

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

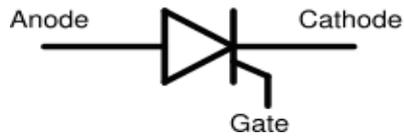
Ex.NO	DATE	NAME OF THE EXPERIMENTS	SIGNATURE
1.		CHARACTERISTICS OF SCR	
2.		CHARACTERISTICS OF TRIAC	
3.		CHARACTERISTICS OF MOSFET & IGBT	
4.		AC TO DC HALF CONTROLLED CONVERTER	
5.		AC TO DC FULLY CONTROLLED CONVERTER	
6.		STEP UP AND STEP DOWN MOSFET BASED CHOPPERS	
7.		IGBT BASED SINGLE PHASE PWM INVERTER	
8.		IGBT BASED THREE PHASE PWM INVERTER	
9.		AC VOLTAGE CONTROLLER	
10.		SWITCHED MODE POWER CONVERTER	
11.		SINGLE PHASE HALF CONTROLLED CONVERTER	
12.		SINGLE PHASE FULLY CONTROLLED CONVERTER	
13.		THREE PHASE HALF CONTROLLED CONVERTER	
14.		THREE PHASE FULLY CONTROLLED CONVERTER	
15.		SINGLE PHASE AC VOLTAGE CONVERTER	
16.		DC-DC CONVERTER	

STAFF IN CHARGE

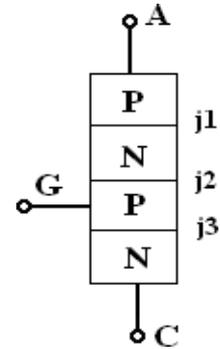
HOD

SILICON CONTROLLED RECTIFIER (SCR):

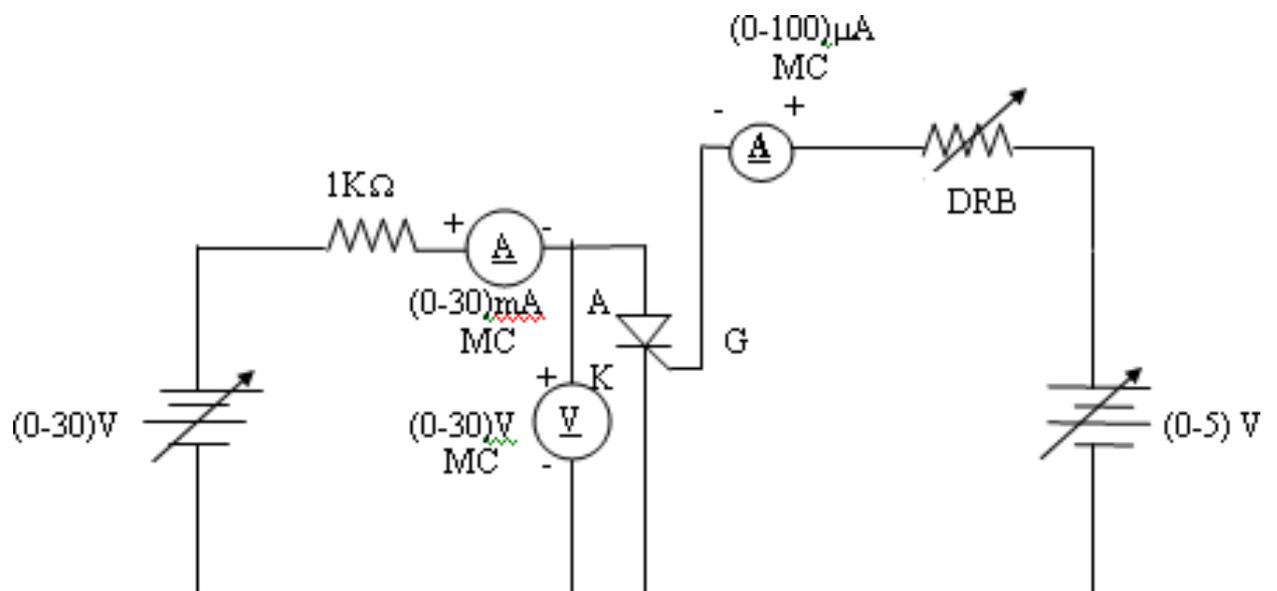
Symbol:



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

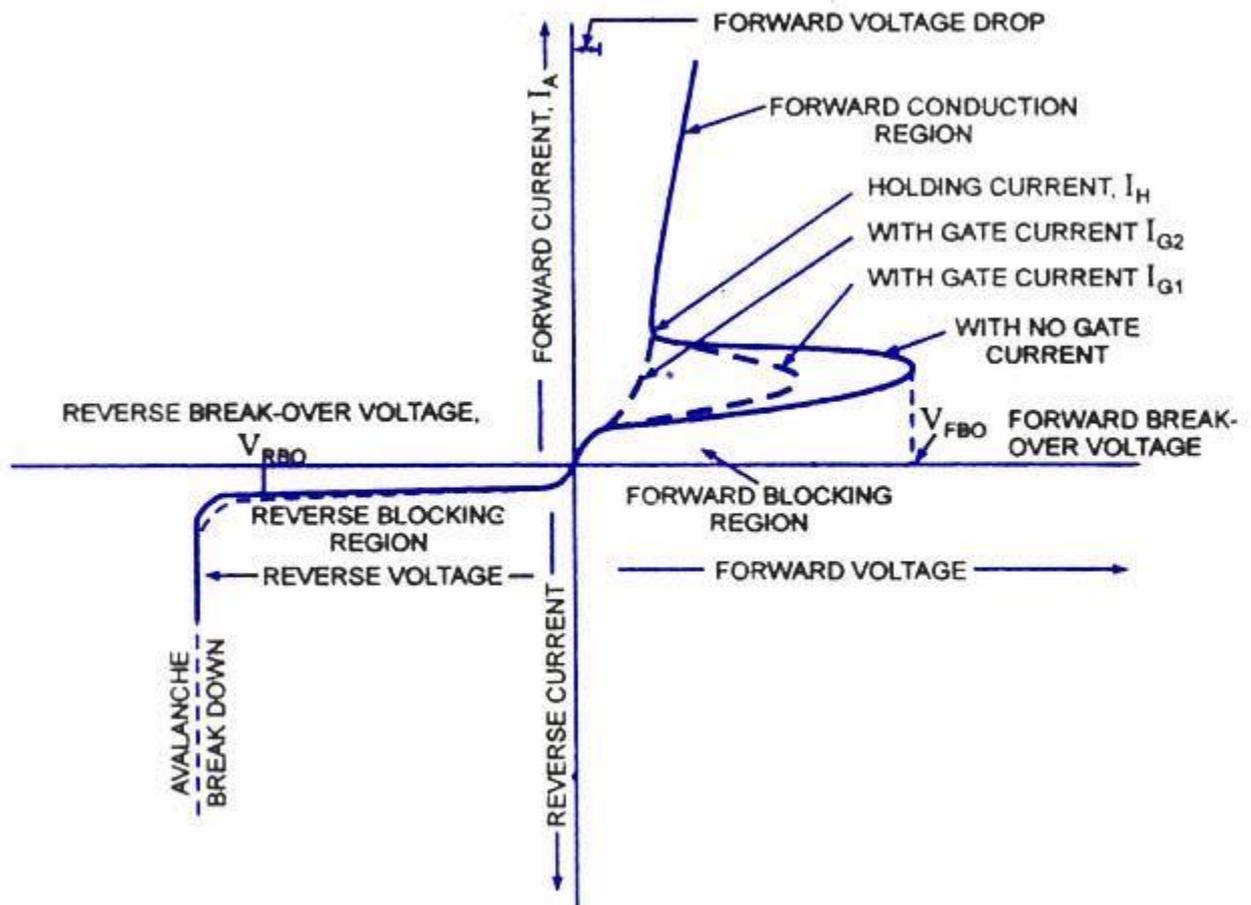
THEORY:

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) :

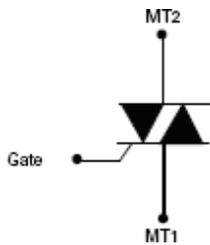
2. Break Down voltage (V_{ak}):

3. Holding Current (I_h) :

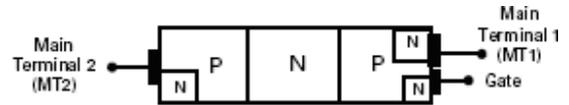
4. Latching Current (I_l) :

TRIODE AC SWITCH (TRIAC):

Symbol:

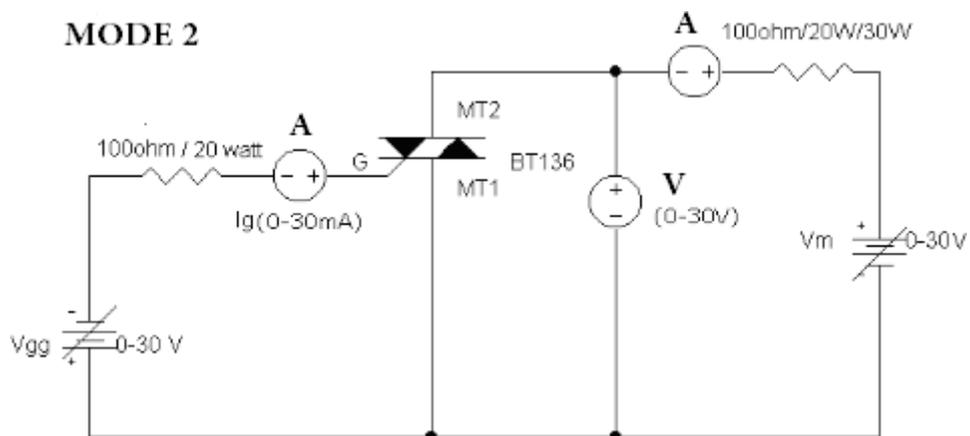
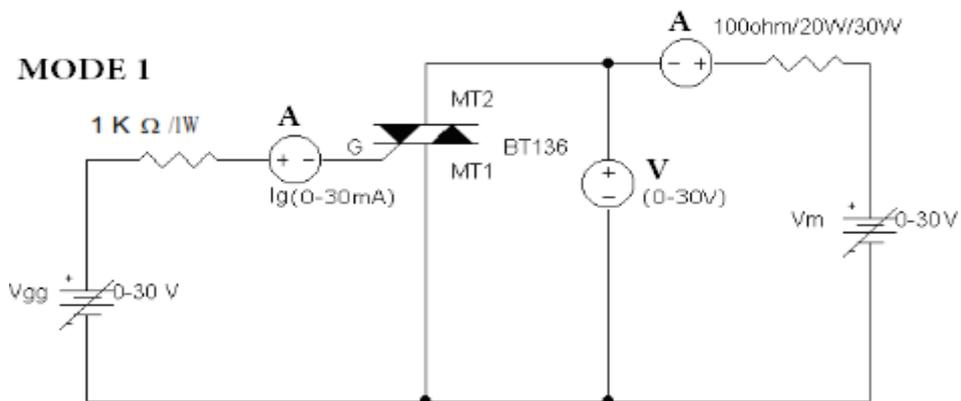


Block Construction:



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:

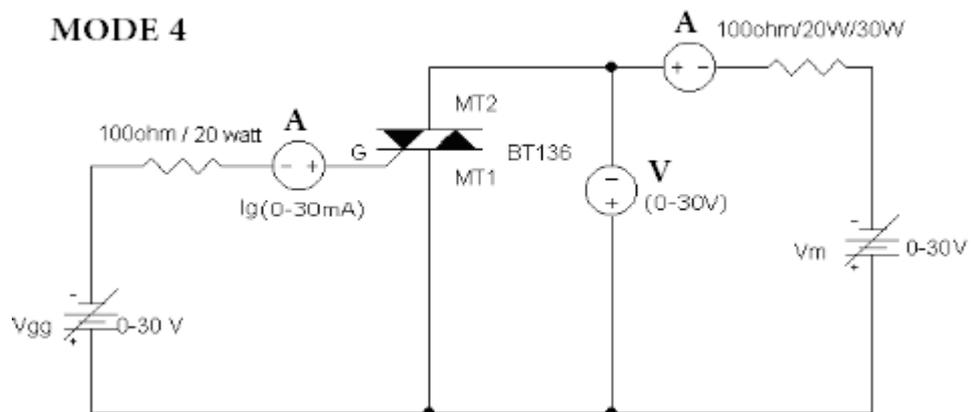
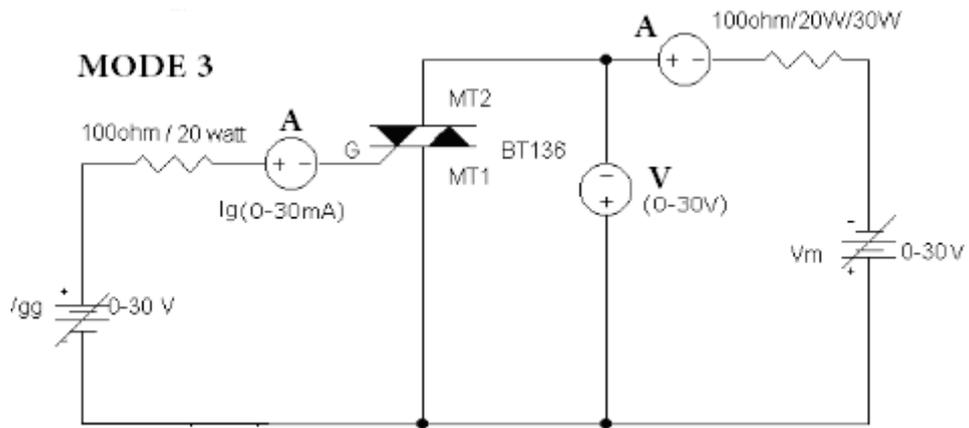
S.No.	APPARATUS	RANGE	TYPE	QUANTITY
1	TRIAC Module kit	220 V / 5 A		1
2	Voltmeter	(0-30) V	MC	1
3	Ammeter	(0-30)mA	MC	1
4	Ammeter	(0-50)mA	MC	1
5	Resistor	1 k Ω		1
6	Patch Chords			10

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 =....(mA)		I_G =....(mA)	
V_{AK}	I_A	V_{AK}	I_A
(V)	(mA)	(V)	(mA)

MODE-2:

I_G =....(mA)		I_G =....(mA)	
V_{AK}	I_A	V_{AK}	I_A
(V)	(mA)	(V)	(mA)

MT2 negative with respect to MT1 positive:

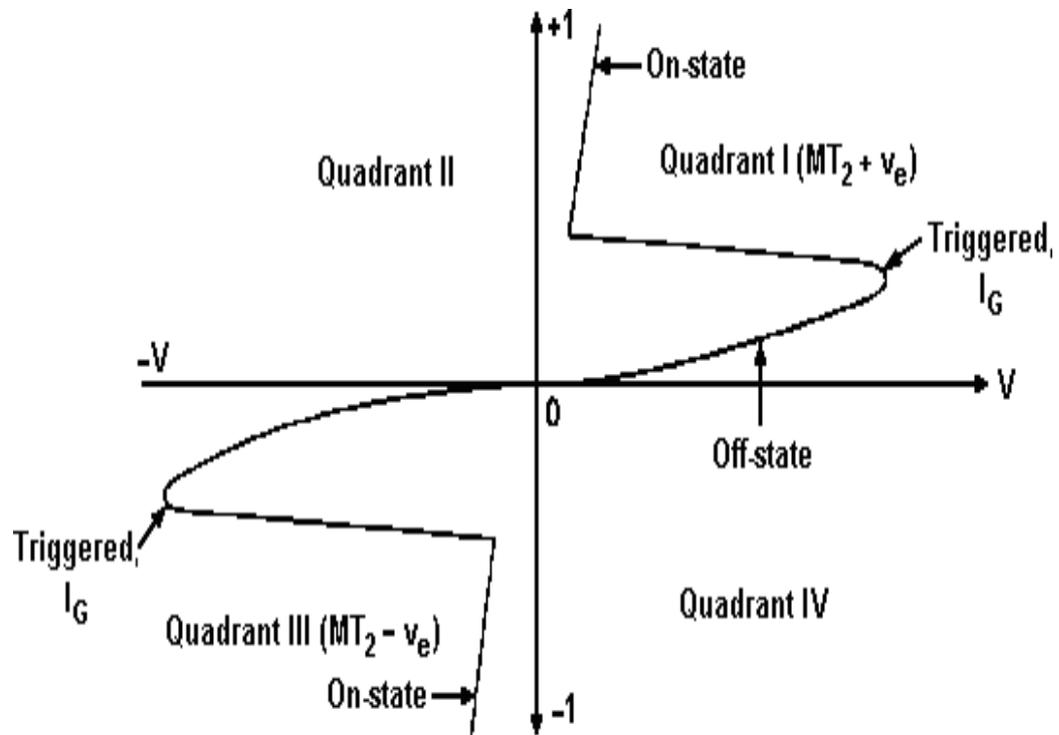
MODE-3:

I_G =....(mA)		I_G =....(mA)	
V_{AK}	I_A	V_{AK}	I_A (mA)
(V)	(mA)	(V)	

MODE-4:

I_G =....(mA)		I_G =....(mA)	
V_{AK}	I_A	V_{AK}	I_A (mA)
(V)	(mA)	(V)	

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:

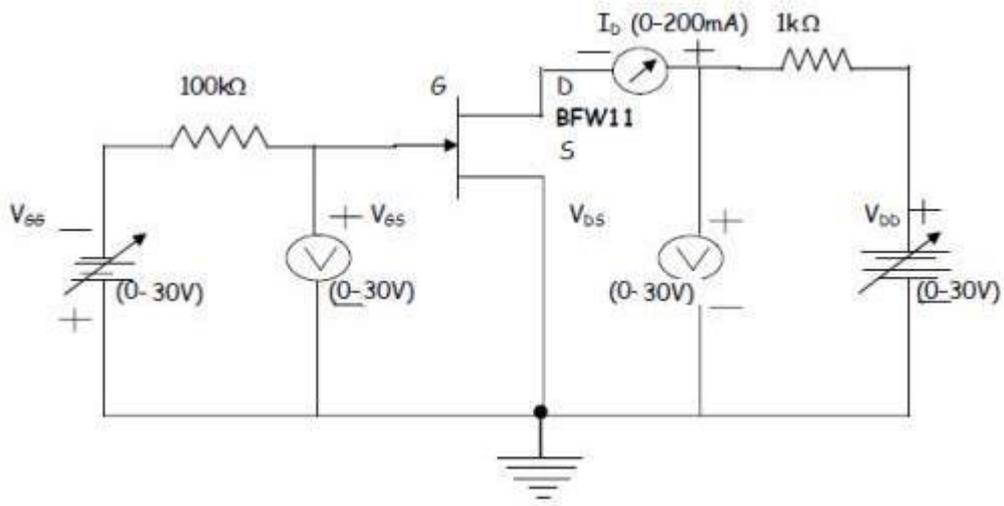
It indicate the variation of drain current I_D as a function of drain–source voltage V_{GS} as a parameter. For low values of V_{DS} , the graph between I_D – V_{DS} is almost linear; this indicates a constant value of on resistance $R_{DS} = V_{DS} / I_D$. For given V_{GS} , if V_{DS} is increased,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

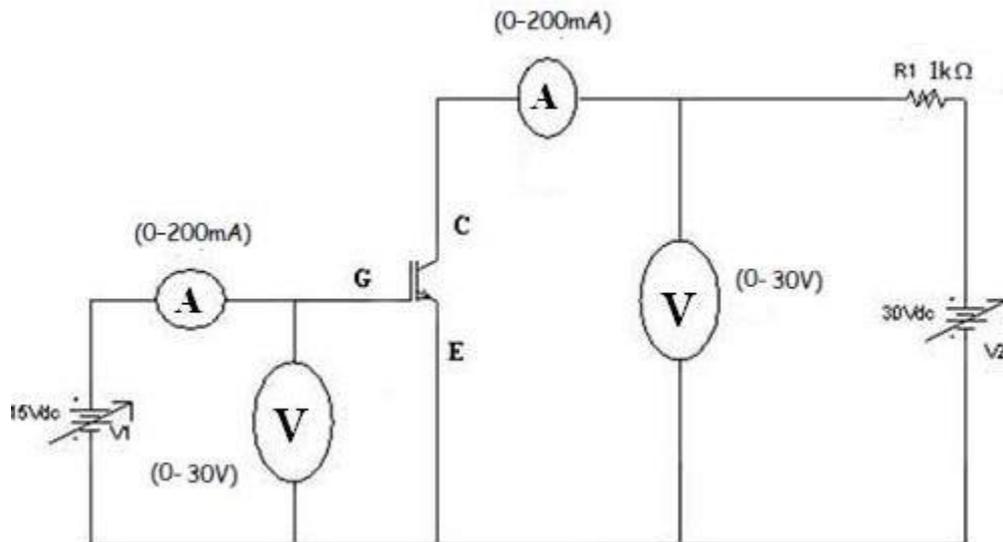
CIRCUIT DIAGRAM:

METAL OXIDE FIELD EFFECT TRANSISTOR (MOSFET):



CIRCUIT DIAGRAM:

INSULATED GATE BASE TRANSISTOR (IGBT):



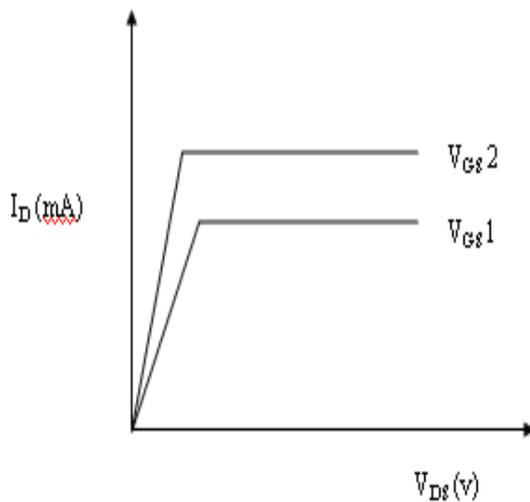
**TABULAR COLUMN (MOSFET):
TRANSFER CHARACTERISTICS:**

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

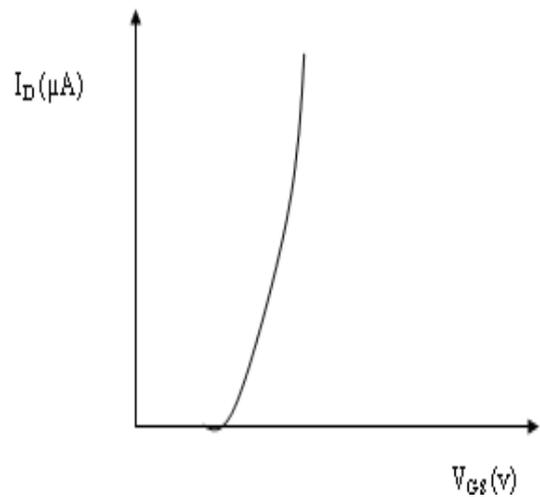
DRAIN CHARACTERISTICS:

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. VRM is the maximum reverse breakdown voltage.

b) Transfer characteristics:

The transfer characteristics of an IGBT is a plot of collector current IC versus Gate-Emitter voltage VGE. This characteristic is identical to that of power MOSFET. When VGE is less than the threshold voltage VGET, 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 VDS (VCE) and ID (IC) is noted .
- 5) The above steps are repeated for different values of IG .
- 6) Vary the voltage across Collector and Emitter and noted down VGE and IC.
- 7) Draw the graph between VGS (VCE) and ID (IC) and VGS (VGE) and ID (IC).

