POWER ELECTRONICS LAB MANUAL

Prepared By:
MUHAMMEDALI SHAFEEQUE K
Al Ameen Engineering College

EXPERIMENT 1

SINGLE PHASE FULLY CONTROLLED SCR BRIDGE CIRCUIT

AIM:

   i.   To study operation of single phase fully controlled rectifier for R load
   ii. To study operation of single phase fully controlled rectifier for RL load with and without freewheeling diode

PRINCIPLE:

  Circuit diagram of single phase fully controlled rectifier consists of four SCRs. During the positive half cycle, SCRs T1 and T2 are forward biased and can be fired for conduction applying a positive voltage to their gate with respect to cathode. At $\omega t = \alpha$, SCRs T1 and T2 are triggered, then the current flows through the P– T1- R load – T2 – N. At $\omega t = \pi$, supply voltage falls to zero and the current also goes to zero. Hence SCRs T1 and T2 turned off. During negative half cycle (π to 2π), SCRs T3 and T4 forward biased and can be fired for conduction applying a positive voltage to their gate with respect to cathode. At $\omega t = \pi + \alpha$, SCRs T3 and T4 are triggered, then current flows through the path N – T3 – R load- T4 – P. At $\omega t = 2\pi$, supply voltage and current goes to zero, SCRs T3 and T4 are turned off. The Fig-3, shows the current and voltage waveforms for this circuit.

PROCEDURE:

   1. Complete the connections as per the circuit diagram.
   2. Keep the firing angle controlling knob at 0^0 position.
   3. Keep switch S1 in internal position for using internal control voltage of gate trigger circuit.
   4. Keep switch S2 and S3 in on position.
   5. Turn on the power switch
   6. Observe the input voltage waveform using CRO
   7. Observe the output voltage waveform, SCR voltage and gate signals for various firing angle
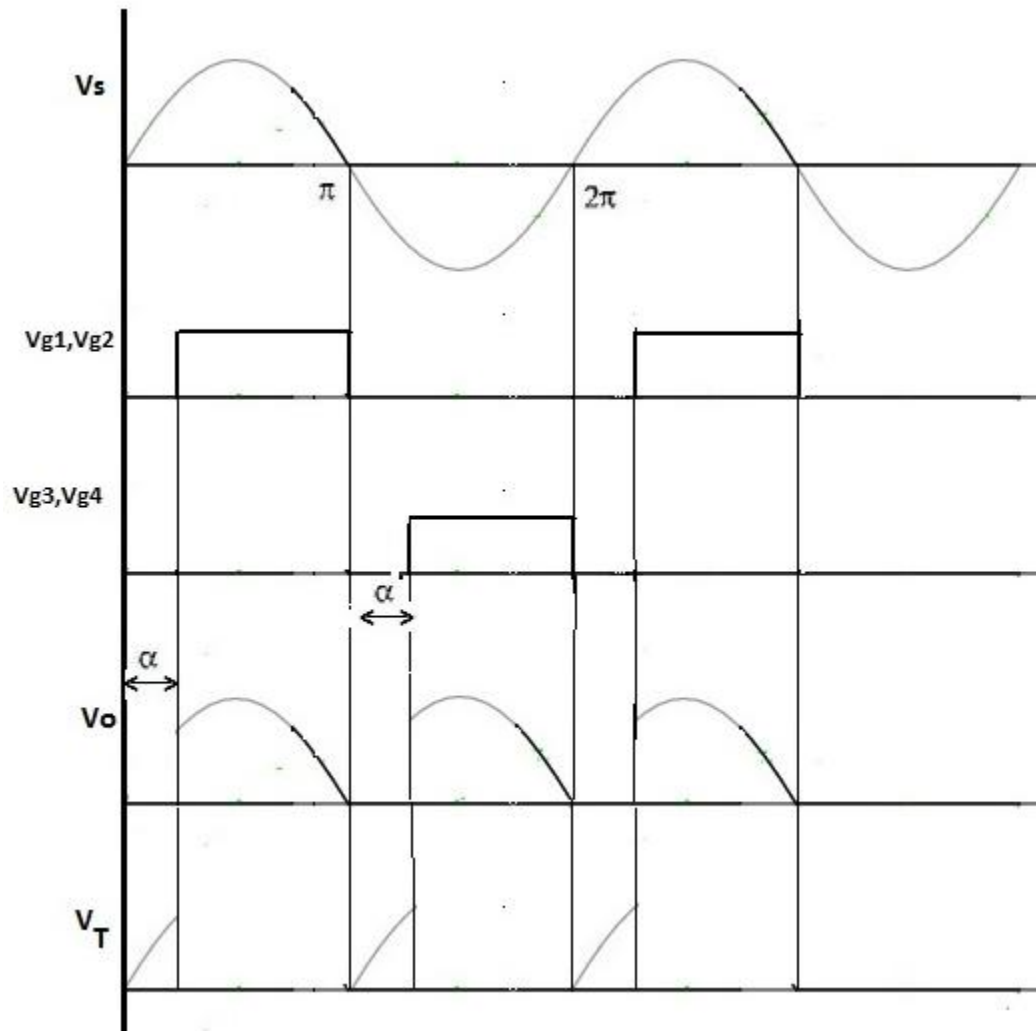
RESULT:

        Studied operation of single phase fully controlled SCR rectifier circuit for both R and RL load.

CIRCUIT DIAGRAM:
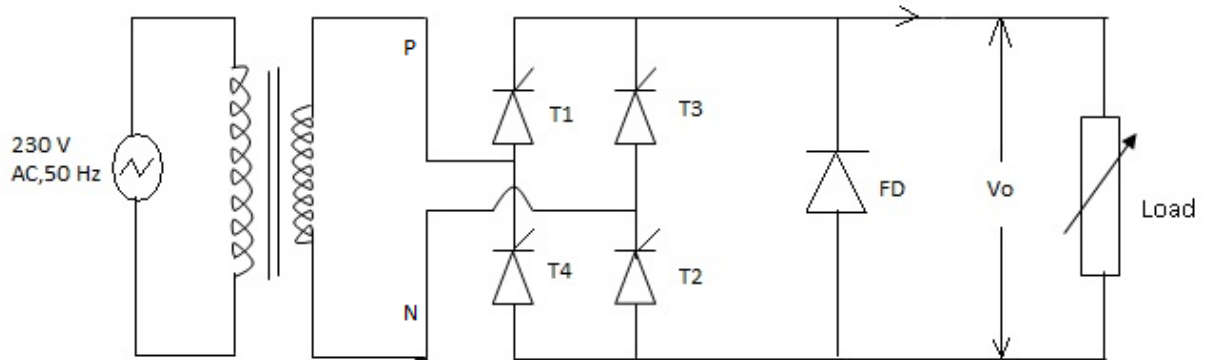
WAVEFORMS:

<div align="center">

EXPERIMENT 2

R AND RC FIRING CIRCUIT FOR SCR

</div>

AIM:

To study operation of R and RC firing circuit for SCR.

THEORY:

RESISTANCE FIRING CIRCUIT:

Resistance trigger circuits are the simplest & most economical method. During the positive half cycle of the input voltage, SCR become forward biased but it will not conduct until its gate current exceeds gate threshold, V_{gt} . Diode D allows the flow of current during positive half cycle only. R1 is the variable resistance and is used to limit the current through the circuit (I_g) and to vary the gate voltage and thus the firing angle. During the positive half cycle current I_g flows. Ig increases and when $V_g=V_{gt}= I_{gmin}R$ the SCR turns ON .The firing angle can be varied from 0 — 90° by varying the resistance R1.

