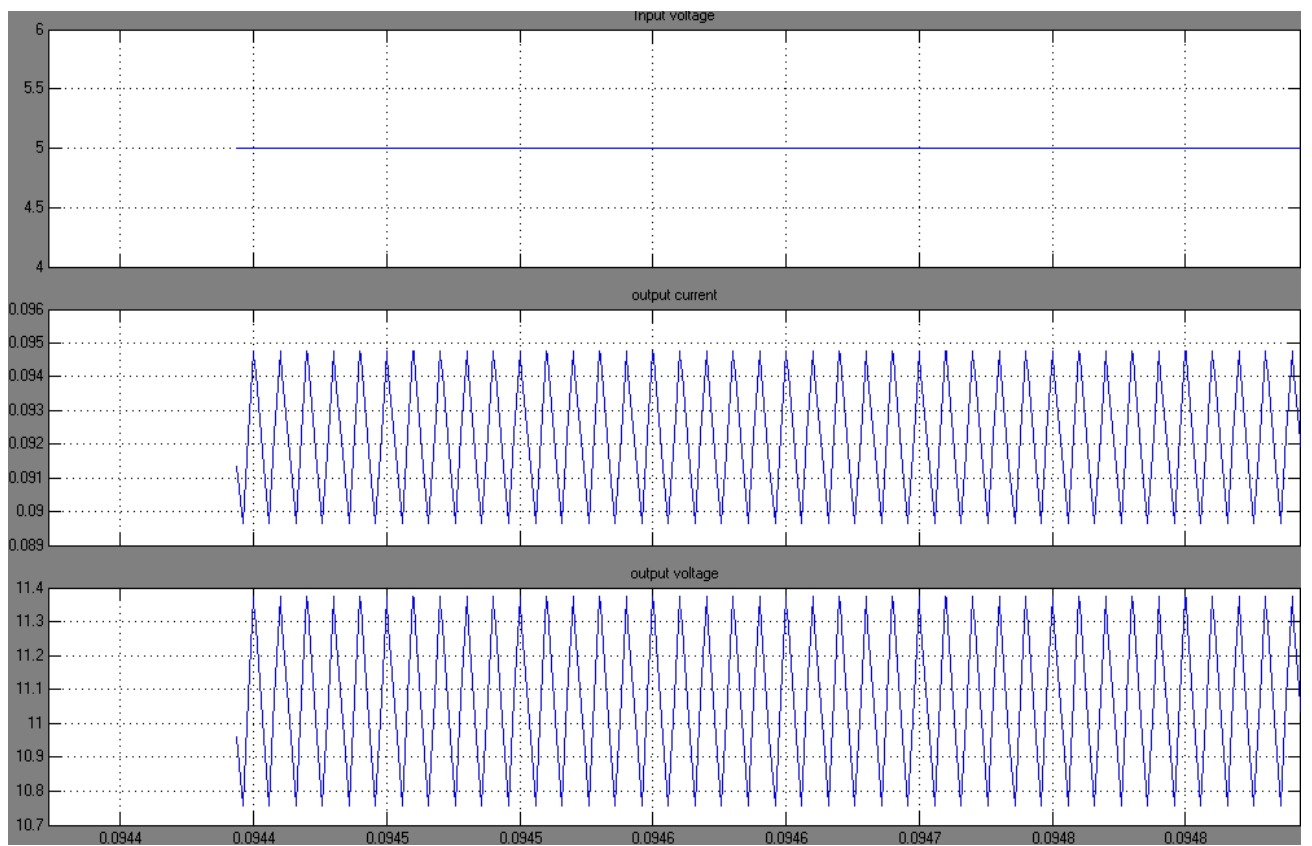
RESULT: -

Thus the single phase AC Voltage converter was simulated with R and RL load of resistance of 10 ohms and inductance of 1 H for the firing angles of 30° , 60° , 90° , 120° , 150° and 180° and the related waveforms were obtained.

BOOST CONVERTER:



MODEL GRAPH:



DC-DC CONVERTER

AIM: -

To simulate and study the DC-DC Converter with R and RL Load in different firing angles.

