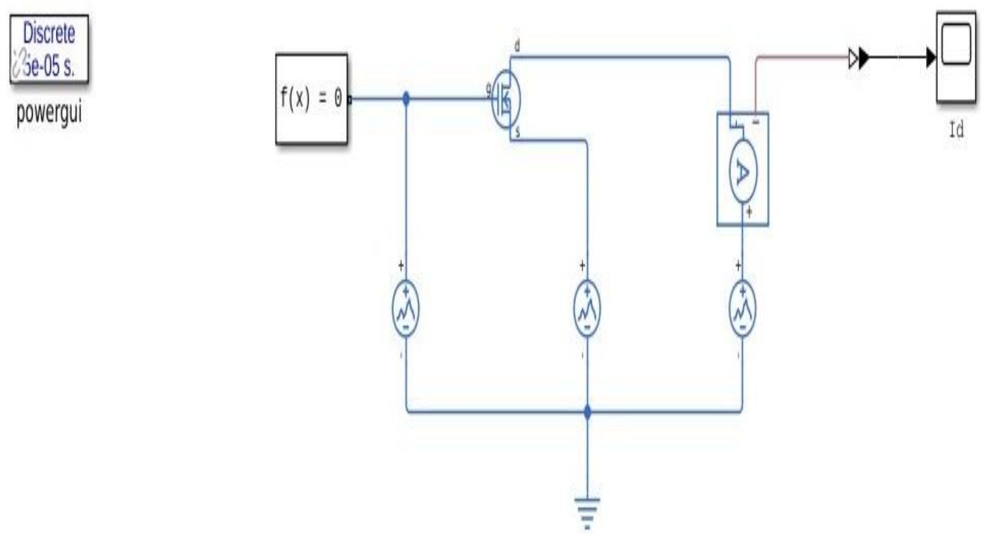
EXPERIMENT NO. 1

STUDY OF CHARACTERISTICS OF MOSFET

AIM: To plot the characteristics of MOSFET

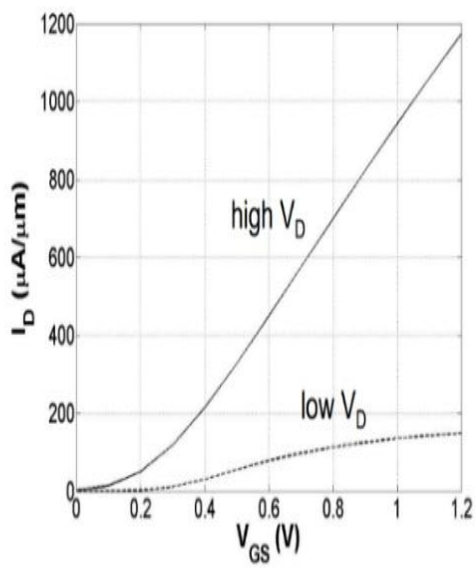
APPARATUS REQUIRED: MATLAB Software

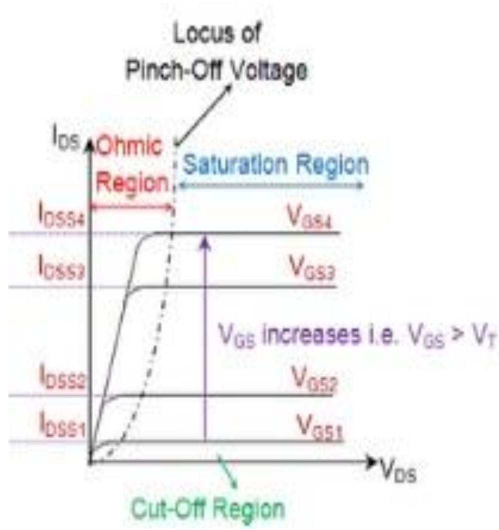
CIRCUIT DIAGRAM:



MOSFET Characteristics

THEORY: The **transfer characteristics** of a MOSFET show the relationship between the **gate-source voltage (V_{GS})** and the **drain current (I_D)** for various fixed values of **drain-source voltage (V_{DS})**. The **output characteristics** of a MOSFET show the relationship between the **drain-source voltage (V_{DS})** and the **drain current (I_D)** for various fixed values of **gate-source voltage (V_{GS})**.





PROCEDURE:

1. Enter into the command window of the MATLAB.
2. Open a new SIMULINK Model page.
3. Develop MATLAB diagram.
4. For transfer characteristics, set V_{DS} at some particular value. Provide a range of values of V_{GS}.
5. Execute the model by pressing Run button.
6. View the results in Scope.
7. Store the data of Scope in Workspace.
8. Repeat steps 4 to 7 for different values of V_{DS}.
9. Plot all the results in same graph.
10. For output characteristics, set V_{GS} at some particular value. Provide a range of values of V_{DS}.
11. Execute the model by pressing Run button.
12. View the results in Scope.
13. Store the data of Scope in Workspace.
14. Repeat steps 10 to 13 for different values of V_{GS}.
15. Plot all the results in same graph.

RESULT: The transfer and output characteristics of MOSFET have been simulated successfully by building a SIMULINK Model.

EXPERIMENT NO. 12
CIRCUIT SIMULATION OF VOLTAGE SOURCE INVERTER AND STUDY OF SPECTRUM ANALYSIS WITH AND WITHOUT FILTER USING MATLAB

AIM:

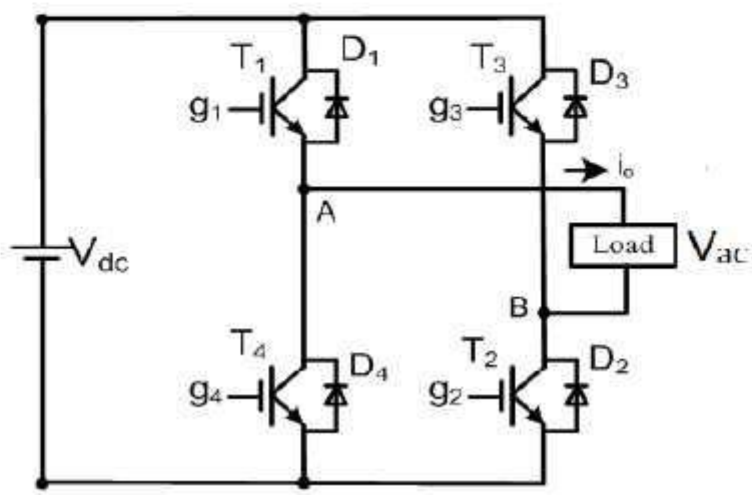
To simulate Voltage Source Inverter and study of spectrum analysis with and without filter

APPARATUSREQUIRED:

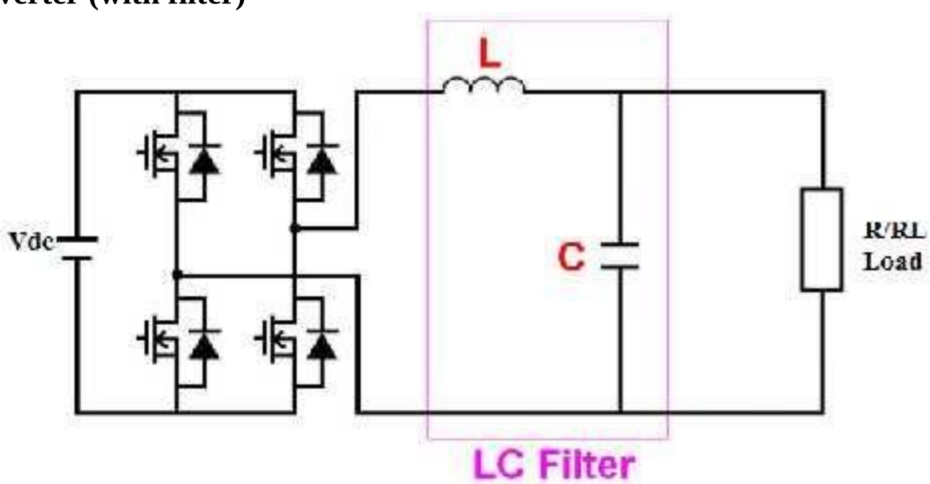
1.MATLAB Software

CIRCUITDIAGRAM:

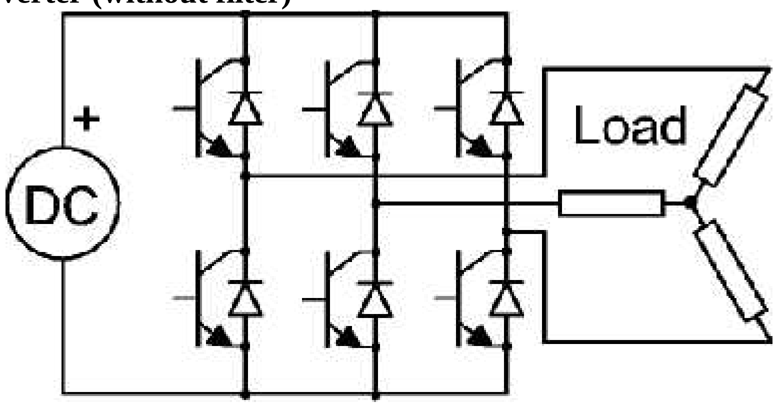
Single phase inverter (without filter)



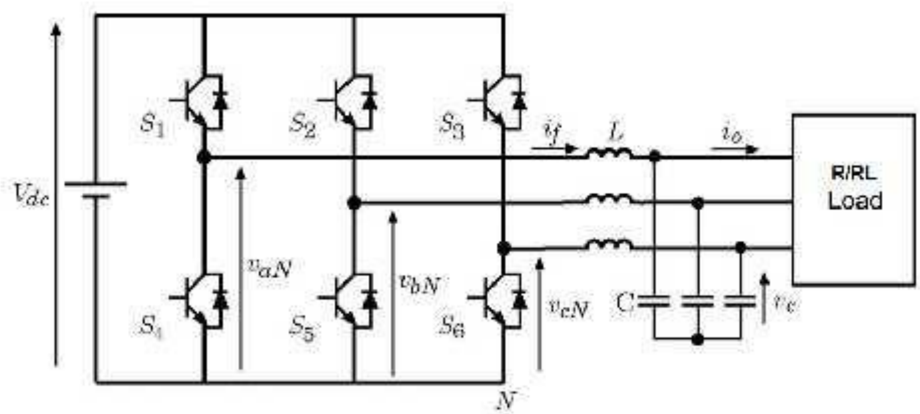
Single phase inverter (with filter)



Three phase inverter (without filter)



Three phase inverter (with filter)



PROCEDURE:

1. Click on File New Model.
2. On resulting window click on library Browser, a Simulink library browser will appear.
3. Make/Model the circuit by placing all its Blocks from its corresponding Library/toolbox, which is clearly shown in the table 1. Right click on the block to rotate mirror etc. to organize the circuit elements.
4. To get any one of the element in series RLC branch, such as R, L or C, click on series RLC branch block in the Simpower System/Element library, and enter the parameters value in such a way that other two elements are invisible. That is to eliminate resistance, inductance, or capacitance of the branch, the R, L and C values must be set respectively to 0, 0 and inf.
5. To change the circuit parameters applicable to the block by double clicking on the block/element and type the values. Keep the values default for some blocks like thyristor, diodes, mosfet etc.
6. To measure/observe the voltage across or current passes through the electrical block/device, connect voltage measurement or current measurement blocks respectively with the electrical block, it is available on the library Simpower System/measurement.
7. To observe the waveform in figure window, scope block is connected with voltage measurement and current measurement blocks. This scope block is available by click on Library browser simulink/source scope.

8. Before simulating the circuit, ensure that all the blocks are connected properly. The most common error encountered is ‘floating node’. This usually means that there is some problem in interconnects.
9. To run the simulation, select simulation start.
10. If any errors are reported here. Correct schematic or the simulation settings and re-run simulation.
11. To view simulation plots on simulation window, double click the scope in the schematic. The scope block corresponding to voltage measurement and current measurement blocks gives voltage and current waveforms respectively with respect to time.

FILTERDESIGN:

$$L = \frac{1}{8}\frac{V_{dc}}{I_{ripple} * f_s}$$

$$C = \frac{1}{2\Pi f R}$$

Where,

L = Filter Inductance

C=FilterCapacitance

V_{dc} = Input dc voltage

I_{ripple} = Ripple current

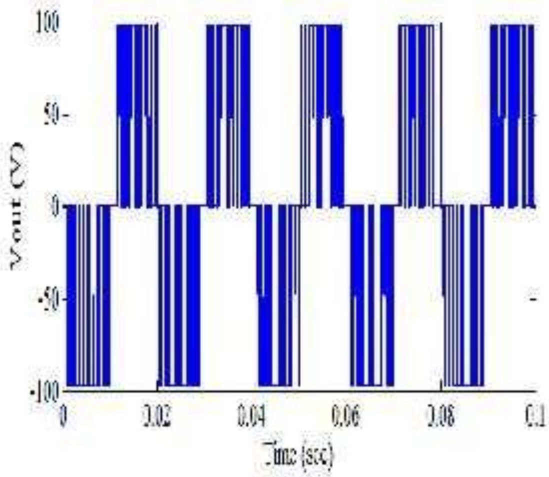
f_s=Switching frequency

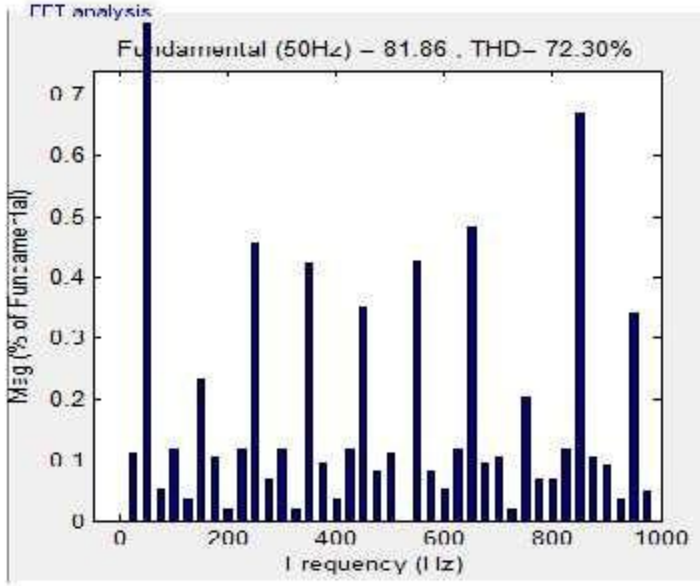
f = Resonant frequency

OUTPUTWAVEFORMS:

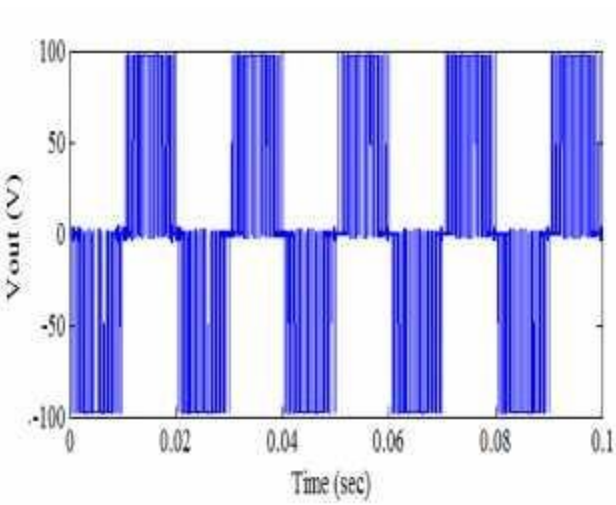
Waveforms and FFT Analysis:

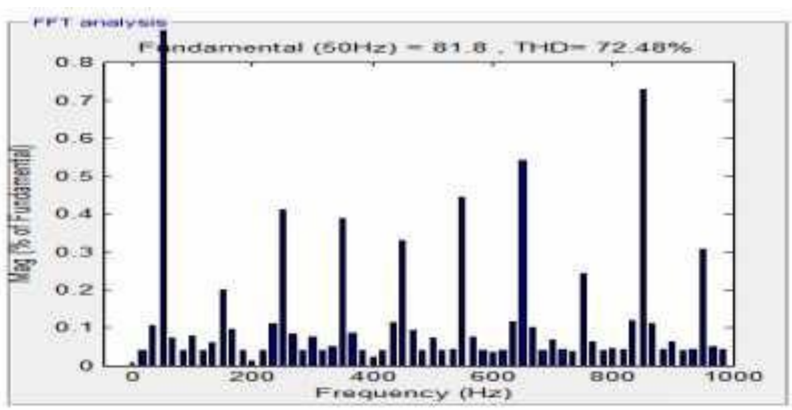
Single phase inverter (R load without filter)