RC FIRING CIRCUIT:

By varying the variable resistance R1, the firing angle can be varied. Capacitor, C charges and discharges with a phase shift. When the capacitor voltage equal to the gate threshold voltage V_{gt},SCR get triggered. Diode D prevents the negative voltage to gate

PROCEDURE:

RESISTANCE FIRING CIRCUIT:

1. Complete the connections as per the circuit diagram.
2. Keep the potentiometer R1 in minimum position (maximum resistance in the circuit)
3. Turn on the power switch
4. Observe the output voltage waveform, SCR voltage and gate signals for various firing angle
5. Observe the range of firing angle control.

RC FIRING CIRCUIT:
1. Complete the connections as per the circuit diagram.
2. Keep the potentiometer R1 in minimum position (maximum resistance in the circuit)
3. Turn on the power switch
4. Observe the output voltage waveform, SCR voltage and gate signals for various firing angle
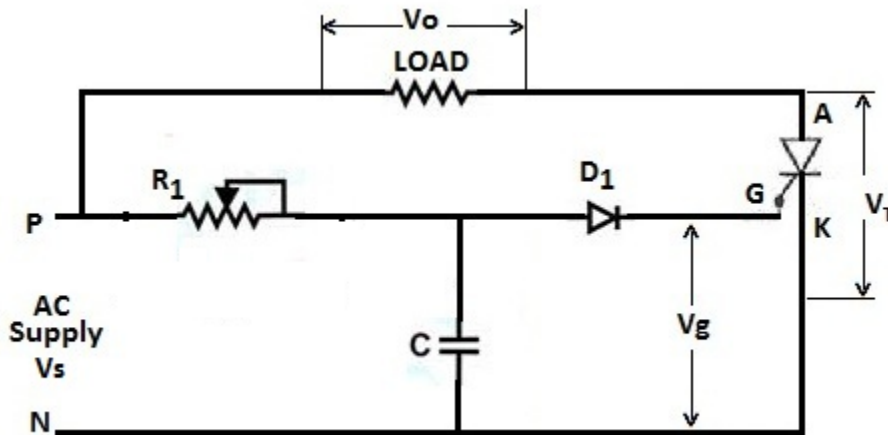5. Observe the range of firing angle control.

RESULT:

Studied the operation of R and RC firing circuit for SCR for different firing angles.
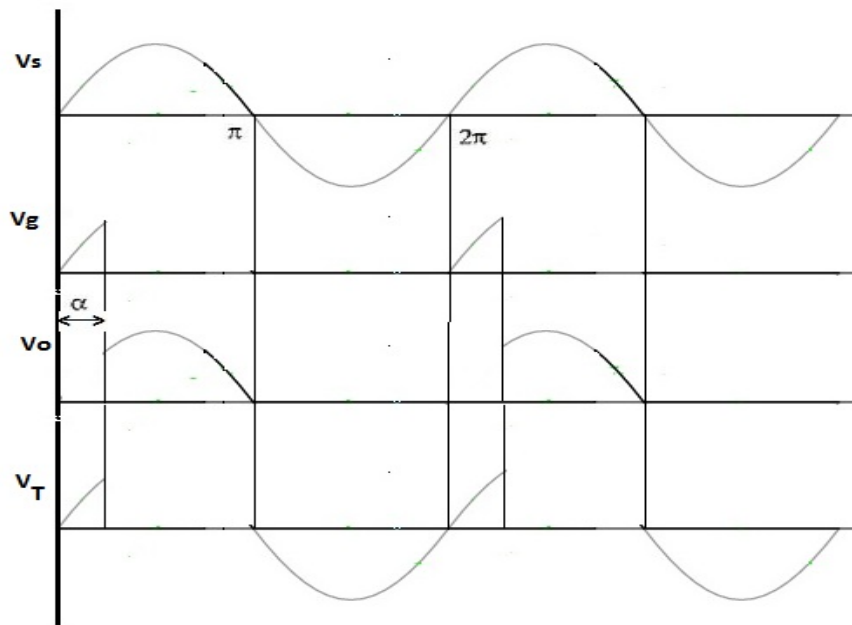
CIRCUIT DIAGRAM:

RESISTANCE FIRING CIRCUIT:
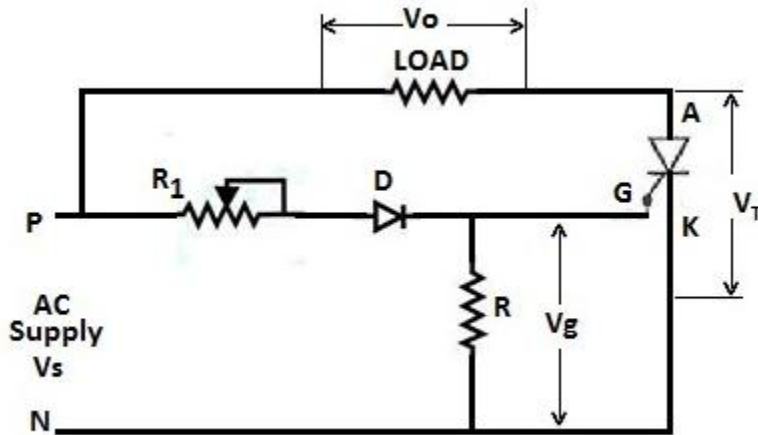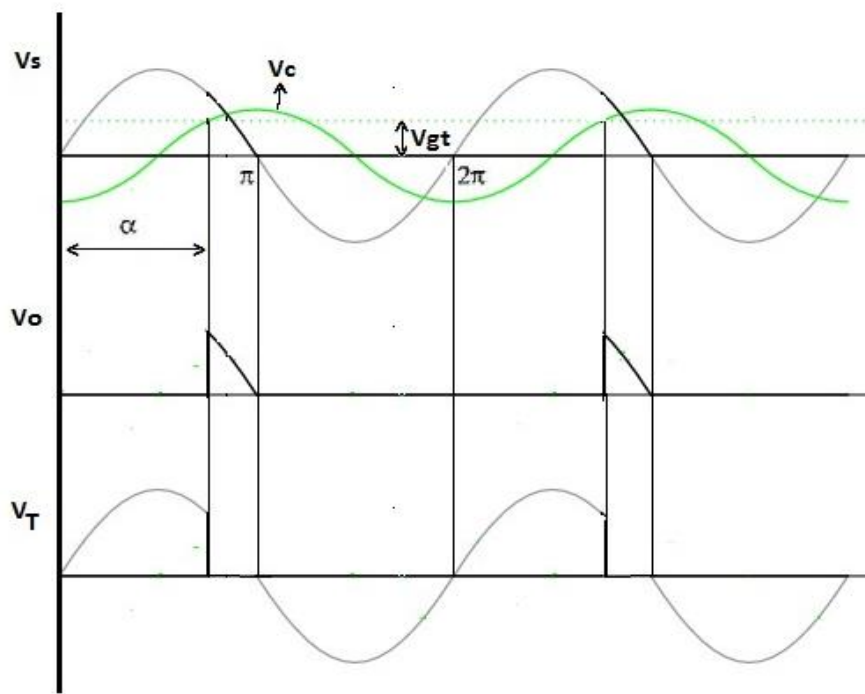

RC FIRING CIRCUIT:

WAVEFORMS:
RESISTANCE FIRING CIRCUIT:

RC FIRING CIRCUIT:

EXPERIMENT 3

UJT TRIGGER CIRCUIT FOR SCR

AIM:

 To study the operation of UJT firing circuit for SCR.