APPARATUS REQUIRED: -

Sl. No.	Description	Quantity
01	Computer	01
02	Printer	01

SOFTWARE REQUIRED: -

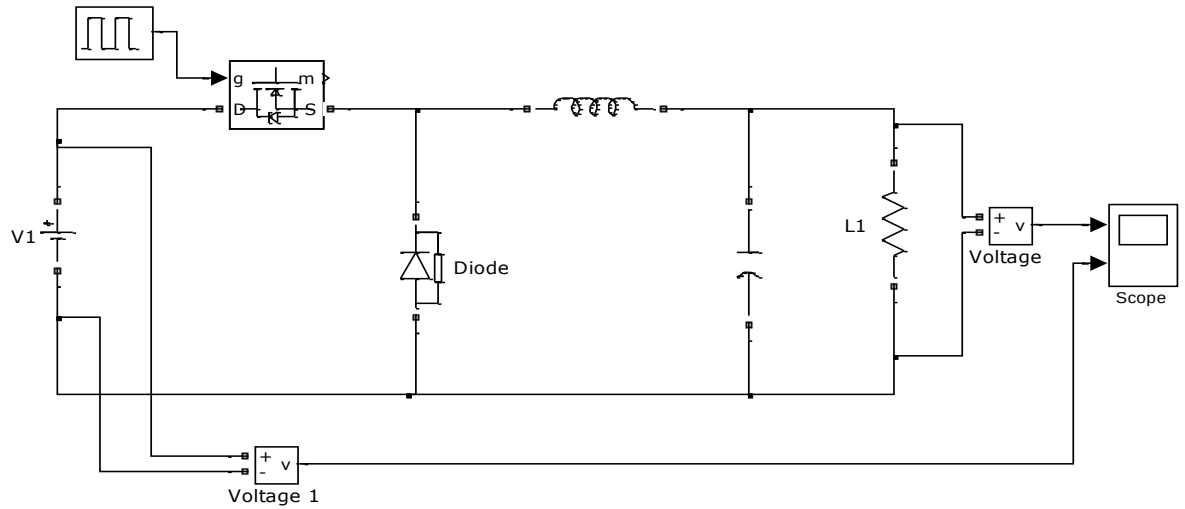
MATLAB 6.1 or above.