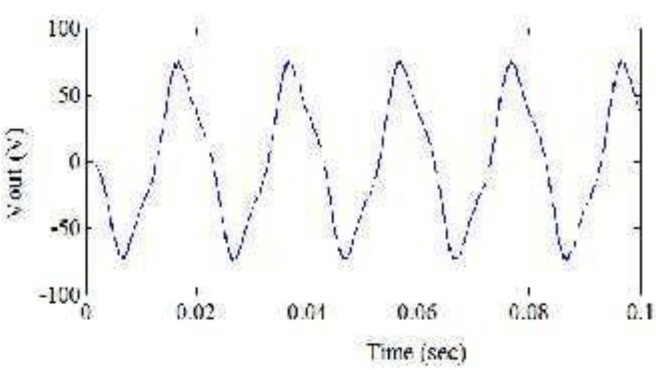


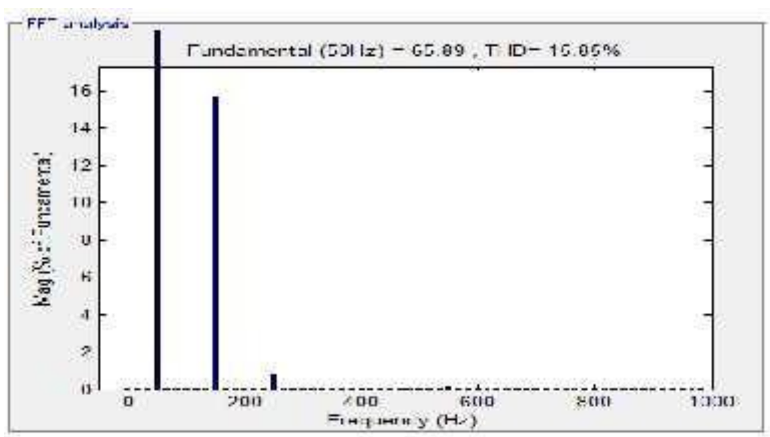
Single phase inverter (RL load without filter)



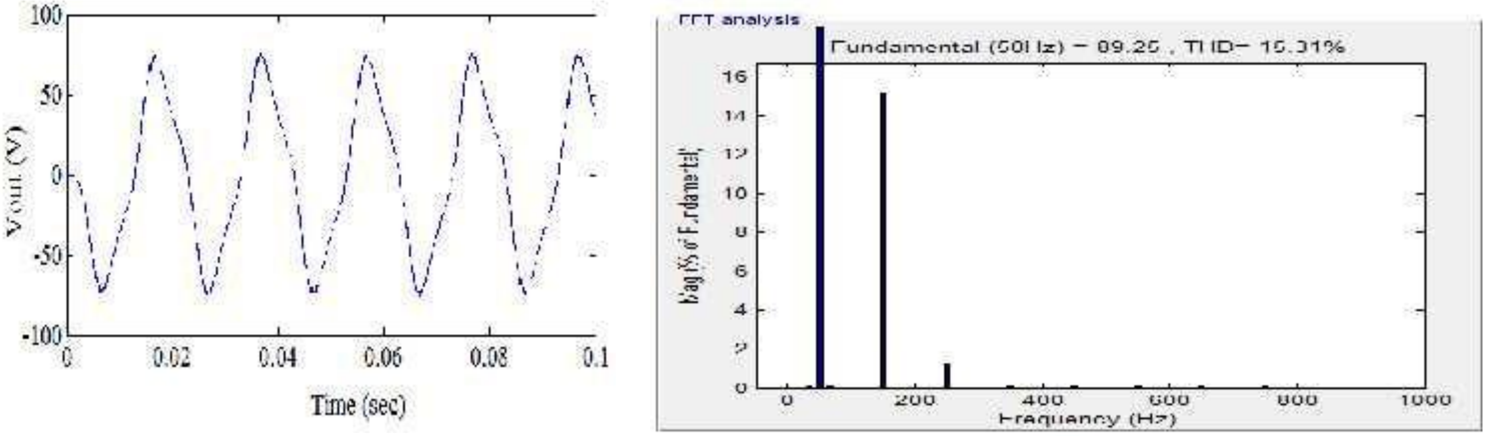


Single phase inverter (R load withfilter)

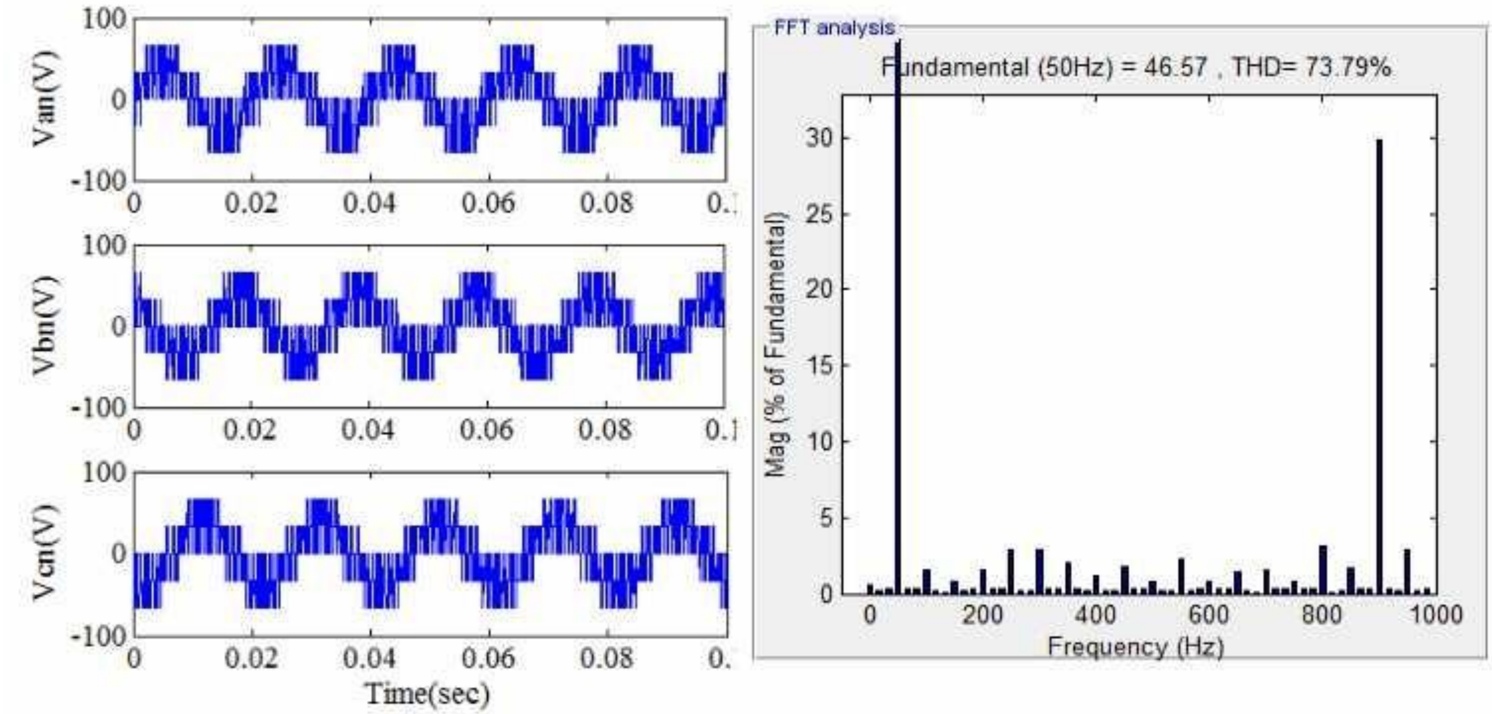




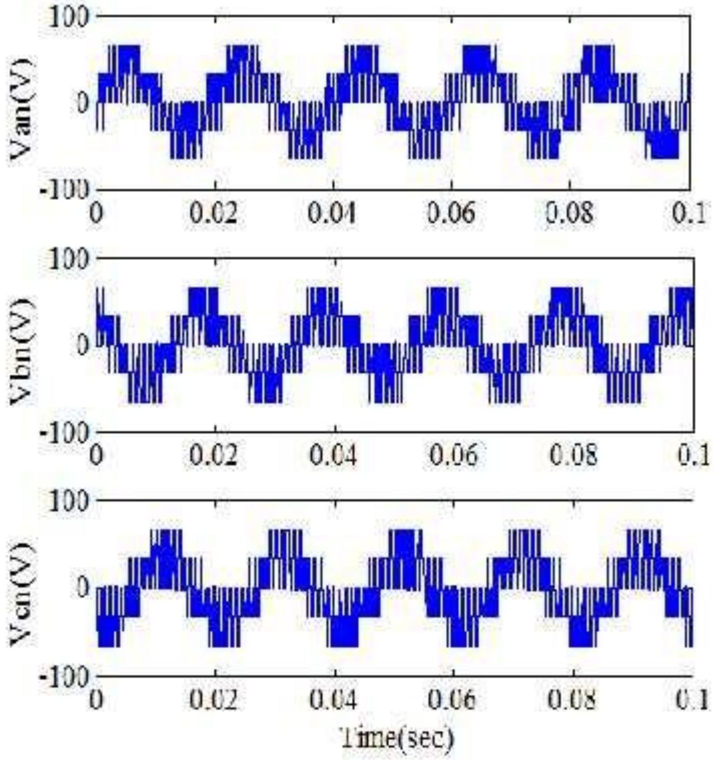
Single phase inverter (RL load with filter)

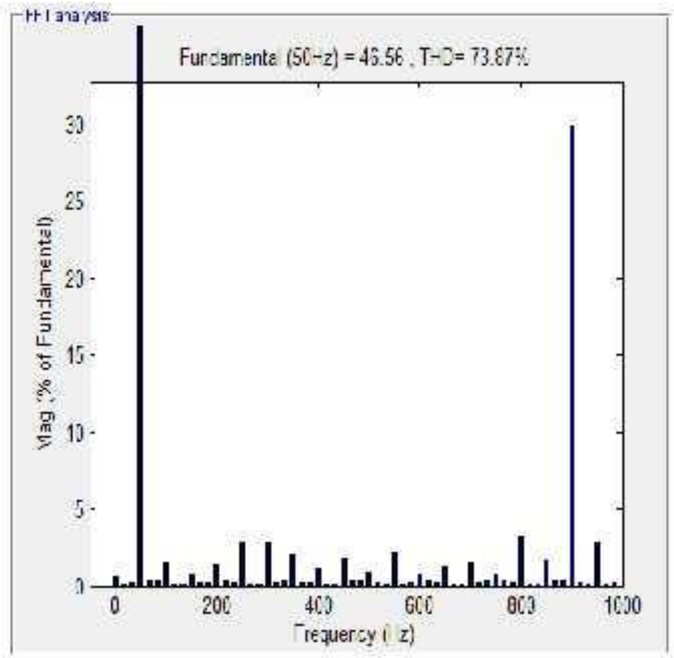


Three phase inverter (R load without filter)

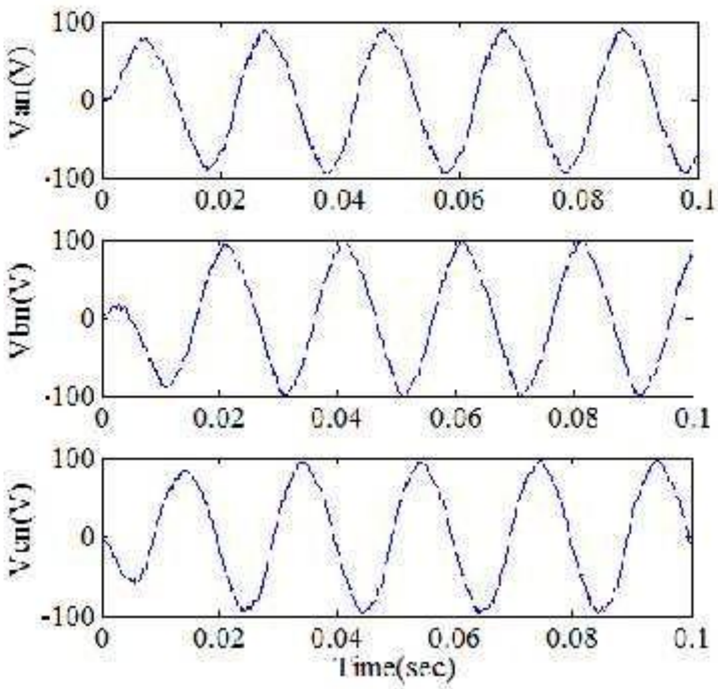


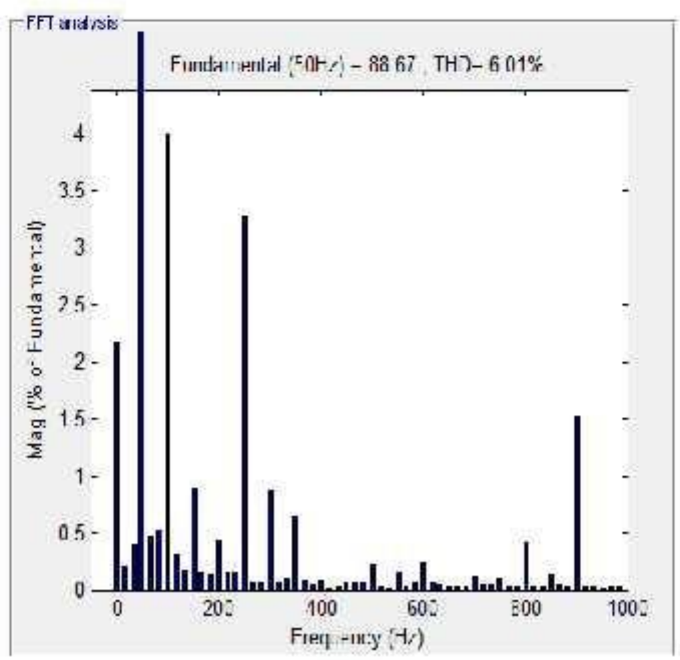
Three phase inverter (RL load without filter)



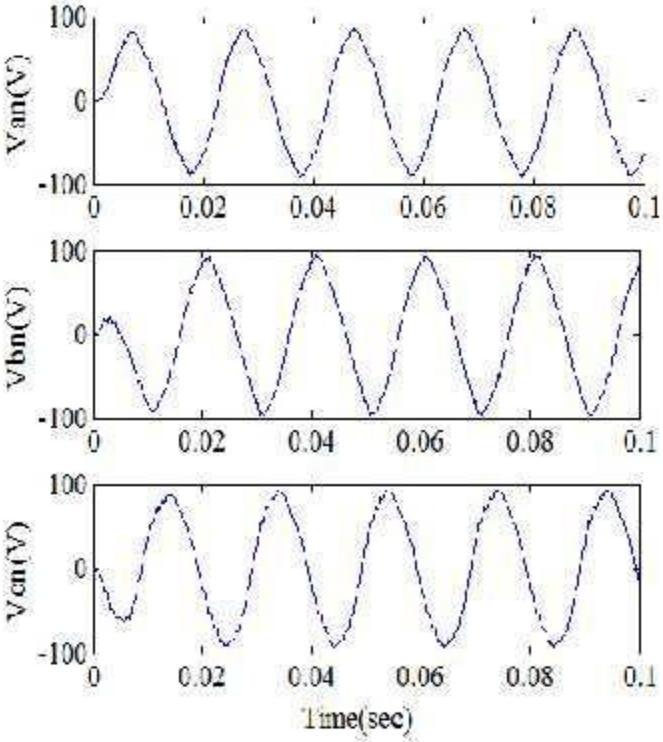


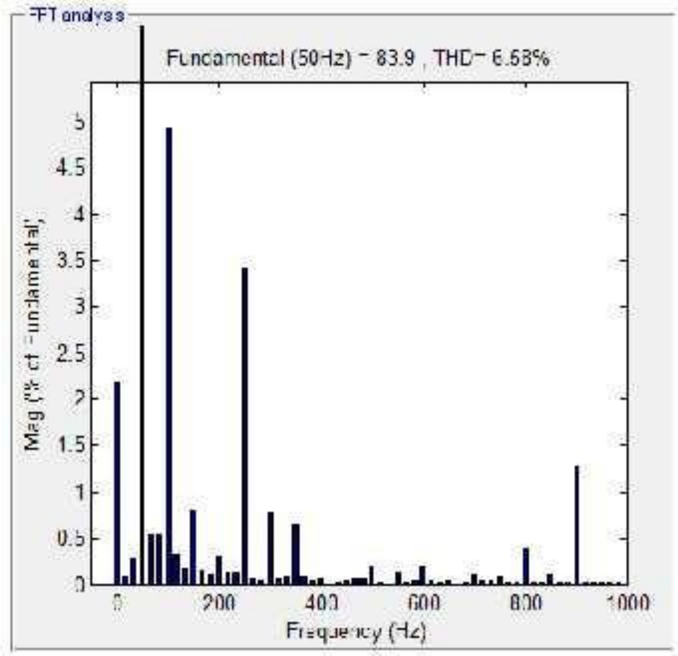
Three phase inverter (R load with filter)





Three phase inverter (RL load with filter)





RESULT:

The simulation of Voltage Source Inverter is done in MATLAB/SIMULINK environment and the waveforms are observed.