THEORY:

 A synchronized UJT triggered circuit using an UJT is shown in the figure. Diode Rectifier converts input ac to dc. Resistor R limits the circuit current to a suitable value for the zener diode and UJT. Zener diode 'Z' functions to clip the rectified voltage to a standard level. The zener voltage V_2 is applied to the RC charging circuit. Charging of capacitor C is at a rate determined by R1 and R3.When voltage across the capacitor,V_3 reaches the unijunction threshold voltage ηV_2, the UJT junction breaks down and the capacitor C discharges through the primary of pulse transformer. As the current is in the form of pulse, windings of the pulse transformer have pulse voltages at their secondary terminals and can turn on the SCR.As the charging rate of capacitor varies by varying R3, firing angle can be controlled by varying R3.Firing angle can be controlled in a range of 0^0 to 180^0 .

PROCEDURE:

1. Complete the connections as per the circuit diagram.
2. Keep the potentiometer R1 in minimum position (maximum resistance in the circuit)
3. Turn on the power switch
4. Observe the output voltage waveform, SCR voltage and gate signals for various firing angle
5. Observe the range of firing angle control.
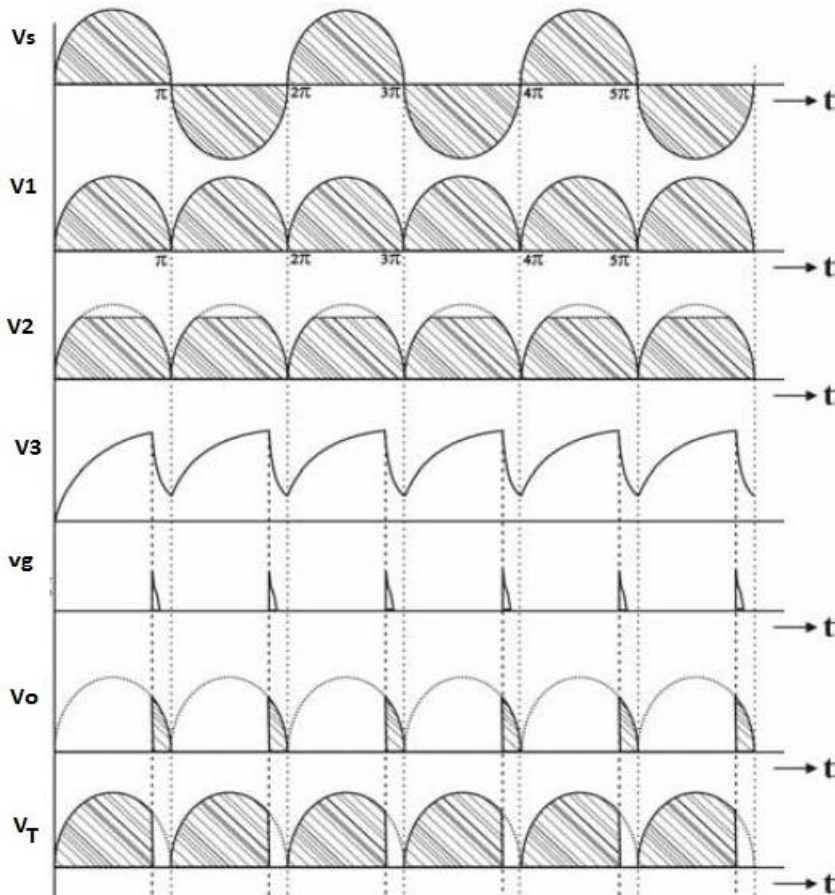
RESULT:

 Studied the operation of UJT firing circuit for SCR for different firing angles.

CIRCUIT DIAGRAM:
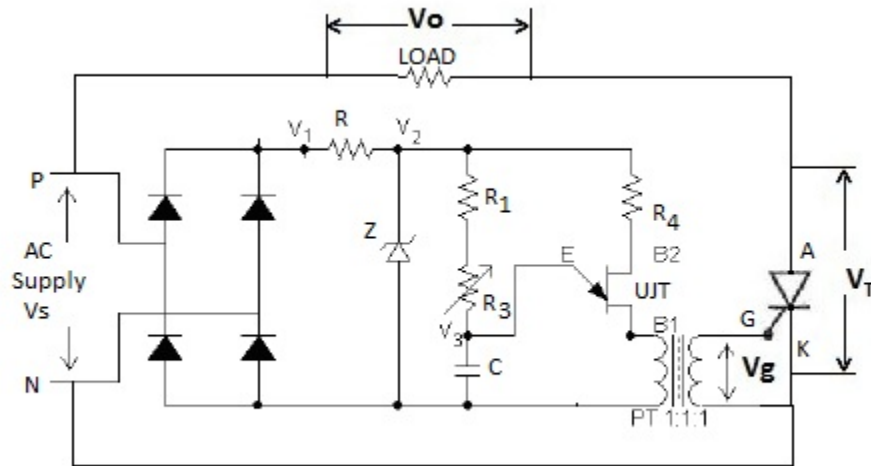
WAVEFORMS:

EXPERIMENT 4

SIMULATION OF THREE PHASE CONTROLLED RECTIFIER CONVERTER

AIM:

i.   To simulate three phase controlled rectifier circuit for R load and R L load in MATLAB
ii.   To compare the theoretical and simulation results for different firing angles

SIMULATION BLOCKS:

AC voltage source, Thyristors,Pulse Generators (for Thyristor gate pulse) , Series RLC block (for R and RL load) ,Voltage and Current measurement blocks, Mean Value block, THD measurement block, Scope and Display blocks, Powergui block (Continuous).
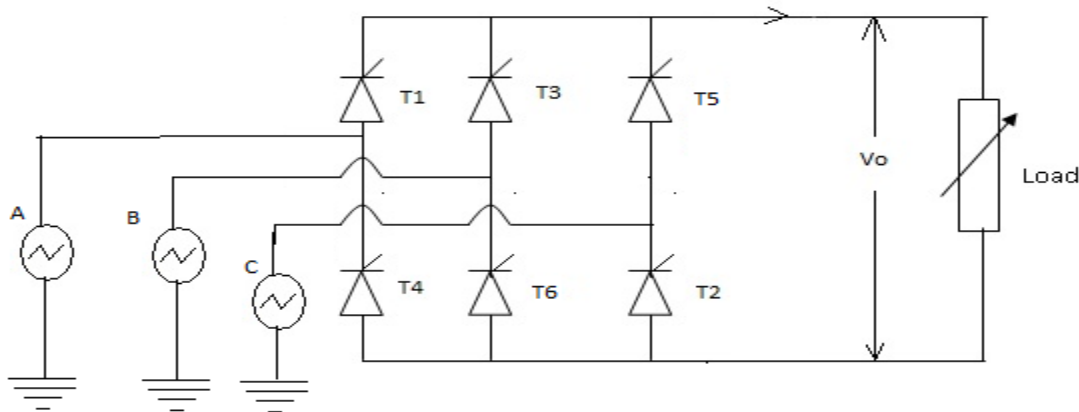
THEORY:

Figure shows the circuit diagram of three phase bridge controlled rectifier. It consist of upper group (T1,T3,T5) and lower group (T2,T4,T5) of thyristors .Thyristor T1 is forward biased ad can be triggered for conduction only when Va is greater than both Vb and Vc. From figure this condition occurs at wt=30^0. Hence T1 can be triggered only at wt=30^0.If firing angle is α, then T1 starts conduction at wt=30+ α and conducts for 120^0 where it get commutated by turning on of next thyristor ie,T3.Similarly triggering instant for T3 and T5 are determined when considering Vb and Vc respectively. For lower group T4,T6 and T2, negative voltages,ie,-Va,-Vb and -Vc respectively are considered. Thus the forward bias instant and triggering instants are obtained as:

Thyristor	Forward Bias instant(degree)	Triggering instant(degree)	Conduction period (degree)
T1	30	30+α	30+α to 150+α
T2	90	90+α	90+α to 210+α
T3	150	150+α	150+α to 270+α
T4	210	210+α	210+α to 330+α
T5	270	270+α	270+α to 390+α
T6	330	330+α	330+α to 450 +α

Average Value of output voltage is given by

$$V_{avg} = \frac{3\sqrt{3}}{\pi} V_m \cos \alpha$$ where V_m is the maximum value of phase to neutral voltage

Average Value of output current is given by

$$I_{avg} = \frac{3\sqrt{3}}{\pi R} V_m \cos \alpha$$ where R is the load resistance

PROCEDURE:

1. Create a new MATLAB model file and create the MATLAB model as shown in figure 2 corresponding to the three phase controlled rectifier circuit.
2. Give the run time value as 0.1 secs to obtain simulation results for 5 input cycles (5x1/50HZ=0.1 sec)
3. In MATLAB command window give the value of variable 'a' (assigned in pulse generator blocks) which is equal to firing angle in degrees.
4. Run the simulation model and obtain the results.
5. Repeat the simulation for different values of firing angle for both R and RL load.