TABULAR COLUMN (IGBT):

TRANSFER CHARACTERISTICS

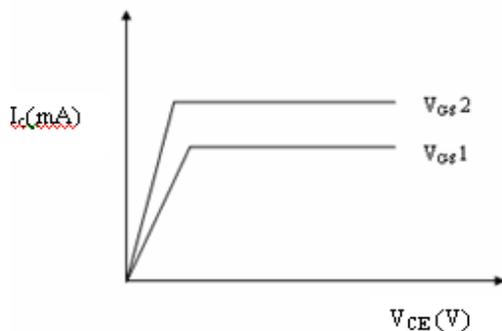
V _{GE} =(V)		V _{GE} =(V)	
V _{CE} (mV)	I _C (mA)	V _{CE} (mV)	I _C (mA)

DRAIN CHARACTERISTICS

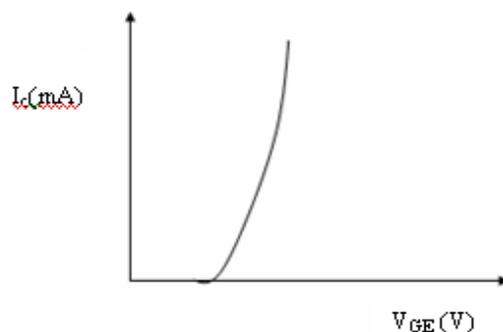
V _{CE} =(V)	
V _{GE} (mV)	I _C (mA)

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 =

Ex. No. 4

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:

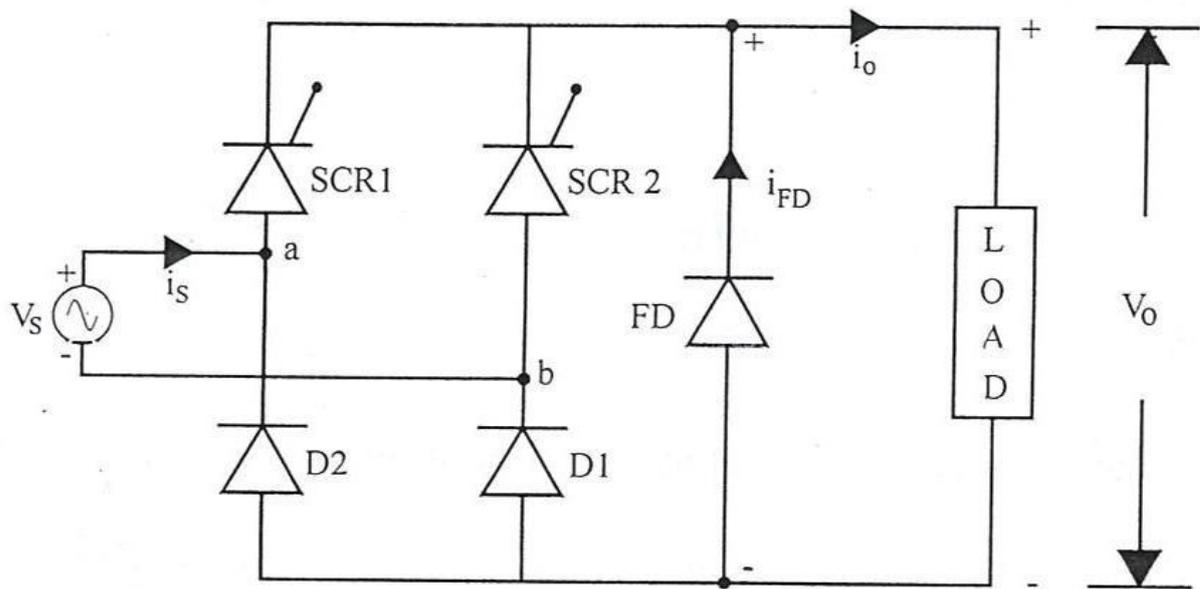
$$\text{Firing angle } \alpha_0 = \frac{t}{10\text{ms}} \times 180$$

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.

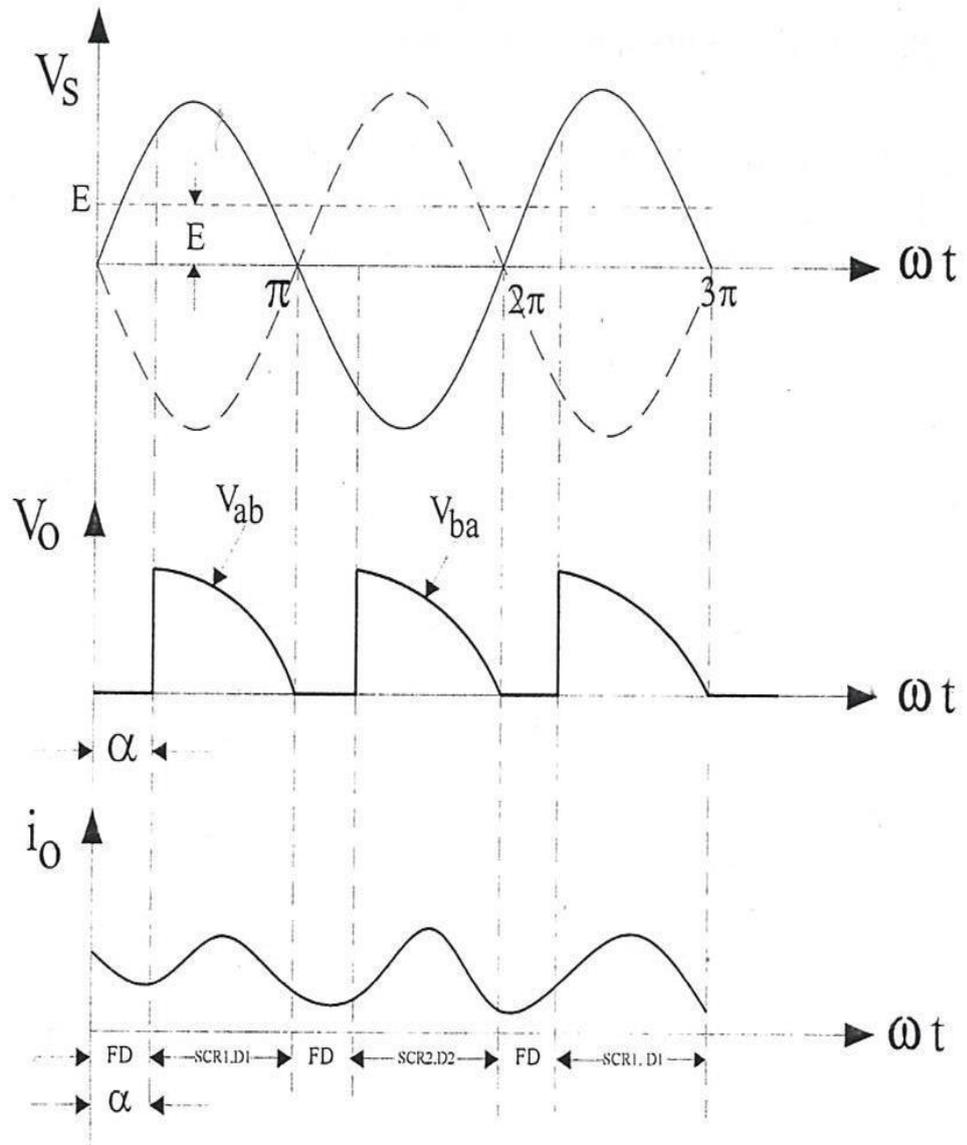
CIRCUIT DIAGRAM :



TABULATOR COLUMN:

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

MODEL GRAPH:

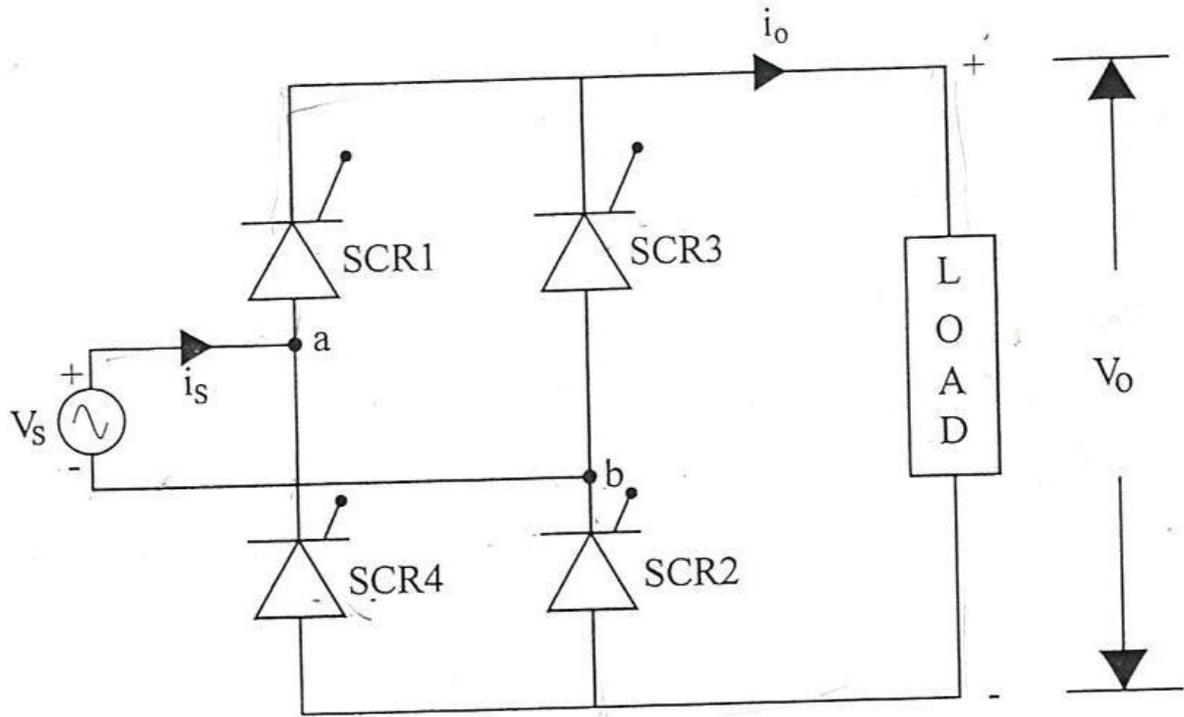


RESULT :

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-Conducting time 't' (ms)	Firing angle α	Measured Voltage(V)	Calculated voltage (V)
R Load						
RL load						
RL with FD						

Ex. No. 5

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 :

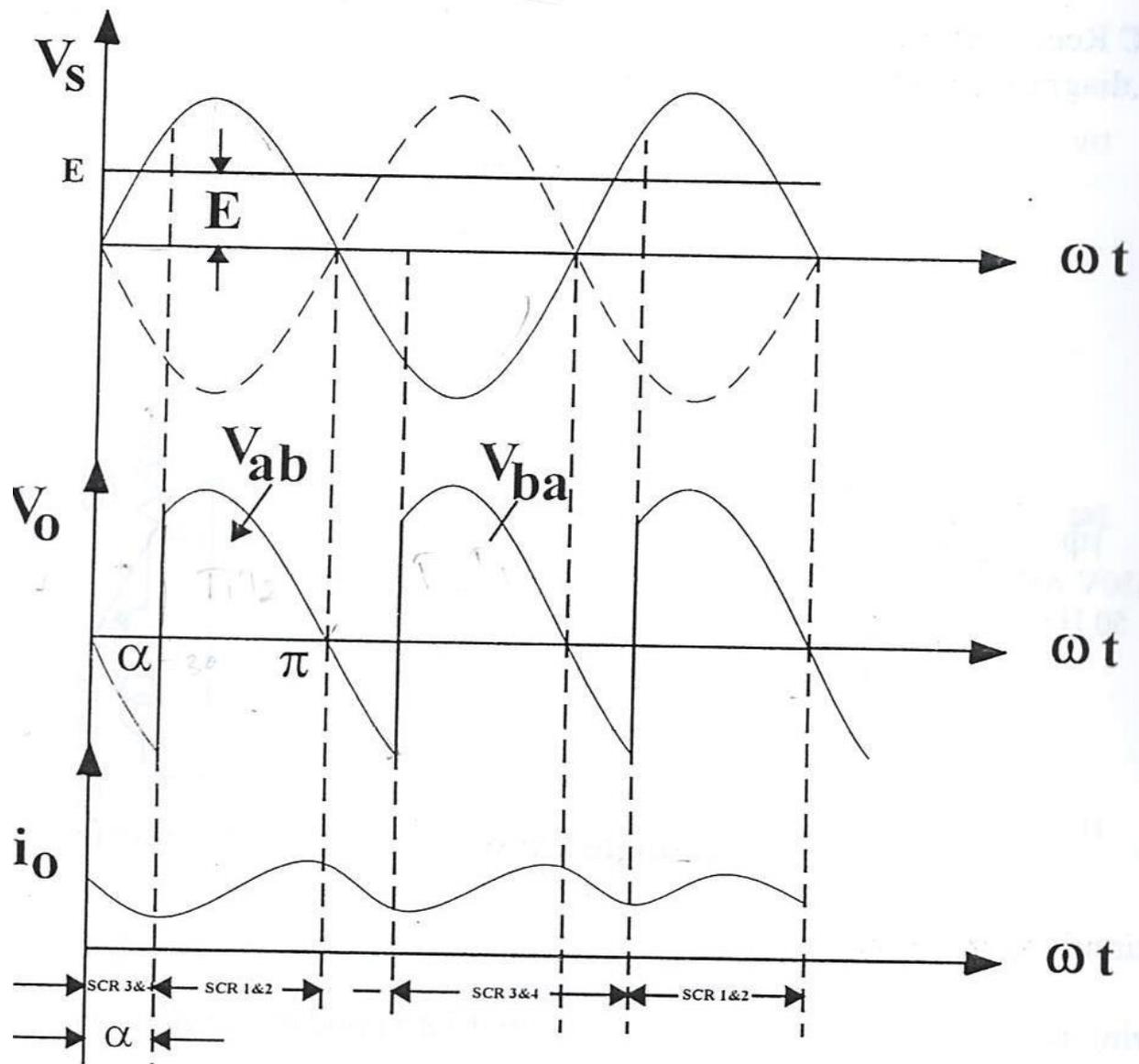
$$\text{Firing angle } \alpha_0 = \frac{t}{10\text{ms}} \times 180$$

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. 6

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:

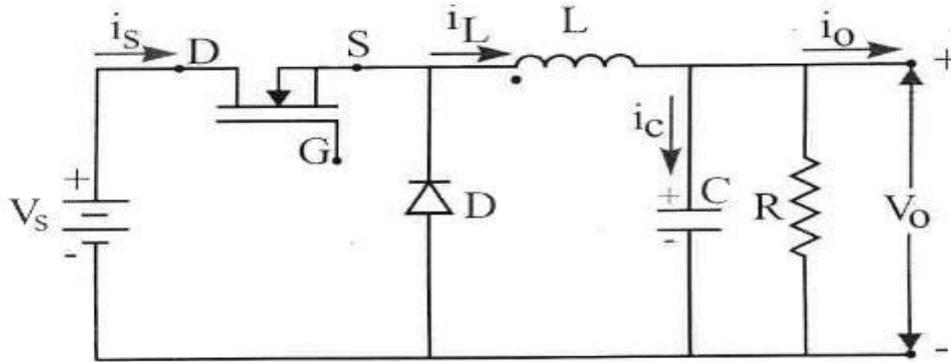
This experiment is intended to study the closed loop operation of Buck-Boost converter. The set voltage to the PWM generator is set at 1V. Feedback voltage from Buck-Boost converter power circuit is connected to the PWM generator circuit. On varying the DC input voltage slowly from 0 to 15V, the output 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 Ton, Toff & output voltage
8. Trace the waveforms of Vo Vs & Io
9. Draw the graph for Vo Vs Duty cycle, K

CIRCUIT DIAGRAM

STEP DOWN CHOPPER :

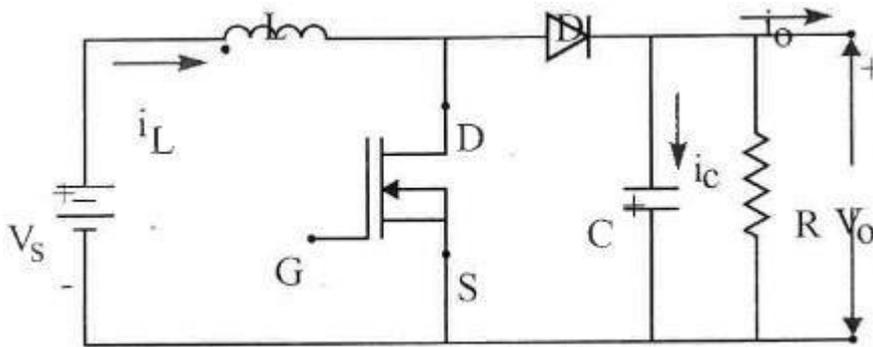


TABULAR COLUMN (STEP UP CHOPPER):

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

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

STEP UP CHOPPER:

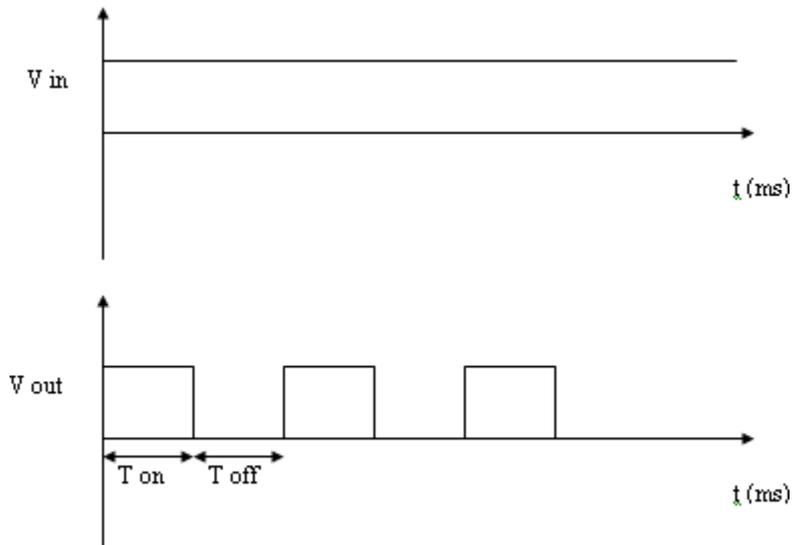


TABULAR COLUMN (STEP DOWN CHOPPER):

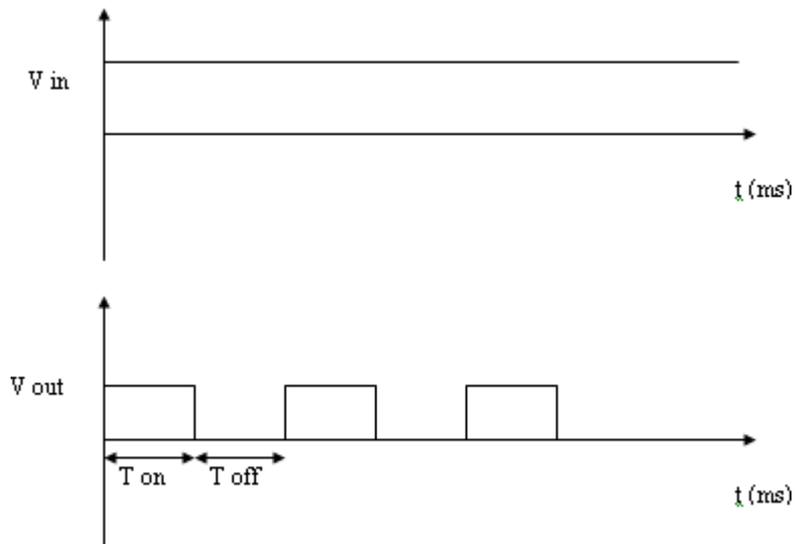
$V_s = \underline{\hspace{2cm}} V$

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

MODEL GRAPH (STEP UP CHOPPER) :



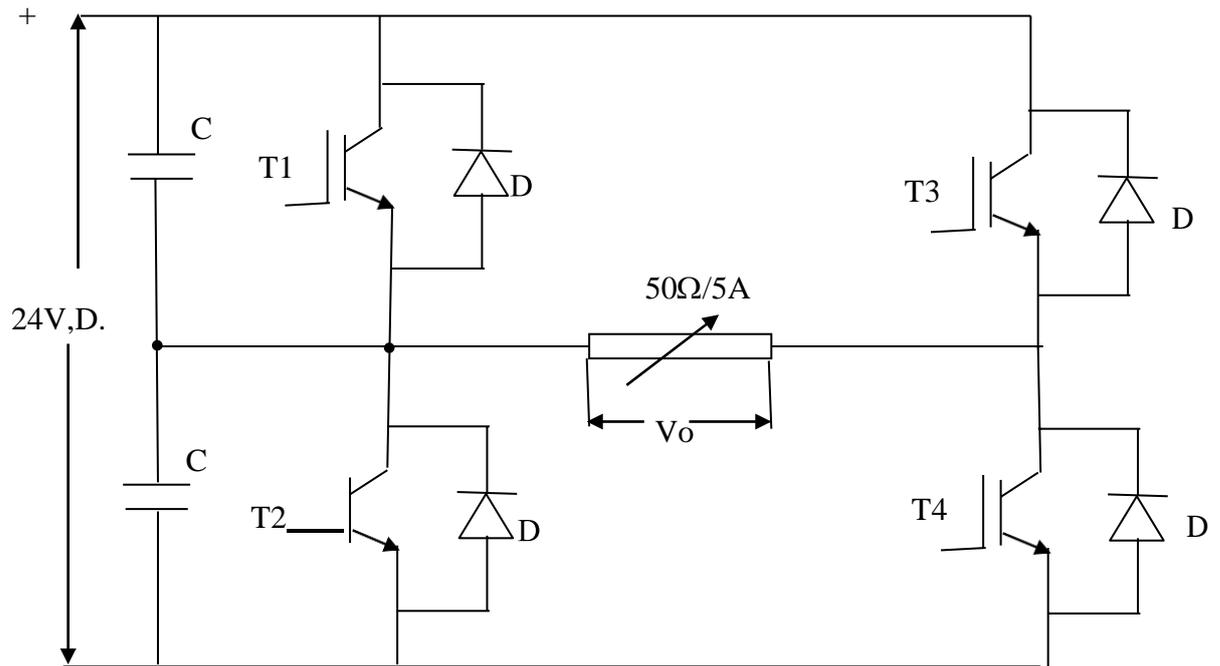
MODEL GRAPH (STEP DOWN CHOPPER) :



RESULT:

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

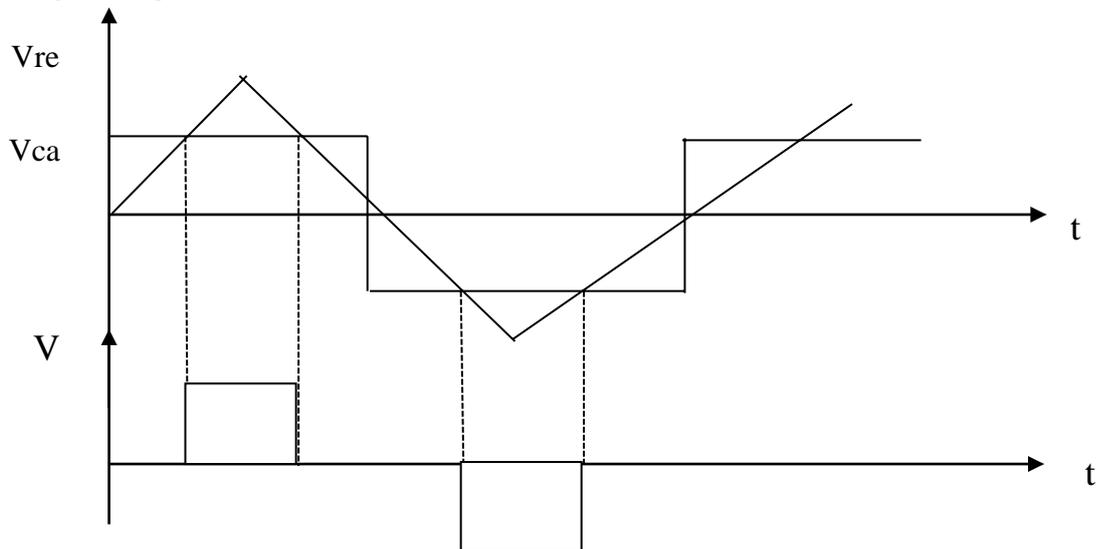
CIRCUIT DIAGRAM:



TABULAR COLUMN:

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

MODEL GRAPH:



Ex. No. 7

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	50Ω/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.

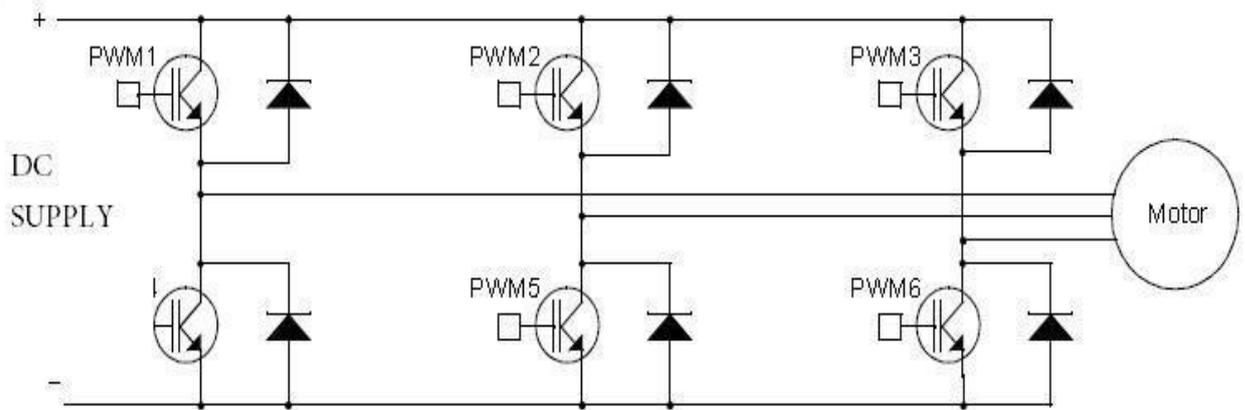
PROCEDURE :

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.

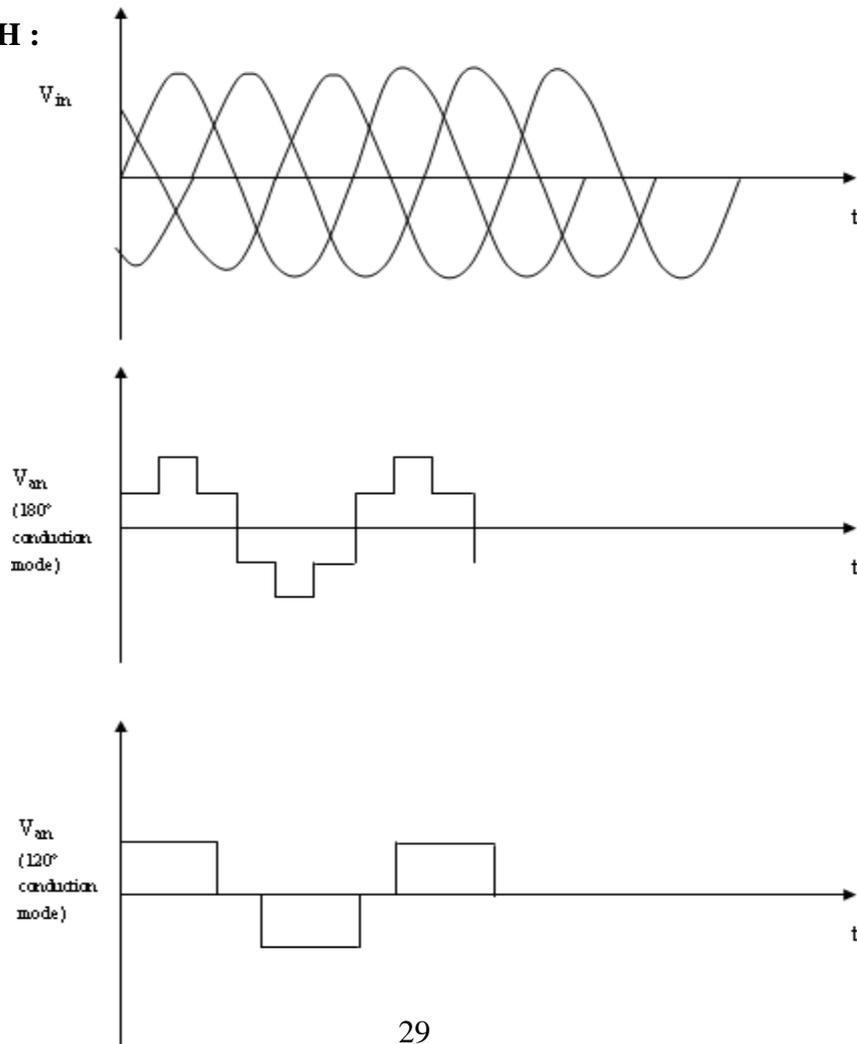
CIRCUIT DIAGRAM:



TABULAR COLUMN:

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

MODEL GRAPH :



Ex. No. 8

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	50Ω/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°.

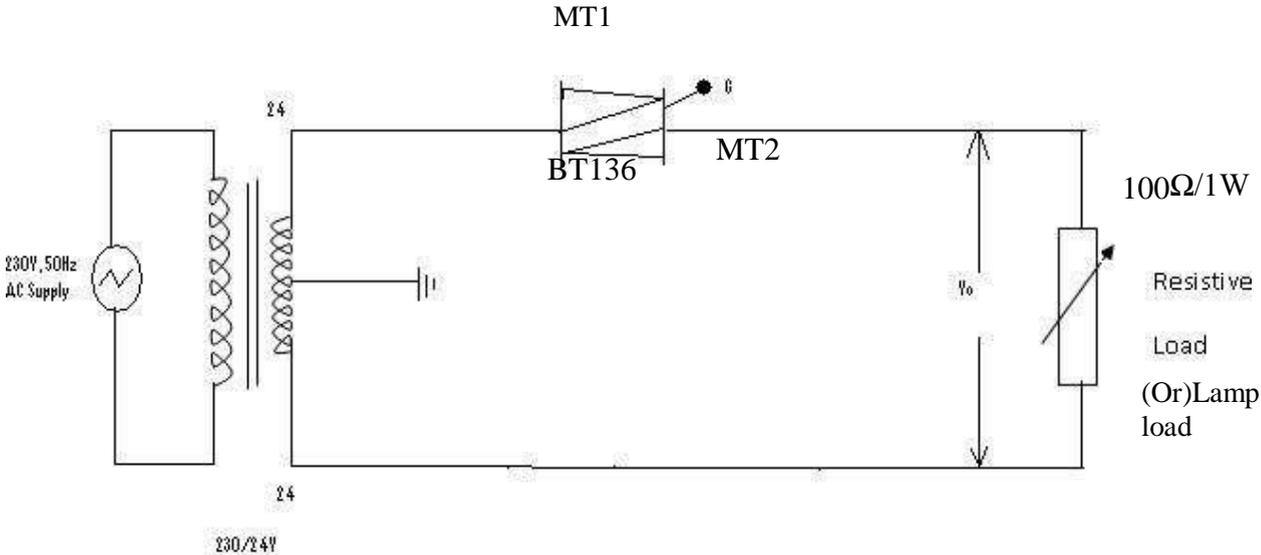
PROCEDURE:

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° conductionmode 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



Date:

Ex. No.9

AC VOLTAGE CONTROLLER

AIM:

To study the Single-phase AC voltage control using TRIAC with DIAC or UJT Firing Circuit.

APPARATUS REQUIRED:

S.No.	APPARATUS	RANGE	TYPE	QUANTITY
1	AC voltage control trainer kit			
2	Lamp	60w		1
3	Resistor	100 ohm/1W 1Kohm		1
4	DIAC	DB3		1
5	TRIAC	BT 136		1
6	CRO			1
7	Patch Chords			15

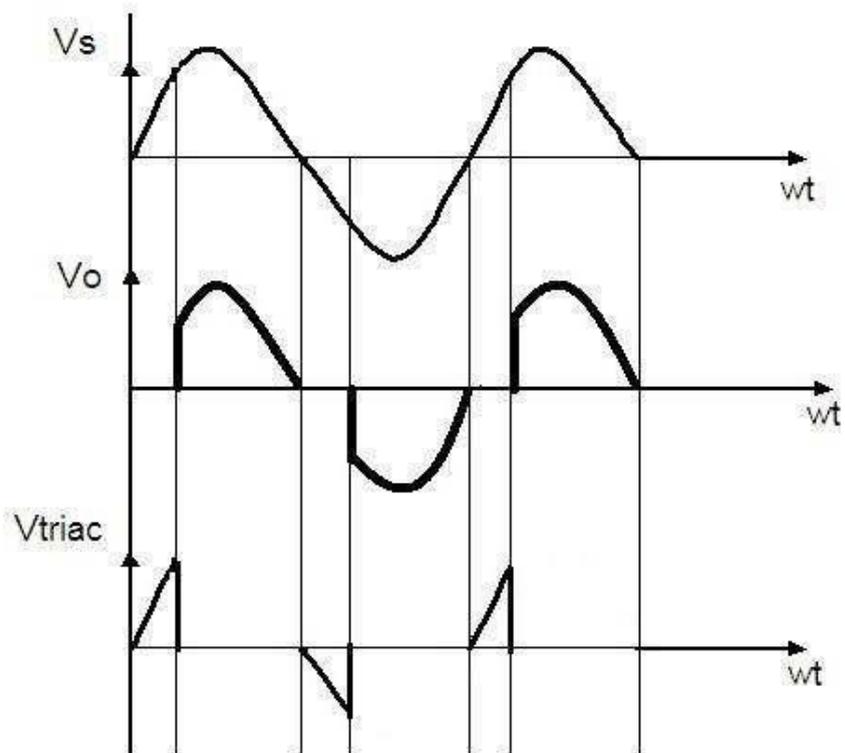
CIRCUIT OPERATION:

1. When potentiometer is in minimum position drop across potentiometer is zero and hence maximum voltage is available across capacitor. This V_c shorts the diac ($V_c > V_{bo}$) and triggers the triac turning triac to ON – state there lamp glows with maximum intensity.
2. When the potentiometer is in maximum position voltage drop across potentiometer is maximum. Hence minimum voltage is available across capacitor ($V_c < V_{bo}$) hence triac is not triggered hence lamp does not glow.
3. When potentiometer is in medium position a small voltage is available across capacitor hence lamp glows with minimum intensity.

TABULAR COLUMN: (DIAC or UJT Firing Circuit)

S.No	V_o (Volts)	V_{in} (Volts)	T_{on} (ms)	T_{off} (ms)	Firing Angle (Degree)

MODAL GRAPH:



PROCEDURE:

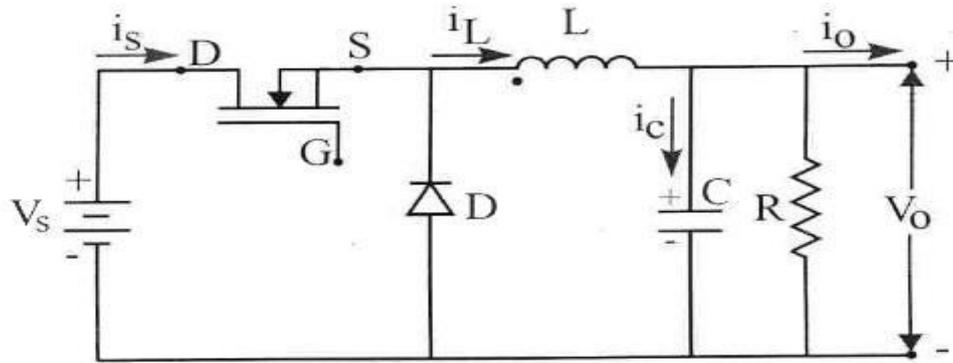
1. Connections are given as per the circuit diagram
2. Initially potentiometer kept at minimum position so lamp does not glow at this instant.
3. Note the voltage across the diac and triac.
4. Capacitor and potentiometer using multimeter and CRO.
5. Potentiometer is now placed at medium and then to minimum position and their voltages were noted.

RESULT:

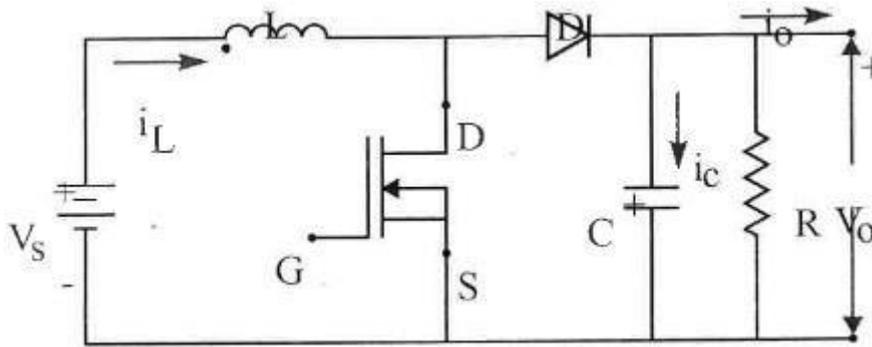
Thus the operation and performance of the single phase AC voltage control using TRIAC is done and output Verified.

CIRCUIT DIAGRAM

BUCK CONVERTER :



BOOST CONVERTER:



Date:

Ex. No. 10

SWITCHED MODE POWER CONVERTER

AIM:

To construct a switched mode power Converter and find its efficiency.