EXPERIMENT NO. 11
MODELING AND SYSTEM SIMULATION OF SCR BASED FULL CONVERTER WITH DIFFERENTTYPES OF LOAD USING MATLAB-SIMULINK

AIM:

To model and simulate

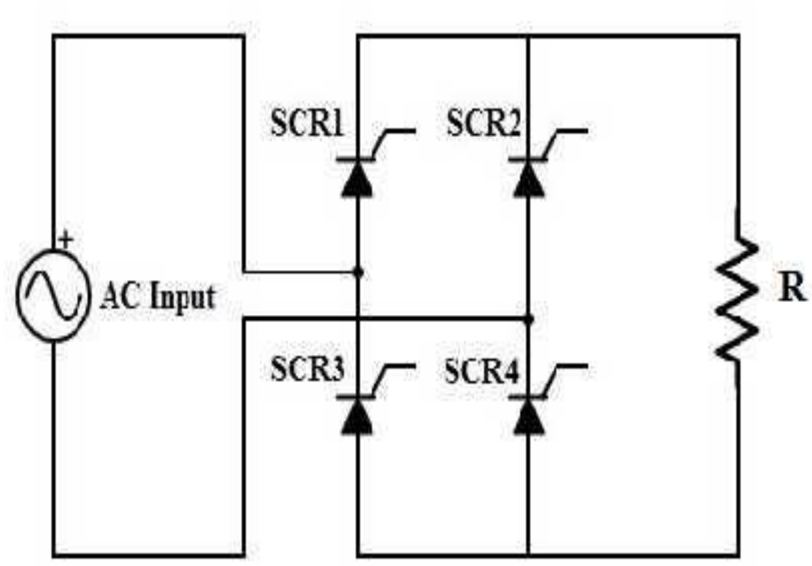
- Full converter fed resistive load
- Full converter fed Resistive-Back Emf (RE) load at different firing angles
- Full Converter fed Resistive-Inductive Load at different firing angles

APPARATUS REQUIRED:

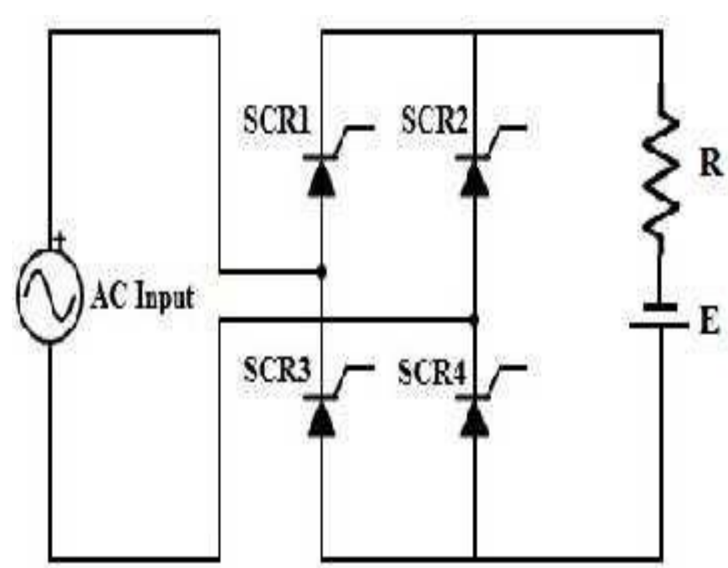
1. MATLAB Software

CIRCUIT DIAGRAM:

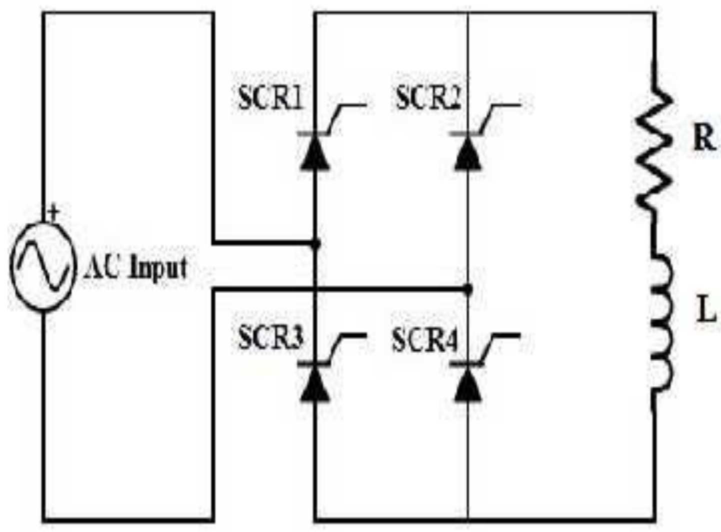
Full converter fed resistive load



Full converter fed Resistive-Back Emf (RE) load at different firing angles



Full Converter fed Resistive-Inductive Load at different firing angles



THEORY:

A single phase full wave converter shown in fig (1) with inductive load so that load current is continuous and ripple free. During positive half cycle thyristors T_1 and T_2 are forward biased, when gating singles are given to gates of T_1 and T_2, thyristors are turn–on. Due to the inductive load thyristors continue to conduct beyond t=∏, even the input voltage is already negative. During negative half cycle of input voltage, thyristors T_3 and T_4 are forward biased, when gate pulses applied to gates of this thyristors they are turned-on .thyristors T_3 and T_4 applies a reverse voltage across T_1 and T_2 as reverse blocking voltage. T_1 and T_2 are turned off due to line or natural commutation.

During the period α to ∏, the input voltage Vs and input current i_S are positive; and the power flows from supply to load the converter operated in rectification mode. During period ∏ to ∏+α, the input voltage negative and input current is positive; and reverse power flow from the load to the source .the converter is said to be operated in inversion mode

The gating sequence for the thyristor is as follows:

1. Generate a pulse-signal at positive zero crossing of the supply voltage Vs.
2. Delay the pulse by desired angle α and apply it between the gate and cathode terminal terminals of T1 through a gate-isolating circuit.

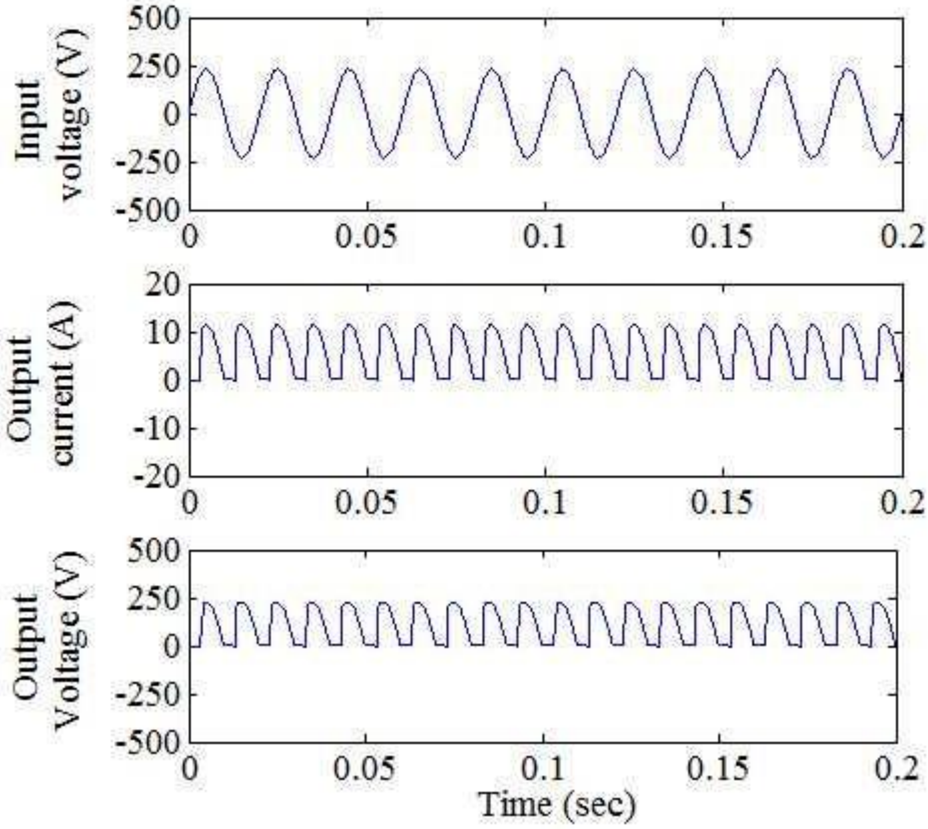
Note: Both the output voltage and input current non-sinusoidal. The performance of the controlled rectifier can be measured by the distortion factor (DF), total harmonic distortion (THD), PF, transformer utilization factor (TUF), and harmonic factor.

PROCEDURE:

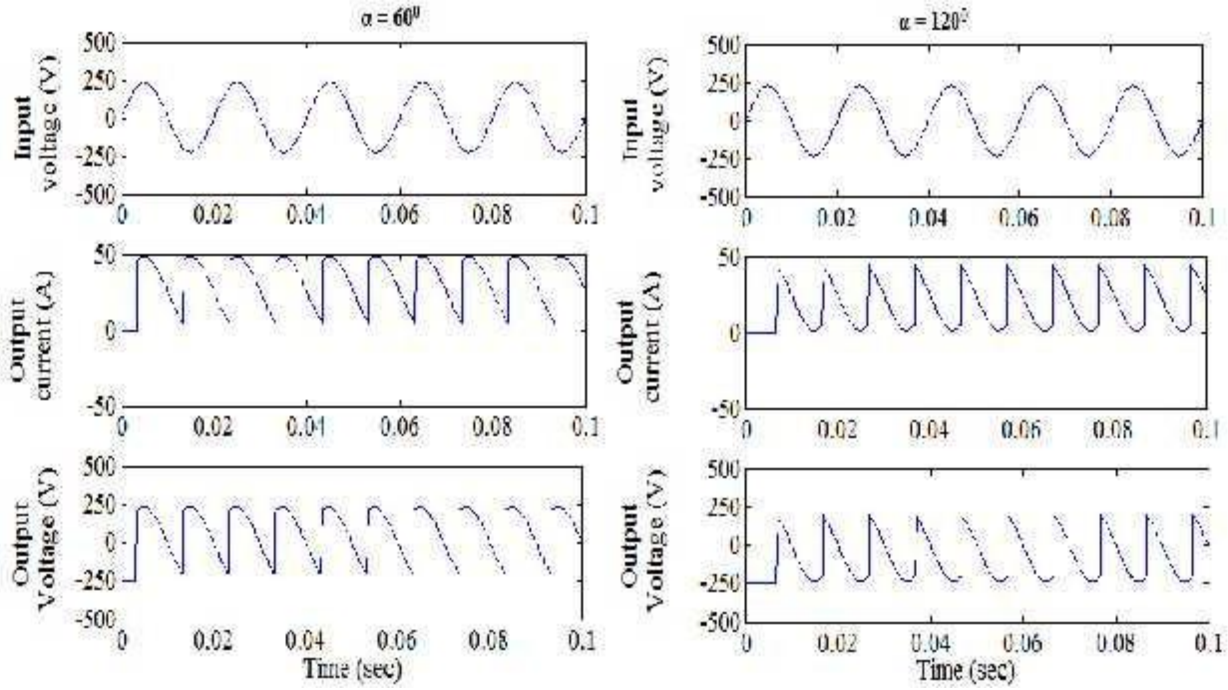
1. Click on File New Model.
2. On resulting window click on library Browser, a Simulink library browser will appear.
3. Make/Model the circuit by placing all its Blocks from its corresponding Library/toolbox, which is clearly shown in the table 1. Right click on the block to rotate mirror etc. to organize the circuit elements.
4. To get any one of the element in series RLC branch, such as R, L, or C, click on series RLC branch block in the Simpower System/Element library, and enter the parameters value in such a way that other two elements are invisible. That is to eliminate resistance, inductance, or capacitance of the branch, the R, L and C values must be set respectively to 0, 0 and inf.
5. To change the circuit parameters applicable to the block by double clicking on the block/element and type the values. Keep the values default for some blocks like thyristor, diodes, mosfet etc.
6. To measure/observe the voltage across or current passes through the electrical block/device, connect voltage measurement or current measurement blocks respectively with the electrical block, it is available on the library Simpower System/measurement.
7. To observe the waveform in figure window, scope block is connected with voltage measurement and current measurement blocks. This scope block is available by click on Library browser simulink/source scope.
8. For single phase full controlled Bridge rectifier with RL load, components required and its parameter values are similar to that of RLE load, except E parameter. That is DC voltage source block must remove from schematic. The schematic arrangement is shown in schematic diagram 2.
9. For single phase full controlled Bridge rectifier with R load, components required and its parameter values are similar to that of RLE load, except L and E parameters. That is L must set to 0, and DC voltage source is removed from the schematic. The schematic arrangement is shown in schematic diagram 1.
10. Before simulating the circuit, ensure that all the blocks are connected properly. The most common error encountered is ‘floating node’. This usually means that there is some problem in interconnects.
11. To run the simulation, select simulation start.
12. If any errors are reported here. Correct schematic or the simulation settings and re-run simulation.
13. To view simulation plots on simulation window, double click the scope in the schematic. The scope block corresponding to voltage measurement and current measurement blocks gives voltage and current waveforms respectively with respect to time.

OUTPUT WAVEFORMS:

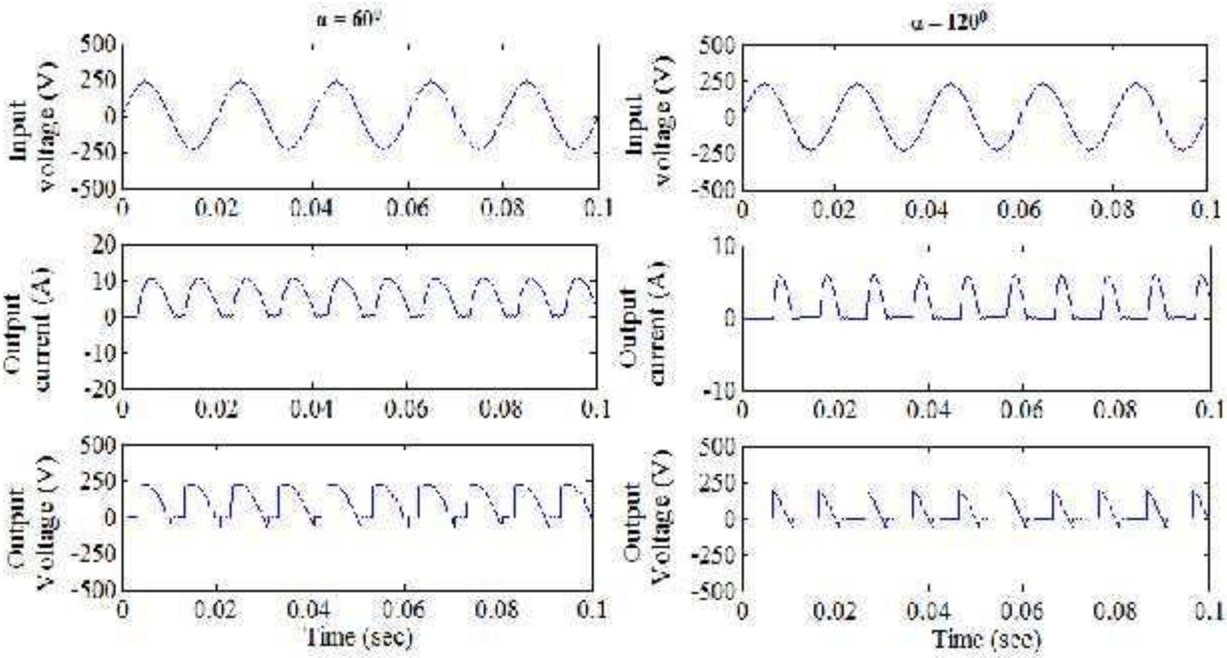
Full converter fed resistive load



Full converter fed Resistive-Back Emf (RE) load at different firing angles



Full Converter fed Resistive-Inductive Load at different firing angles



RESULT:

The simulation of SCR based full converter with different types of load is done in MATLAB/SIMULINK environment and the waveforms are observed.