RESULT:

   i.   Simulated single phase controlled rectifier circuit for both R and RL load using MATLAB
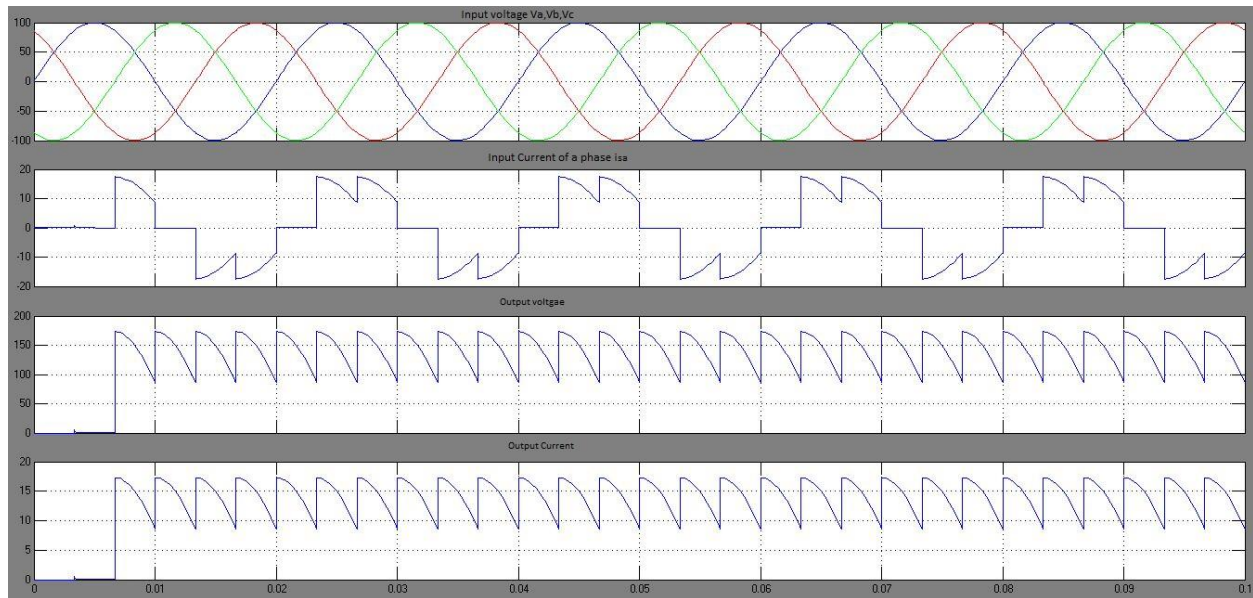   ii.   Compared the simulation results obtained with theoretical calculations.

MATLAB MODEL:
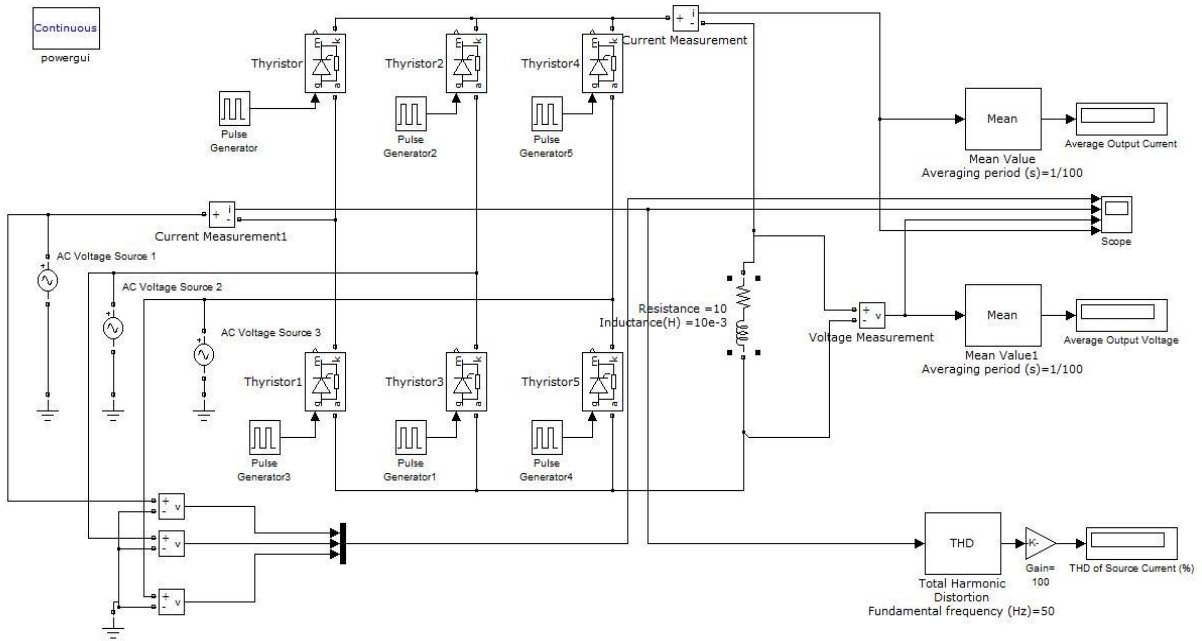
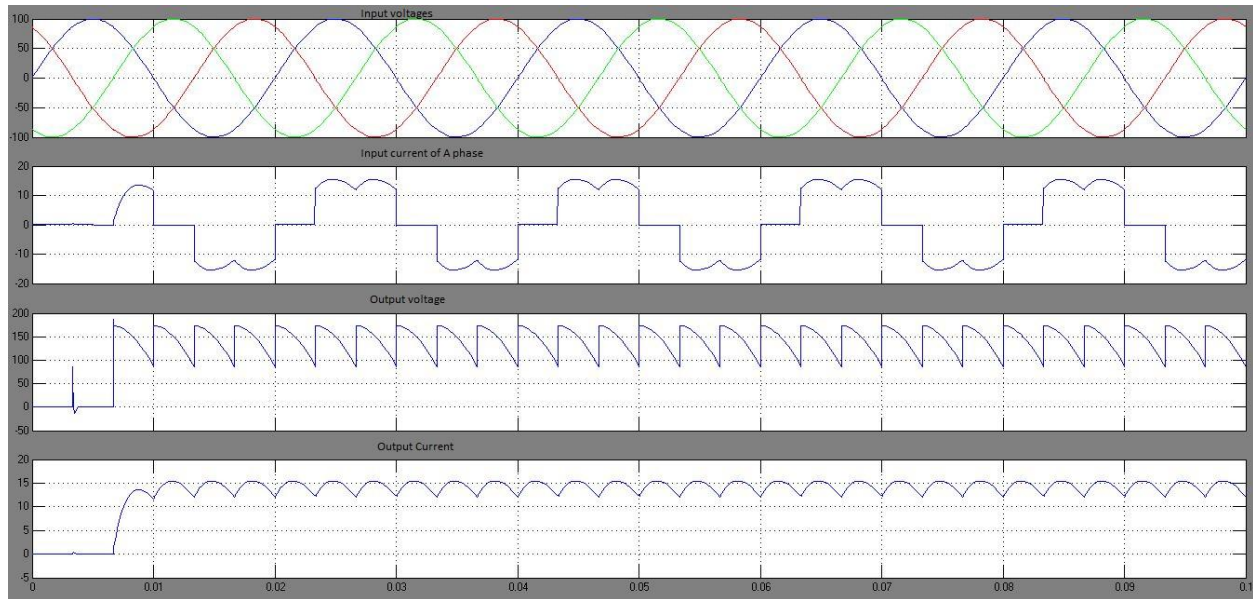## OBSERVATIONS:

Input Peak Voltage =100 V, Frequency=50Hz

Load Type	Firing Angle (in deg)	Theoretical Calculations		Simulation Results		
		Average Output Voltage (V)	Average Output Current (A)	Average Output Voltage (V)	Average Output Current (A)	THD of Source Current (in %)
R Load	0					
	30					
	60					
	90					
RL Load	0					
	30					
	60					
	90					

WAVEFORMS:

(a) R load: α=30^0

(b) RL load: α=30^0

EXPERIMENT 5

SIMULATION OF THREE PHASE SINE PWM INVERTER

AIM:

i.   To simulate Three Phase Sine PWM Inverter for R Load in MATLAB
ii.   To compare the theoretical and simulation results for different values of modulation index.

SIMULATION BLOCKS:

DC voltage source, MOSFET , Series RLC block (for RL load) ,Voltage and Current measurement blocks, Mean Value block, THD measurement block, Scope and Display blocks, Powergui block (Continuous).
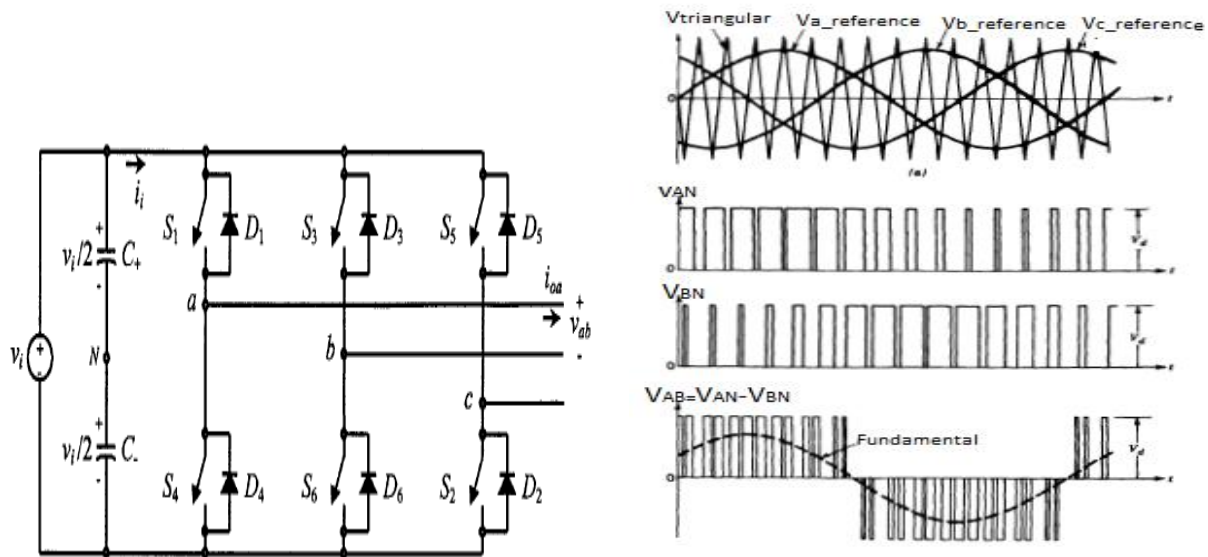
THEORY:

   Figure shows the three phase inverter circuits with on and off controlled switches such as MOSFET or IGBT. Sine PWM pulses are used to gate the switches. Upper switches are gated with signals obtained by comparing three reference sine waves each are phase shifted with 120^0 with a high frequency triangular carrier wave. Thus switches are ON when amplitude of corresponding reference sine wave is greater than amplitude of triangular carrier wave. Lower switches are gated with a gate signal which is complement of upper switches of same leg.

Rms Value of phase to neutral output voltage is given by

$$V_{ph.rms} = \frac{m.V_{dc}}{2\sqrt{2}}$$

Rms Value of line to line output voltage is given by

$$V_{Line.rms} = \frac{\sqrt{3}\ m.V_{dc}}{2\sqrt{2}}$$

PROCEDURE:

1. Create a new MATLAB model file and create the MATLAB model as shown in figure 2 corresponding to the three phase inverter circuit.
2. Give the run time value as 0.1 secs to obtain simulation results for 5 output ac cycles (5x1/50HZ=0.1 sec)
3. In MATLAB command window give the value of variable 'm' (assigned in reference sine blocks) which is equal to modulation index.
4. Run the simulation model and obtain the results.
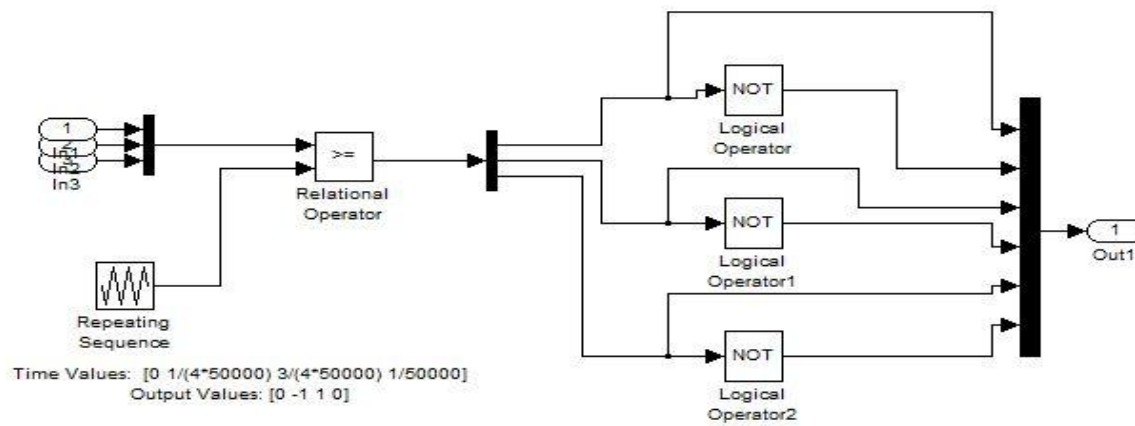5. Repeat the simulation for different values of modulation index.

RESULT:

i.   Simulated three phase sine PWM inverter circuit using MATLAB
ii.   Compared the simulation results obtained with theoretical calculations.