APPARATUS REQUIRED:

S.NO	ITEM	RANGE	QUANTITY
1	Buck-boost kit		1
2	CRO	20 MHZ	1
3	Patch chords		15

PROCEDURE (buck and boost converter) :

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 V_s & I_o
9. Draw the graph for V_o V_s Duty cycle, K

TABULAR COLUMN (Boost converter):

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

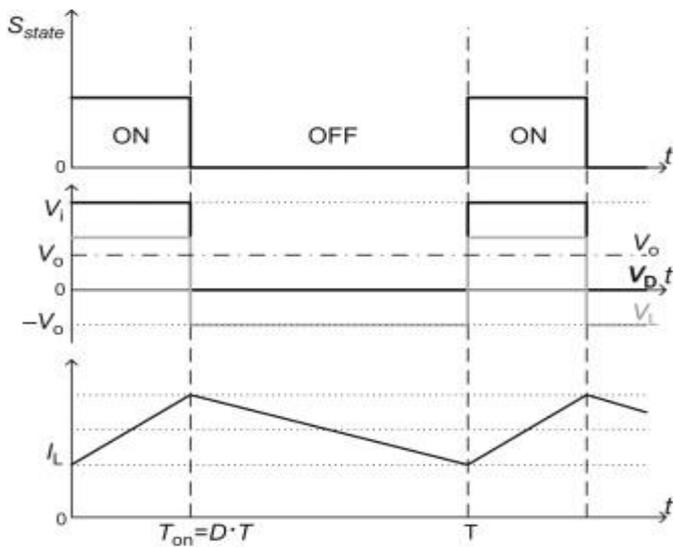
S.NO	T _{ON} (sec)	T _{OFF} (sec)	T (sec)	Duty Ratio, $k=T_{ON} / T$	Measured Voltage (V)	Calculated $V_o=1/1-kV_s(V)$

TABULAR COLUMN (Buck converter):

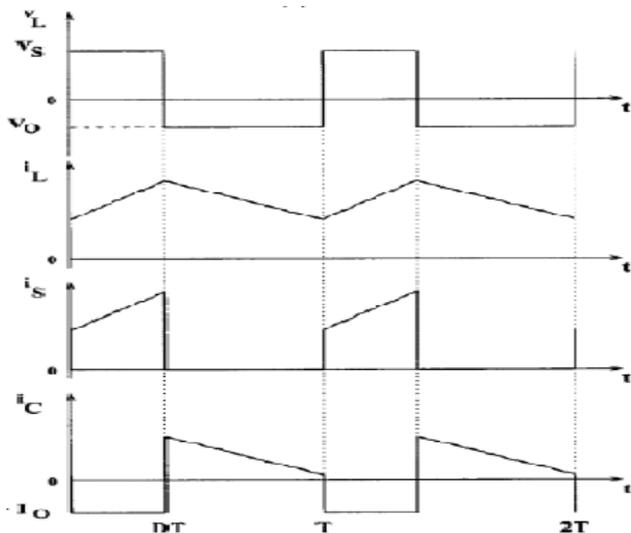
$V_s = \underline{\hspace{2cm}} \text{ V}$

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

BUCK CONVERTER:



BOOST CONVERTER:

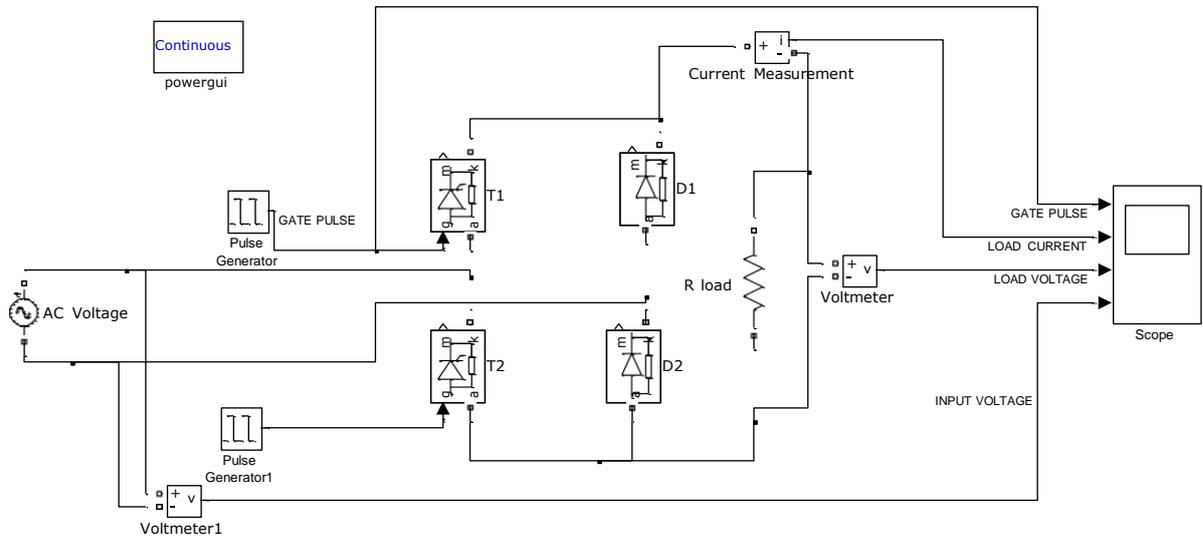


RESULT:

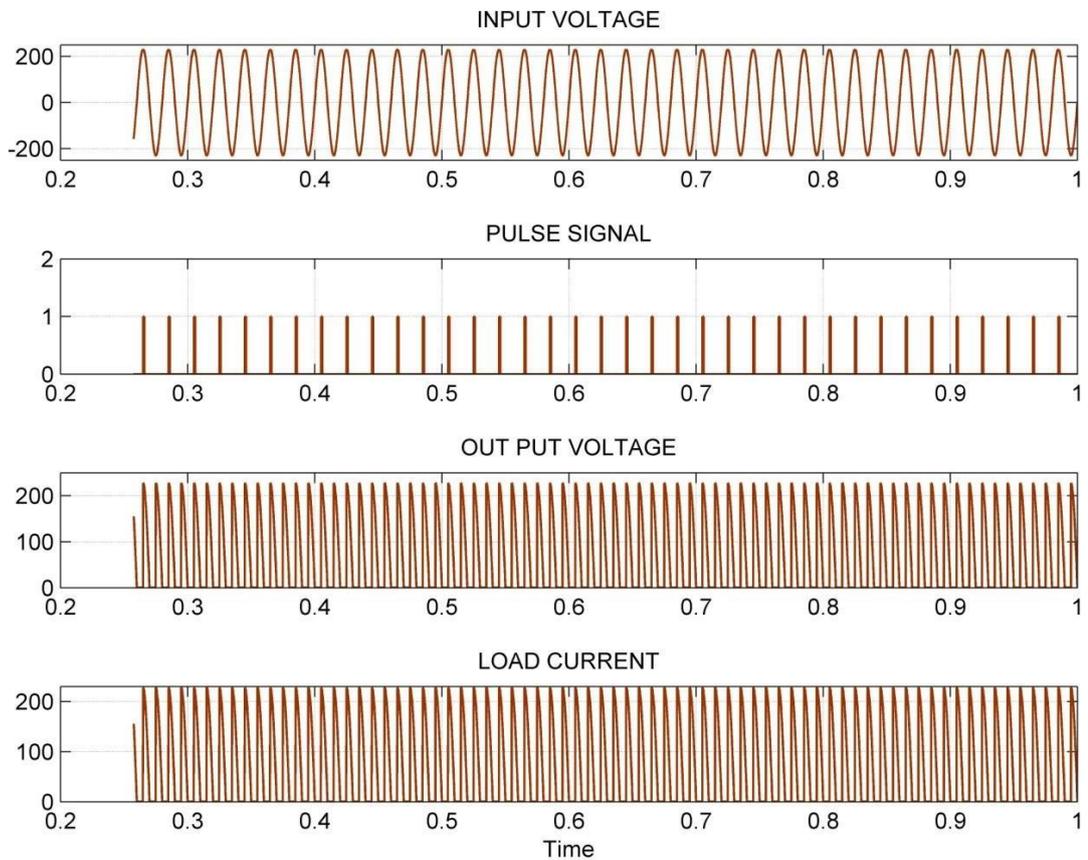
Thus the output response of Buck and Boost converter were drawn.

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

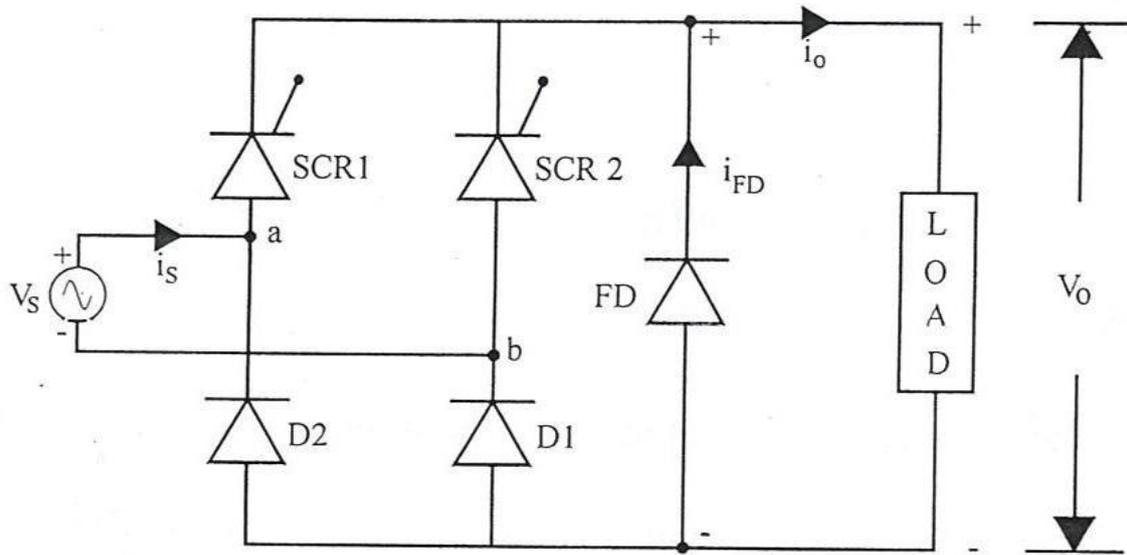
SOFTWARE REQUIRED: -

MATLAB 6.1 or above.

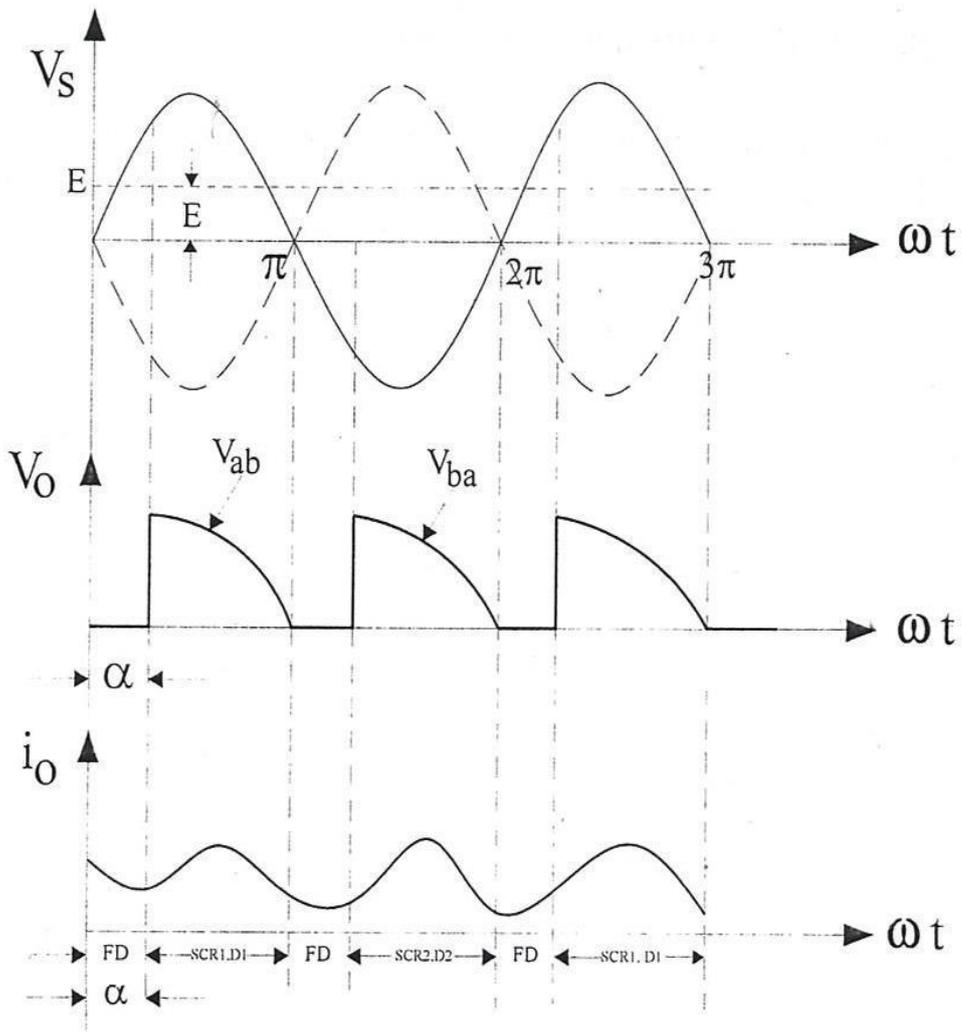
PROCEDURE FOR R LOAD: -

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., $\alpha = 30^\circ$

Firing angle delay for pulse generator 1 = $\frac{(0 + 30) * 0.02}{360}$ sec

= 0.001667 sec.

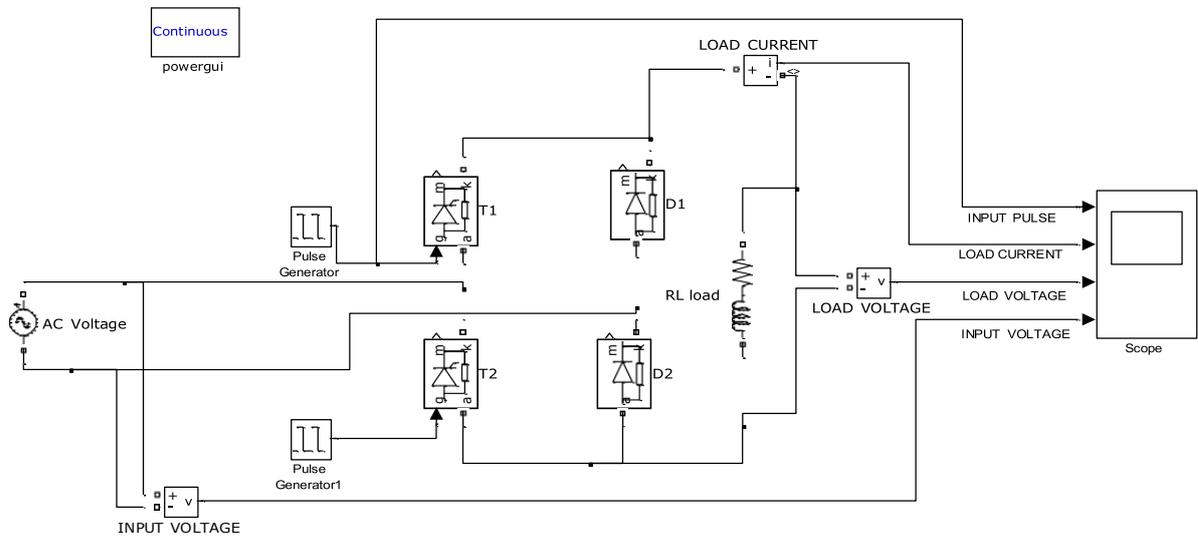
Firing angle delay for pulse generator 2 = $\frac{(180 + 30) * 0.02}{360}$ sec

= 0.011667 sec.

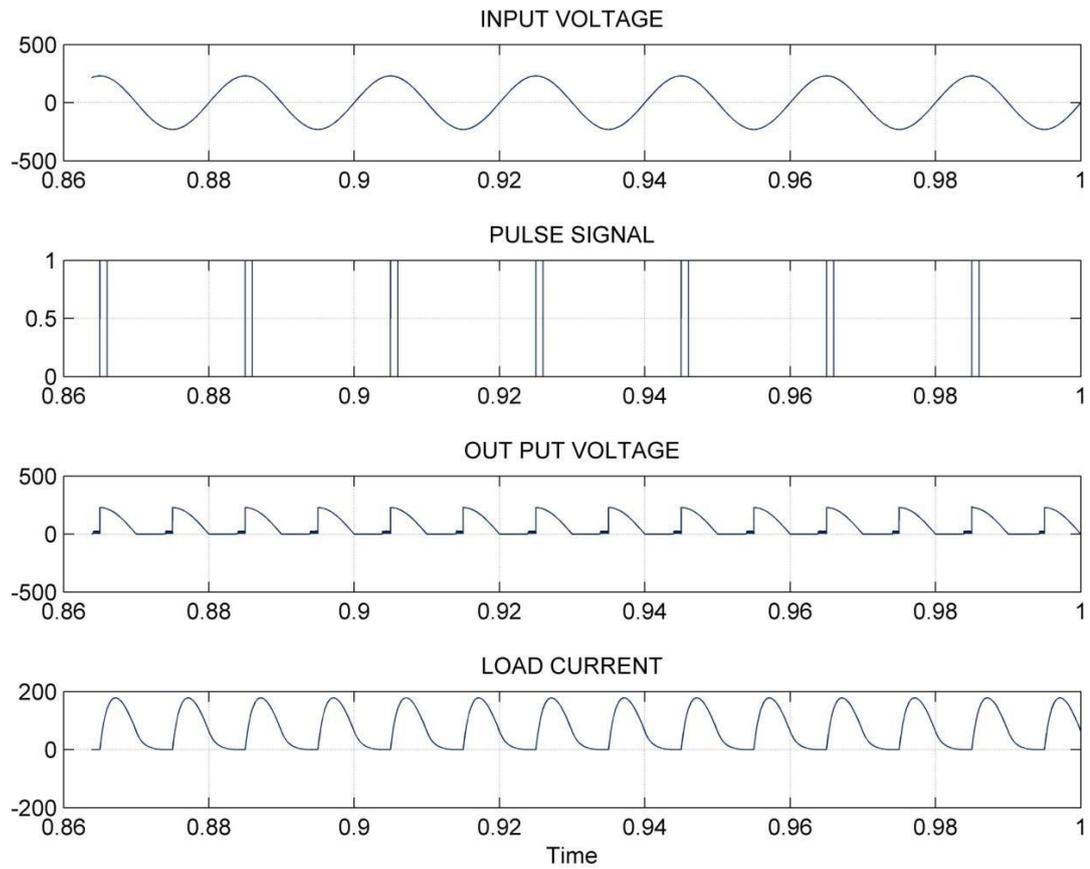
Firing angle delays in sec for pulse generators 1 & 2 for various firing angles can be calculated and tabulated as follows.

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: -

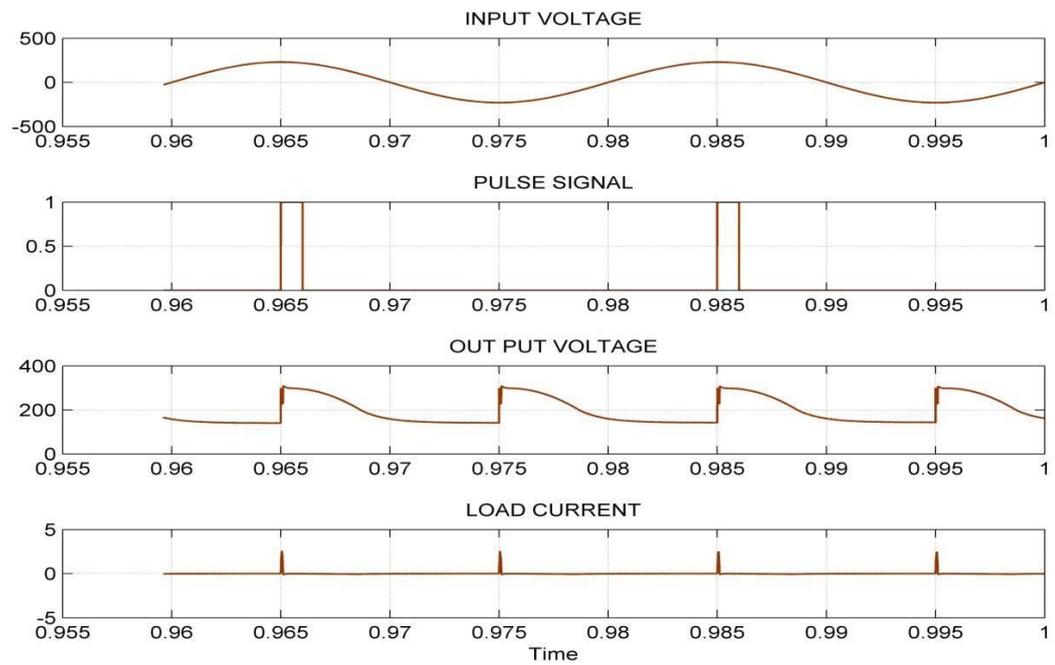


MODEL WAVEFORMS: -



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.: 12

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.

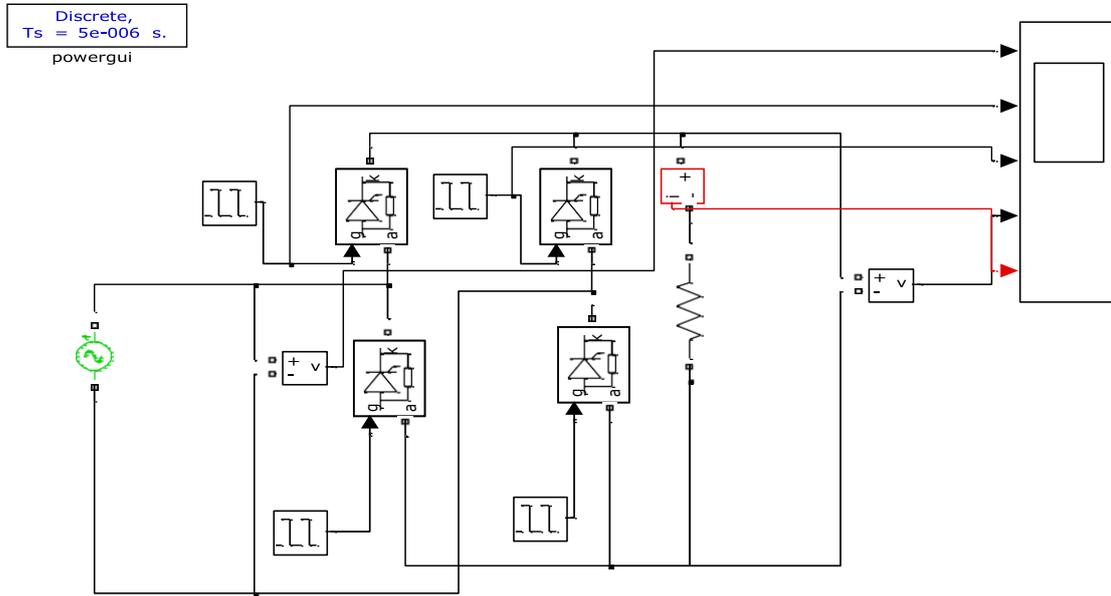
PROCEDURE FOR R LOAD: -

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.

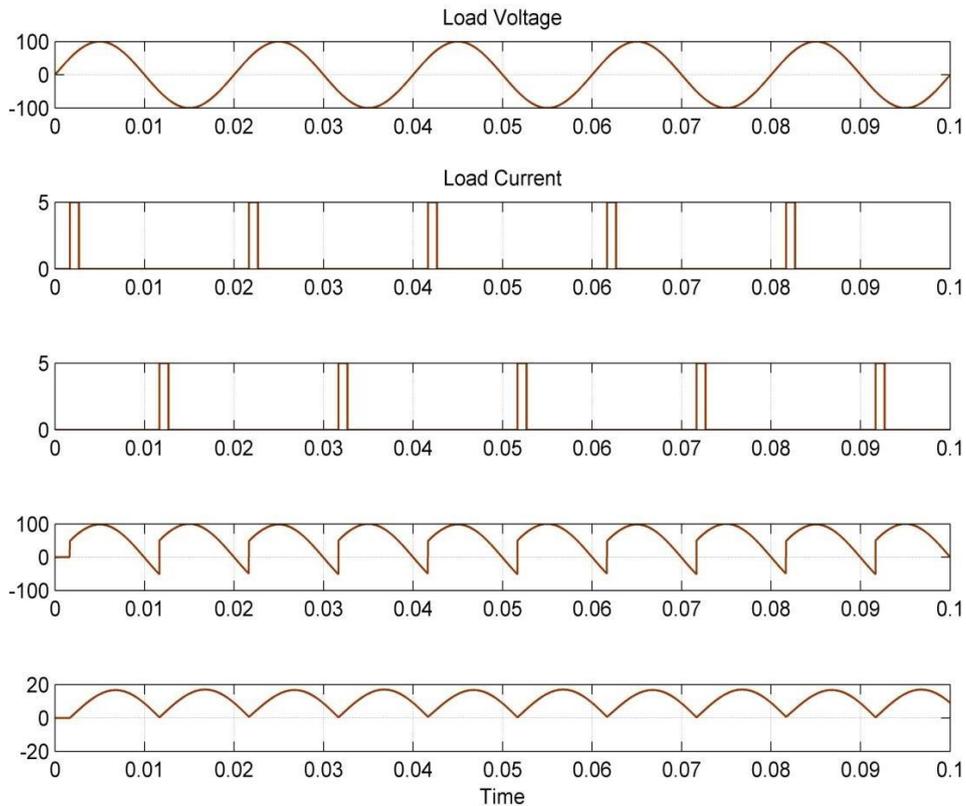
SINGLE PHASE FULLY CONTROLLED CONVERTER WITH R LOAD

SIMULATION DIAGRAM FOR R LOAD: -

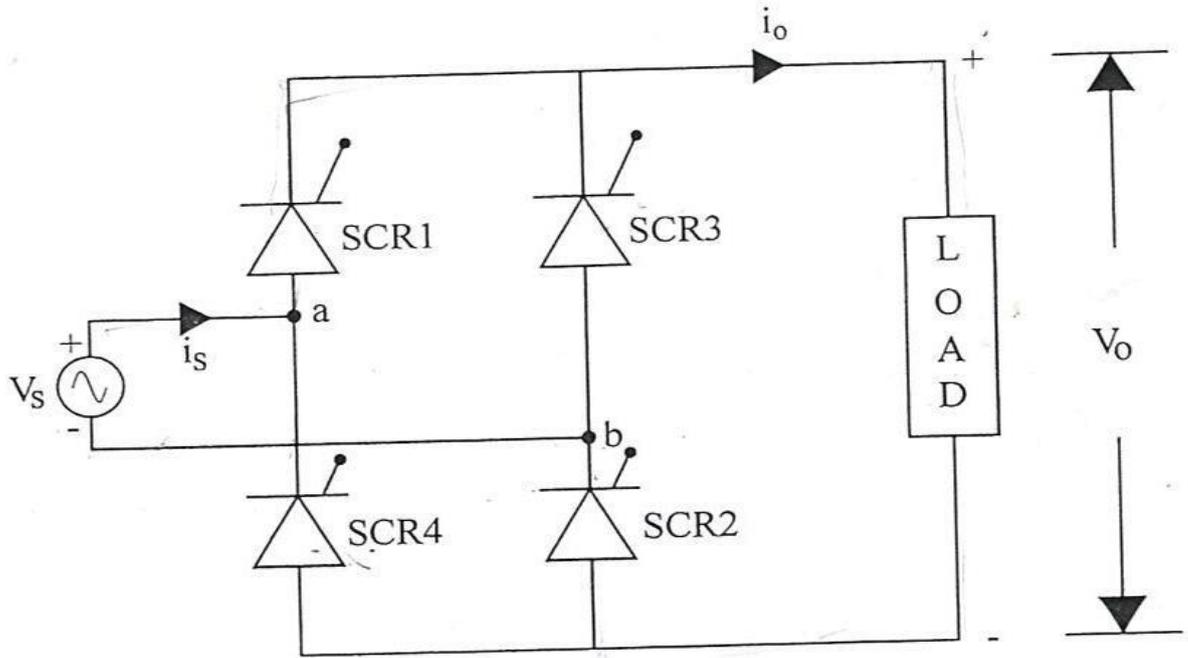
Single phase Bridge Contolled Rectifier



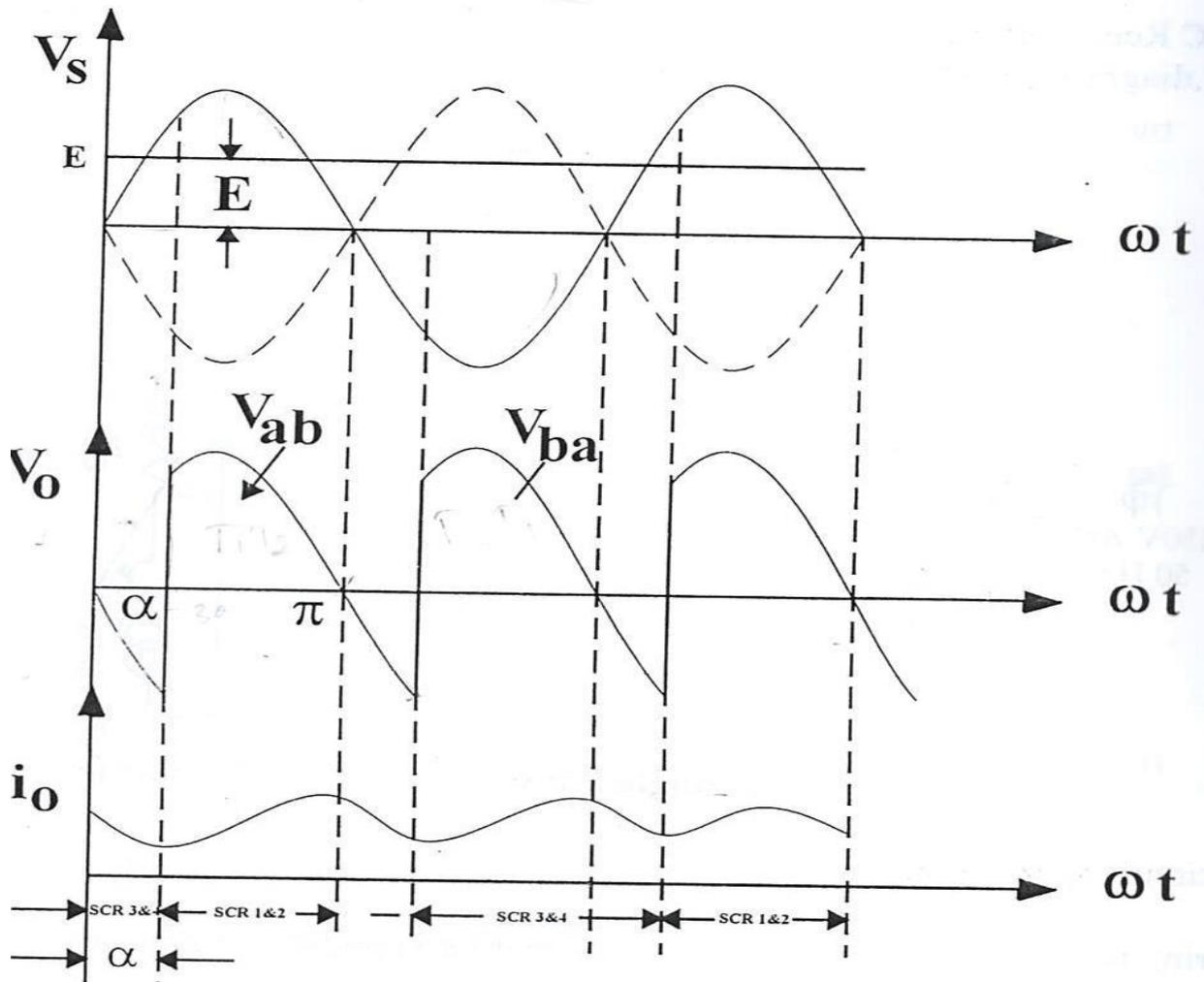
WAVEFORMS OF R LOAD



CIRCUIT DIAGRAM: -



MODEL WAVEFORMS: -



FORMULA USED FOR DESIGN OF PHASE DELAY: -

1. Firing angle delay for pulse generator 1 for T₁ & T₂ = $\frac{(0 + \alpha) * 0.02}{360}$ sec

2. Firing angle delay for pulse generator 2 for T₃ & T₄ = $\frac{(180 + \alpha) * 0.02}{360}$ 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., $\alpha = 30^\circ$

Firing angle delay for pulse generator 1 for T₁ & T₂ = $\frac{(0 + 30) * 0.02}{360}$ sec

= 0.001667 sec.

Firing angle delay for pulse generator 2 for T₃ & T₄ = $\frac{(180 + 30) * 0.02}{360}$ sec

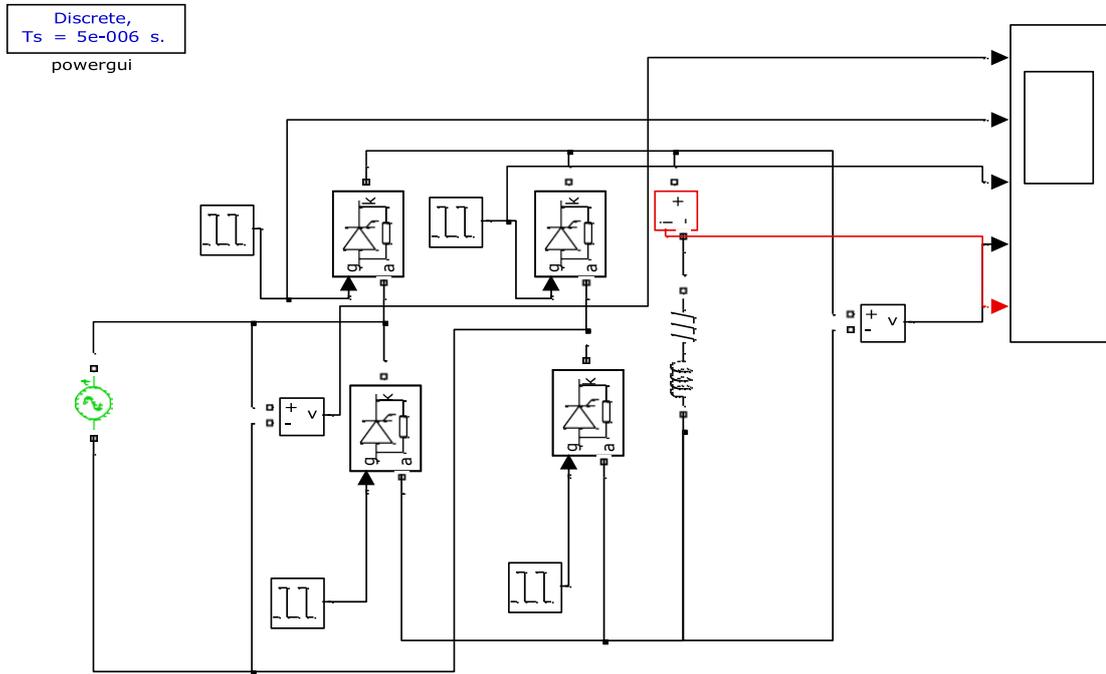
= 0.011667 sec.

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

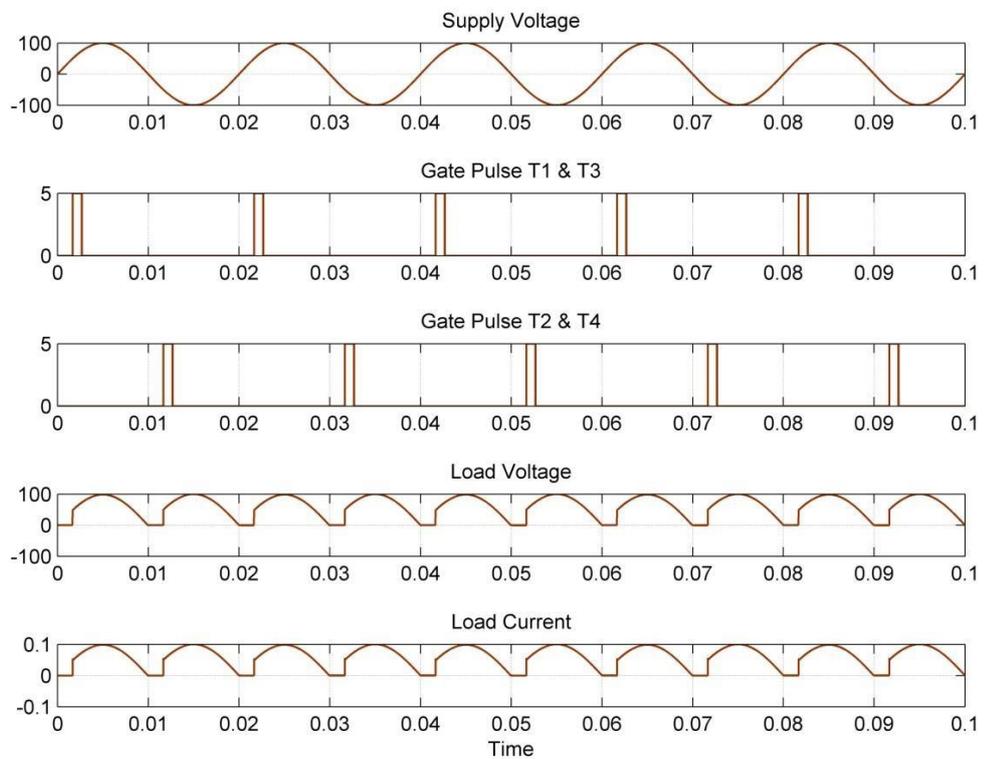
FIRING ANGLE (α) in deg	Phase angle delay for (in sec.)	
	Pulse Generator 1	Pulse Generator 2
0	0	0.01
30	0.001667	0.011667
60	0.00333	0.01333
90	0.005	0.015
120	0.00667	0.01667
150	0.00833	0.01833
180	0.01	0.02

SIMULATION DIAGRAM FOR RL LOAD: -

Single phase Bridge Contolled Rectifier with RL Load



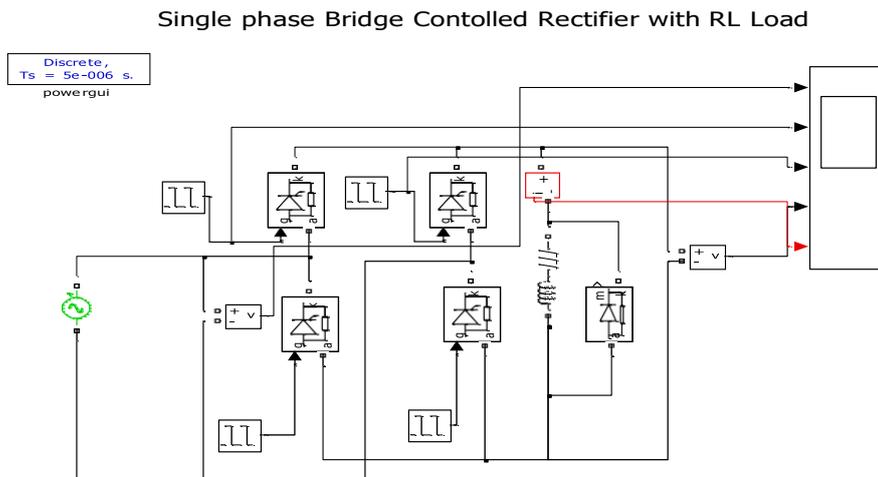
MODEL WAVEFORMS: -



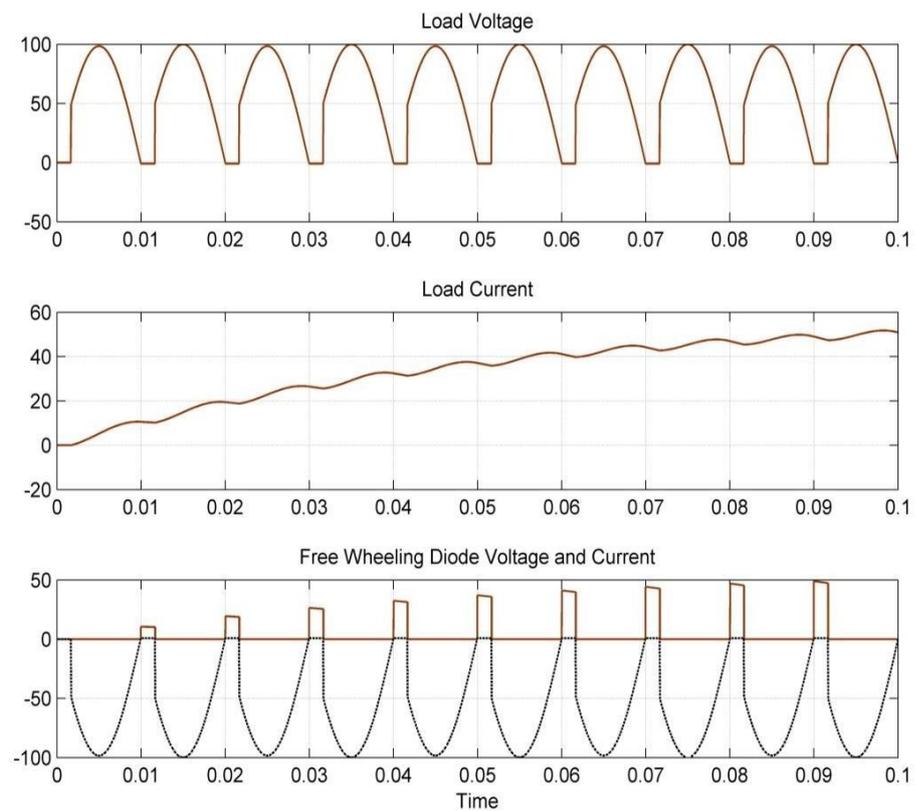
PROCEDURE FOR RL LOAD: -

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:-

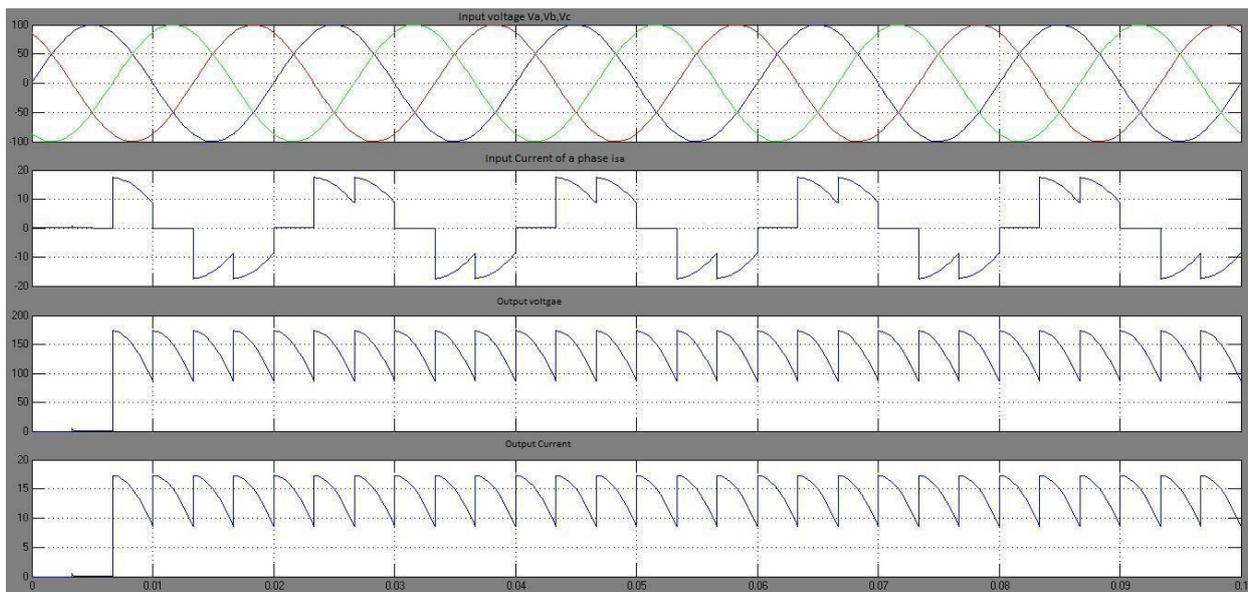
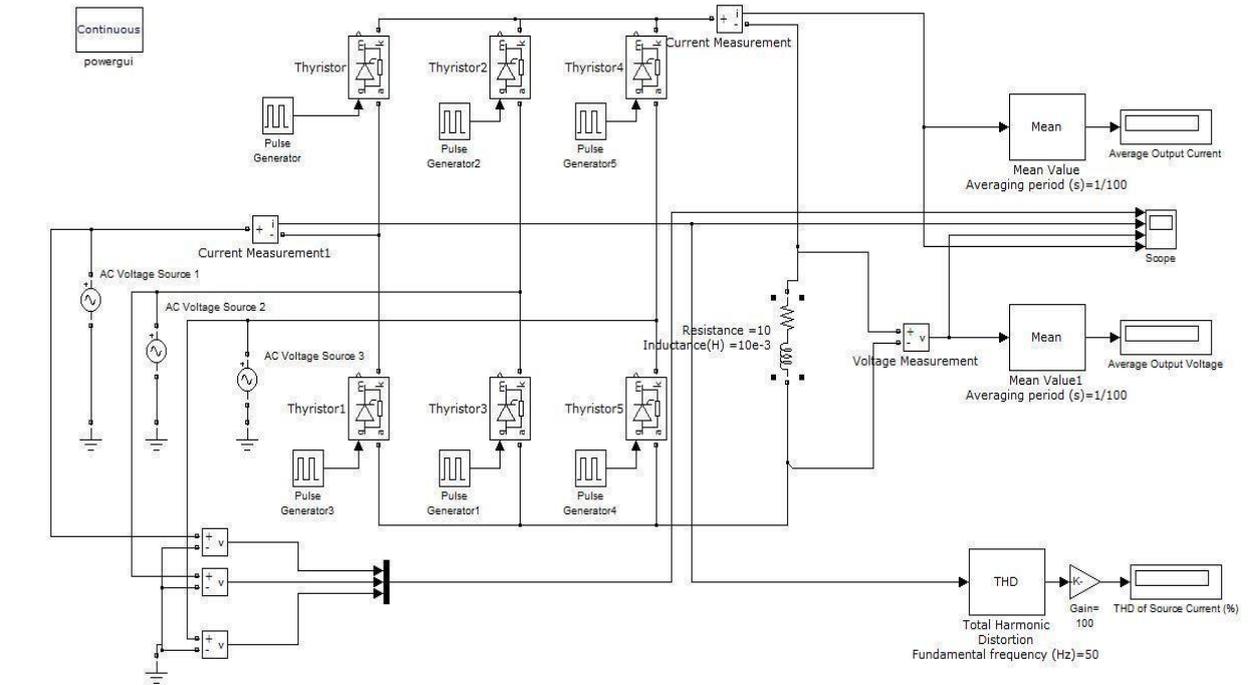


RESULT:-

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

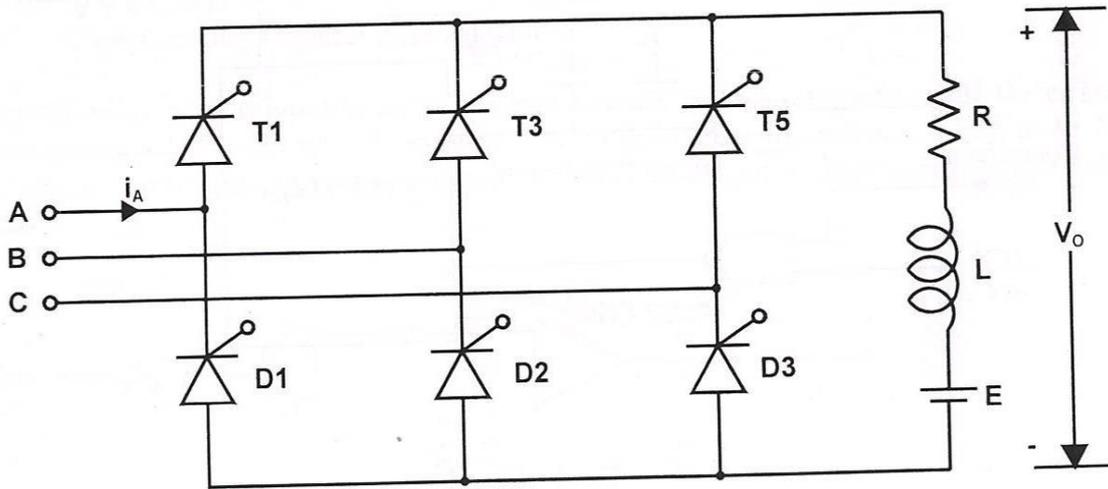
SOFTWARE REQUIRED: -

MATLAB 6.1 or above.

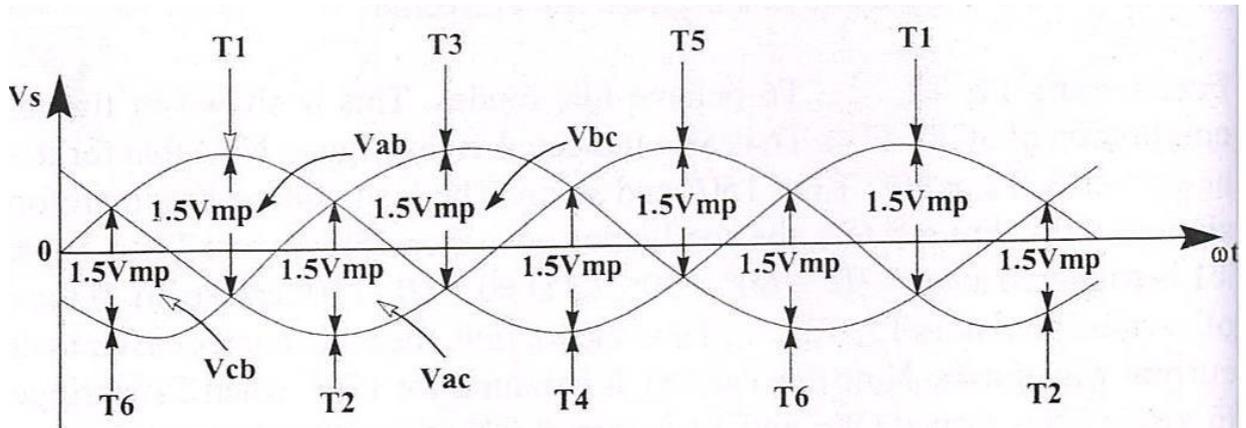
PROCEDURE FOR R LOAD: -

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.

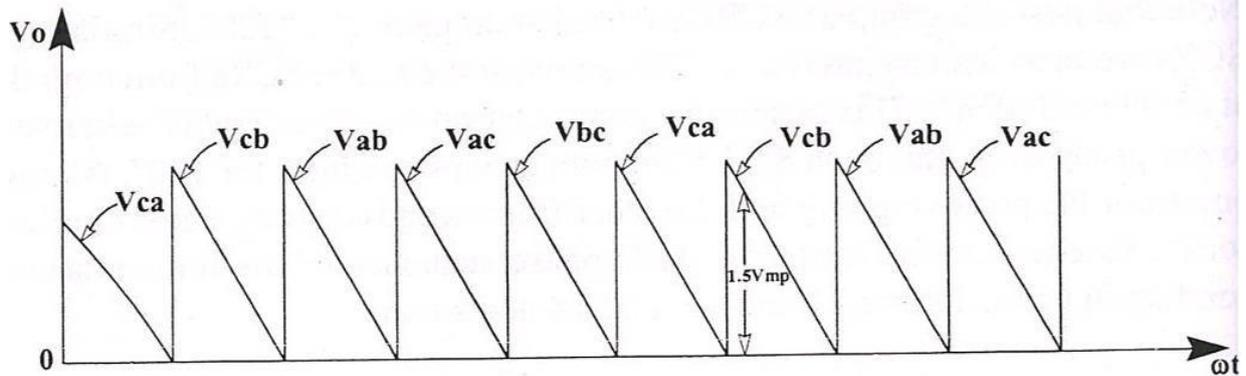
CIRCUIT DIAGRAM: -



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., $\alpha = 30^\circ$

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

$$\begin{aligned} \text{Firing angle delay for pulse generator 2 for } T_2 &= \frac{(30 + 120 + 30) * 0.02}{360} \text{ sec} \\ &= 0.01 \text{ sec.} \end{aligned}$$

$$\begin{aligned} \text{Firing angle delay for pulse generator 2 for } T_2 &= \frac{(30 + 240 + 30) * 0.02}{360} \text{ sec} \\ &= 0.01667 \text{ sec.} \end{aligned}$$

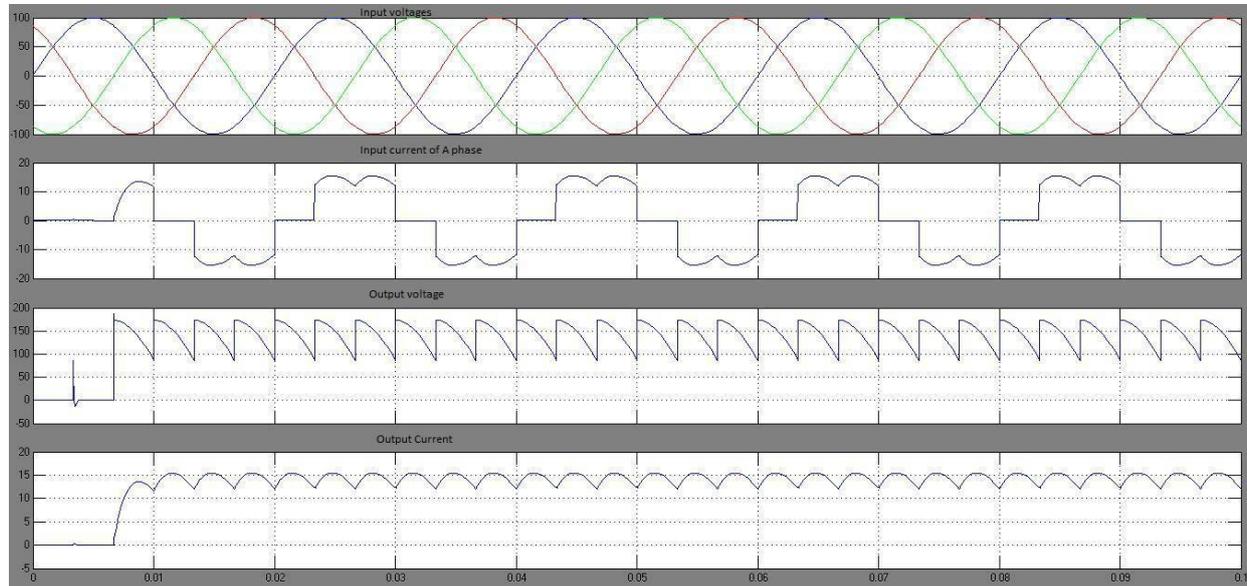
Firing angle delays in sec for pulse generators 1, 2 & 3 for various firing angles can be calculated and tabulated as follows.

FIRING ANGLE (α) in deg	Phase angle delay for (in sec.)		
	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.

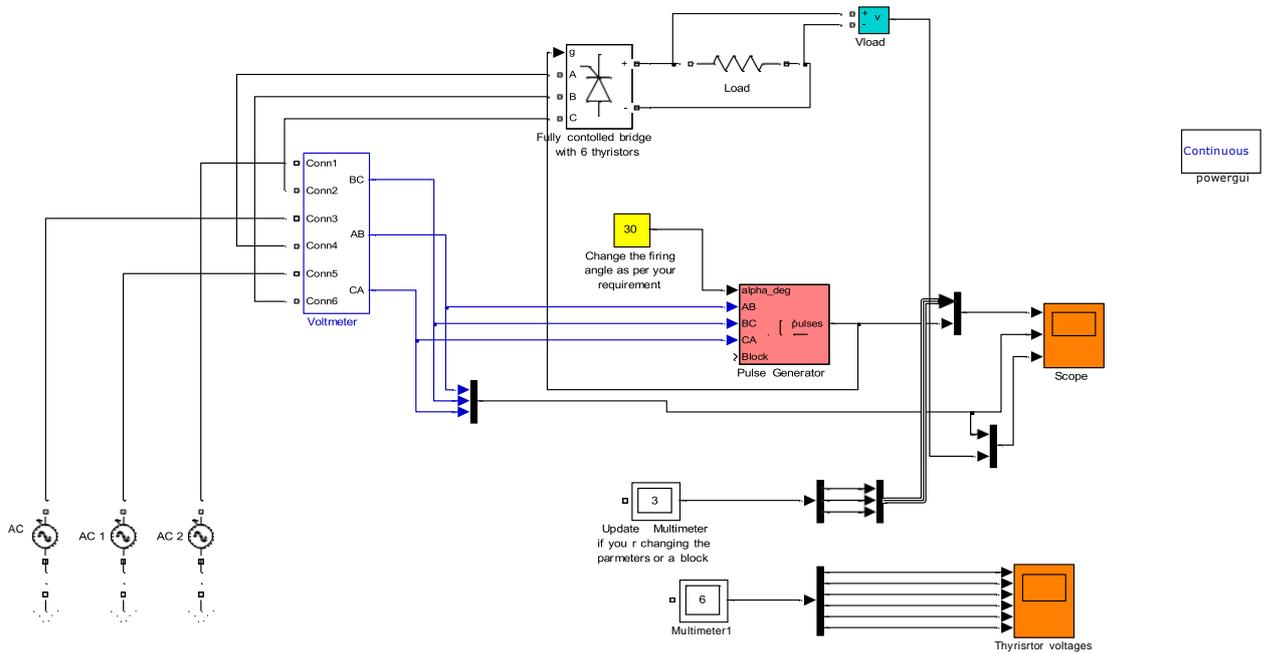
13. Set the pulse generator parameters such as pulse type, amplitude, period, pulse width, phase delay, etc. as per circuit design formula.
14. Set the scope axis as per the circuit requirements like input voltage, output voltage, output current, etc.
15. Set the load resistance and inductance values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

RESULT: -

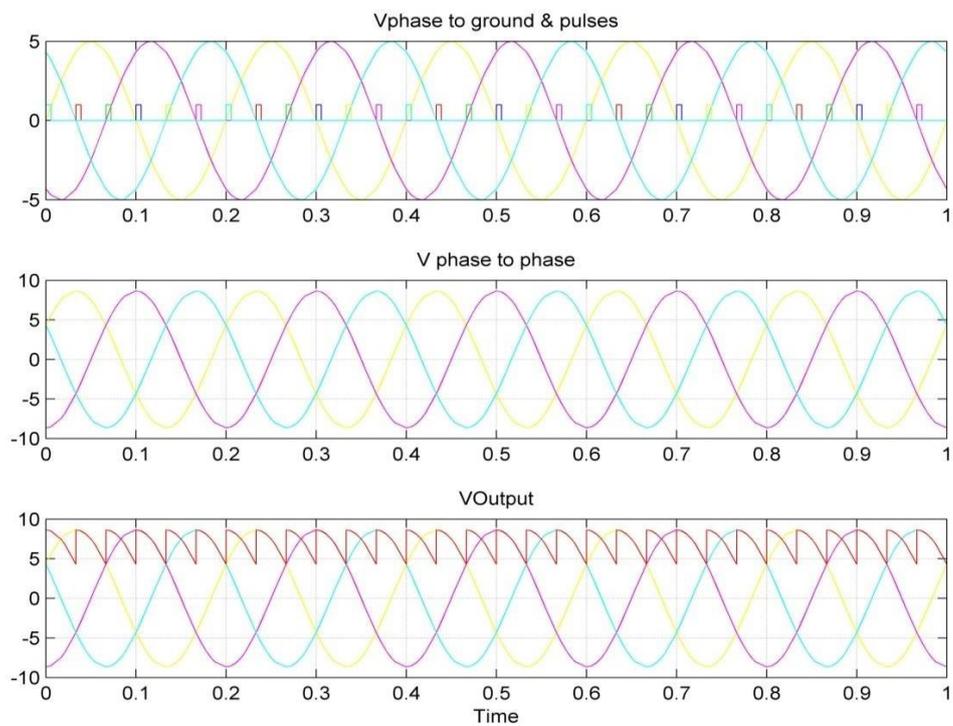
Thus the three phase half controlled converter was simulated with R and RL load of resistance of 10 ohms and inductance of 1 mH for the firing angles of 30° , 60° , 90° and 120° and the related waveforms were obtained.

THREE PHASE FULLY CONTROLLED CONVERTER WITH R AND RL LOAD

SIMULATION DIAGRAM FOR R LOAD: -



MODEL WAVEFORM:



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

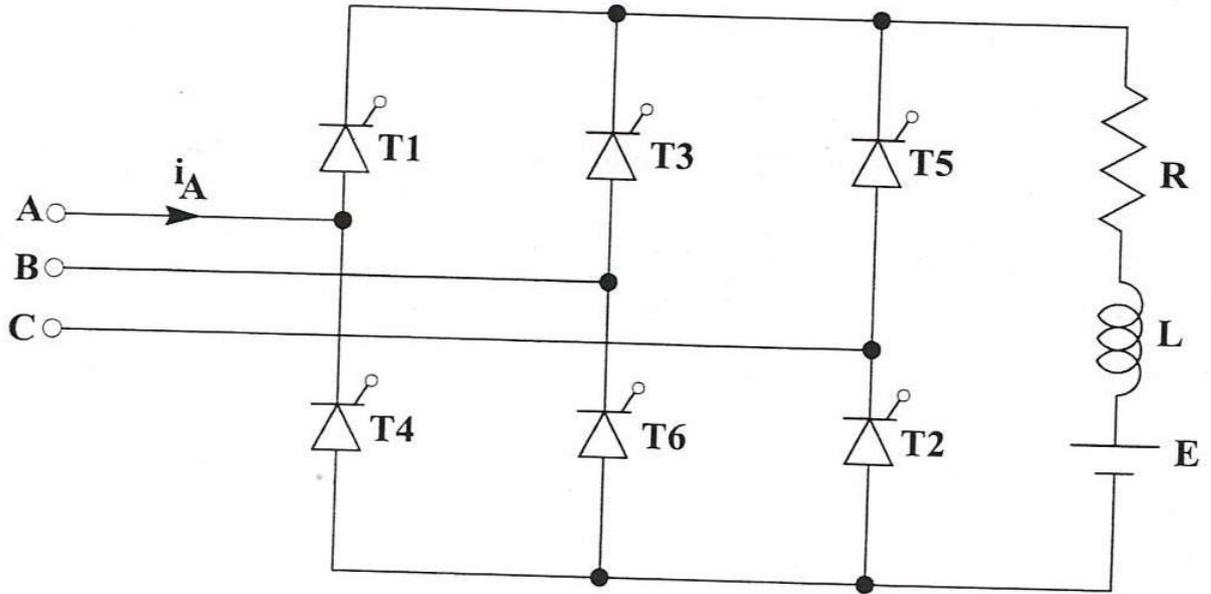
SOFTWARE REQUIRED: -

MATLAB 6.1 or above.

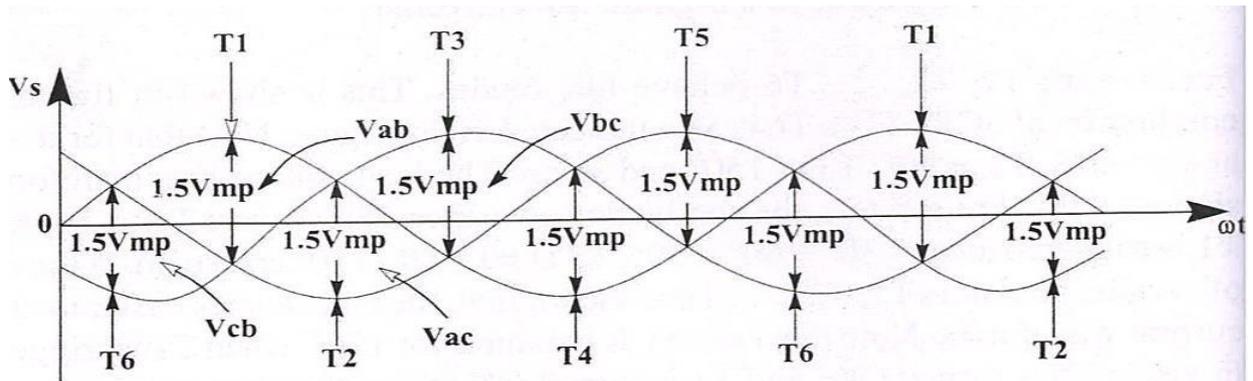
PROCEDURE FOR R LOAD: -

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 T₁, T₂, T₃, T₄, T₅ & T₆ 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.

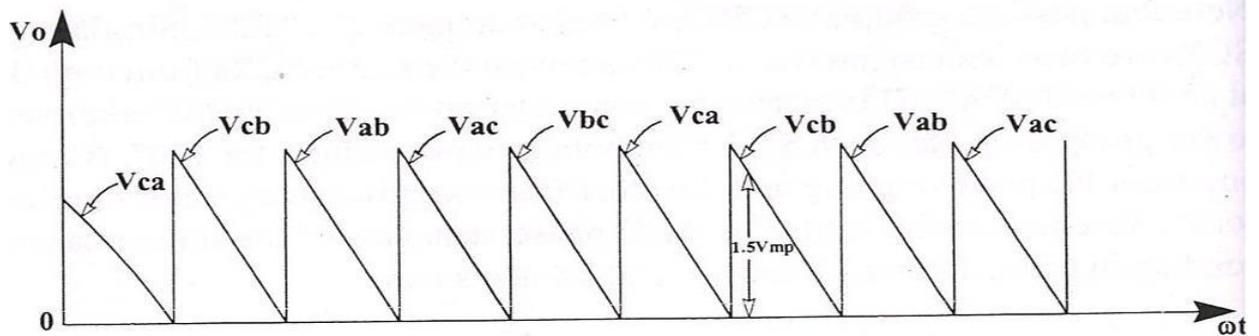
CIRCUIT DIAGRAM: -



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

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

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

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

$$= 0.00667 \text{ sec.}$$

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

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

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

$$= 0.01333$$

$$\text{Firing angle delay for pulse generator 5 for } T_5 = \frac{(30 + 240 + 30) * 0.02}{360}$$

$$= 0.01667$$

Firing angle delay for pulse generator 6 for $T_6 = \frac{(30 + 300 + 30) * 0.02}{360}$
 $= 0.02$

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

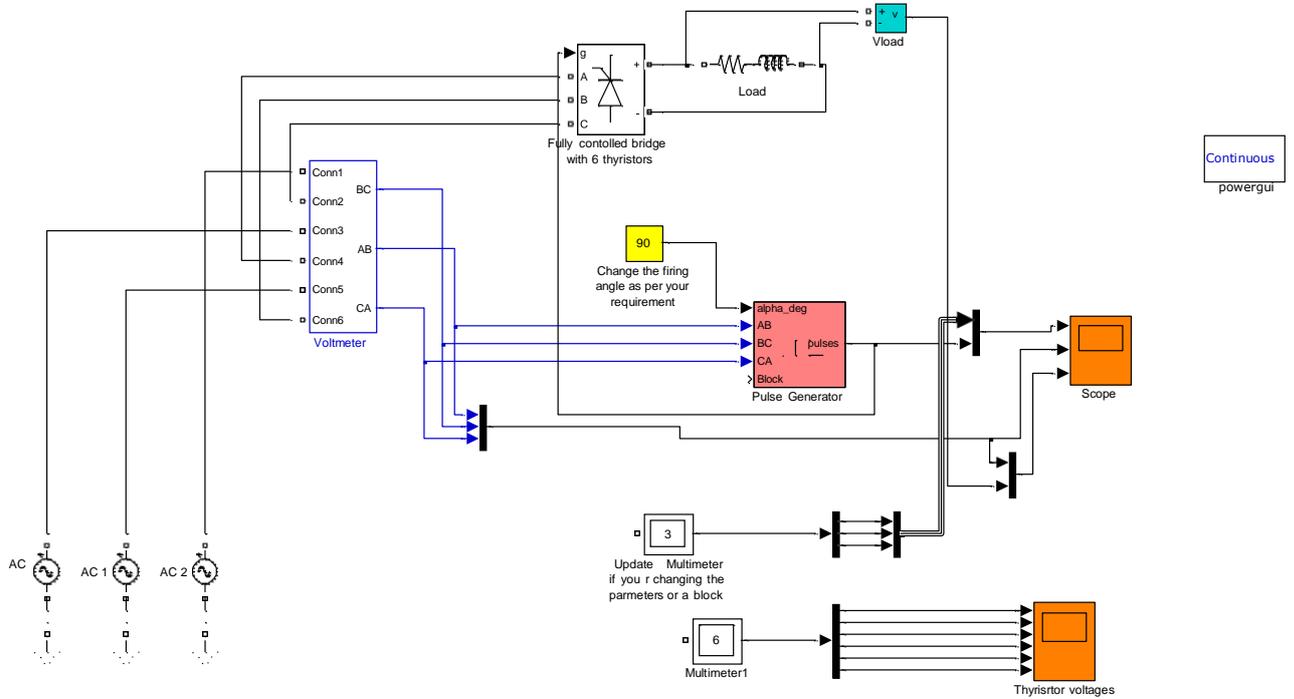
FIRING ANGLE (α) in deg	Phase angle delay for (in sec.)					
	PG 1	PG 2	PG 3	PG 4	PG 5	PG 6
0	0.00167	0.005	0.00833	0.01167	0.015	0.01833
30	0.00333	0.00667	0.01	0.01333	0.01667	0.02
60	0.005	0.00833	0.01167	0.015	0.01833	0.02167
90	0.00667	0.01	0.01333	0.01667	0.02	0.02333
120	0.00833	0.01167	0.015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

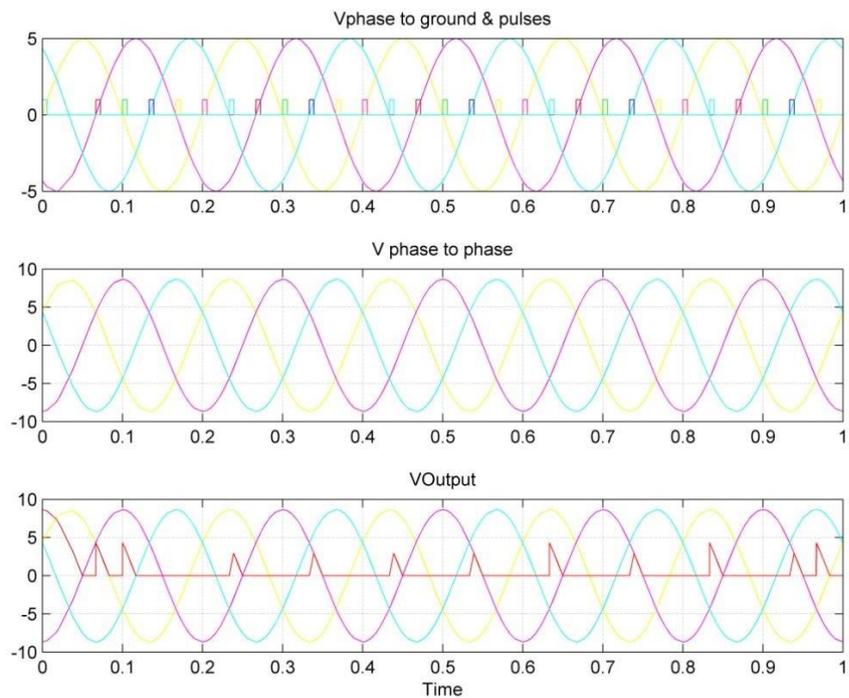
Thus the three phase half controlled converter was simulated with R and RL load of resistance of 10 ohms and inductance of 1 mH for the firing angles of 30°, 60°, 90° and 120° and the related waveforms were obtained.

THREE PHASE FULLY CONTROLLED CONVERTER WITH R AND RL LOAD

SIMULATION DIAGRAM FOR RL LOAD: -



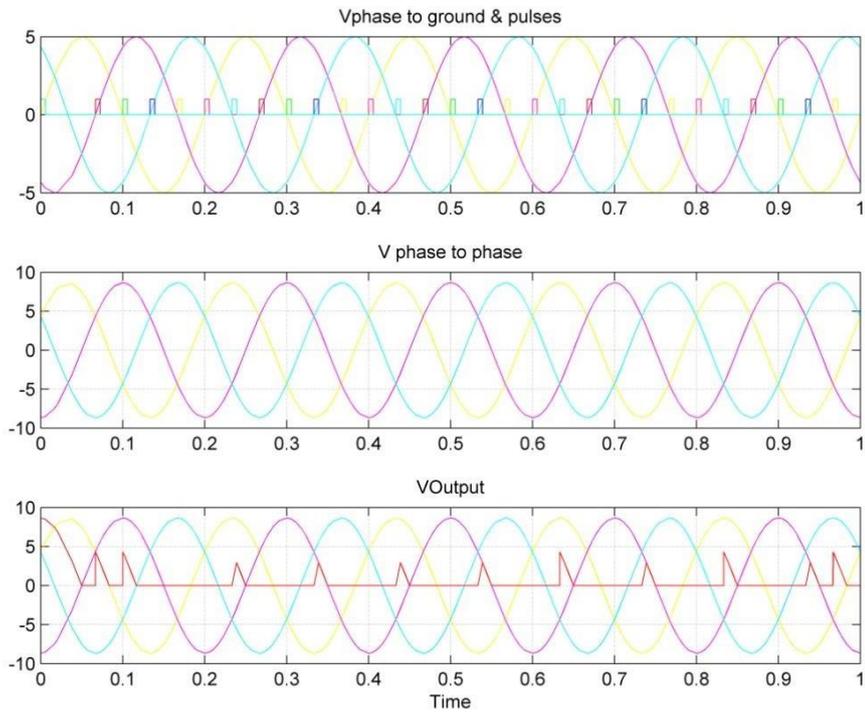
MODEL GRAPH:



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: -

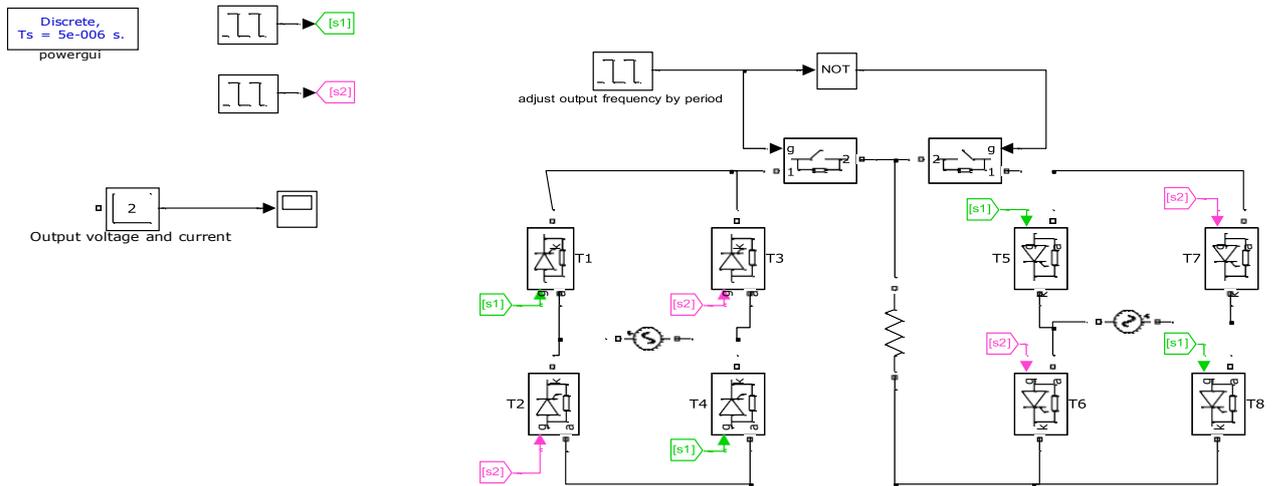
MATLAB 6.1 or above.

PROCEDURE FOR R LOAD: -

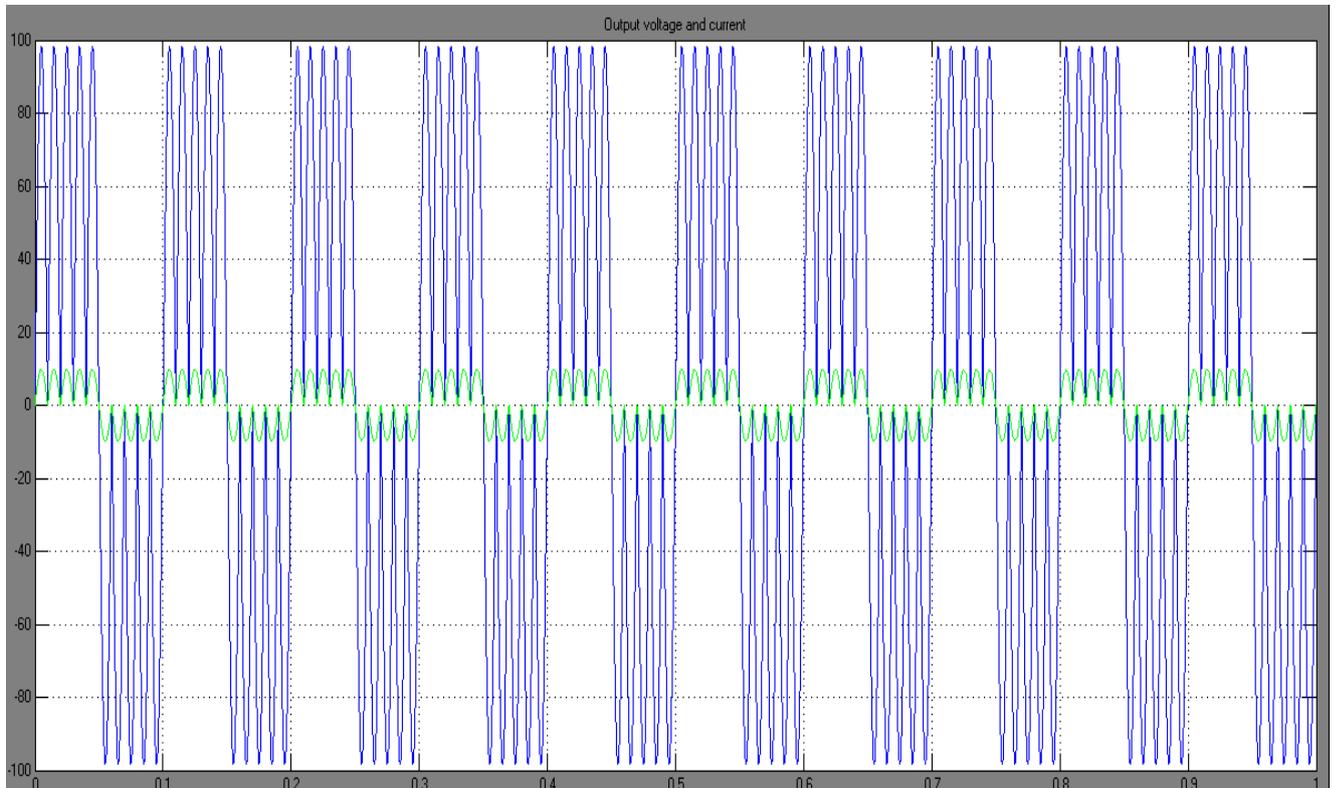
1. Study the circuit diagram, operation and model waveforms of a single phase AC Voltage controller.
2. Open the MATLAB software by double-clicking the MATLAB icon on the desktop and click the shortcut "Simulink" to open the "Simulink Library Browser".
3. Create a new model and click on "SimPowerSystems" 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., $\alpha = 30^\circ$

Firing angle delay for pulse generator 1 = $\frac{(0 + 30) * 0.02}{360}$ sec

= 0.001667 sec.

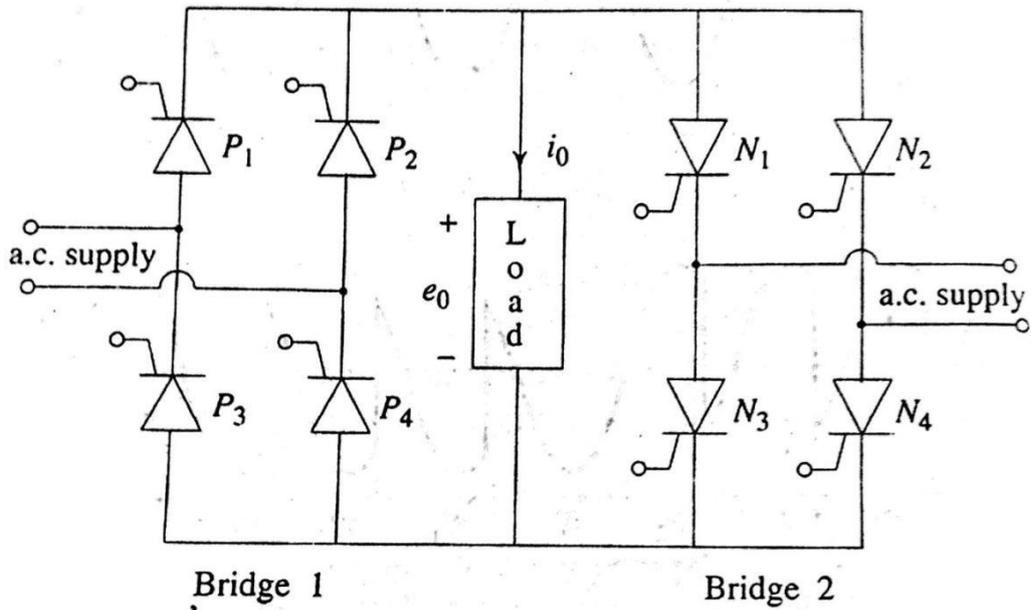
Firing angle delay for pulse generator 2 = $\frac{(180 + 30) * 0.02}{360}$ sec

= 0.011667 sec.

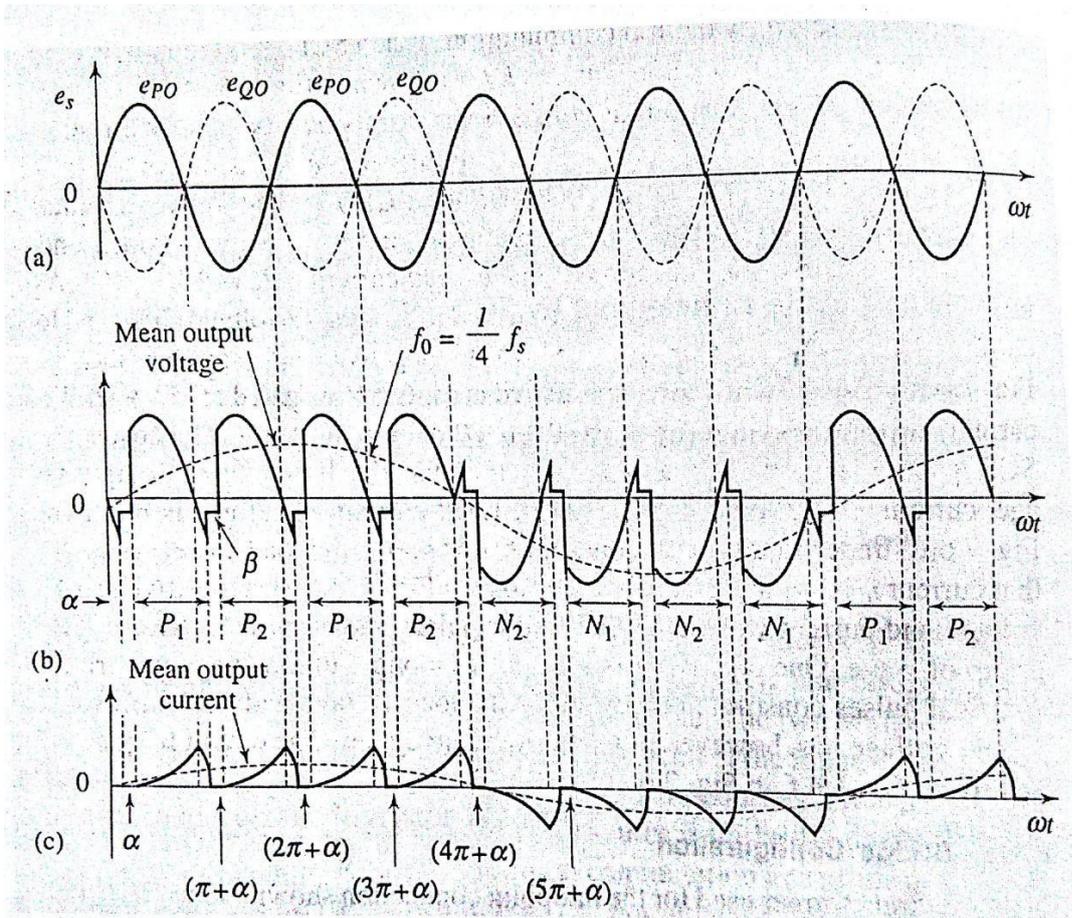
Firing angle delays in sec for pulse generators 1 & 2 for various firing angles can be calculated and tabulated as follows.

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: -

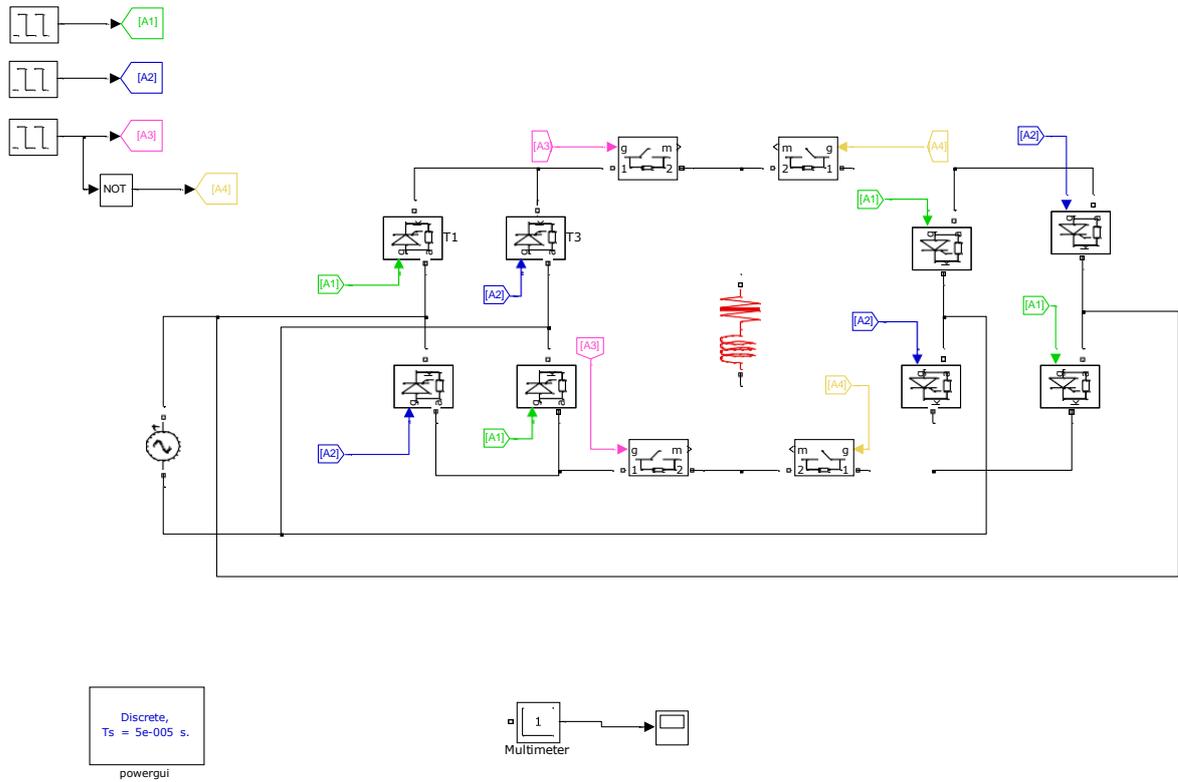


MODEL WAVEFORMS: -

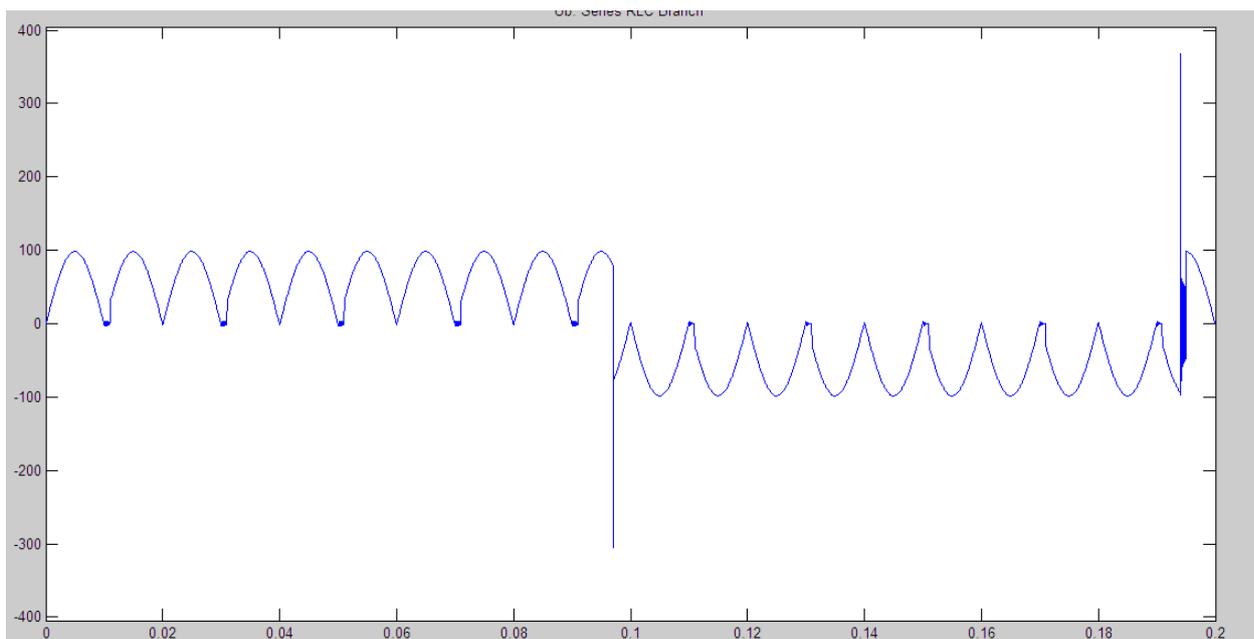


SINGLE PHASE AC VOLTAGE CONVERTER WITH R AND RL LOAD

SIMULATION DIAGRAM FOR RL LOAD: -



GRAPH:



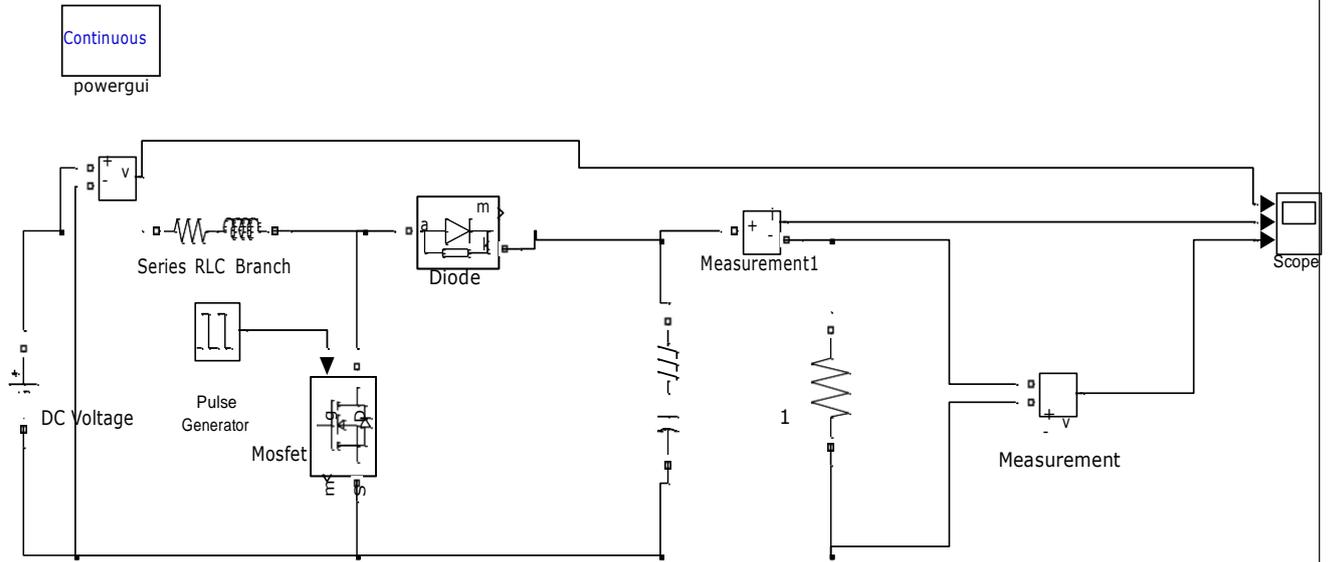
PROCEDURE FOR RL LOAD: -

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.

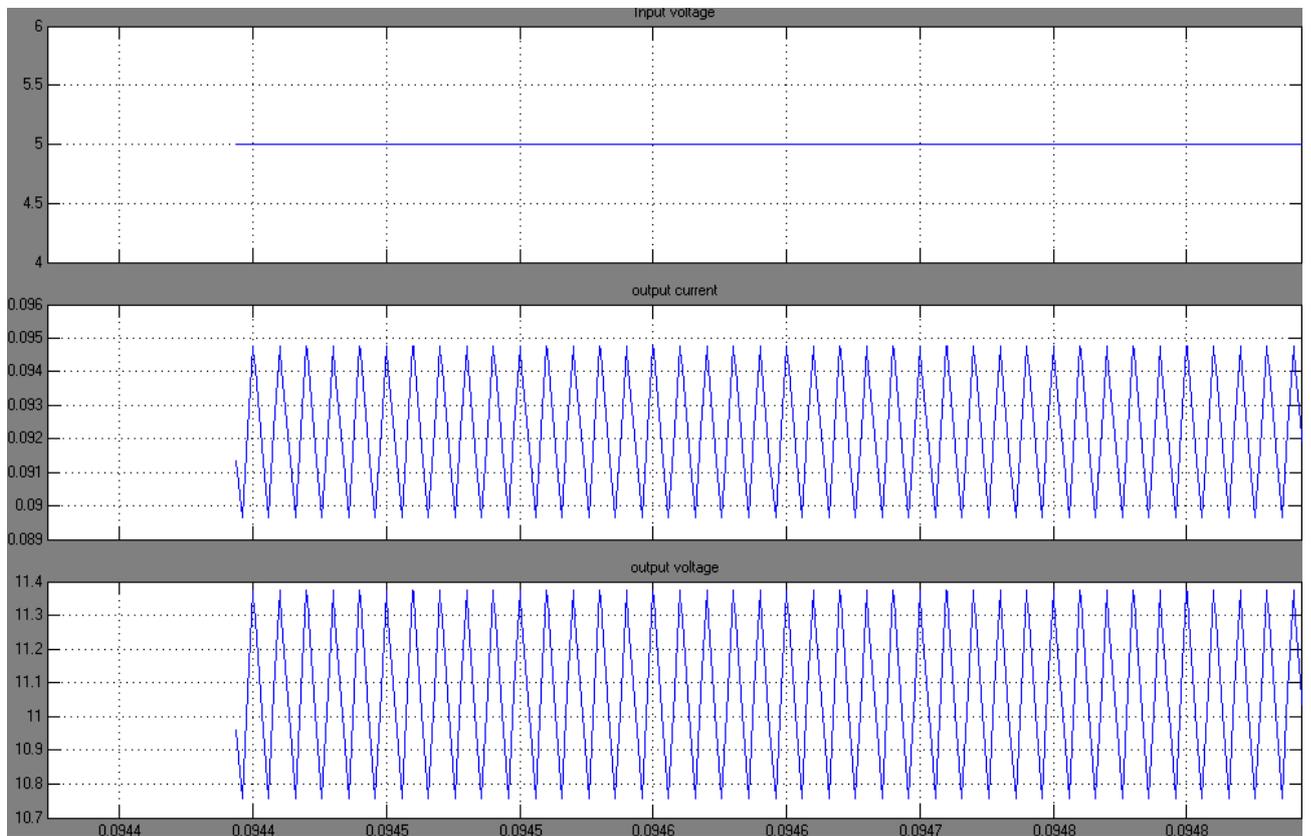
RESULT: -

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:



Expt. No.:16

Date:

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

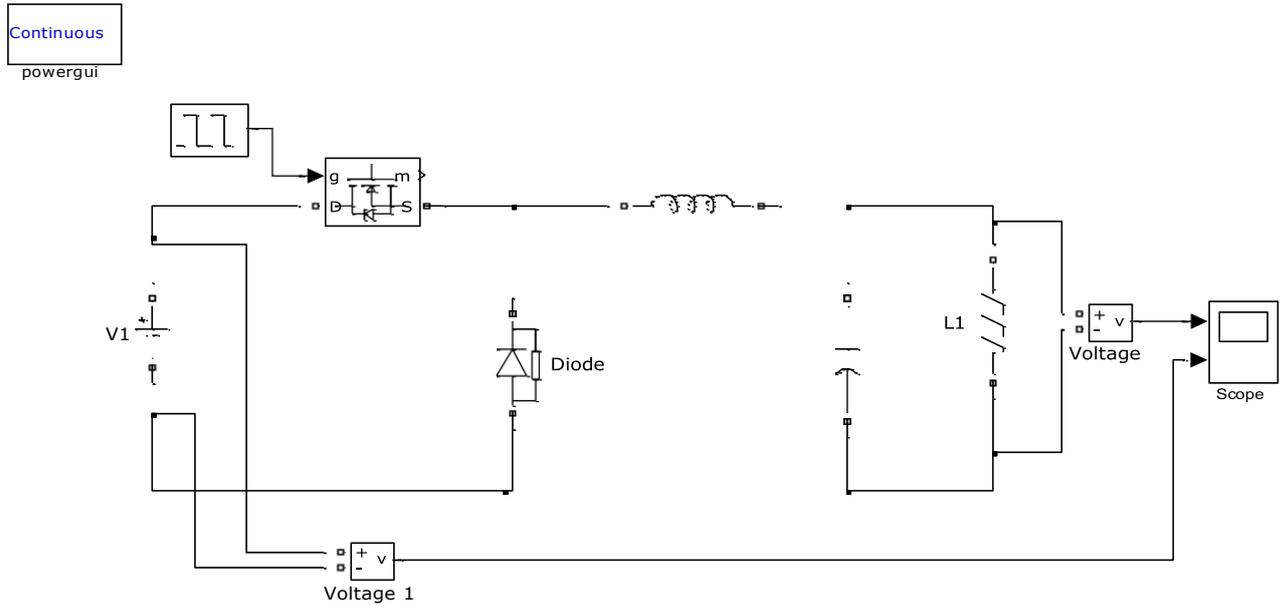
SOFTWARE REQUIRED: -

MATLAB 6.1 or above.

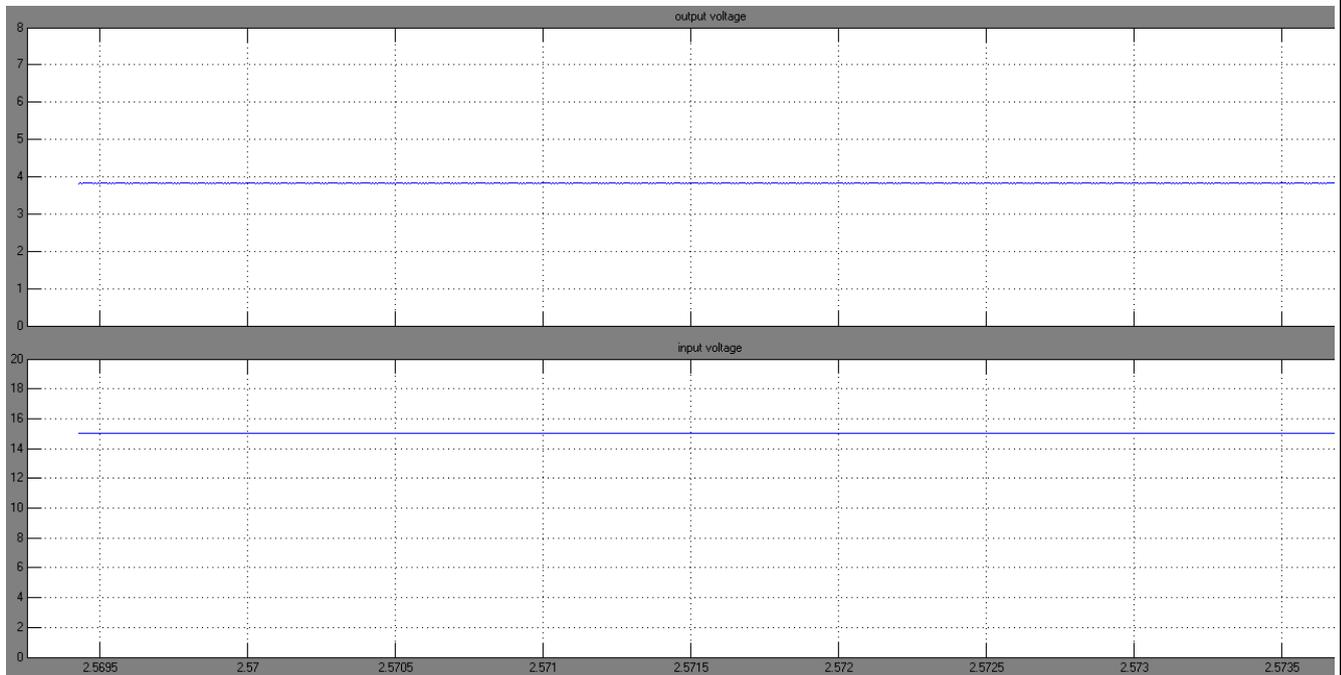
PROCEDURE FOR R LOAD: -

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:



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., $\alpha = 30^\circ$

Firing angle delay for pulse generator 1 = $\frac{(0 + 30) * 0.02}{360}$ sec

= 0.001667 sec.

Firing angle delay for pulse generator 2 = $\frac{(180 + 30) * 0.02}{360}$ sec

= 0.011667 sec.

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.