EXPERIMENT NO. 10

MODELING AND SYSTEM SIMULATION OF SCR BASED SEMI CONVERTER WITH DIFFERENTTYPES OF LOAD USING MATLAB-SIMULINK

AIM:

To model and simulate

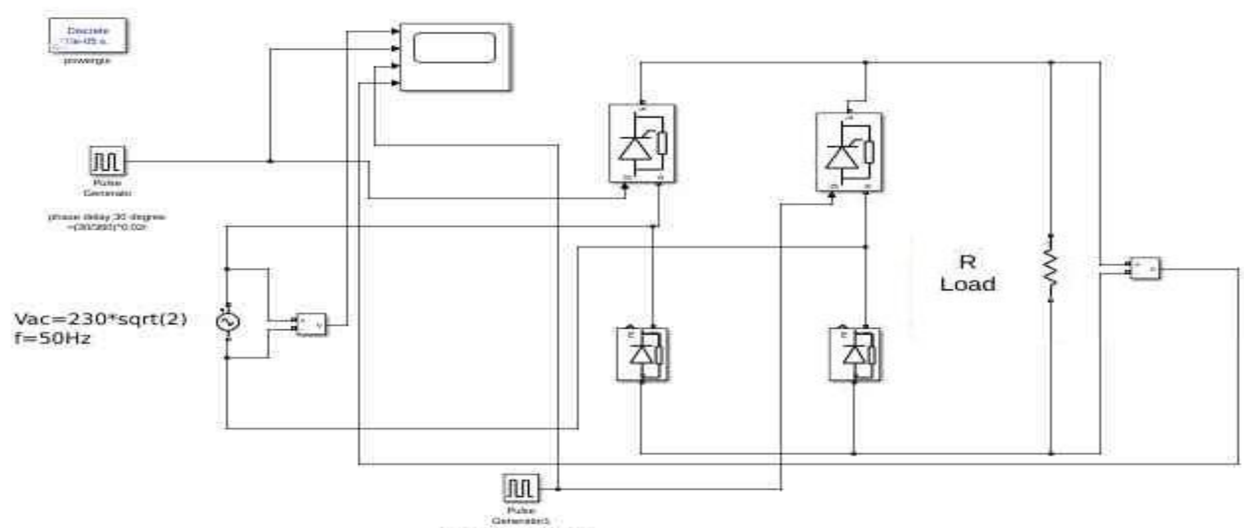
- Full converter fed resistive load
- Full Converter fed Resistive-Inductive Load

APPARATUS REQUIRED:

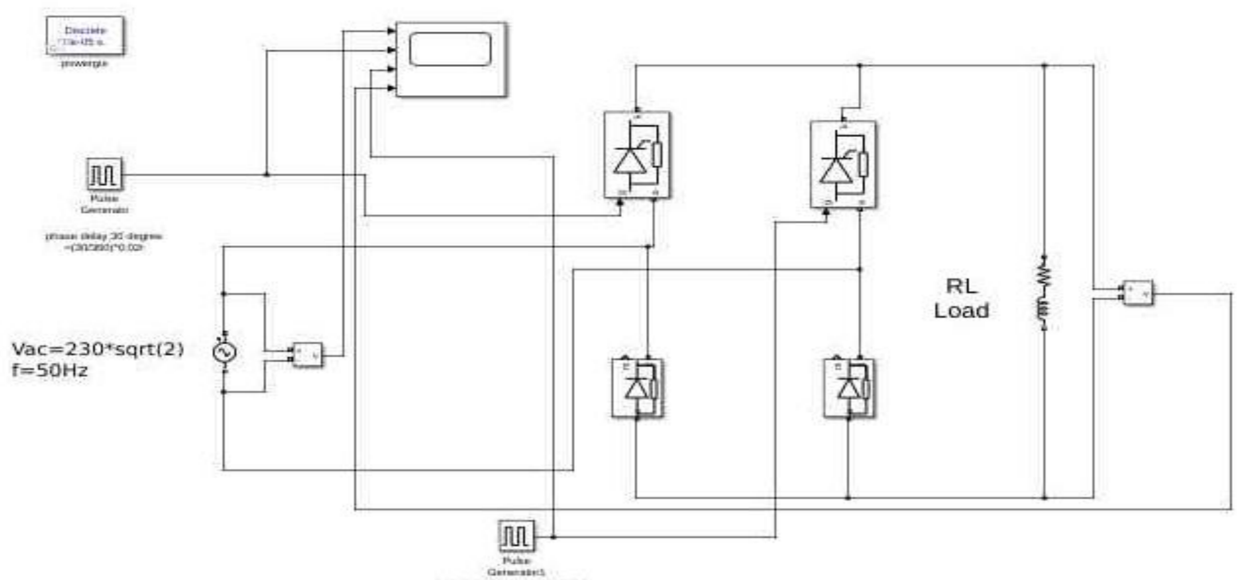
1. MATLAB Software

CIRCUIT DIAGRAM:

Single Phase Semi Controlled Bridge Rectifier With R Load



Single Phase Semi Controlled Bridge Rectifier With RL Load

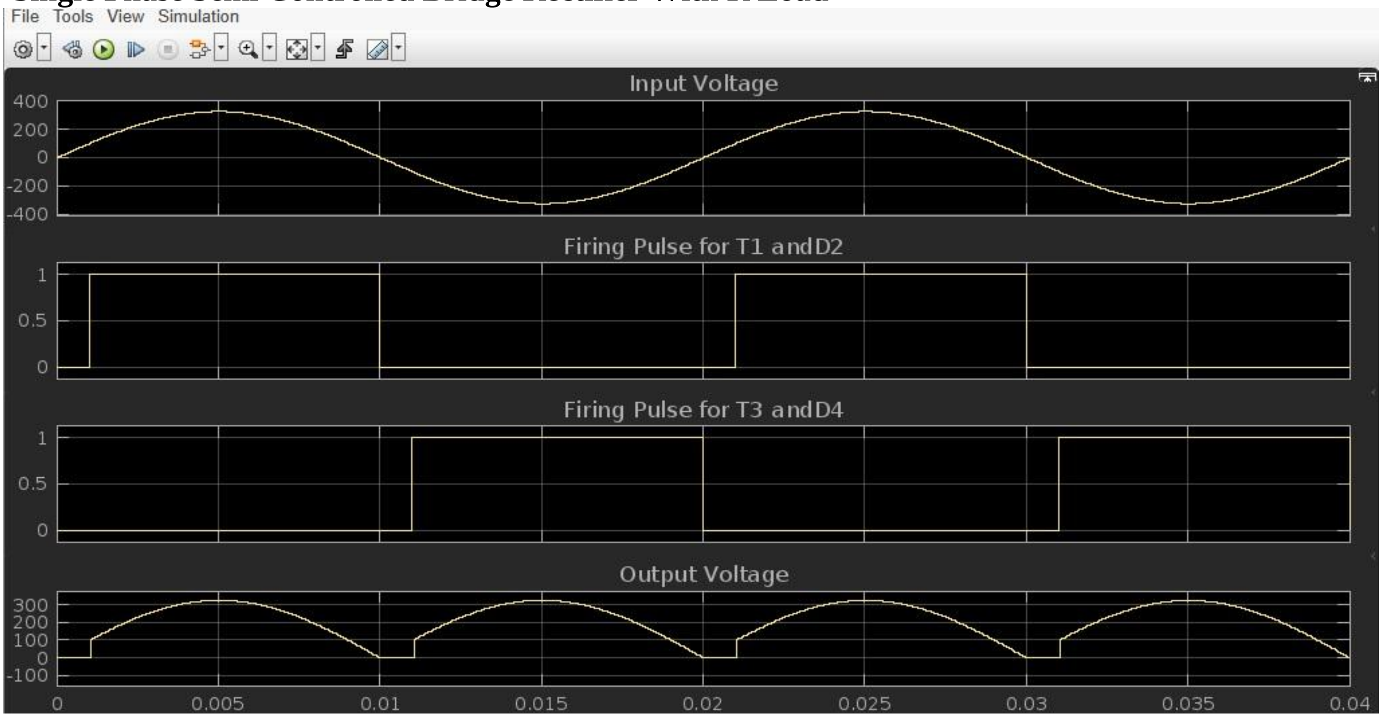


PROCEDURE:

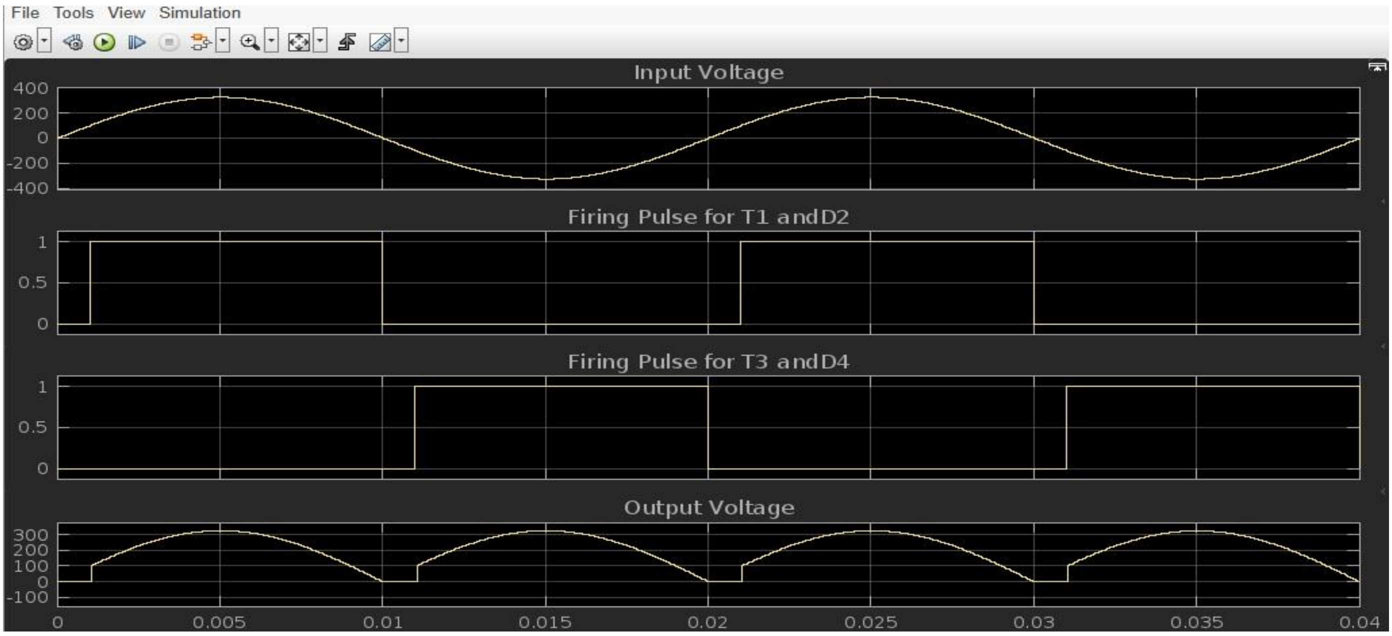
1. Click on File New Model.
2. On resulting window click on library Browser, a Simulink library browser will appear.
3. Make/Model the circuit by placing all its Blocks from its corresponding Library/toolbox, which is clearly shown in the table 1. Right click on the block to rotate mirror etc. to organize the circuit elements.
4. To get any one of the element in series RLC branch, such as R, L, or C, click on series RLC branch block in the Simpower System/Element library, and enter the parameters value in such a way that other two elements are invisible. That is to eliminate resistance, inductance, or capacitance of the branch, the R, L and C values must be set respectively to 0, 0 and inf.
5. To change the circuit parameters applicable to the block by double clicking on the block/element and type the values. Keep the values default for some blocks like thyristor, diodes, mosfet etc.
6. To measure/observe the voltage across or current passes through the electrical block/device, connect voltage measurement or current measurement blocks respectively with the electrical block,it is available on the library Simpower System/measurement.
7. To observe the waveform in figure window, scope block is connected with voltage measurement and current measurement blocks. This scope block is available by click on Library browser simulink/source scope.
8. For single phase semi controlled Bridge rectifier with RL load, components required and its parameter values are similar to that of RLE load, except E parameter. That is DC voltage source block must remove from schematic. The schematic arrangement is shown in schematic diagram 2.
9. For single phase semi controlled Bridge rectifier with R load, components required and its parameter values are similar to that of RLE load, except L and E parameters. That is L must set to 0, and DC voltage source is removed from the schematic. The schematic arrangement is shown inschematic diagram 1.
10. Before simulating the circuit, ensure that all the blocks are connected properly. The most common error encountered is ‘floating node’. This usually means that there is some problem in interconnects.
11. To run the simulation, select simulation start.
12. If any errors are reported here. Correct schematic or the simulation settings and re-run simulation.
13. To view simulation plots on simulation window, double click the scope in the schematic. The scope block corresponding to voltage measurement and current measurement blocks gives voltage and current waveforms respectively with respect to time.

OUTPUT WAVEFORMS:

Single Phase Semi Controlled Bridge Rectifier With R Load



Single Phase Semi Controlled Bridge Rectifier With RL Load



RESULT:

The simulation of SCR based semiconverter with different types of load is done in MATLAB/SIMULINK environment and the waveforms are observed

EXPERIMENT NO. 9

AIM: **AC SOURCE WITH FOUR DIODE FED RESISTIVEAND RESISTIVE-INDUCTIVE LOAD**

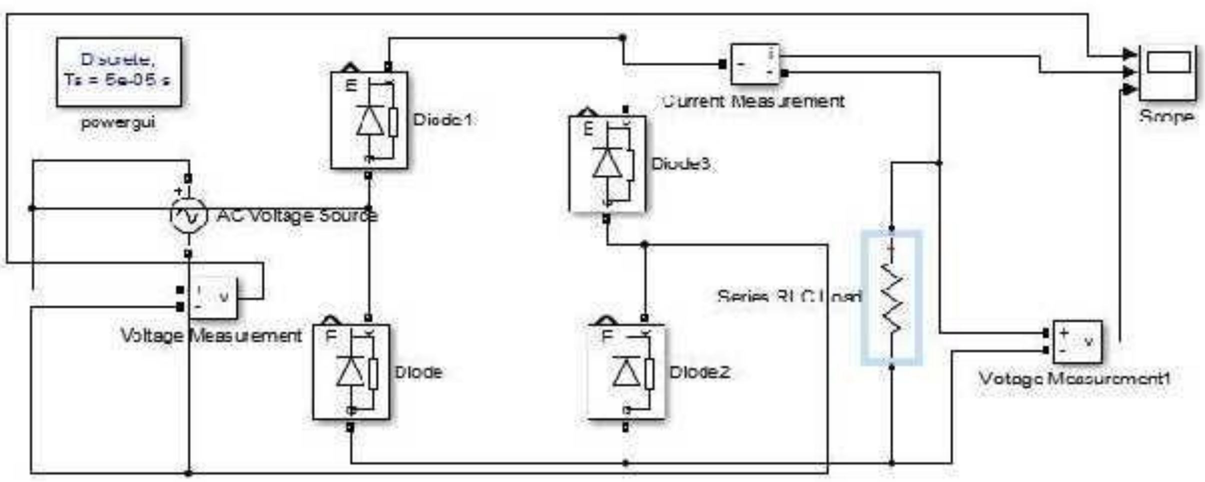
To study the simulation of Full wave Uncontrolled rectifier with R & RL-load.

APPARATUS REQUIRED:

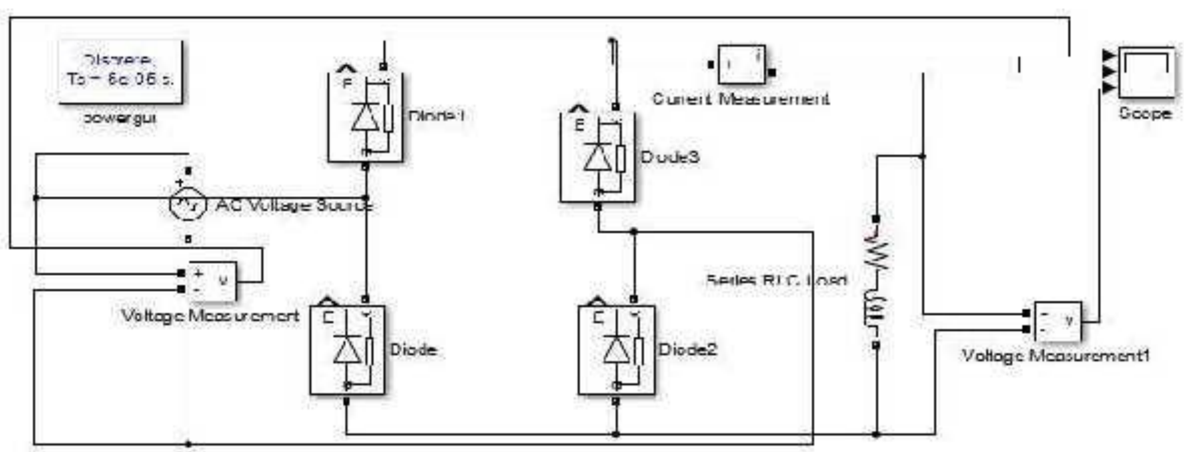
1. MATLAB Software

CIRCUIT DIAGRAM:

Full wave uncontrolled rectifier with R-load



Full wave uncontrolled rectifier with RL-load



THEORY:

A single phase full wave diode converter shown in fig with resistive and resistive-inductive load. During positive half cycle diodes D1 and D2 are forward biasedwith repect to cathode.. During negative half cycle of input voltage,diodes D3 and D4 are forward biased with respect to anode.