PROCEDURE FOR R LOAD: -

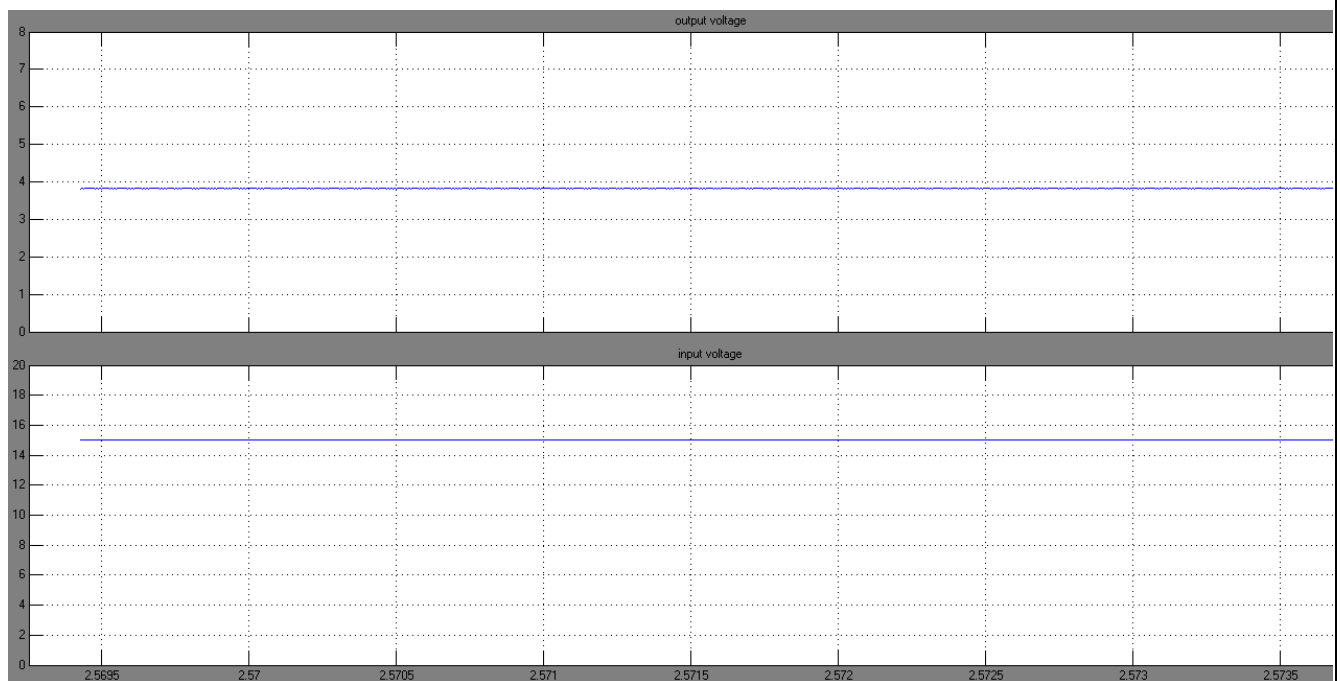
1. Study the circuit diagram, operation and model waveforms of a DC-DC Converter Voltage controller.
2. Open the MATLAB software by double-clicking the MATLAB icon on the desktop and click the shortcut "Simulink" to open the "Simulink Library Browser".
3. Create a new model and click on "SimPowerSystems" block in "Simulink Library Browser".
4. Drag the "powergui" block in SimPowerSystems submenu to the new model.
5. Double click on the "Electrical Source" block in SimPowerSystems submenu to select the single phase supply source "DC Voltage Source" and drag it to the new model. Set the peak amplitude of voltage as 110 Volts.
6. In the SimPowerSystems submenu, select "Power Electronics" block to pick the "Thyristor" and drag them to the new model.
7. In the SimPowerSystems submenu, select "Elements" block to pick the load "Series RLC branch" and drag it to the new model. Set the values of L & C as zero by double-clicking on the load.
8. In the SimPowerSystems submenu, select "Measurements" block to pick the "Voltage Measurement" & "Current Measurement" and drag them to the new model.
9. Also find the "RMS" block for converting the output signal to rms value in the "measurements" submenu under Extra Library in SimPowerSystems menu and drag it to the new model.
10. Find the "Display" block for displaying the rms value of the output signal in the "Sinks" submenu under "Simulink" menu and drag it to the new model.
11. In the Simulink submenu, select "Commonly Used Blocks" to pick the "SCOPE" and drag it to the new model.
12. In the Simulink submenu, select "Sources" to pick the "Pulse Generator" for T_1 & T_2 and drag it to the new model.
13. Arrange the components as per the circuit diagram and connect them.
14. Set the values of input source voltage parameters such as amplitude, phase angle and frequency.
15. Set the pulse generator parameters such as pulse type, amplitude, period, pulse width, phase delay, etc. as per circuit design formula.

BOOST CONVERTER:

Continuous
powergui



MODEL GRAPH:



FORMULA USED FOR DESIGN OF PHASE DELAY: -

1. Firing angle delay for pulse generator 1 = $\frac{(0 + \alpha) * 0.02}{360}$ sec
2. Firing angle delay for pulse generator 2 = $\frac{(180 + \alpha) * 0.02}{360}$ sec

Where, α = firing angle delay in degrees.

Frequency = 50 Hz, i.e. time period = 0.02 sec.

PHASE DELAY CALCULATIONS: -

Let the firing angle be 30°.

i.e., $\alpha = 30^\circ$

$$\begin{aligned}\text{Firing angle delay for pulse generator 1} &= \frac{(0 + 30) * 0.02}{360} \text{ sec} \\ &= 0.001667 \text{ sec.}\end{aligned}$$

$$\begin{aligned}\text{Firing angle delay for pulse generator 2} &= \frac{(180 + 30) * 0.02}{360} \text{ sec} \\ &= 0.011667 \text{ sec.}\end{aligned}$$

Firing angle delays in sec for pulse generators 1 & 2 for various firing angles can be calculated and tabulated as follows.

FIRING ANGLE (α) in deg	Phase angle delay for (in sec.)	
	Pulse Generator 1	Pulse Generator 2
0	0	0.01
30	0.001667	0.011667
60	0.00333	0.01333
90	0.005	0.015
120	0.00667	0.01667
150	0.00833	0.01833
180	0.01	0.02

RESULT: -

Thus the single phase DC-DC Voltage converter was simulated with R and RL load of resistance of 10 ohms and inductance of 1 H for the firing angles of 30°, 60°, 90°, 120°, 150° and 180° and the related waveforms were obtained.