MATLAB MODEL:
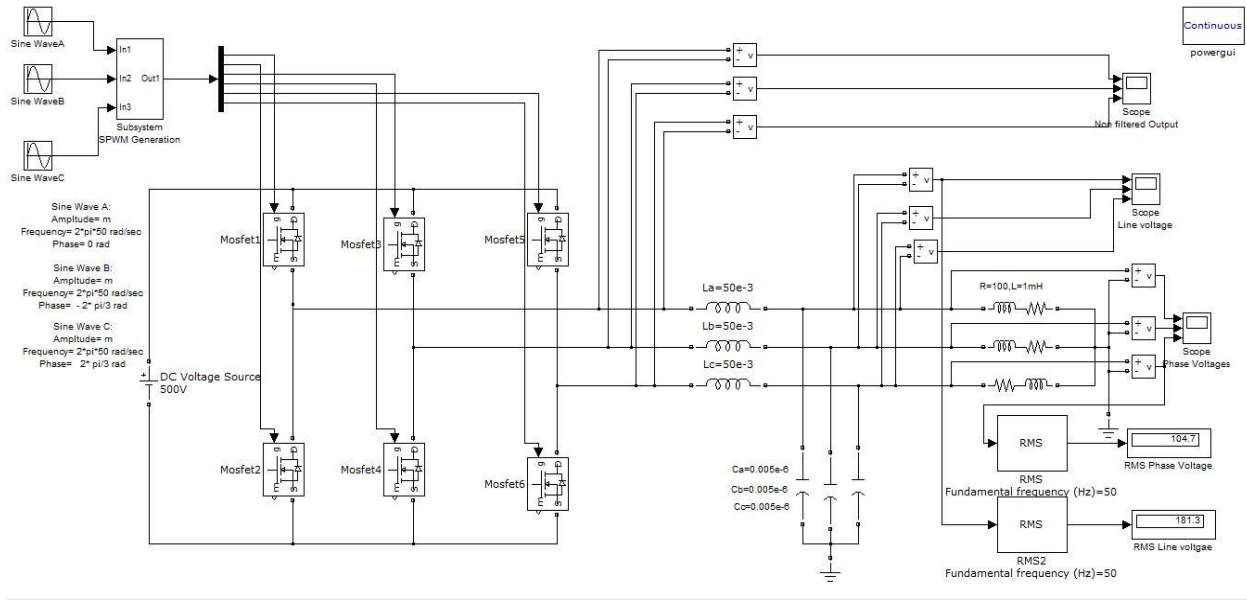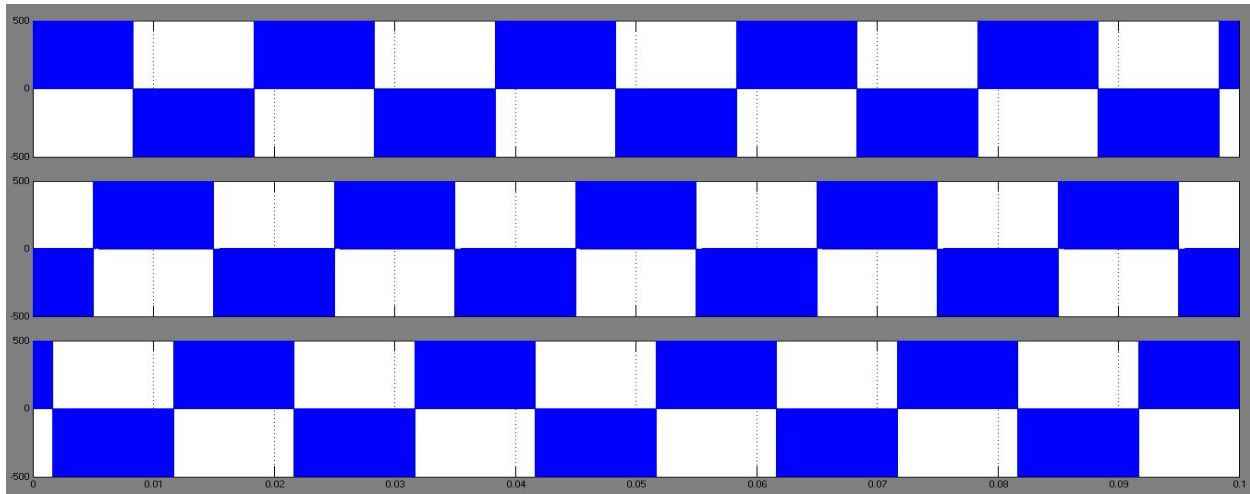

Subsystem (SPWM Generator):

OBSERVATIONS:

Input DC Voltage =500 V

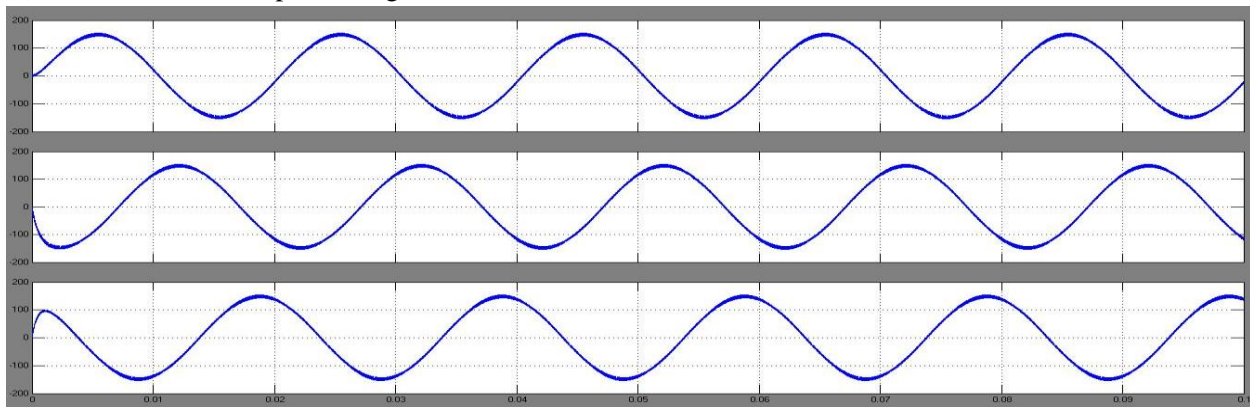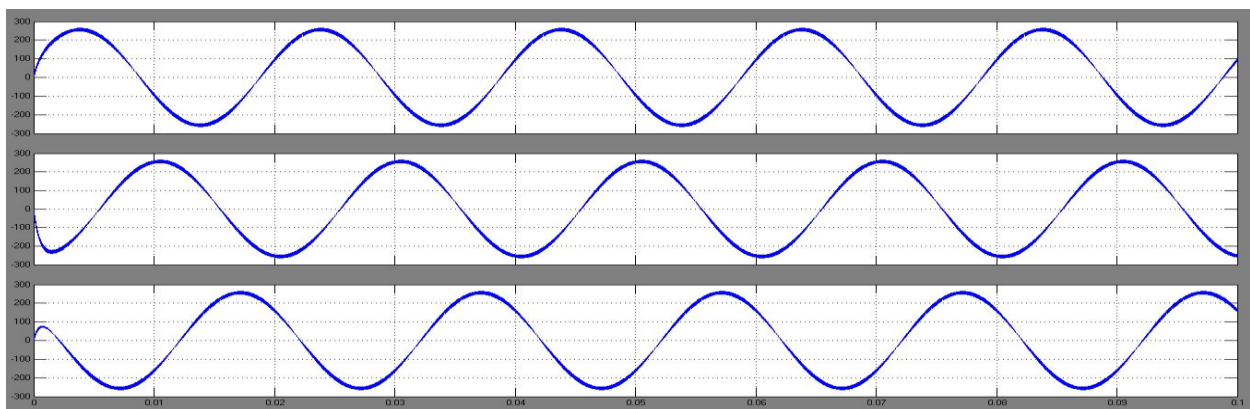| Modulation index (m)	Theoretical Calculations		Simulation Results	
	RMS Output Phase Voltage (V)	RMS Output Line Voltage (V)	RMS Output Phase Voltage (V)	RMS Output Line Voltage (V)
0.85				
0.75				
0.6				
0.5				

WAVEFORMS:

(a) Non filtered Line to Line output voltages (Va,Vb,Vc)

(b) Phase to neutral output Voltages:

(c) Line to Line output voltages:

EXPERIMENT 6

DESIGN AND SIMULATION OF BUCK CONVERTER

AIM:

  i.   To design DC-DC buck converter and simulate it with MATLAB
  ii.   To compare the theoretical and simulation results for different values of duty cycle.