PROCEDURE:

1.Enter the command window of the MATLAB
2.Create a new SIMULINK Model page.
3.Develop MATLAB diagram.
4.Execute the model by pressing Run.
5. View the results in scope

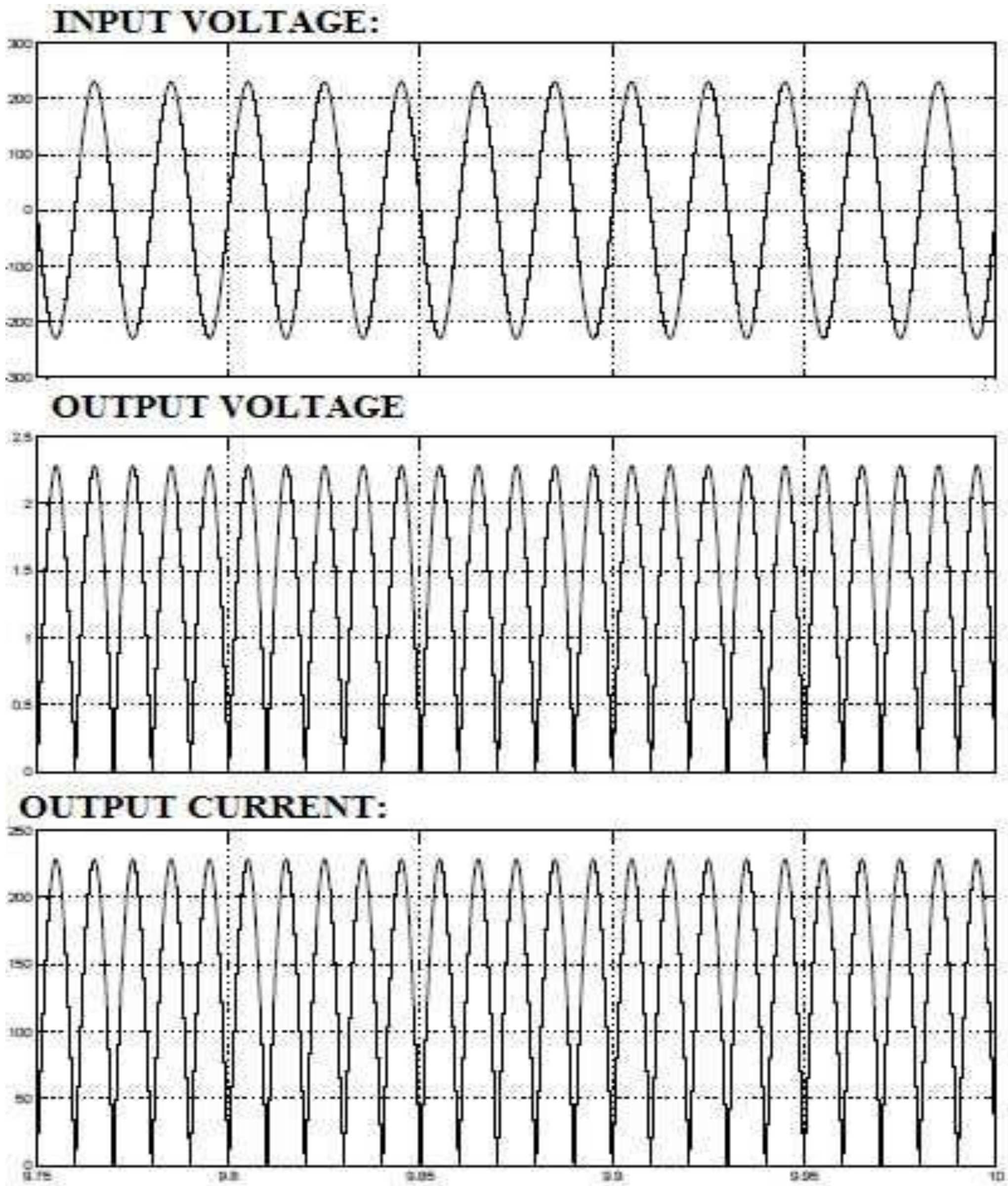
CALCULATIONS :

The average output voltage for full wave uncontrolled rectifier with R and RL load,

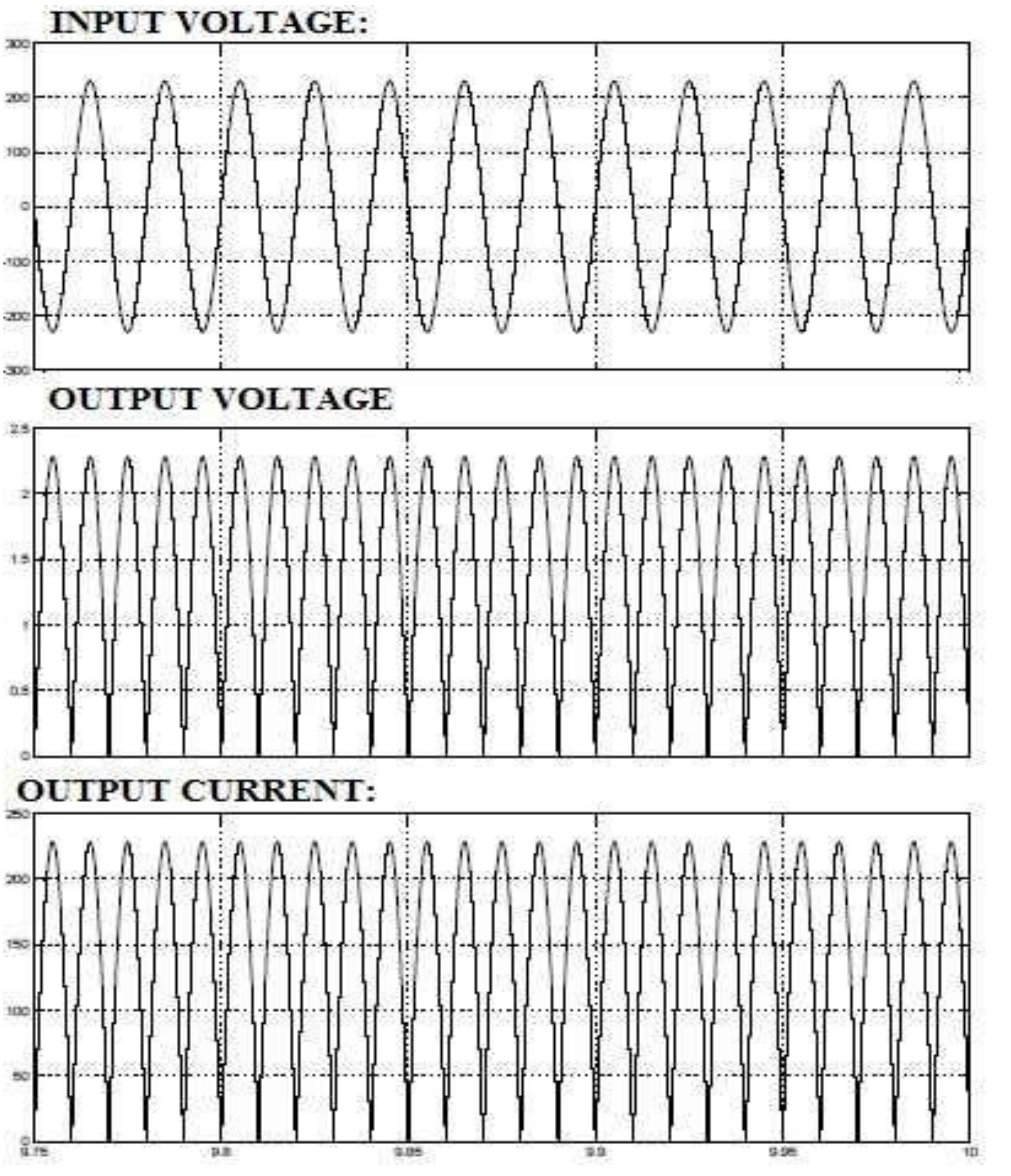
$$v_{o}=\frac{2v_m}{\pi}$$

OUTPUT WAVEFORMS:

Full wave rectifier with r load:



FullWAVERECTIFIER WITHRL LOAD:



RESULT:

The AC source with full wave uncontrolled rectifier fed Resistive and Resistive-Inductive Load was simulated using MATLAB

EXPERIMENT NO. 8

SINGLE PHASE HALF WAVE CONTROLLED RECTIFIER WITH R AND RL LOAD

AIM:

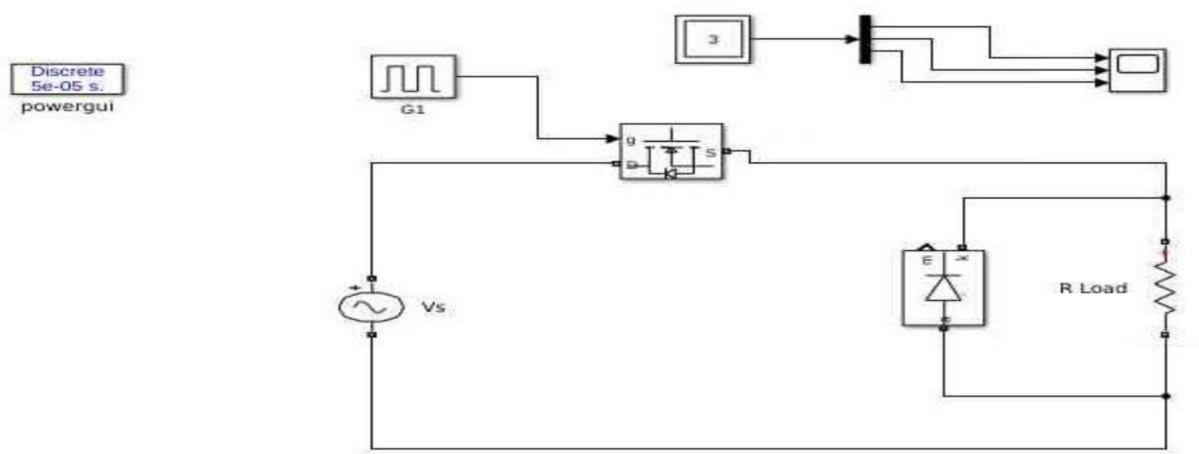
To study the simulation of half wave controlled rectifier with R & RL-load.

APPARATUS REQUIRED:

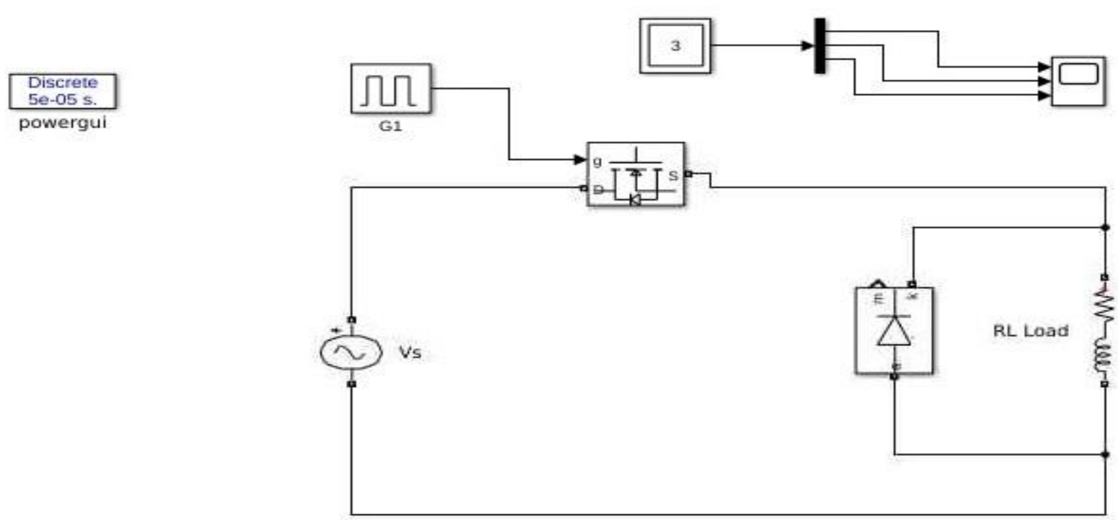
1. MATLAB Software

CIRCUIT DIAGRAM:

Half Wave Controlled Rectifier With R-Load



Half Wave Controlled Rectifier With RL-Load



THEORY:

A **single-phase half-wave controlled rectifier** is a type of rectifier circuit that uses a thyristor, typically an SCR, to convert AC power into DC power. By controlling the triggering angle of the SCR, the output DC voltage can be regulated. During the positive half-cycle of the AC input, the anode of the SCR is positive relative to the cathode. If a gate signal is applied at a specific angle (firing angle, α), the SCR conducts, allowing current to flow through the load. During the negative half-cycle, the SCR is reverse-biased and does not conduct, so no current flows through the load in this cycle.

PROCEDURE:

1.Enter the command window of the MATLAB

2.Create a new SIMULINK Model page.

3.Develop MATLAB diagram.

4.Execute the model by pressing Run.

5. View the results in scope

CALCULATIONS :

The average output voltage for half wave rectifier with R load:

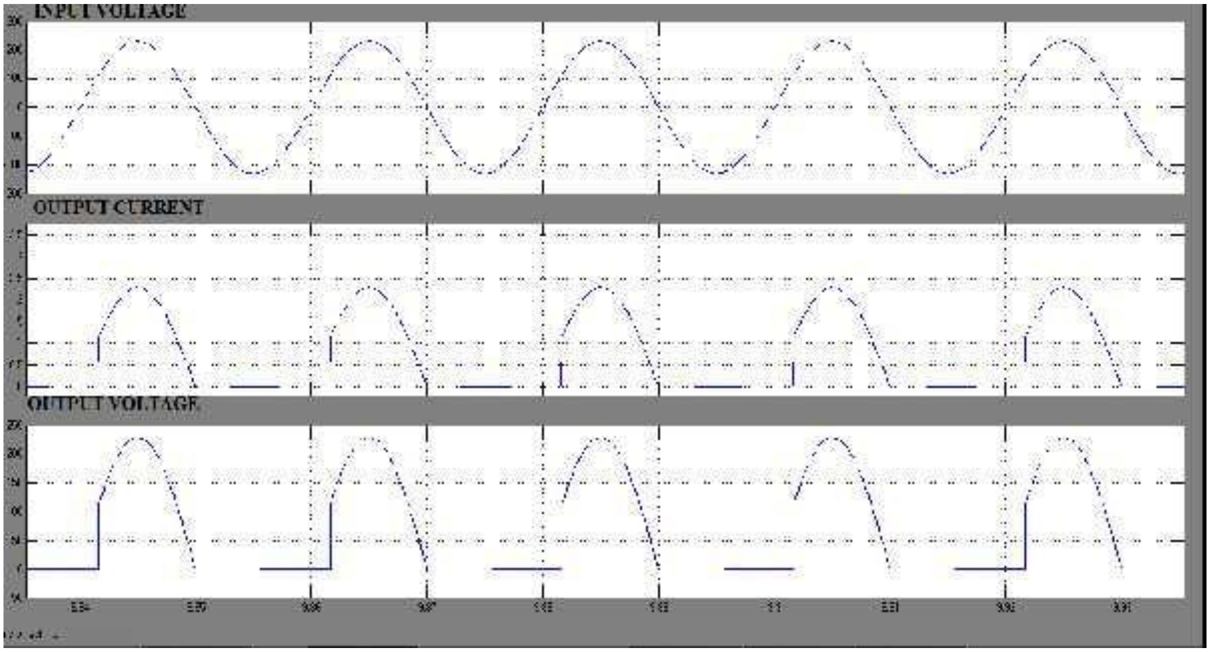
$$v0 = \frac{Vm}{\pi}(1 + cos\alpha)$$

The average output volage for half wave rectifier with RL load with freewheeling diode:

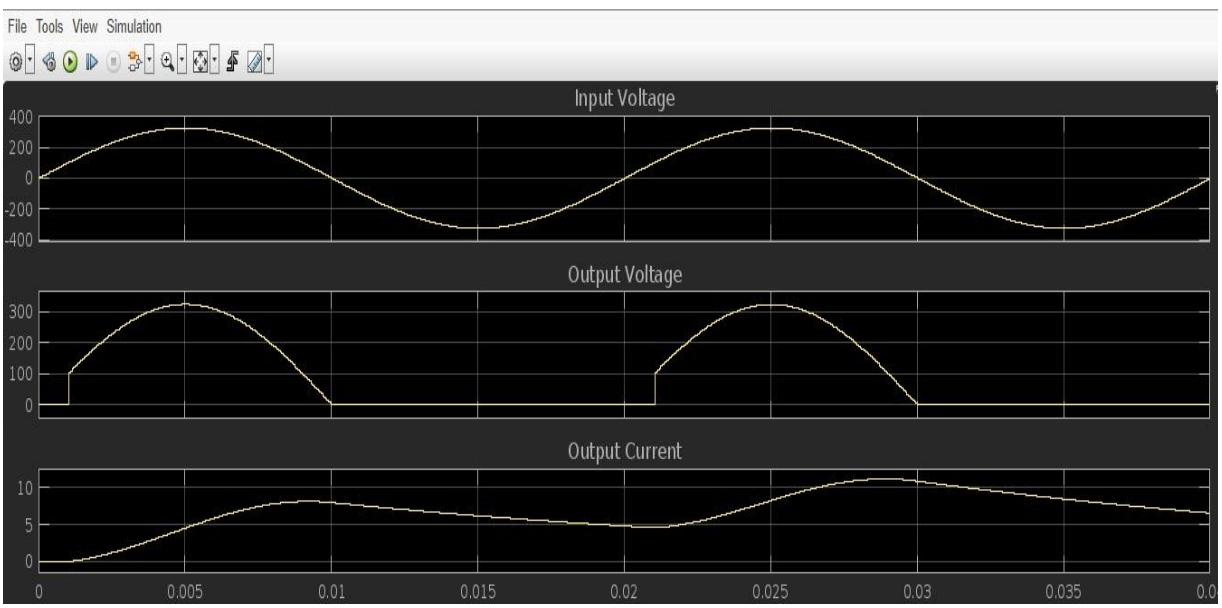
$$v0 = \frac{Vm}{\pi}(1 + cos\alpha)$$

OUTPUT WAVEFORMS:

Half wave rectifier with r load:



Half wave rectifier with RL load:



RESULT:

The single-phase half-wave controlled rectifier with resistive and resistive-inductive load was simulated by building a SIMULINK Model.

EXPERIMENT NO. 8

SINGLE PHASE HALF WAVE UNCONTROLLED RECTIFIER WITH R AND RL LOAD

AIM:

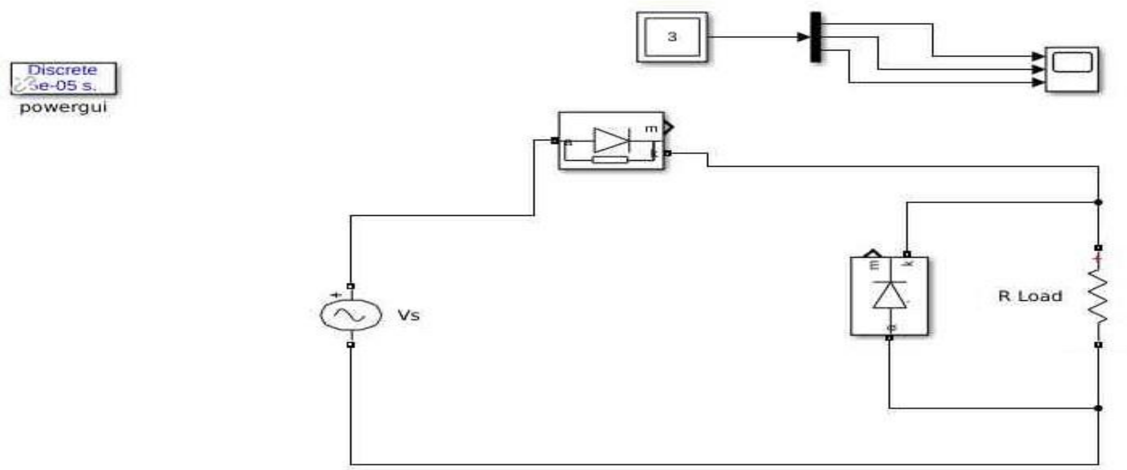
To study the simulation of half wave uncontrolled rectifier with R & RL-load.