SIMULATION BLOCKS:

  DC voltage source, MOSFET , Series RLC block ,Voltage and Current measurement blocks, Scope,
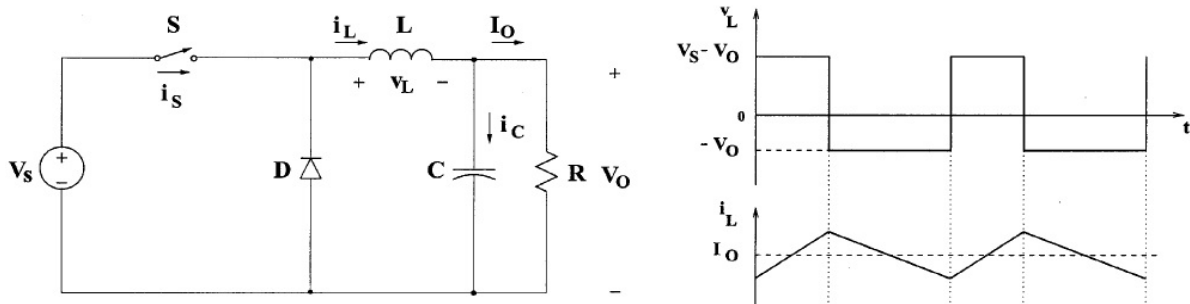 Powergui block (Continuous).

THEORY:

   Figure shows the circuit diagram of step down DC-DC converter, commonly known as buck converter.
When switch S is ON, diode D is reverse biased and voltage across inductor will be Vs-Vo. So inductor
current increases and attains apeak..When switch S is OFF, diode D is forward biased and voltage across
inductor will be   -Vo. So inductor current decreases. Hence inductor with diode ensures an uninterrupted
current flow and hence a constant output voltage with capacitor for removing voltage ripples.

                Output voltage is given by

$$V_o = D\ V_s$$

                Where *D* is duty cycle and V_S is input voltage

PROCEDURE:

  1. Create a new MATLAB model file and create the MATLAB model as shown in figure 2
      corresponding to DC-DC buck converter.
  2. Give the run time value as 0.5 secs .
  3. Run the simulation model and obtain the results.
  4. Repeat the simulation for different values of duty cycles.

RESULT:

  i.   Simulated DC-DC buck converter circuit using MATLAB
  ii.   Compared the simulation results obtained with theoretical calculations.

DESIGN OF CIRCUIT COMPONENTS:

Input voltage, V_S=100V

Output Voltage requiredV_O=60 V
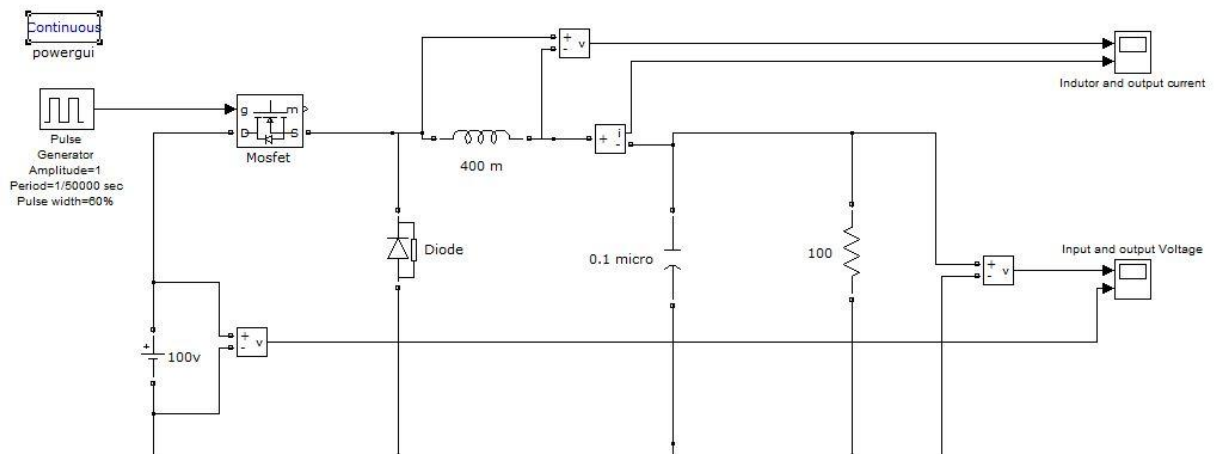
Output current I_O= 0.6 A

Switching frequency, f_S=50kHZ

Ripple allowed in inductor current, $\Delta I = 2mA$

Ripple allowed in capacitor voltage, ΔV_c= 50mV
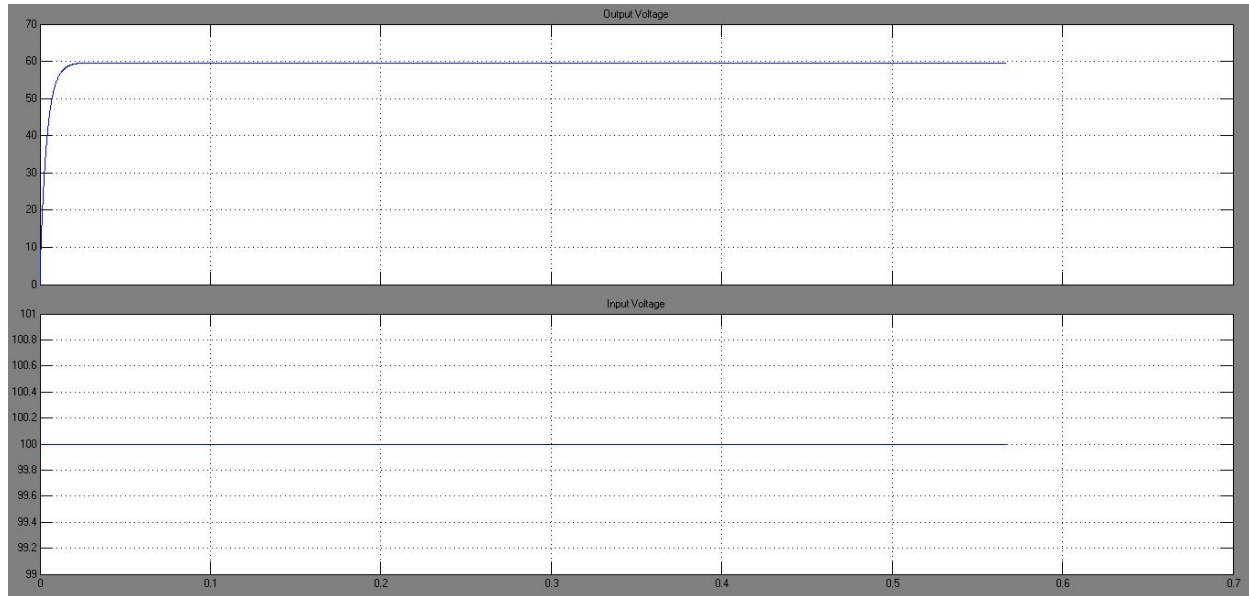
Inductor,$L = \dfrac{V_O(1-D)}{f_s\,\Delta I}=$