APPARATUS REQUIRED:

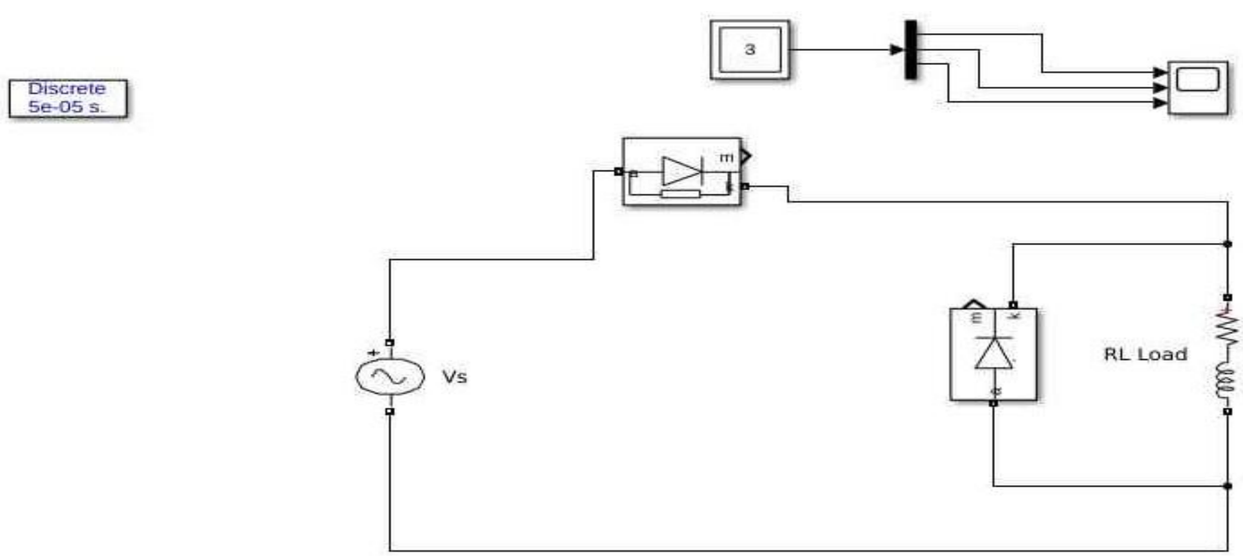
1. MATLAB Software

CIRCUIT DIAGRAM:

Half Wave Uncontrolled Rectifier With R-Load



Half Wave Uncontrolled Rectifier With RL-Load



THEORY:

A **single-phase half-wave uncontrolled rectifier** is a type of rectifier circuit that uses a diode to convert AC power into DC power. The output DC voltage cannot be regulated. During the positive half-cycle of the AC input, the anode of the diode is positive relative to the cathode therefore diode is forward-biased and does conduct, so current flows through the load in this cycle. During the negative half-cycle, the diode is reverse-biased and does not conduct, so no current flows through the load in this cycle.

PROCEDURE:

1.Enter the command window of the MATLAB

2.Create a new SIMULINK Model page.

3.Develop MATLAB diagram.

4.Execute the model by pressing Run.

5. View the results in scope

CALCULATIONS :

The average output voltage for half wave rectifier with R load:

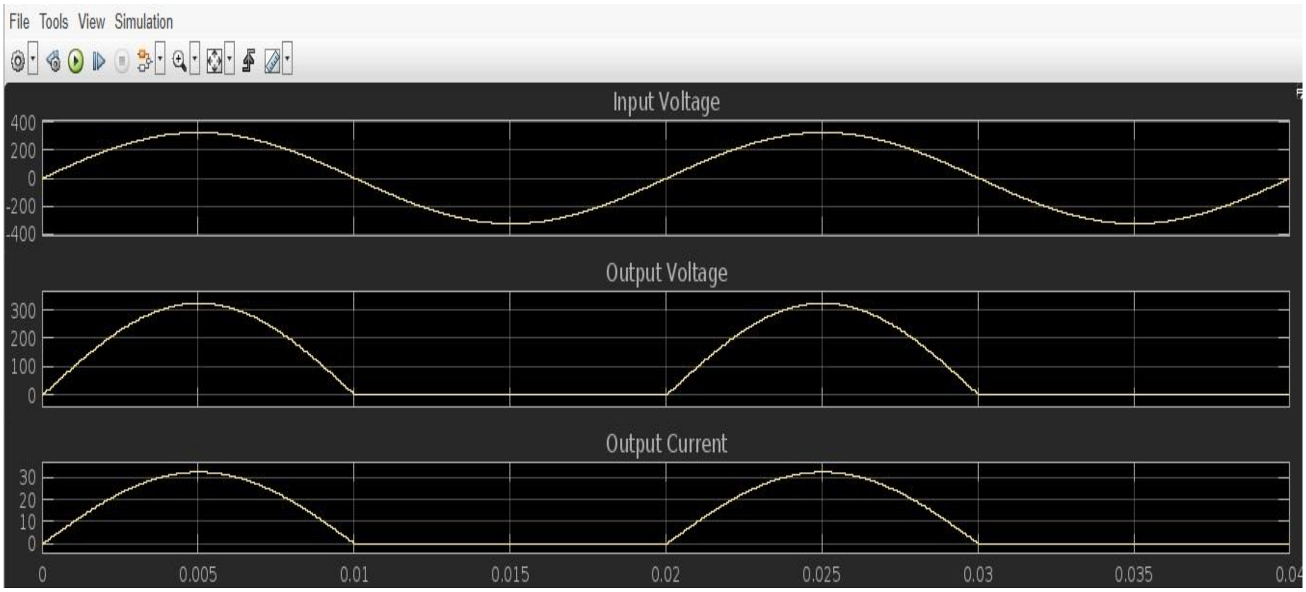
$$v0 = \frac{Vm}{\pi}$$

The average output volage for half wave rectifier with RL load with freewheeling diode:

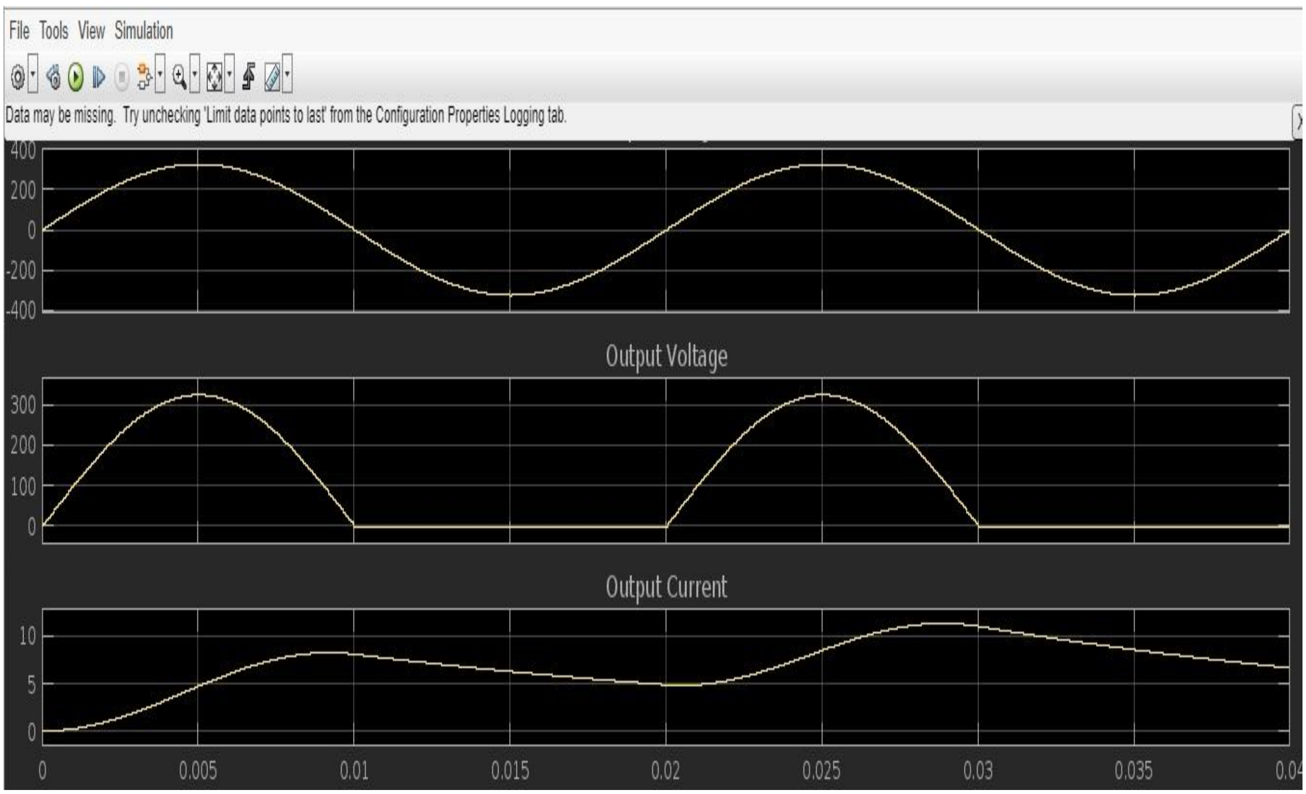
$$v0 = \frac{Vm}{\pi}$$

OUTPUT WAVEFORMS:

Half wave rectifier with R load:



Half wave rectifier with RL load:



RESULT:

The single-phase half-wave uncontrolled rectifier with resistive and resistive-inductive load was simulated by building a SIMULINK Model.

SIMULATION OF BUCK BOOST CONVERTER

AIM:

To construct DC to DC buck boost converter simulation using MATLAB.

APPARATUS:

S. No	SOFTWARE USED	DESKTOP QUANTITY
1	MATLAB	10

CIRCUIT DIAGRAM:

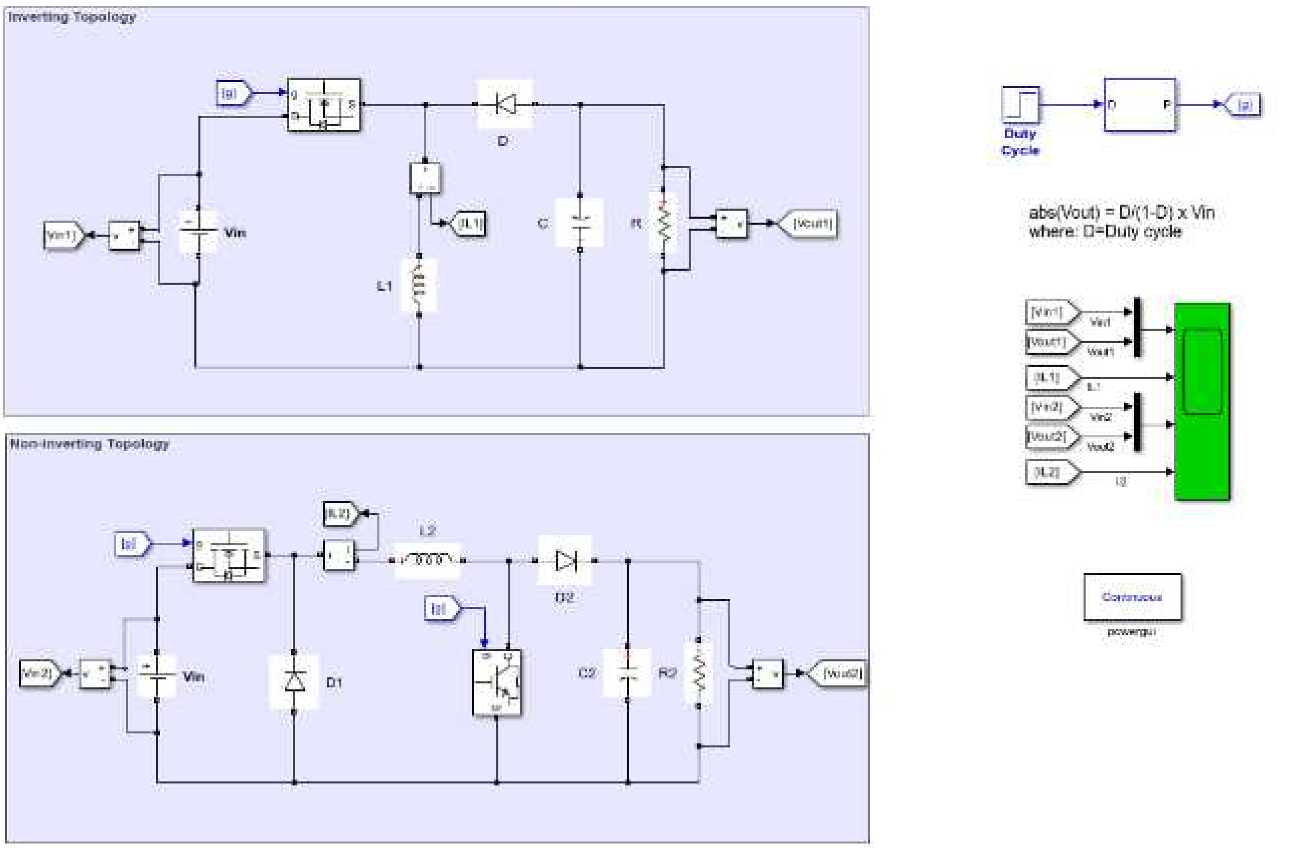
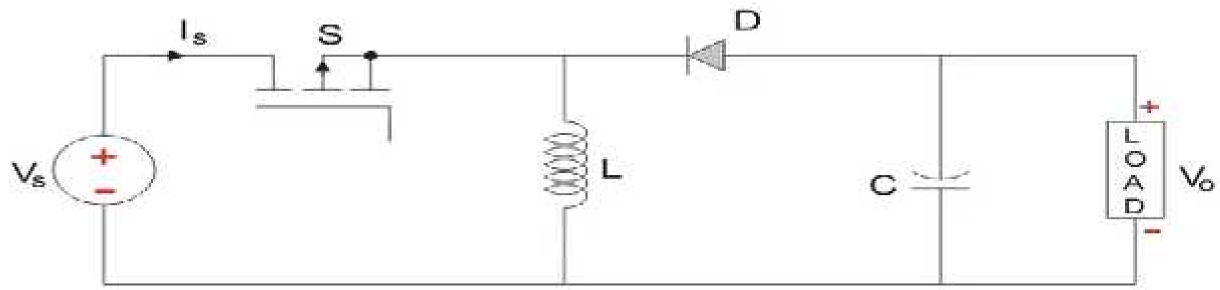


Fig .BuckBoost Converter simulation using MATLAB

DESCRIPTION:

The buck–boost converter is a type of DC-to-DC converter (also knownas a chopper) that has an output voltage magnitude that is either greater than or less than the input voltage magnitude. It is used to “step up” the DC voltage, similar to a transformer for AC circuits. It is equivalent to a flyback converter using a single inductor instead of a transformer. Two different topologies are called buck–boost converter.

Buck Boost converter which can operate as a DC-DC Step-Down converter or a DC-DC Step-Up converter depending upon the duty cycle, D. A typical Buck-Boost converter is shown below.

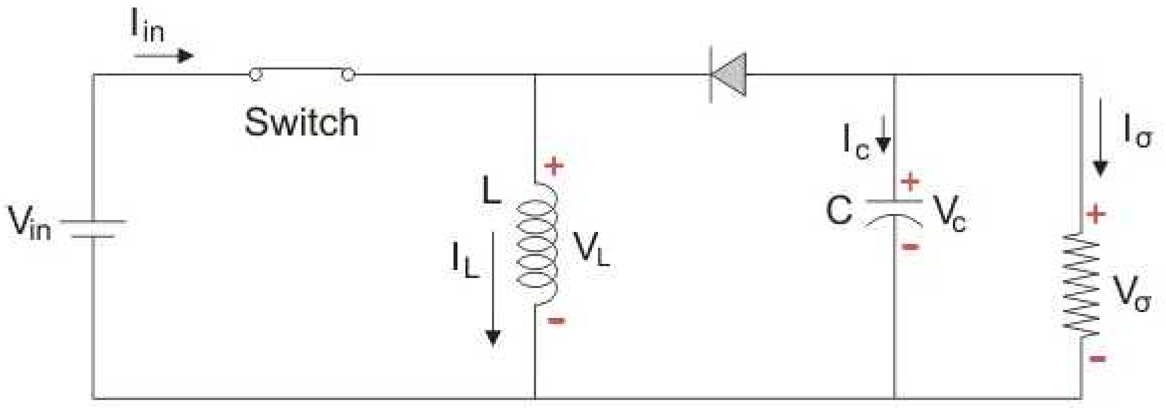


The input voltage source is connected to a solid state device. The second switch used is a diode. The diode is connected, in reverse to the direction of power flow from source, to a capacitor and the load and the two are connected in parallel as shown in the figure above.

The controlled switch is turned on and off by using Pulse Width Modulation (PWM). PWM can be time based or frequency based.

Mode I : Switch is ON, Diode is OFF

The Switch is ON and therefore represents a short circuit ideally offering zero resistance to the flow of current so when the switch is ON all the current will flow through the switch and the inductor and back to the DC input source.



The inductor stores charge during the time the switch is ON and when the solid state switch is OFF the polarity of the Inductor reverses so that current flows through the load and through the diode and back to the inductor. So the direction of current through the inductor remains the same.

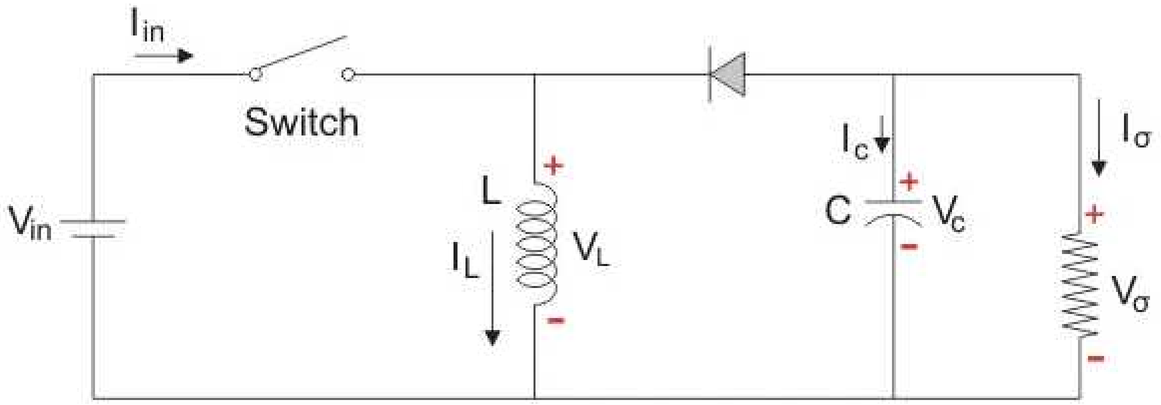
$$\therefore V_{in} = V_L$$
$$\therefore V_L = L\frac{di_L}{dt} = V_{in}$$
$$\frac{di_L}{dt} = \frac{\Delta i_L}{\Delta t} = \frac{\Delta i_L}{DT} = \frac{V_{in}}{L}$$

Since the switch is closed for a time $T_{ON} = DT$ we can say that $\Delta t = DT$.

$$(\Delta i_L)_{closed} = \left(\frac{V_{in}}{L}\right) DT$$

Mode II : Switch is OFF, Diode is ON

In this mode the polarity of the inductor is reversed and the energy stored in the inductor is released and is ultimately dissipated in the load resistance and this helps to maintain the flow of current in the same direction through the load and also step-up the output voltage as the inductor is now also acting as a source in conjunction with the input source.



$$\therefore V_L = V_o$$

$$\therefore V_L = L\frac{di_L}{dt} = V_o$$

$$\frac{di_L}{dt} = \frac{\Delta i_L}{\Delta t} = \frac{\Delta i_L}{(1-D)T} = \frac{V_o}{L}$$

Since the switch is open for a time

$$T_{OFF} = T - T_{ON} = T - DT = (1-D)T$$

we can say that.

$$\Delta t = (1-D)T$$

$$(\Delta i_L)_{open} = \left(\frac{V_o}{L}\right)(1-D)T$$

It is already established that the net change of the inductor current over any one complete cycle is zero.

$$\therefore (\Delta i_L)_{closed} + (\Delta i_L)_{open} = 0$$

$$\left(\frac{V_o}{L}\right)(1-D)T + \left(\frac{V_{in}}{L}\right)DT = 0$$

$$\frac{V_o}{V_{in}} = \frac{-D}{1-D}$$

We know that D varies between 0 and 1. If D > 0.5, the output voltage is larger than the input; and if D < 0.5, the output is smaller than the input. But if D = 0.5 the output voltage is equal to the input voltage.

SIMULATION:

The buck boost converter is a DC/DC converter with the output voltage magnitude that is either greater than or less than the input voltage magnitude. It is comparable to a flyback converter where an inductor is used in place of a transformer. The theoretical transfer function of the buck boost converter is:

$$(Vout) = D/(1-D) * Vin$$

an output voltage that is of the opposite polarity as the input voltage. The output voltage is determined by the duty cycle of the MOSFET transistor.

PROCEDURE:

1. Make the connections as shown in the diagram by using MATLAB Simulink
2. Enter the rated values of the particular element
3. Draw the respective wave forms

RESULT:

Thus the Buck Boost converter simulation using MATLAB was constructed.

EXPERIMENT NO. 5

SIMULATION OF BOOST CONVERTER

AIM:

To construct DC to DC step up chopper simulation using MATLAB.

APPARATUS:

S. No	SOFTWARE USED	DESKTOP QUANTITY
1	MATLAB	10

CIRCUIT DIAGRAM:

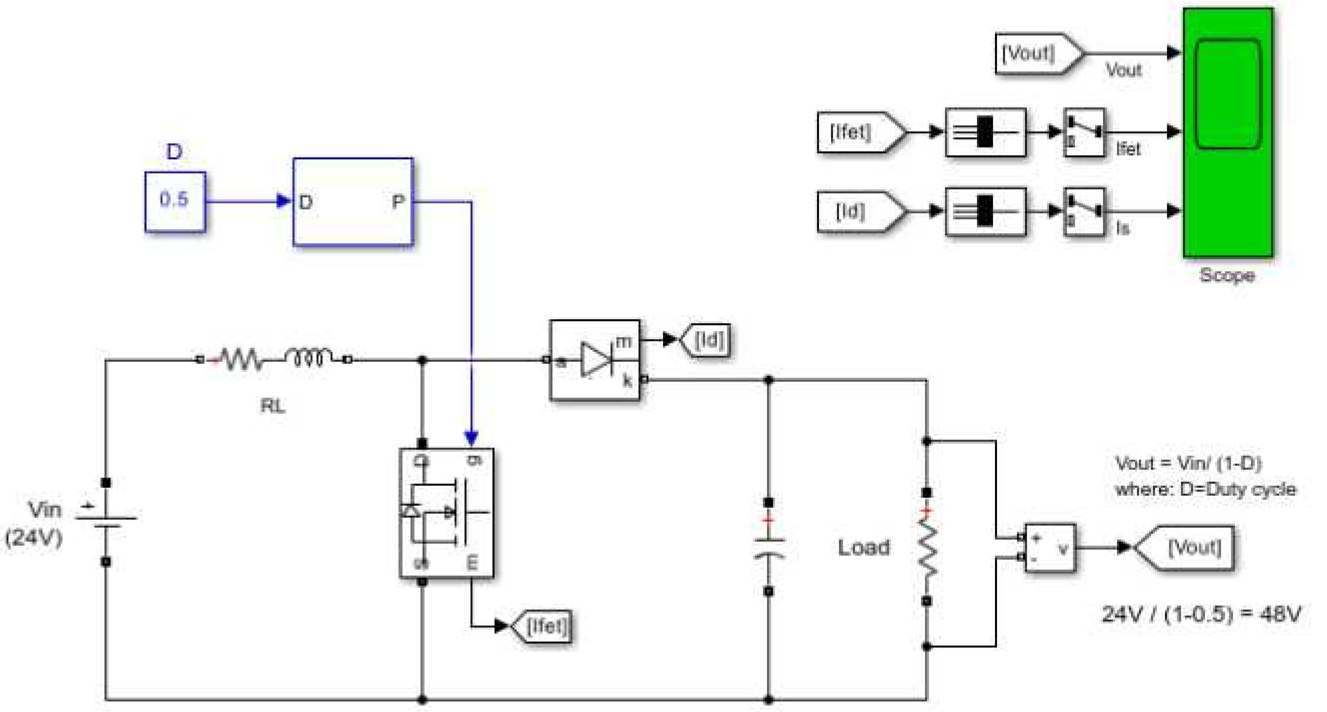


Fig . Boost Converter simulation using MATLAB

DESCRIPTION:

A boost converter's output voltage is always higher than the input voltage. Figure 1 shows the schematics of a boost converter. It has a dc input voltage, a transistor working as a switch, an inductor and a capacitor forming a low pass filter to smooth out the output voltage, and a load resistor. The diode provides a path for the inductor current when the switch is opened and is reverse biased when the switch is closed.

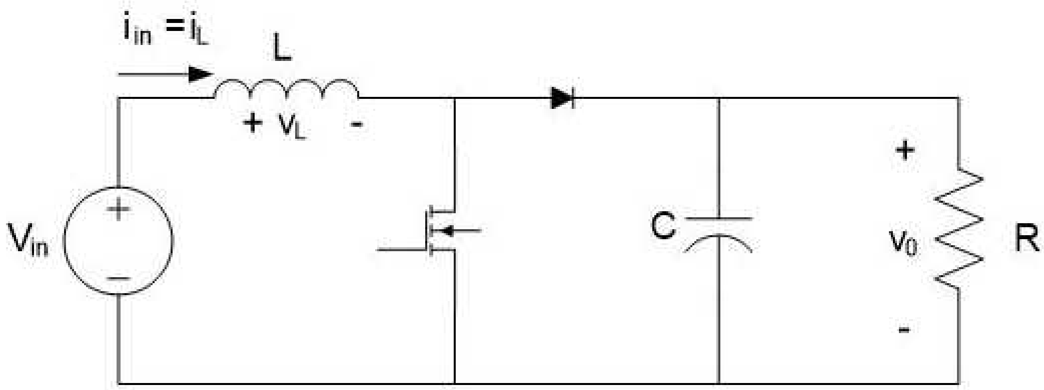


Figure 1. Schematics of a boost converter

When the switch is closed, the diode is reverse biased. The equivalent circuit is shown in Figure 2.

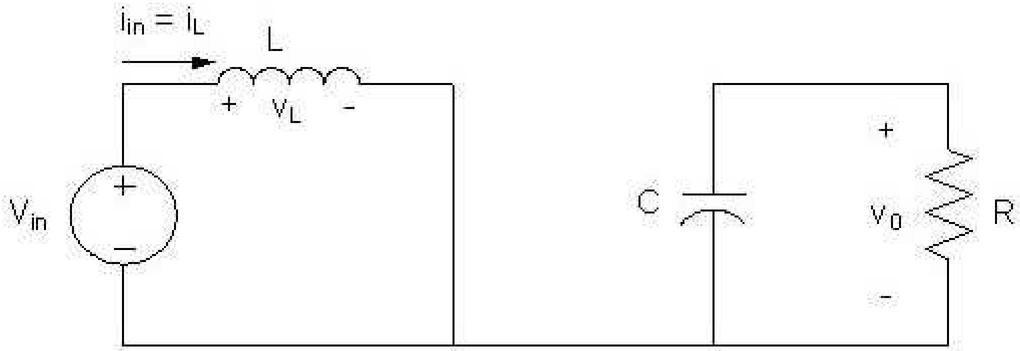


Figure 2. Equivalent circuit when the switch is closed

When the switch is open, the diode is forward biased. The equivalent circuit is shown in Figure 3.

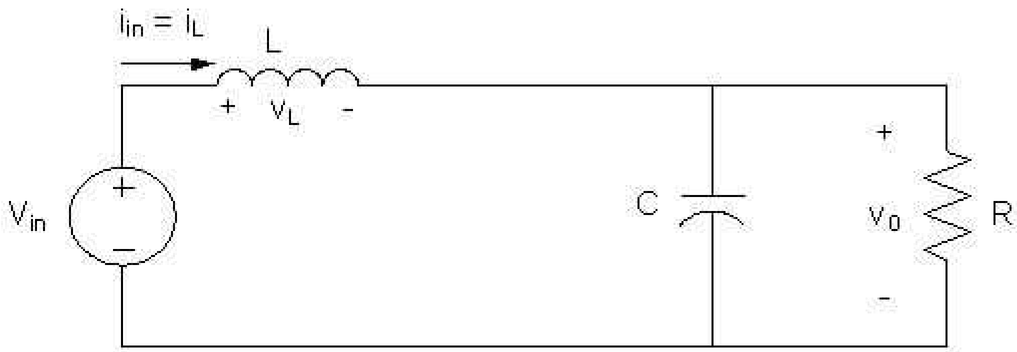


Figure 3. Equivalent circuit when the switch is open

Output voltage of a boost converter is controlled by a pulse width modulated (PWM) signal shown in figure 4. The duty cycle is the ratio between the on time and the switching period shown in (1). By adjusting the duty cycle, we can obtain desired output voltage, which is shown in (2). When D increases, the output voltage increases. When D decreases, the output voltage decreases.

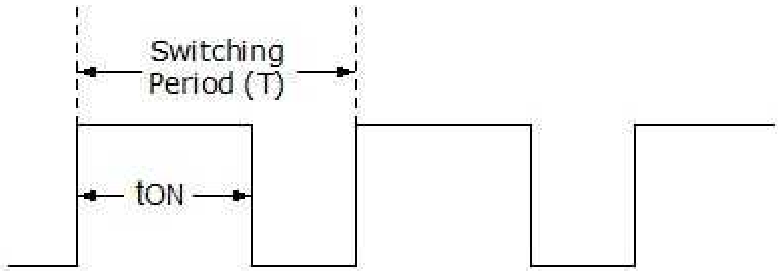


Figure 4. Pulse width modulated (PWM) Signal

$$D = \frac{T_{on}}{T} \quad (1)$$

$$\frac{V_o}{V_{in}} = \frac{1}{1-D} \quad (2)$$

If the inductor current is continuous, the boost converter is operating in continuous conduction mode. If the inductor current is reduced to zero, the boost converter is operating in discontinuous mode. We would like to keep the boost converter operating in continuous conduction mode. The minimum inductance to keep the boost converter in continuous conduction mode is the critical inductance. Value of the critical inductance is shown in (3).

$$L_{min} = \frac{D(1-D)^2 R}{2f} \quad (3)$$

Average inductor current is the same as the average load current in the load resistor. Value of the average inductor current is shown in (4).

$$I_L = I_R = \frac{V_{in}}{(1-D)^2 R} \quad (4)$$

The peak-to-peak inductor current is often a design criterion in the design of a boost converter. It is also called the current ripple. Equation (5) shows the value of the current ripple. The current ripple is inversely proportional to the inductance and the switching frequency. To reduce the current ripple, we can increase the inductance or the switching frequency.

$$\Delta i_L = \frac{V_{in} D T}{L} \quad (5)$$

The maximum inductor current is the average inductor current plus half of the current ripple, shown in (6). The minimum inductor current is the average inductor current minus half of the current ripple, shown in .

$$I_{\max} = I_L + \frac{\Delta i_L}{2} = \frac{V_{in}}{(1-D)^2 R} + \frac{V_{in} DT}{2L} \qquad (6)$$

$$I_{\min} = I_L - \frac{\Delta i_L}{2} = \frac{V_{in}}{(1-D)^2 R} - \frac{V_{in} DT}{2L} \qquad (7)$$

SIMULATION:

A boost converter is a DC/DC power converter which steps up voltage from its input (source) to its output (load). In continuous conduction mode (current through the inductor never falls to zero), the theoretical transfer function of the boost converter is:

$$Vout/Vin = 1/(1-D)$$

In this example, the converter is feeding an RC load from a 24 V source and the PWM frequency is set to 20 kHz.

Run the simulation and observe waveforms on Scope. Verify that the mean value of the load voltage (Vout) is very close to the theoretical value of: Vout = 24 / (1 - 0.5) = 48 V.

PROCEDURE:

1. Make the connections as shown in the diagram by using MATLAB Simulink
2. Enter the rated values of the particular element
3. Draw the respective wave forms

RESULT:

Thus the Boost converter simulation using MATLAB was constructed.

EXPERIMENT No. 4

SIMULATION OF BUCK CONVERTER

AIM:

To construct DC to DC step down chopper simulation using MATLAB.

APPARATUS:

S. No	SOFTWARE USED	DESKTOP QUANTITY
1	MATLAB	10

CIRCUIT DIAGRAM:

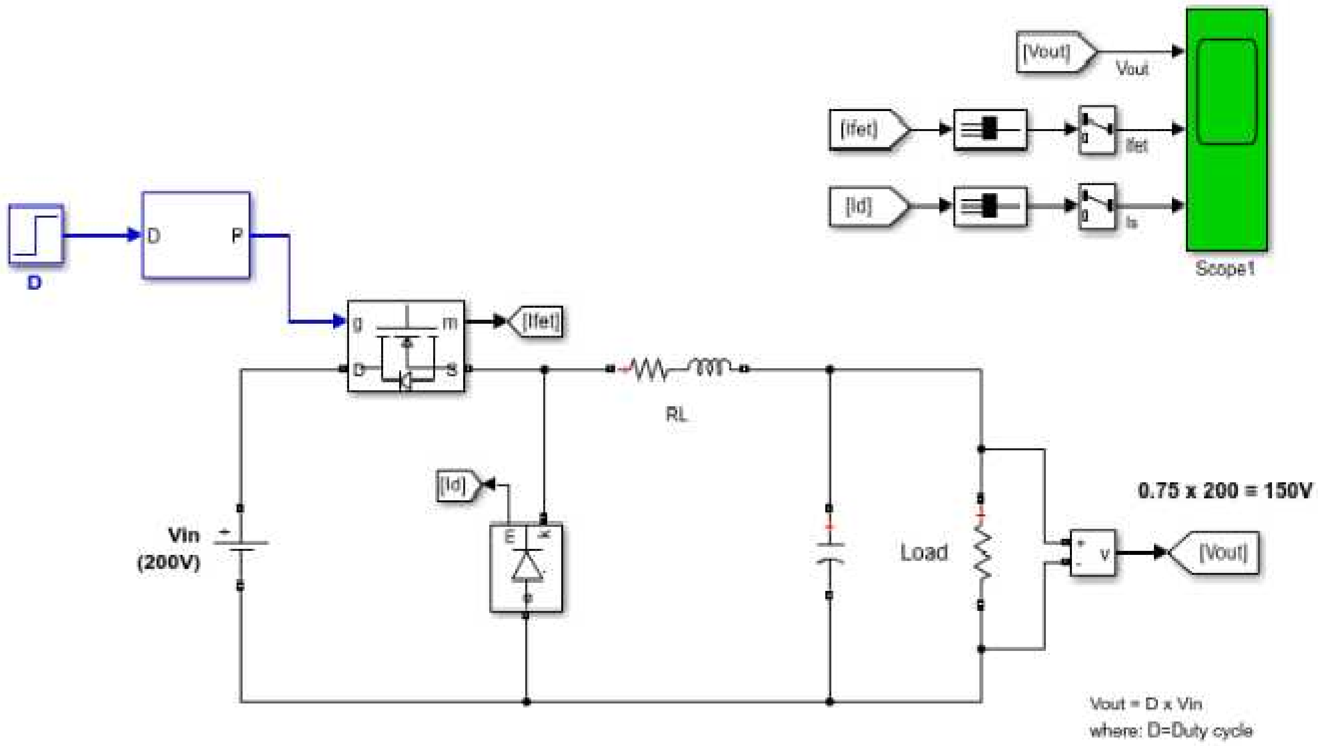


Fig .Buck Converter simulation using MATLAB

DESCRIPTION:

A buck converter's output voltage is always lower than the input voltage. Figure 2 shows the schematics of a buck converter. It has a dc input voltage, a transistor working as a switch, an inductor and a capacitor forming a low pass filter to smooth out the output voltage, and a load resistor. The diode provides a path for the inductor current when the switch is opened and is reverse biased when the switch is closed.

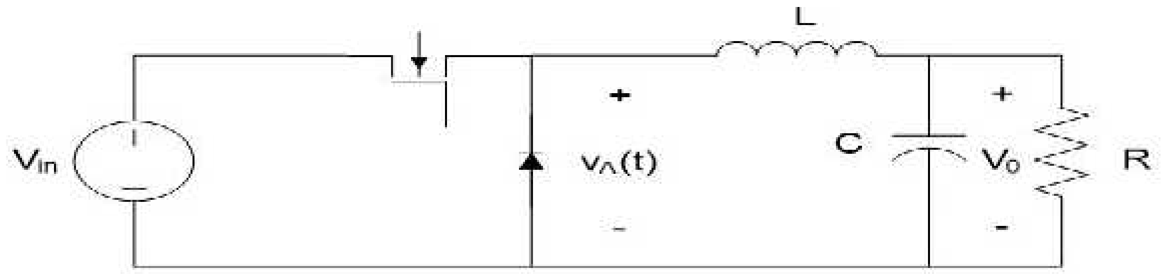


Figure 2. Schematics of a buck converter

When the switch is closed, the diode is reverse biased. The equivalent circuit is shown in Figure 3.

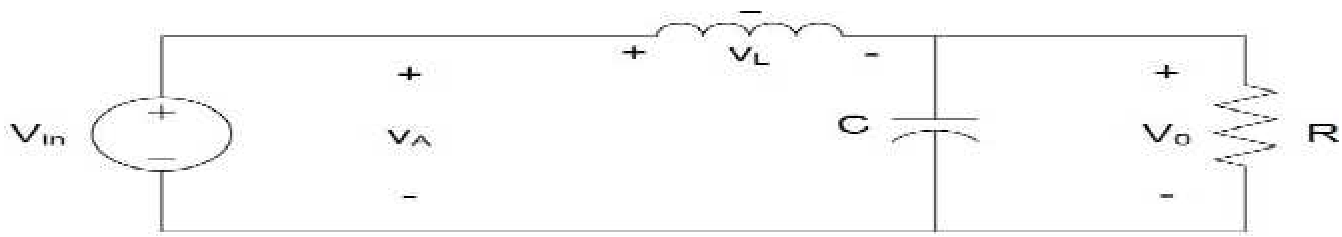


Figure 3. Equivalent circuit when the switch is closed

When the switch is open, the diode is forward biased. The equivalent circuit is shown in Figure 4.

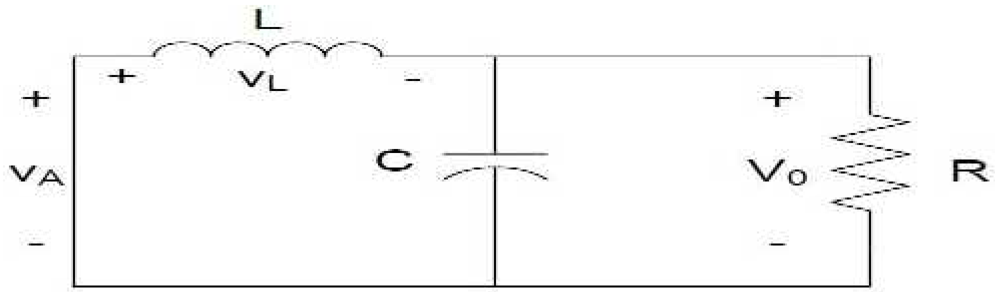


Figure 4. Equivalent circuit when the switch is open

Output voltage of a buck converter is controlled by a pulse width modulated (PWM) signal. The duty cycle is the ratio between the on time and the switching period shown in (1). By adjusting the duty cycle, we can obtain desired output voltage, which is shown in (2). When D increases, the output voltage increases. When D decreases, the output voltage decreases.

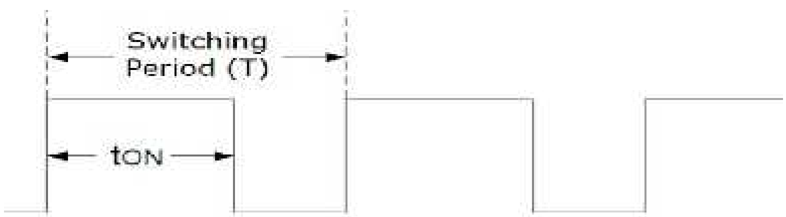


Figure 5. Duty Cycle

If the inductor current is continuous, the buck converter is operating in continuous conduction mode. If the inductor current is reduced to zero, the buck converter is operating in discontinuous mode. We would like to keep the buck converter operating in continuous conduction mode. The minimum inductance to keep the buck converter in continuous conduction mode is the critical inductance. Value of the critical inductance is shown in

$$D = \frac{T_{on}}{T} \tag{1}$$

$$\frac{V_o}{V_{in}} = D \tag{2}$$

$$L_{min} = \frac{(1-D)R}{2f} \tag{3}$$

Average inductor current is the same as the average load current in the load resistor. Value of the average inductor current is shown in (4).

$$I_L = I_R = \frac{V_O}{R} \tag{4}$$

The peak-to-peak inductor current is often a design criterion in the design of a buck converter. It is also called the current ripple. Equation (5) shows the value of the current ripple. The current ripple is inversely proportional to the inductance and the switching frequency. To reduce the current ripple, we can increase the inductance or the switching frequency.

$$\Delta i_L = \frac{V_O(1-D)}{Lf} \tag{5}$$

SIMULATION:

Run the simulation and observe waveforms on Scope1. At 3.5 ms, the duty cycle is increased from 0.5 to 0.75. Verify that the mean value of the load voltage (Vout) at the end of the simulation is very close to the theoretical value of: 0.75 * 200 = 150 V.

PROCEDURE:

1. Make the connections as shown in the diagram by using MATLAB Simulink
2. Enter the rated values of the particular element
3. Draw the respective wave forms

RESULT:

Thus the Buck converter simulation using MATLAB was constructed.

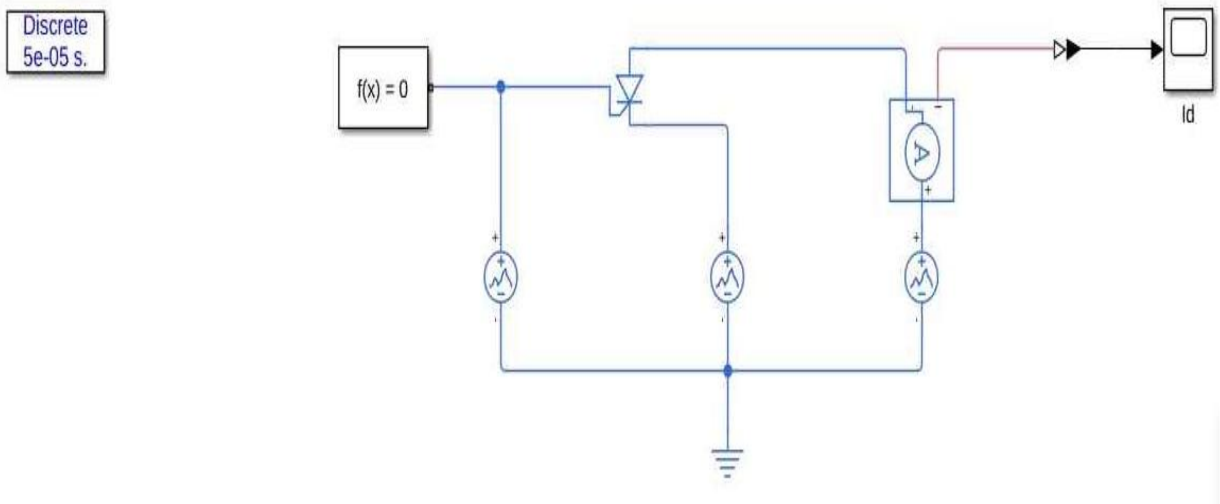
EXPERIMENT NO. 3

STUDY OF CHARACTERISTICS OF SCR

AIM: To plot the characteristics of SCR

APPARATUS REQUIRED: MATLAB Software

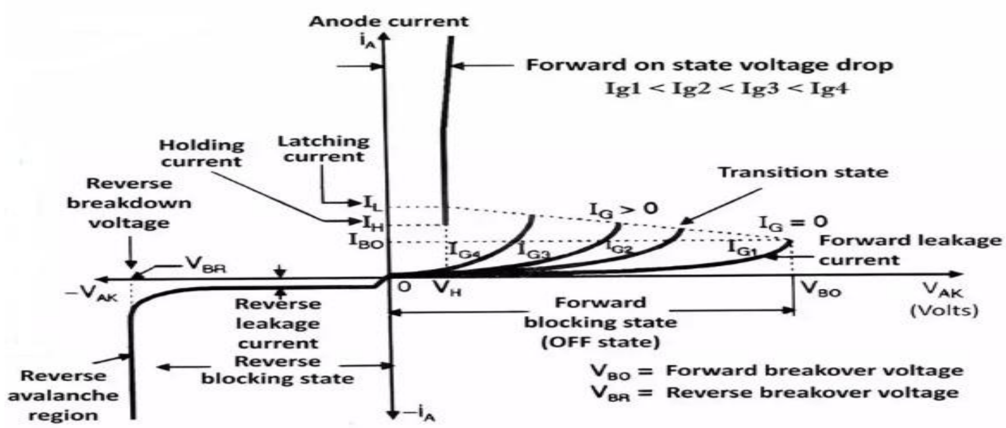
CIRCUIT DIAGRAM:



SCR Characteristics

THEORY: The **V-I characteristics** of a **Silicon Controlled Rectifier (SCR)** describe the relationship between the **anode-cathode voltage (V_{AK})** and the **anode current (I_A)** for different gate currents. The V-I characteristics can be divided into the following regions:

- **Forward Blocking Region**: SCR is off, blocking current flow with minimal leakage.
- **Forward Conduction Region**: SCR is on, conducting high current after being triggered by the gate.
- **Reverse Blocking Region**: SCR blocks reverse current, acting like a diode in reverse bias.



V-I Characteristics of SCR

PROCEDURE:

1. Enter into the command window of the MATLAB.
2. Open a new SIMULINK Model page.
3. Develop MATLAB diagram.
4. For VI characteristics, set V_{GK} at some particular value. Provide a range of values of V_{AK}.
5. Execute the model by pressing Run button.
6. View the results in Scope.
7. Store the data of Scope in Workspace.
8. Repeat steps 4 to 7 for different values of V_{GK}.
9. Plot all the results in same graph.

RESULT: The VI characteristics of SCR have been simulated successfully by building a SIMULINK Model.

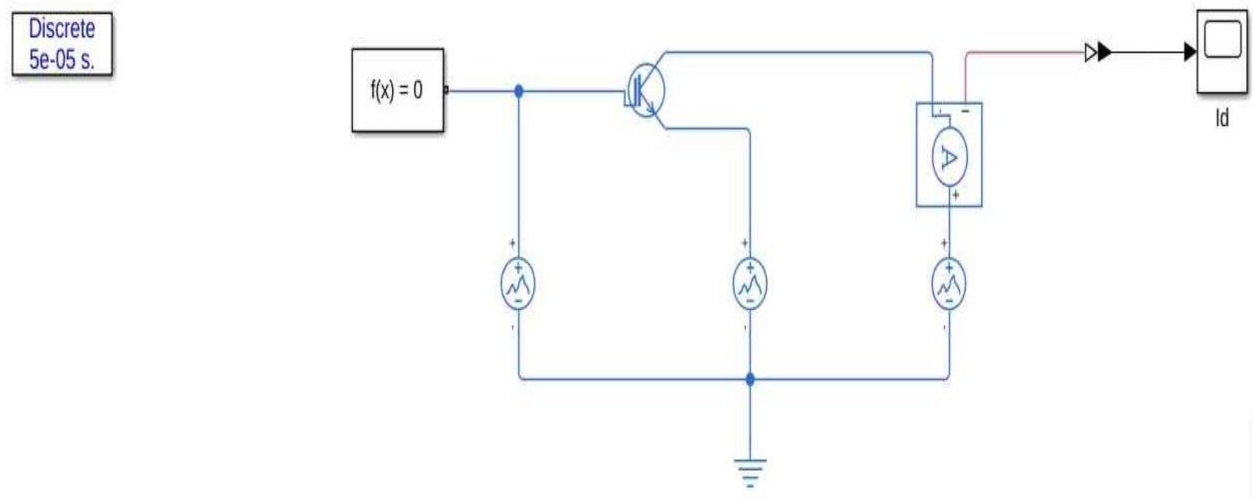
EXPERIMENT NO. 2

STUDY OF CHARACTERISTICS OF IGBT

AIM: To plot the characteristics of IGBT

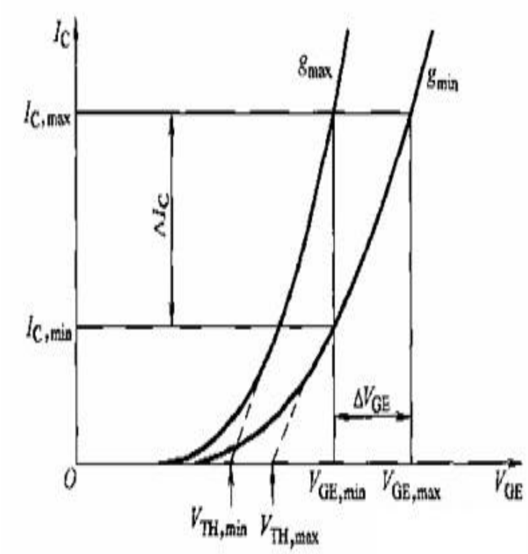
APPARATUS REQUIRED: MATLAB Software

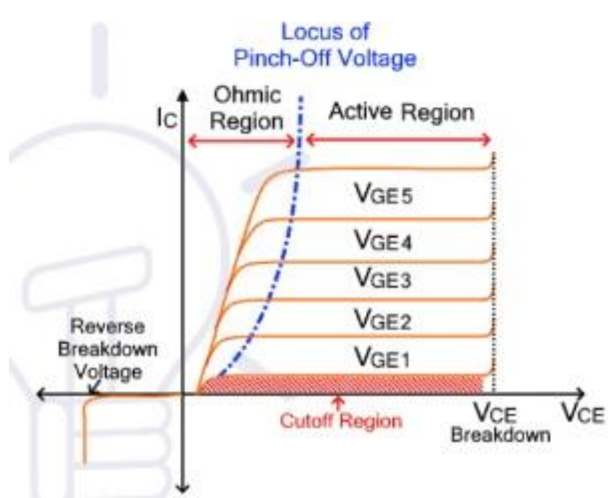
CIRCUIT DIAGRAM:



IGBT Characteristics

THEORY: The **transfer characteristics** of a IGBT show the relationship between the **gate-emitter voltage (V_{GE})** and the **collector current (I_C)** for various fixed values of **collector-emitter voltage (V_{CE})**. The **output characteristics** of a IGBT show the relationship between the **collector-emitter voltage (V_{CE})** and the **collector current (I_c)** for various fixed values of **gate-emitter voltage (V_{GE})**.





PROCEDURE:

1. Enter into the command window of the MATLAB.
2. Open a new SIMULINK Model page.
3. Develop MATLAB diagram.
4. For transfer characteristics, set V_{CE} at some particular value. Provide a range of values of V_{GE}.
5. Execute the model by pressing Run button.
6. View the results in Scope.
7. Store the data of Scope in Workspace.
8. Repeat steps 4 to 7 for different values of V_{CE}.
9. Plot all the results in same graph.
10. For output characteristics, set V_{GE} at some particular value. Provide a range of values of V_{CE}.
11. Execute the model by pressing Run button.
12. View the results in Scope.
13. Store the data of Scope in Workspace.
14. Repeat steps 10 to 13 for different values of V_{GE}.
15. Plot all the results in same graph.

RESULT: The transfer and output characteristics of IGBT have been simulated successfully by building a SIMULINK Model.