Capacitor ,C=$\dfrac{V_O(1-D)}{8\,f_s{}^2 \Delta V_c} =$
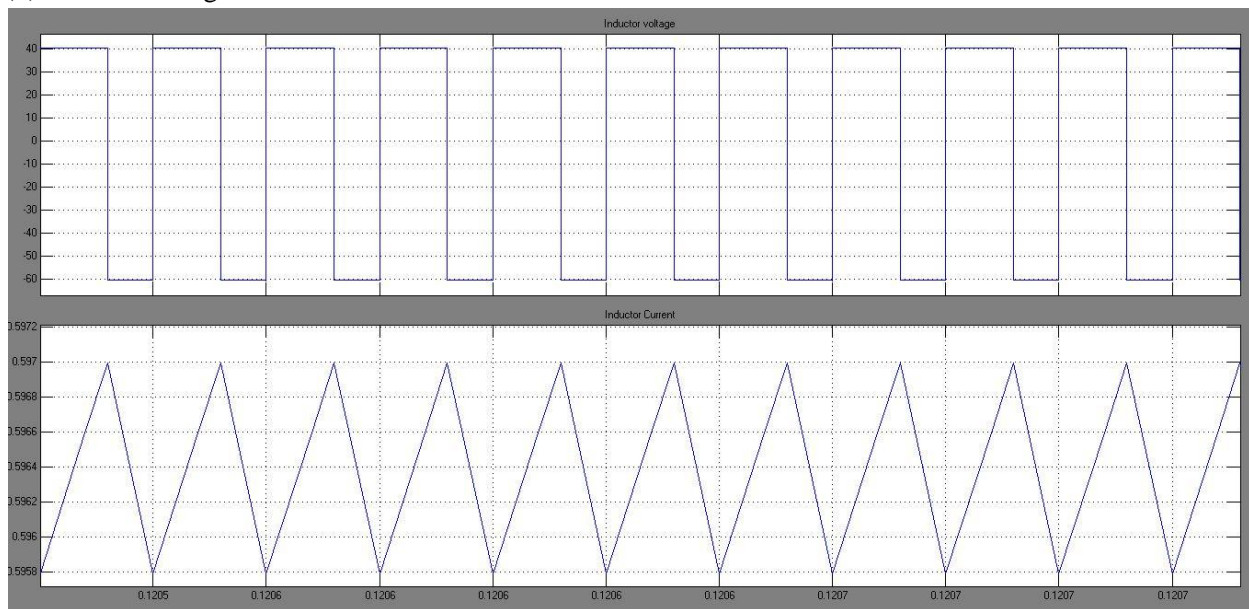
Load Resistance $R = \dfrac{V_O}{I_O} =$

MATLAB MODEL:

WAVEFORMS:

(a) Output and input voltages:

(b)Inductor voltage and current:

OBSERVATIONS:

Duty Cycle (D)	Theoretical Calculations	Simulation Results
	Output Voltage (V)	Output Voltage (V)
0.6		
0.5		
0.4		

EXPERIMENT 7

SIMULATION OF THREE PHASE INDUCTION MOTOR DRIVES WITH V/F CONTROL

AIM:

To simulate Three Phase Induction motor with V/f control.

SIMULATION BLOCKS:

Inducion motor,DC voltage source, Universal Bridge(IGBT) ,PWM Generator block,Voltage and Current measurement blocks, Scope and Display blocks, Powergui block (Continuous).

THEORY:

Figure shows the three phase inverter circuits with on and off controlled switches such as MOSFET or IGBT. Sine PWM pulses are used to gate the switches. Upper switches are gated with signals obtained by comparing three reference sine waves each are phase shifted with 120^0 with a high frequency triangular carrier wave. Thus switches are ON when amplitude of corresponding reference sine wave is greater than amplitude of triangular carrier wave. Lower switches are gated with a gate signal which is complement of upper switches of same leg.

PROCEDURE:

1. Create a new MATLAB model file and create the MATLAB model for induction motor fed with IGBT DC-AC inverter .
2. Give the run time value as 1 secs.
3. Set speed as rated (1430 rpm) and run the simulation model and obtain the results for various loads by changing torque.
4. Set speed as 1300 rpm and run the simulation model and obtain the results for various loads by changing torque.

RESULT:

i.   Simulated three phase induction motor with V/F control using MATLAB
ii.   Analyzed speed regulation with the control scheme.

MATLAB MODEL:
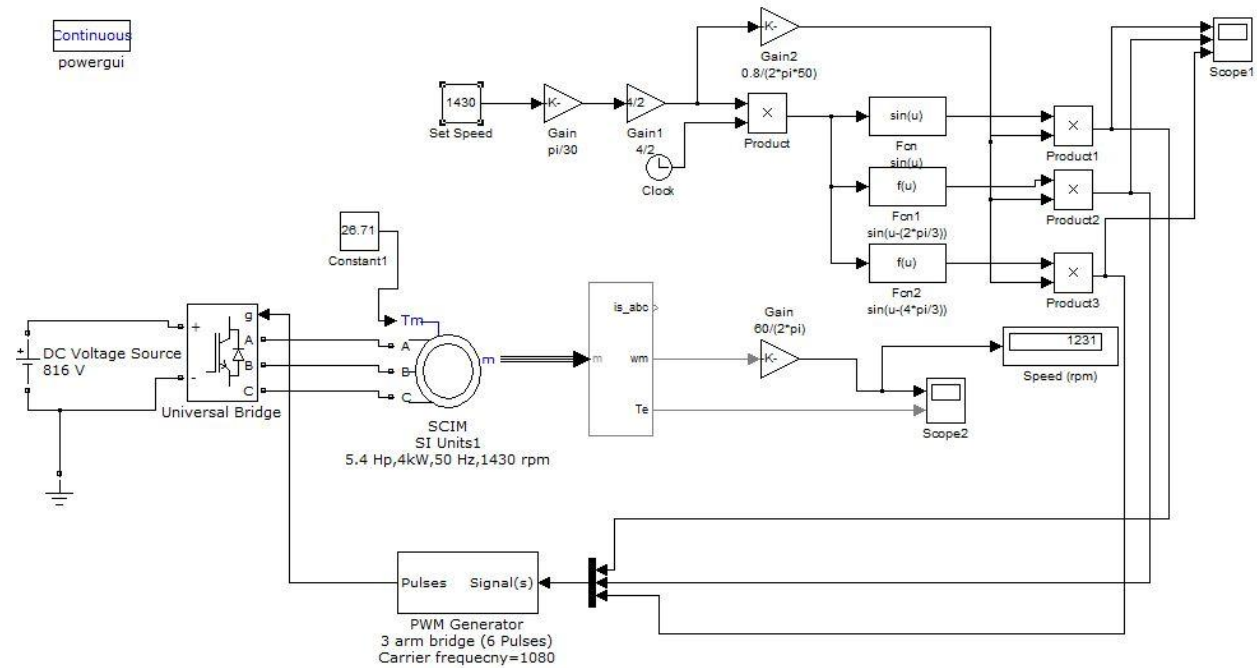
OBSERVATIONS:

Set Speed =1430			
Load Fraction	Torque	Speed of motor (rpm)	Speed regulation(%)
No load			
Full load			
½ of full load			
¼ of full load			
Set Speed =1430			
Load Fraction	Torque	Speed of motor (rpm)	Speed regulation(%)
No load			
Full load			
½ of full load			
¼ of full load			

Motor Selected is 400V 4 kW,50HZ,1430 rpm

Input DC voltage required $,V_{dc} = \dfrac{2\sqrt{2}}{\sqrt{3}} \dfrac{V_{Line\,.rms}}{m.}$ =816 V

Where , *m*=modulation index=0.8 and $V_{Line\,.rms} = 400\,V$

Let X be the load fraction

Then, Torque ,$T = \dfrac{XP}{2\pi\,N}60$

Where N= set speed, P=4 kW

Regulation = $\dfrac{V_{No\ load} - V_{loaded}}{V_{No\ load}}$ x 100

WAVEFORMS:

Speed and Torque: