DEPARTMENT
OF
ELECTRICAL AND ELECTRONICS ENGINEERING

ELECTRICAL SYSTEMS SIMULATION LAB

2022

G. Narayanamma Institute of Technology & Science (for Women)

(Autonomous)

Shaikpet, Hyderabad – 500 104

G. Narayanamma Institute of Technology & Science (For Women)

Department of Electrical and Electronics Engineering
ELECTRICAL SYSTEMS SIMULATION LAB
List of Experiments

1. Simulation of buck, boost and buck-boost converter with open loop operation.

2. Single Phase Inverter using PWM controller with RL load.

3. Three phase fully controlled Converter using RL &RLE loads.

4. Three phase Inverter with SPWM Controller.

5. Simulation of z-source inverter.

6. Simulation of Separately Excited DC Motor to study the Dynamic behavior

of the machine for change in Load Torque.

7. Mathematical Modeling of separately excited DC motor.

8. Modeling and Simulation of Three Phase Induction Motor in three different

9. Simulation of three phase inverter with 120°, 150°, 180°mode of operation. frames.

10. Load (Power) Flow studies using Mi-Power.

11. Stability analysis using Root locus, Bode plot and Nyquist plot.

12. Simulation and Analysis of 3-ϕ PWM inverter fed Induction Motor.

13. Resonant Converter – ZVS and ZCS operation.

Mrs.Y.Priyanka   Dr.N.Malla Reddy
  Lab In-Charge   HOD-EEE
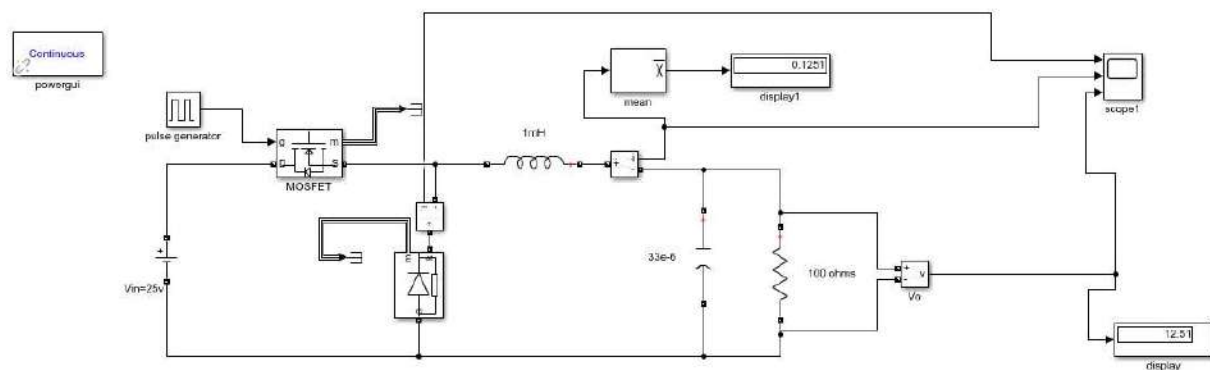
List of Experiments

1. Simulation of buck, boost and buck-boost converter with open loop operation.

2. Single Phase Inverter using PWM controller with RL load.

3. Three phase fully controlled Converter using RL &RLE loads.

4. Three phase Inverter with SPWM Controller.

5. Simulation of z-source inverter.

6. Simulation of Separately Excited DC Motor to study the Dynamic behavior of the machine for change in Load Torque.

7. Mathematical Modeling of separately excited DC motor.

8. Modeling and Simulation of Three Phase Induction Motor in three different frames.

9. Simulation of three phase inverter with 120^0, 150^0, 180^0 mode of operation.

10. Load (Power) Flow studies using Mi-Power.

11. Stability analysis using Root locus, Bode plot and Nyquist plot.

12. Simulation and Analysis of 3-ϕ PWM inverter fed Induction Motor.

13. Resonant Converter – ZVS and ZCS operation.

Electrical Systems Simulation Lab   I M.Tech – I Sem

SIMULATION OF BUCK, BOOST AND BUCK-BOOST CONVERTER WITH OPEN LOOP OPERATION

Aim: To Simulate a buck converter with R load in open loop.

Software: MATLAB/SIMULINK

Circuit diagram:

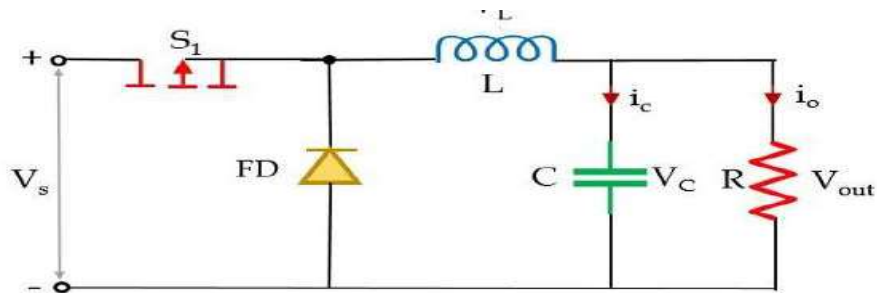**Specifications:**

Voltage Source =25V

Inductance =1mH

Capacitor= 33µF

Resistor =100Ω

Frequency= 25 KHz

1

[EXP 1)

Electrical Systems Simulation Lab   I M.Tech – I Sem

Theory & Operation: The Buck converter is a step down converter and produces a lower average output voltage than the dc input voltage.

In the above figure, it is clearly shown that along with the power electronics solid-state device which acts as a switch for the circuit, there is another switch in the circuit which is a freewheeling diode. The combination of these two switches forms a connection with a low-pass LC filter in order to reduce current or voltage ripples. This helps in generating regulated dc output. A pure resistor is connected across this whole arrangement that acts as a load of the circuit. The whole operation of the circuit takes place in two modes. The first mode is the one when the power MOSFET i.e., switch S_1 is closed. In this mode of operation, switch S_1 is in closed condition thus allows the flow of current to take place through it.
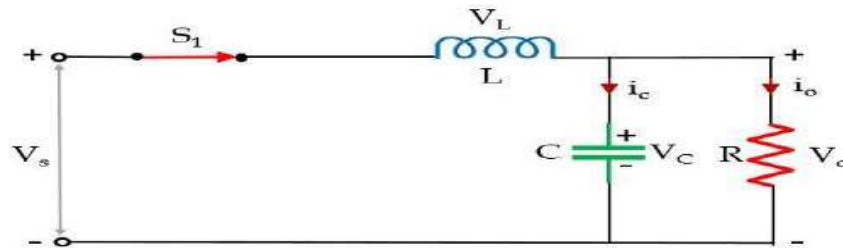
When S_1 is closed

Initially when a fixed dc voltage is applied across the input terminal of the circuit then in the closed condition of switch S_1 current flows in the circuit. Due to this flowing current, the inductor in the path stores energy in the form of a magnetic field. Also, there is a capacitor in the circuit and current flows through it also, therefore, it will store the charge and the voltage across it will appear across the load. However, due to Lenz's law, the energy stored within the inductor will oppose the cause which has produced it and so an induced current will get generated and the polarity across the inductor will get reversed. Here the total time period is a combination of

T_{on} and T_{off} time. $T = T_{on} + T_{off}.$ The duty cycle is written as: $D = \dfrac{T_{on}}{T}$ On applying KVL, in the above-given circuit,

[EXP 1)

Electrical Systems Simulation Lab   I M.Tech – I Sem

$$V_s = V_L + V_{out}$$
$$V_L = V_s - V_{out}$$

$$V_L = L\frac{di_L}{dt} = V_s - V_{out}$$
$$\frac{di_L}{dt} = \frac{V_s - V_{out}}{L}$$

When S1 is in closed condition then T_{on} = DT thus Δt = DT. Therefore, we can write,

$$\frac{\Delta i_L}{\Delta t} = \frac{V_s - V_{out}}{L}$$
$$\frac{\Delta i_L}{DT} = \frac{V_s - V_{out}}{L}$$

$$\Delta i_t = (\frac{V_s - V_{out}}{L})DT$$

Now, the second mode of operation takes place when switch S_2 is closed and S_1 gets open. However, you must be thinking about how automatically, the switch S_2 will be closed. So, as we have discussed that the inductor in the circuit will store the energy so, once S_1 will get open the inductor in the circuit will start acting as the source. In this mode, the inductor releases the energy which is stored in the previous mode of operation. As we have discussed that the polarity of the inductor will get reversed therefore this causes the freewheeling diode to come in a forward-biased state which was earlier present in a reverse-biased state due to the applied dc input.

When S₂ is closed

[EXP 1)

Electrical Systems Simulation Lab   I M.Tech – I Sem
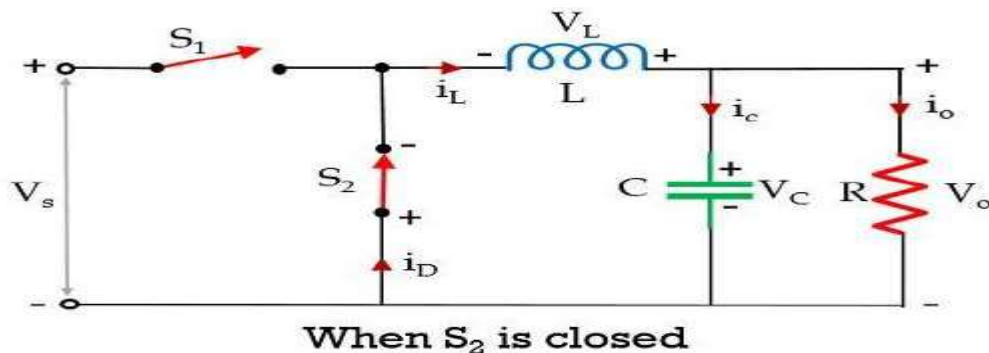
This flow of current will take place till the time the stored energy within the inductor gets completely collapsed. As once the inductor gets completely discharged, the diode comes in reverse biased condition leading to cause opening of switch S_2, and instantly switch S_1 will get closed and the cycle continues. Now, let us apply KVL, in the above circuit,

$$0 = V_L + V_{out}$$

$$V_L = L\frac{di_L}{dt} = -V_{out}$$

$$T = T_{on} + T_{off}$$

$$T = DT + T_{off}$$

$$T_{off} = T - DT$$

$$T_{off} = (1 - D)T$$

$$V_L = L\frac{\Delta i_L}{\Delta t} = -V_{out}$$
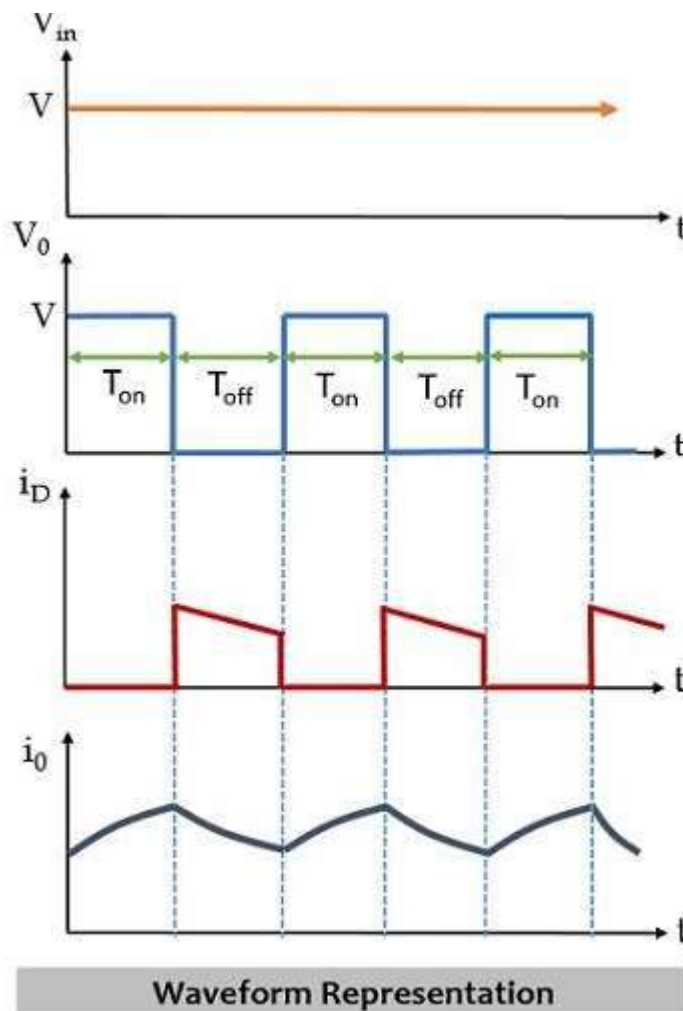
$$T_{off} = \Delta t = (1 - D)T$$

$$L\frac{\Delta i_L}{(1 - D)T} = -V_{out}$$

$$\Delta i_L = -\frac{V_{out}}{L}(1 - D)T$$

$$\Delta i_{L(S1-closed)} + \Delta i_{L(S1-open)} = 0$$

$$\frac{V_s - V_{out}}{L}\,DT + \left\{-\frac{V_{out}}{L}(1 - D)T\right\} = 0$$

[EXP 1)

Electrical Systems Simulation Lab   I M.Tech – I Sem

$$\frac{V_s DT}{L} - \frac{V_{out} DT}{L} - \frac{V_{out} T}{L} + \frac{V_{out} DT}{L} = 0$$

$$\left(\frac{V_s DT}{L}\right) = \frac{V_{out} T}{L}$$

$$V_{out} = DV_s$$

Waveform Representation

Procedure:
1)Rig up the circuit as per the circuit diagram in MATLAB/SIMULINK Model platform.
2)Select the appropriate values to the elements
3) Simulate the circuit & observe the output waveforms for different values of Duty cycle

5

[EXP 1)

Electrical Systems Simulation Lab   I M.Tech – I Sem

Observations:

S.NO	Input Voltage V_{in} (V)	Duty cycle D(%)	Output Voltage V_o (V)	Output Current I_o (A)

Simulation Results:
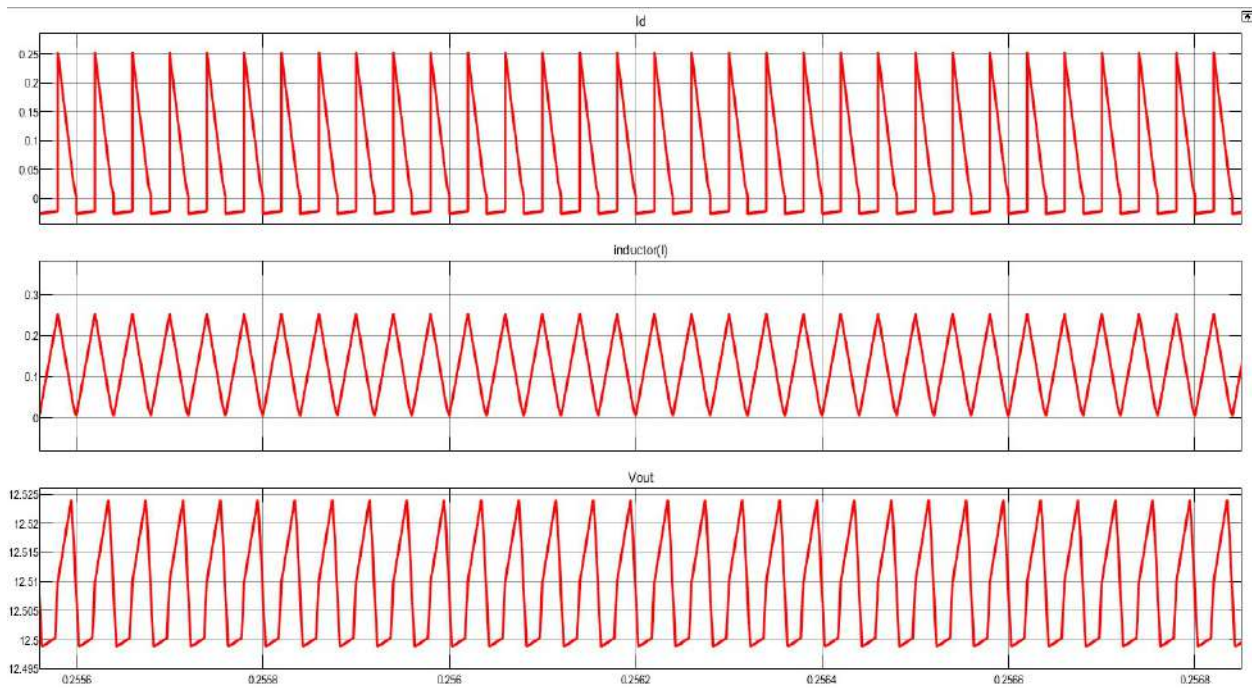
Output waveforms for Duty cycle 50%

Results:

Hence the Simulation of Buck converter for R load is done in MATLAB/SIMULINK and the waveforms are observed.

[EXP 1)

Electrical Systems Simulation Lab   I M.Tech – I Sem

Aim: To Simulate a boost converter with R load in open loop.
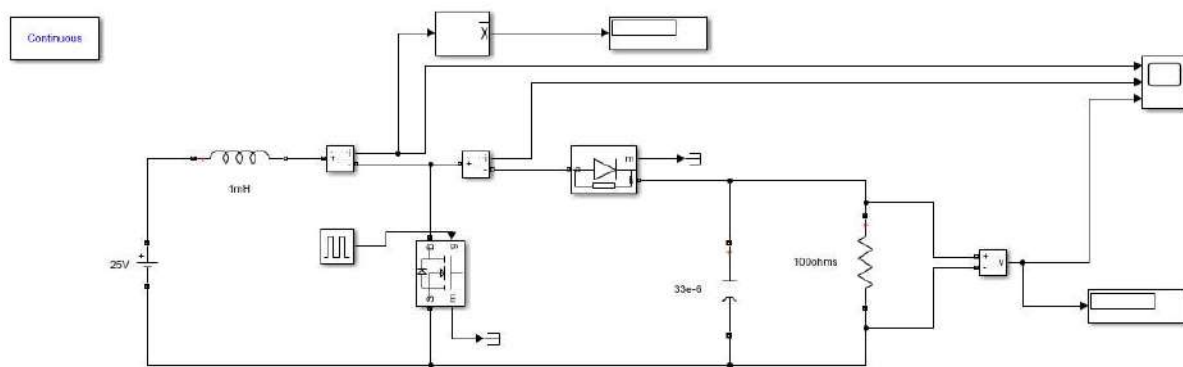
Software: MATLAB/SIMULINK

Circuit diagram:

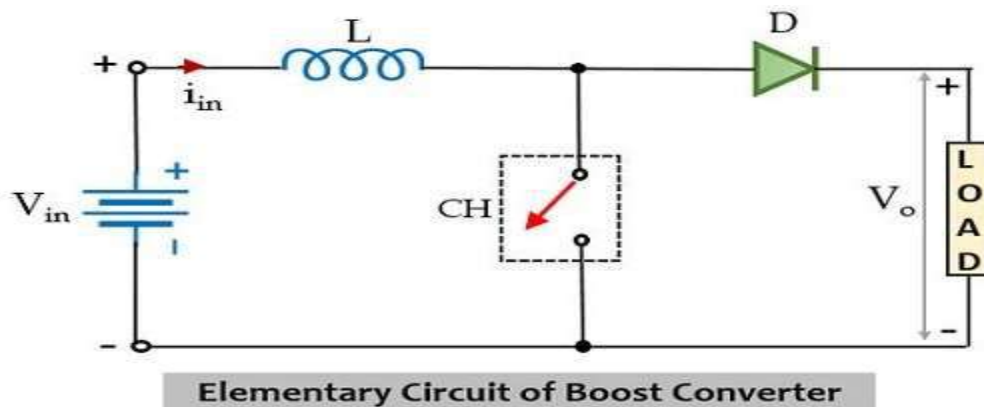**Specifications:**

Voltage Source =25V

Inductance =1mH

Capacitor= 33µF

Resistor =100Ω

Frequency= 25 KHz

7

[EXP 1)

Electrical Systems Simulation Lab   I M.Tech – I Sem

Theory & Operation: The Boost converter is a step up converter and produces a high average output voltage than the dc input voltage

Elementary Circuit of Boost Converter

The circuit here is an elementary form of step-up chopper which necessarily requires a large inductor L in series connection with the voltage source. The whole circuit arrangement operates in a way that it helps in maintaining a regulated dc signal at the output. Let us understand how the given circuit operates in order to provide an increased dc signal at the load. Initially, when the chopper CH is in on state, then in the presence of supply dc input current begins to flow through the closed path of the circuit i.e., passing through the inductor as shown in the figure below.
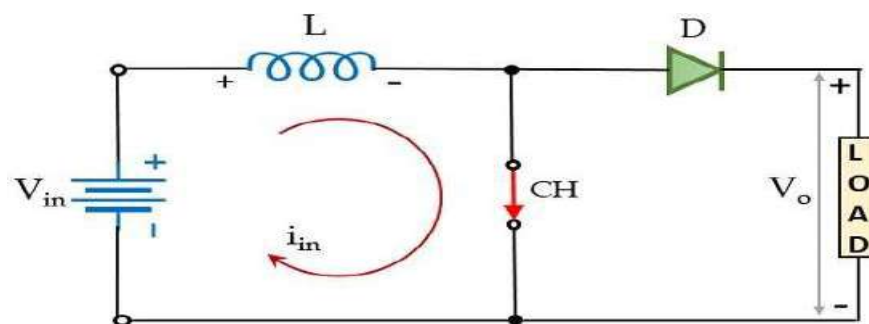

Here, the polarity of the inductor will be according to the direction of the flow of current. In this particular case, the diode in the configuration is in reverse biased condition and so current will not be allowed to flow through that particular part of the circuit during on state of the chopper. Resultantly, the voltage across the chopper will appear across the load.

[EXP 1)

Electrical Systems Simulation Lab   I M.Tech – I Sem
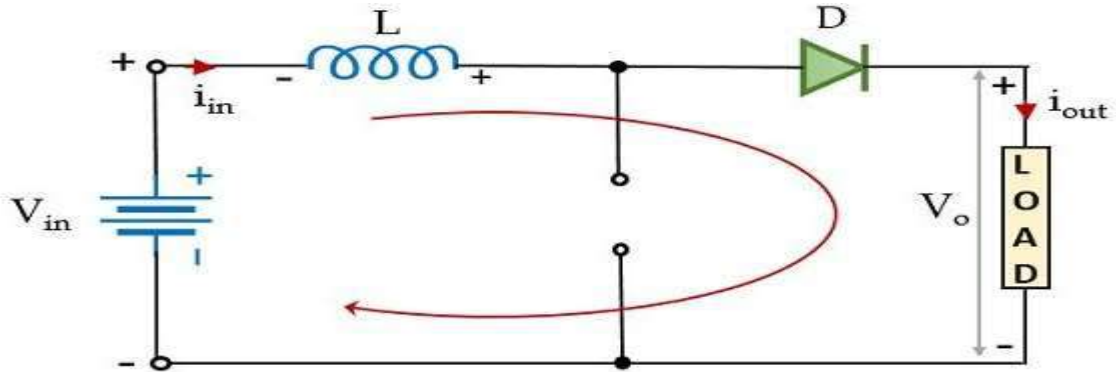
Furthermore, at the instant when CH is in the off state, then the part of the circuit through which the current was flowing earlier will not be active in this case. However, as the inductor stores, the energy in the form of a magnetic field and so the current through it will not die out instantly. Also, we know according to Lenz's law a reverse current will be induced that will oppose the cause which has produced it. And so, due to the induced current, the polarity of the inductor will get reversed. This reverse polarity of the inductor forward biases the diode present in the circuit. This provides the path for the current through the diode that flows through the load during the off state of the chopper i.e., T_{off}. However, we must note here that the current through the inductor is of decreasing nature and will die out after a point in time.Thus, the total voltage across the load will be given as: $V_{out} = V_{in} + V_L$ .This means that the output voltage exceeds the applied input voltage. Thus, performs step-up conversion as the energy stored within the inductor during the T_{on} period is released during the T_{off} period. During the T_{on} period, the voltage across the inductor will be given as:

$$V_L = L\frac{di}{dt}$$

[EXP 1)

Electrical Systems Simulation Lab   I M.Tech – I Sem
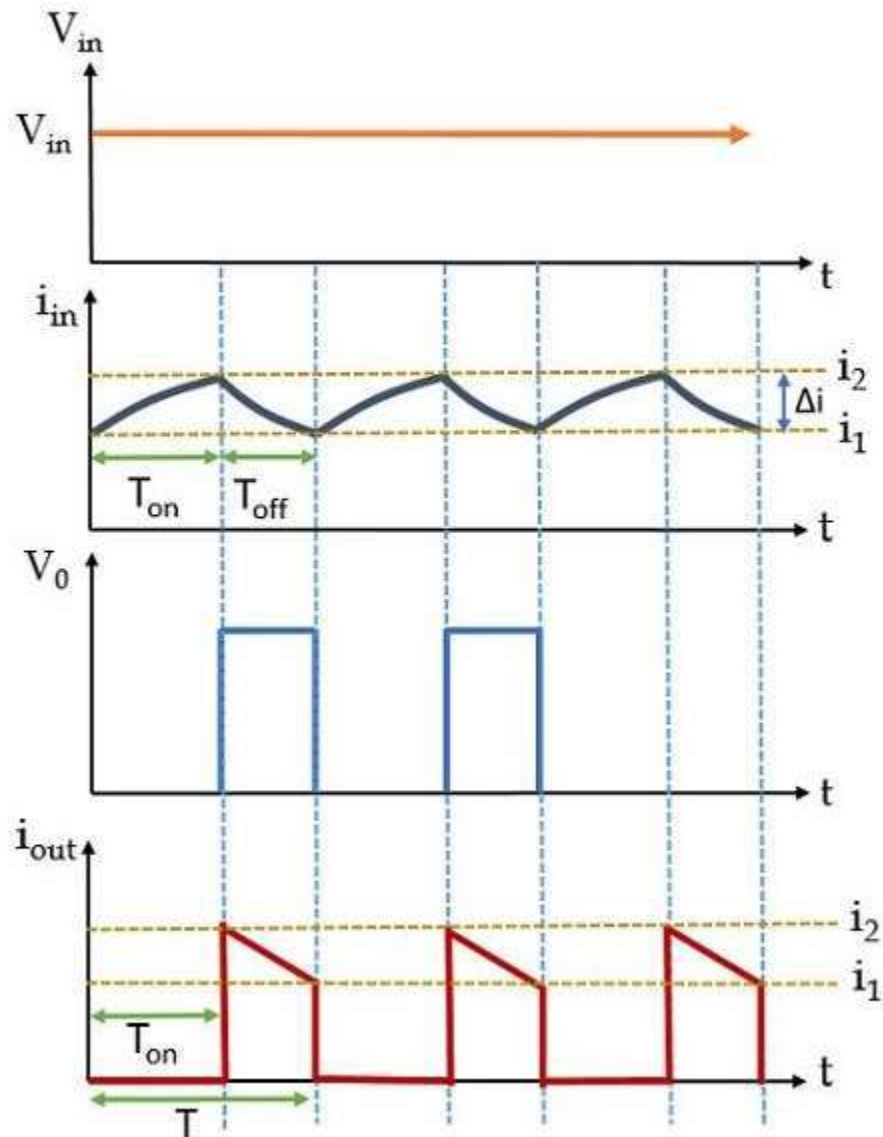
During the T_{on} period, the current through the inductor will change from i_1 to i_2 this is clearly shown above. While during the T_{off} period, the inductor current will change from i_2 to i_1. Now, talking about voltage, so during the turn-on period, the voltage across the inductor will be equal to the supply input voltage. But when CH gets off then on applying KVL in the figure shown above, we will get,

$$V_L - V_0 + V_{in} = 0$$

[EXP 1)

Electrical Systems Simulation Lab   I M.Tech – I Sem

$$V_L = V_0 - V_{in}$$

Considering that output current is varying linearly, the energy input provided by the source to the inductor, when CH is on, is given as:

W_{on} = (voltage across the inductor)(average current through the inductor)T_{on}

$$W_{on} = V_{in} \left(\frac{i_1 + i_2}{2} \right) T_{on}$$

Further, the energy that the inductor releases to the load when CH is off is given as:

W_{off} = (voltage across the inductor)(average current through the inductor)T_{off}

$$W_{off} = V_{out} - V_{in} \left(\frac{i_1 + i_2}{2} \right) T_{off}$$

For a lossless system, comparing the two energies, we will have,

$$V_{in} \left(\frac{i_1 + i_2}{2} \right) T_{on} = V_{out} - V_{in} \left(\frac{i_1 + i_2}{2} \right) T_{off}$$

On simplifying,

$$V_{in} T_{on} = V_{out} T_{off} - V_{in} T_{off}$$
$$V_{out} T_{off} = V_{in} T_{on} + V_{in} T_{off}$$
$$V_{out} T_{off} = V_{in} (T_{on} + T_{off})$$

Since we know, T = T_{on} + T_{off}, therefore,

[EXP 1)

Electrical Systems Simulation Lab   I M.Tech – I Sem

$$V_{Out} \, T_{off} = V_{in} T$$

$$V_{Out} = V_{in} \frac{T}{T_{off}}$$

$$V_{Out} = V_{in} \frac{T}{T - T_{on}}$$

$$V_{Out} = V_{in} \frac{1}{\left(\frac{T}{T} - \frac{T_{on}}{T}\right)}$$

Since, we know, duty cycle i.e., $\alpha = T_{on}/ T$

$$V_{Out} = V_{in} \frac{1}{(1 - \alpha)}$$

Thus, we can conclude here that the average load voltage can be stepped up with the change in the duty cycle.

Procedure:
1)Rig up the circuit as per the circuit diagram in MATLAB/SIMULINK Model platform.
2)Select the appropriate values to the elements
3) Simulate the circuit & observe the output waveforms for different values of Duty cycle
Observations:

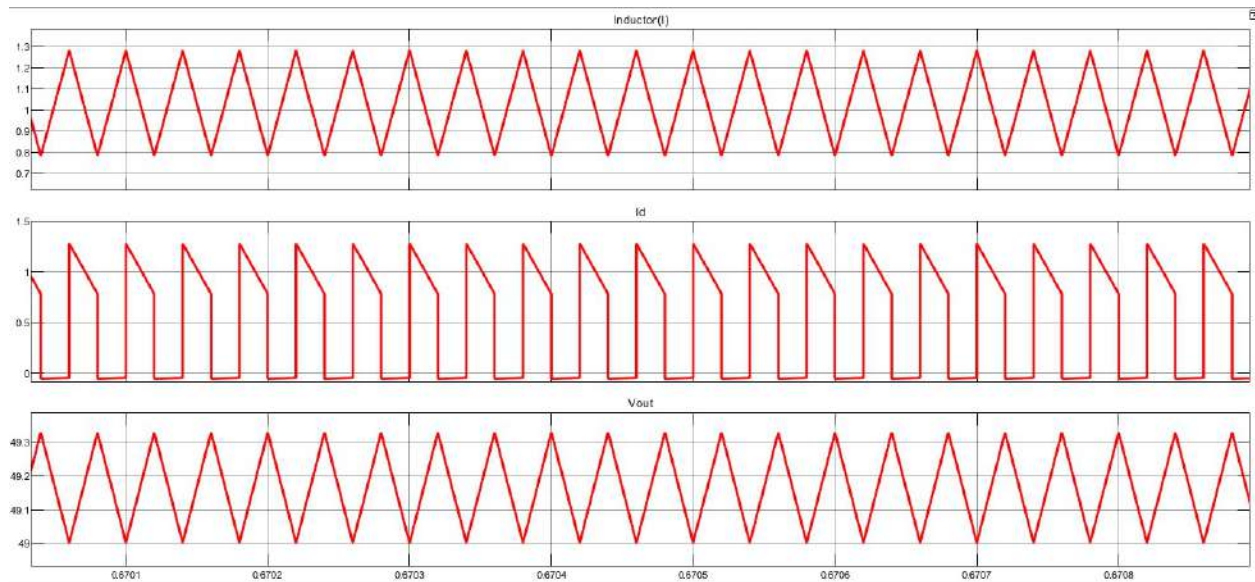| S.NO	Input Voltage V_{in} (V)	Duty cycle α(%)	Output Voltage V_o (V)	Output Current I_o (A)

[EXP 1)

Electrical Systems Simulation Lab   I M.Tech – I Sem

Simulation Results:

Output waveforms for duty cycle 50%

Results:

Hence the Simulation of Boost converter for R load is done in MATLAB/SIMULINK and the waveforms are observed.
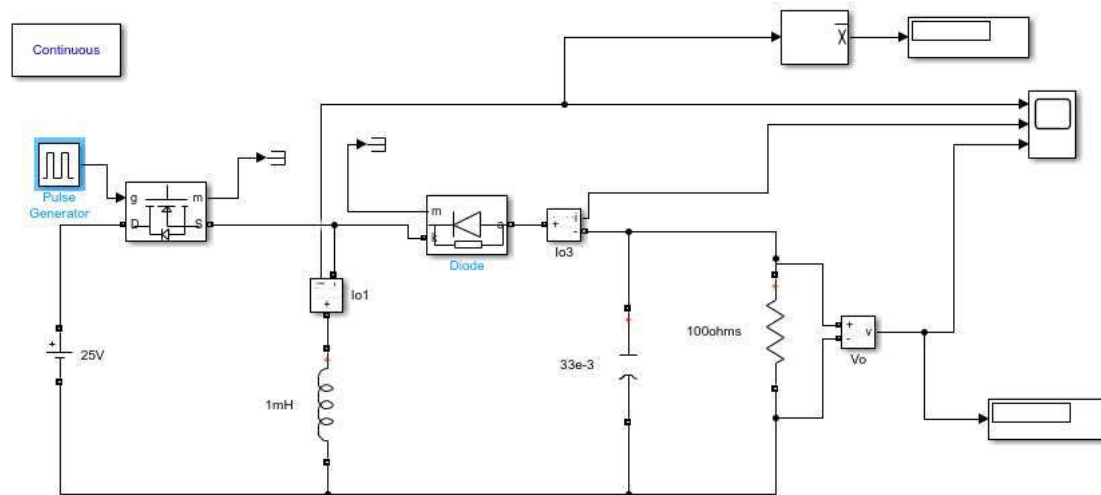
[EXP 1)

Electrical Systems Simulation Lab   I M.Tech – I Sem

Aim: To Simulate a Buck-Boost converter with R load in open loop.

Software: MATLAB/SIMULINK

Circuit diagram:

Specifications:

Voltage Source =25V

Inductance =1mH

Capacitor= 33μF

Resistor =100Ω

Frequency= 25 KHz

14

[EXP 1)

Electrical Systems Simulation Lab   I M.Tech – I Sem

Theory & Operation: It is a type of DC to DC converter and it has a magnitude of output voltage. It may be more or less than equal to the input voltage magnitude. The buck boost converter is equal to the fly back circuit and single inductor is used in the place of the transformer.

The output voltage of the magnitude depends on the duty cycle. These converters are also known as the step up and step down transformers and these names are coming from the analogous step up and step down transformer. The input voltages are step-up/down to some level of more than or less than the input voltage. By using the low conversion energy, the input power is equal to the output power. The following expression shows the low of a conversion.

Input power (Pin) = Output power (Pout)

For the step up mode, the input voltage is less than the output voltage (Vin < Vout). It shows that the output current is less than the input current. Hence the buck booster is a step up mode.

Vin < Vout and Iin > Iout

In the step down mode the input voltage is greater than the output voltage (Vin > Vout). It follows that the output current is greater the input current. Hence the buck boost converter is a step down mode.

Vin > Vout and Iin < Iout

There are two types of converters in the buck boost converter that are buck converter and the other one is boost converter. These converters can produce the range of output voltage than the input voltage. The following diagram shows the basic buck boost converter.
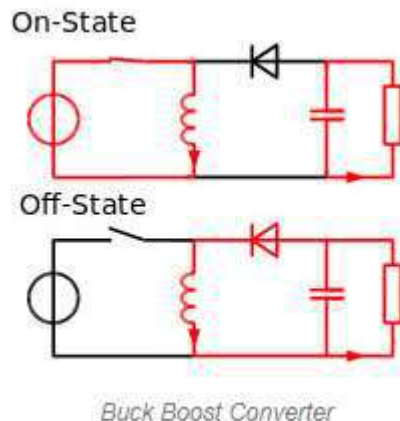
Buck Boost Converter

15

[EXP 1)

Electrical Systems Simulation Lab   I M.Tech – I Sem

The working operation of the DC to DC converter is the inductor in the input resistance has the unexpected variation in the input current. If the switch is ON then the inductor feed the energy from the input and it stores the energy of magnetic energy. If the switch is closed it discharges the energy. The output circuit of the capacitor is assumed as high sufficient than the time constant of an RC circuit is high on the output stage. The huge time constant is compared with the switching period and make sure that the steady state is a constant output voltage $V_o(t)$ = Vo(constant) and present at the load terminal.
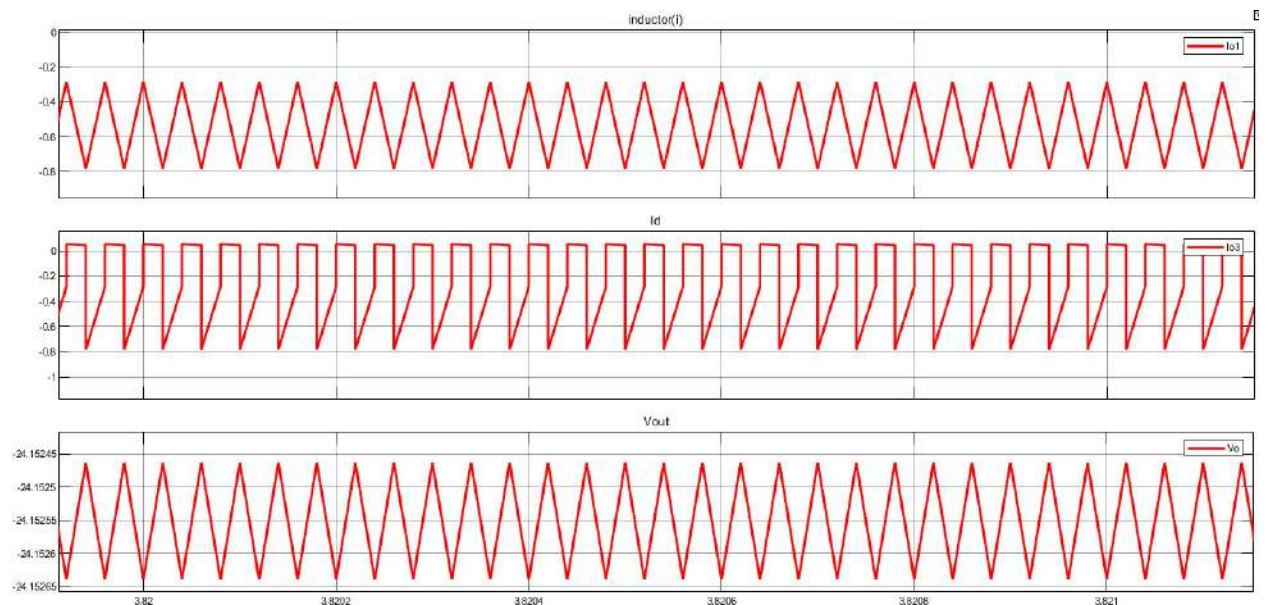
Procedure:
1)Rig up the circuit as per the circuit diagram in MATLAB/SIMULINK Model platform.
2)Select the appropriate values to the elements
3) Simulate the circuit & observe the output waveforms for different values of Duty cycle

Observations:

S.NO	Input Voltage V_{in} (V)	Duty cycle D(%)	Output Voltage V_o (V)	Output Current I_o (A)

Simulation Results:

Output waveforms for duty cycle 50%

[EXP 1)

Electrical Systems Simulation Lab   I M.Tech – I Sem

Result:

Hence the Simulation of Buck-Boost converter for R load is done in MATLAB/SIMULINK and the waveforms are observed.

(Mrs.Y.Priyanka)   (Dr N. Malla Reddy)

 Asst. Prof. – EEE   HOD - EEE

 [EXP 1)

G. Narayanamma Institute of Technology & Science
(For Women)
Shaikpet, Hyderabad – 500 104
Department of Electrical and Electronics Engineering

Electrical Systems Simulation Lab   I M.Tech – I Sem

SINGLE PHASE INVERTER USING PWM CONTROLLER WITH RL LOAD

Aim: To simulate single phase inverter with and without PWM technique.
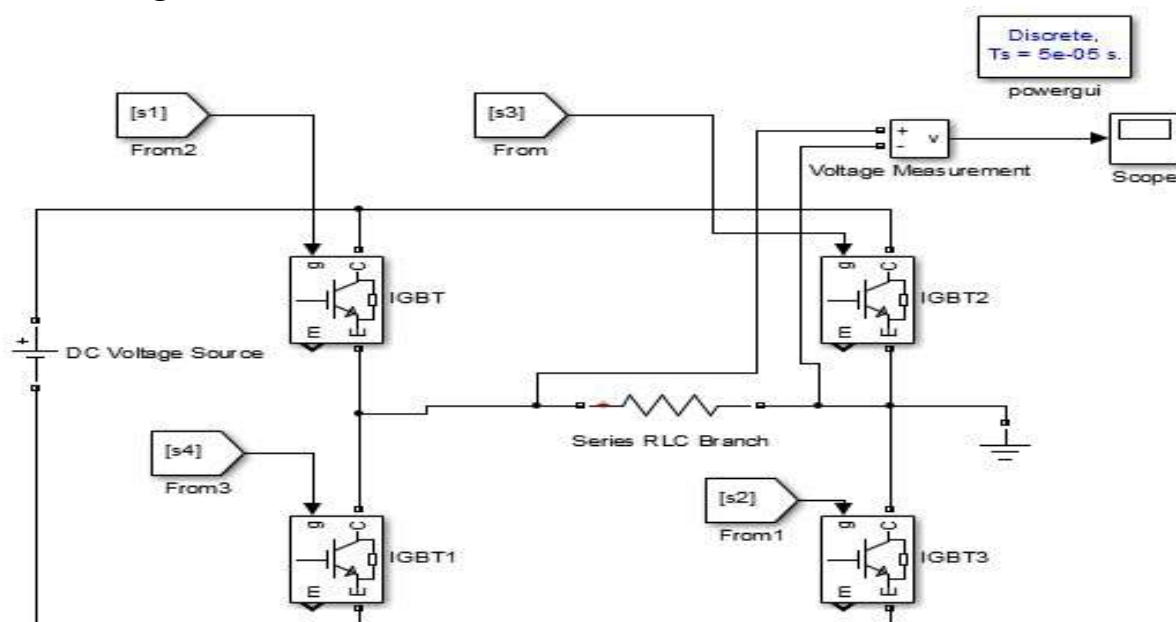
 Software: MATLAB/SIMULINK

Circuit Diagram:

Fig.1. Single Phase Inverter

Theory:

Pulse Width Modulation Control of Single phase inverter:

A fixed DC input voltage is given to the inverter and a controlled AC output voltage is obtained by adjusting the ON and OFF periods of the inverter switches and this method is termed as "Pulse Width Modulation Control". Using this technique, the output voltage control can be obtained without any additional components. Also, lower order harmonics can be eliminated along with its output voltage control. The higher order harmonics can be eliminated easily using filters.

1

[EXP2]

Electrical Systems Simulation Lab   I M.Tech – I Sem

In this experiment, 'Single Pulse Modulation Technique' is adopted. The output modulated wave consists of a pulse of width '2d' located symmetrical about 'π/2' and another pulse located symmetrical about '3π/2'

The Fourier series of the output waveform is

$V_0 = 4V_S/\pi$ [sin(d)sin(ωt)–1/3sin(3d)sin(3ωt) + 1/5 sin(5d) sin(5ωt) …………..]

$V_0 = \sum_{n=1,3,5}^{\infty} \frac{4Vs}{n} \sin \frac{n\pi}{2} \sin(nd) \sin(n\omega t)$

If d = π/n, the nth order harmonic is eliminated from the inverter output voltage, i.e. in order to eliminate third harmonic content, the pulse width must be equal to 2π/3 = T/3 , where T is the time period.

Operation: a) Without Pulse Width Modulation:

In the single phase bridge inverter shown in Fig.1, the switches S1 and S2 are triggered simultaneously at '0' and S3 and S4 at 'π'.
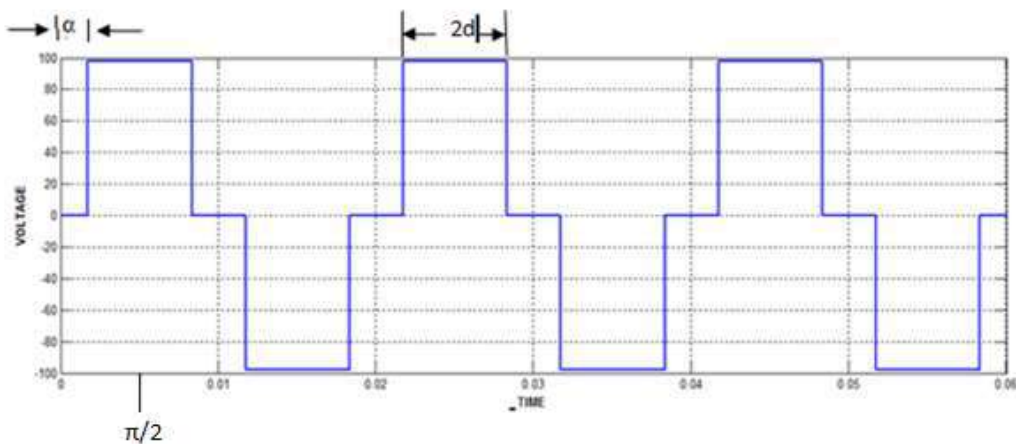

b) With Pulse Width Modulation:

Fig. 2 Output voltage using PWM

2

[EXP2]

Electrical Systems Simulation Lab   I M.Tech – I Sem

In case, third harmonic is to be eliminated from the output voltage, then the pulse width must be $2d = 2\pi/3 = T/3$. The delay is $\square = (\pi/2 - d)$.

The simulation circuit explaining the pulse generation for single phase inverter using PWM is shown in the following figure.
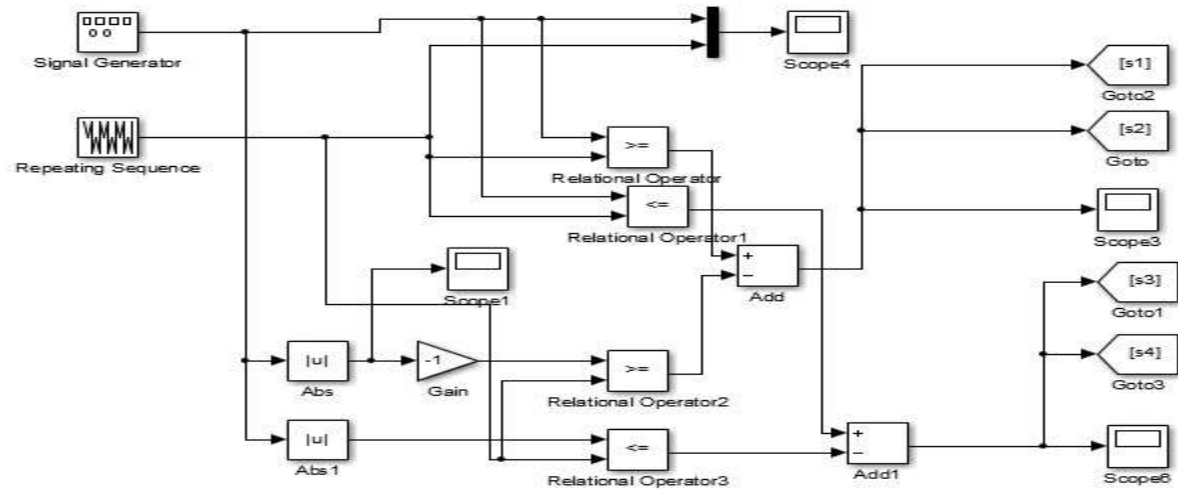

Fig. 3 Pulse Generation using PWM

Procedure:

1. Rig up the circuit as given in the diagram.

2. Select source voltage and set the specifications as required.

3. Set the Pulse voltage specifications as required.

4. Choose RLC series branch to represent load and select suitable element/ elements.

5. Run the simulation and trace the output waveforms.

[EXP2]

Electrical Systems Simulation Lab   I M.Tech – I Sem

Simulation Results:

	S1		S2		S3		S4		Harmonic content	
	□	PW	□	PW	□	PW	□	PW	3rd	5th
Without modula-tion										
With Modula-tion										

i)   Output voltage waveform without modulation
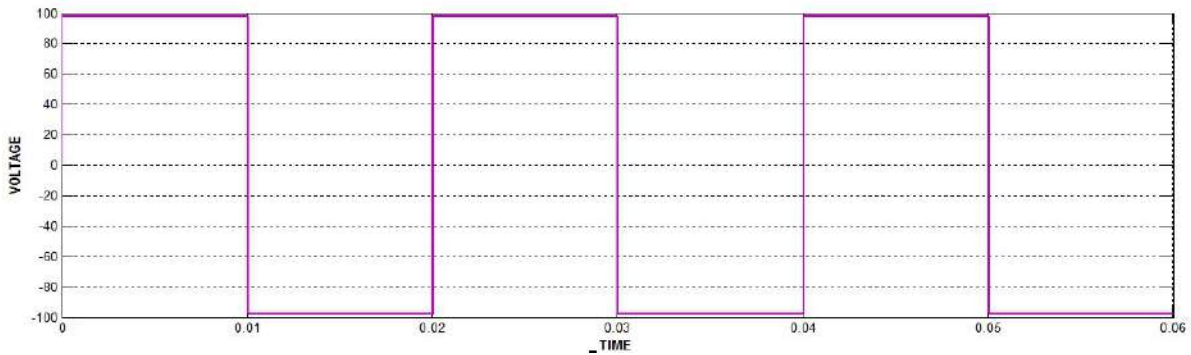
ii)   Output voltage waveform eliminating 'Third' harmonic

4

[EXP2]

Electrical Systems Simulation Lab   I M.Tech – I Sem
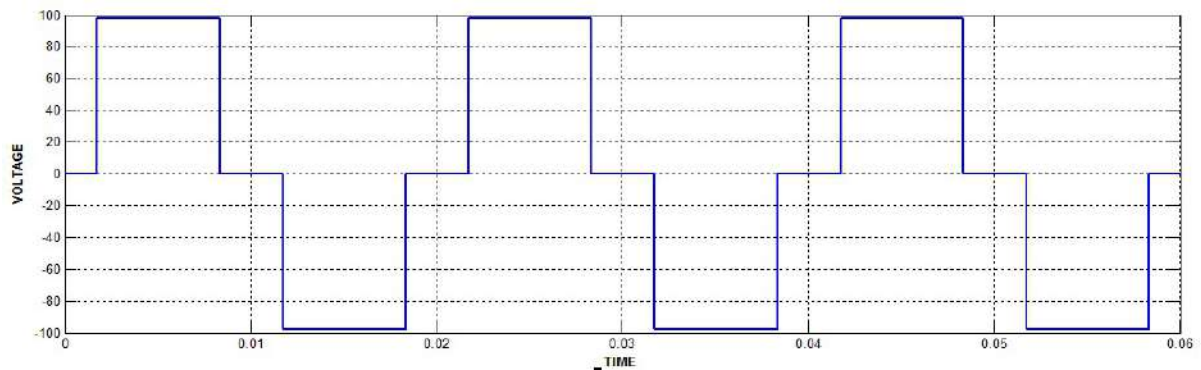
iii)   Output voltage waveform eliminating 'Fifth' harmonic
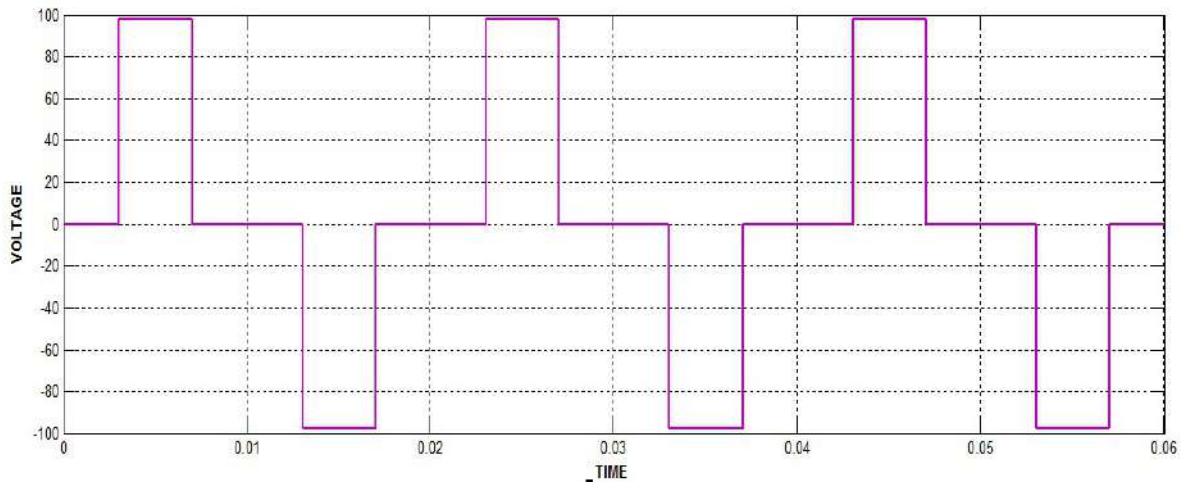
Result:

The Single phase inverter is simulated with and without pulse width modulation and the harmonic content in the output voltage is compared for both cases.

(Dr G. Annapurna)   (Dr N. Malla Reddy)
Professor. – EEE   HOD –EEE

[EXP2]

Electrical Systems Simulation Lab   I M.Tech – I Sem

THREE PHASE FULLY CONTROLLED CONVERTER USING RL &RLE LOADS

Aim: Simulation of Three Phase Full Converter with R ,R-L and R-L-E loads for finding out average output voltage . The simulated results are to be verified with theoretical calculations.

Software : MATLAB/ SIMULINK
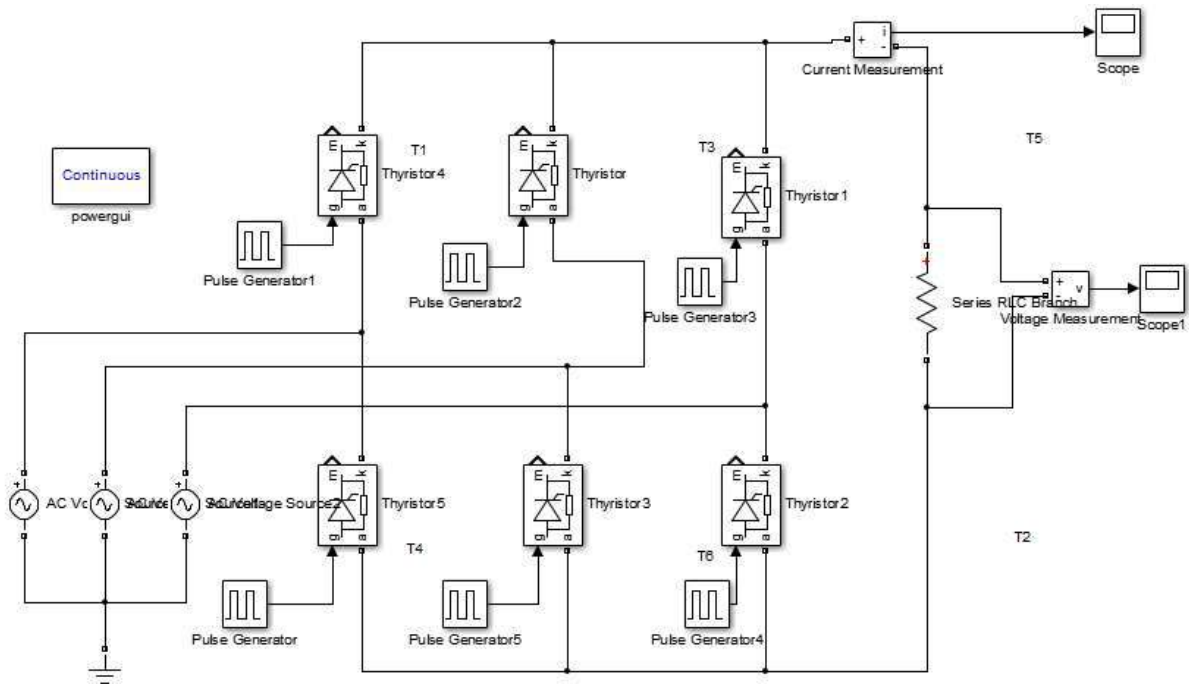
Circuit Diagram:

Fig.1. Circuit Diagram of Three Phase Full Converter

Electrical Systems Simulation Lab   I M.Tech – I Sem

Theory & Operation:

The three phase full converter acts like ac – dc converter for firing angle $0^0 < \alpha \leq 90^0$ and as line commutated inverter for $90^0 < \alpha < 180^0$. For $\alpha = 0^0$, T_1 to T_6 behave like diodes. The conduction sequence of thyristors is T_1 to T_6. The upper group SCRs are fired at an interval of 120^0 and similarly lower group SCR s are fired with an interval of 120^0 amongst them. The SCRs from both groups are fired at an interval of 60^0. Hence, commutation occurs for every 60^0 alternatively in upper and lower group of SCRs. Each SCR from both group conducts for 120^0. At any instant, two SCRs , one from upper group and one from lower group conducts.

The firing angle is measured from the cross over of the phase or line voltages. Ex. For α = 0 ^{0} , thyristor T_1 is triggered at $\alpha = 30^0$ when phase voltages are taken as reference an likewise at $\alpha = 60^0$ when line voltages are considered as reference.

The switching   instants of thyristors for   various firing angles are indicated in the following table.

α	T_1	T_2	T_3	T_4	T_5	T_6
0°	30°	90°	150°	210°	270°	330°
30°	60°	120°	180°	240°	300°	360°
60°	90°	150°	210°	270°	330°	390(30°)
72°	102°	162°	220°	282°	342	402(12°)

Design Considerations:

Vs =440 Volts, f = 50 Hz, R = 10 Ω, L = 4mH, E = 50 Volts.

Procedure:

Electrical Systems Simulation Lab   I M.Tech – I Sem

1. Rig up the circuit as given in the diagram.

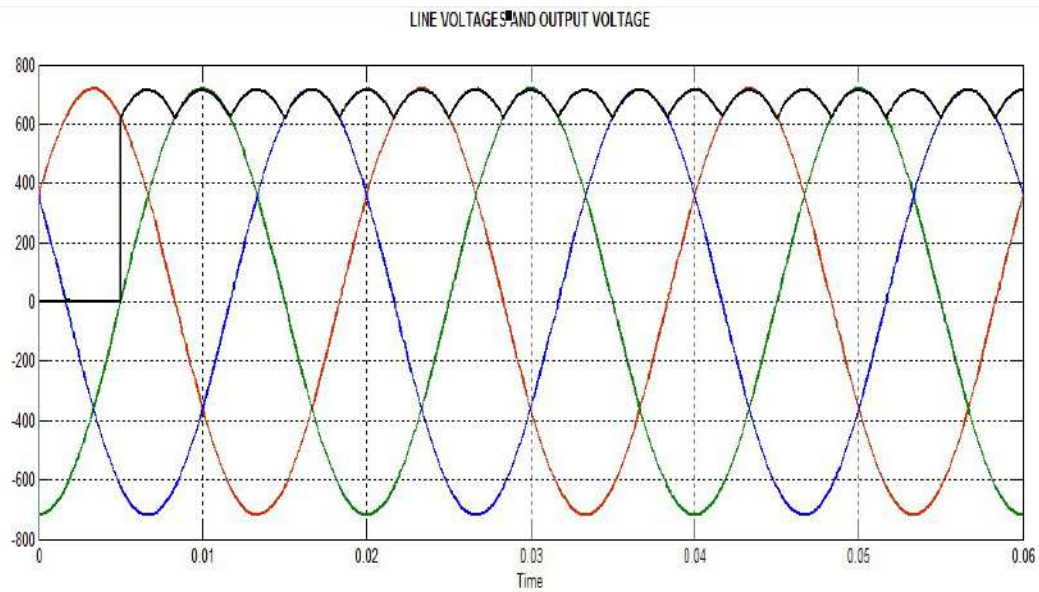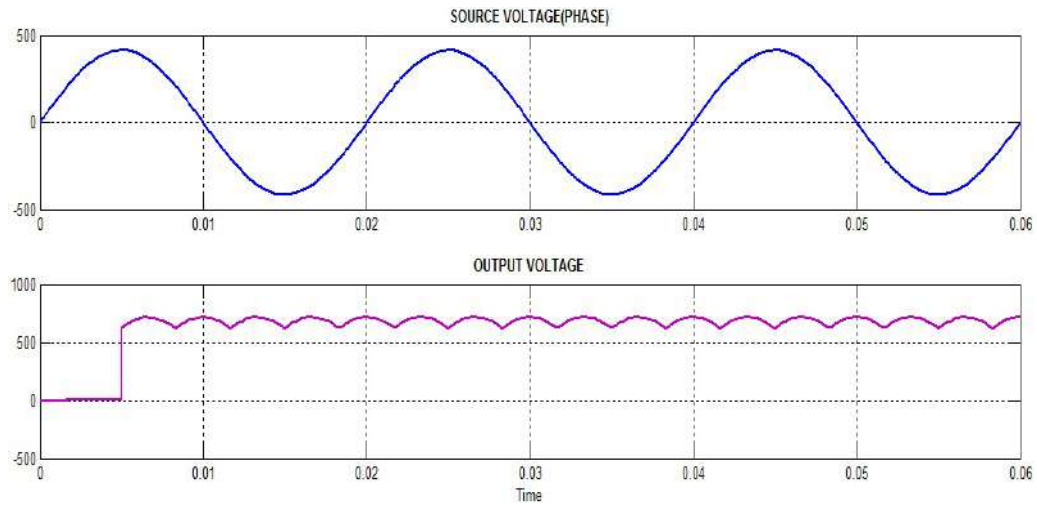2. Select source voltage and set the specifications as required.

S.NO	Input Voltage (V)	α	R(Ω)	L(mH)	E(Volts)	Output Voltage	
						Calculated	Simulated
1.							
2.							
3.							

3. Set the Pulse voltage specifications for a given firing angle.

4. Choose RLC series branch to represent load and select suitable element/ elements.

5. Run the simulation and note down the average output voltage value and trace the output waveforms.
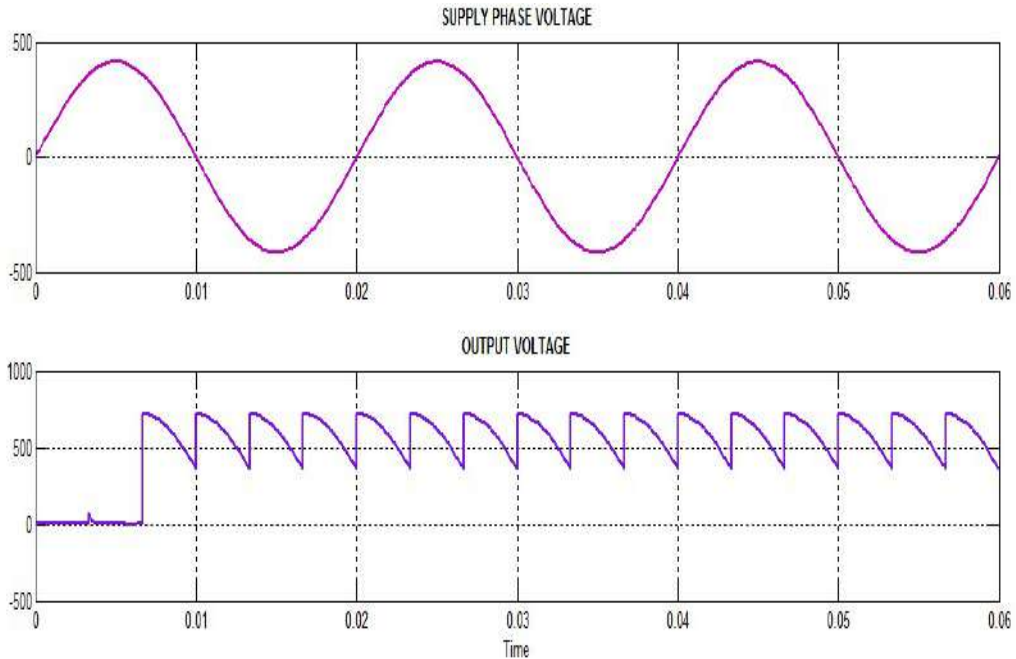
Simulation Results:

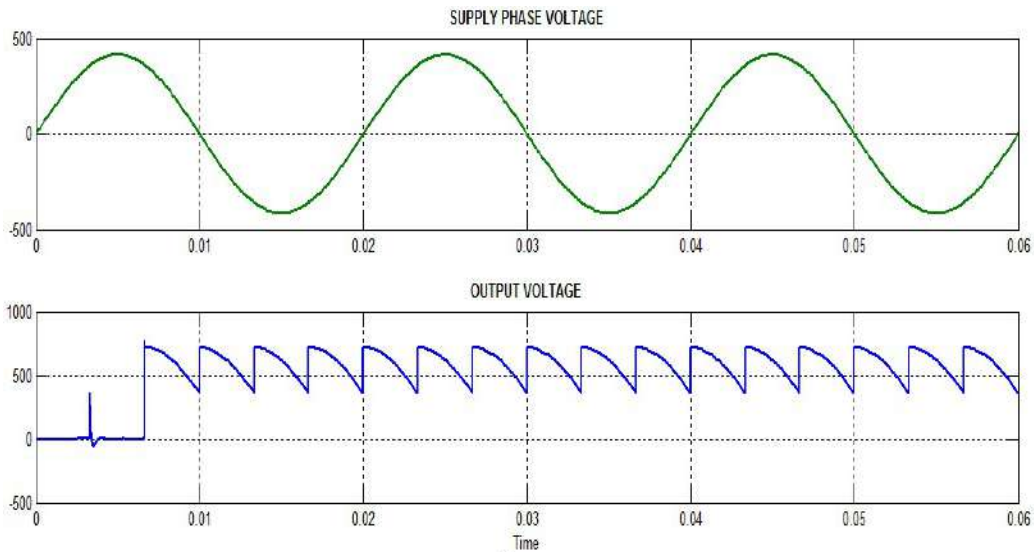**Output Waveforms for R – LOAD**

a) **For firing angle $\alpha = 0^0$**

G. Narayanamma Institute of Technology & Science
(For Women)
Shaikpet, Hyderabad – 500 104
Department of Electrical and Electronics Engineering

Electrical Systems Simulation Lab   I M.Tech – I Sem

SOURCE VOLTAGE(PHASE)

OUTPUT VOLTAGE

LINE VOLTAGES AND OUTPUT VOLTAGE

b) For firing angle α = 30⁰

Electrical Systems Simulation Lab   I M.Tech – I Sem

Output Waveforms for RL- LOAD

    a)   **For firing angle α = 30⁰**

b)For firing angle α = 72⁰
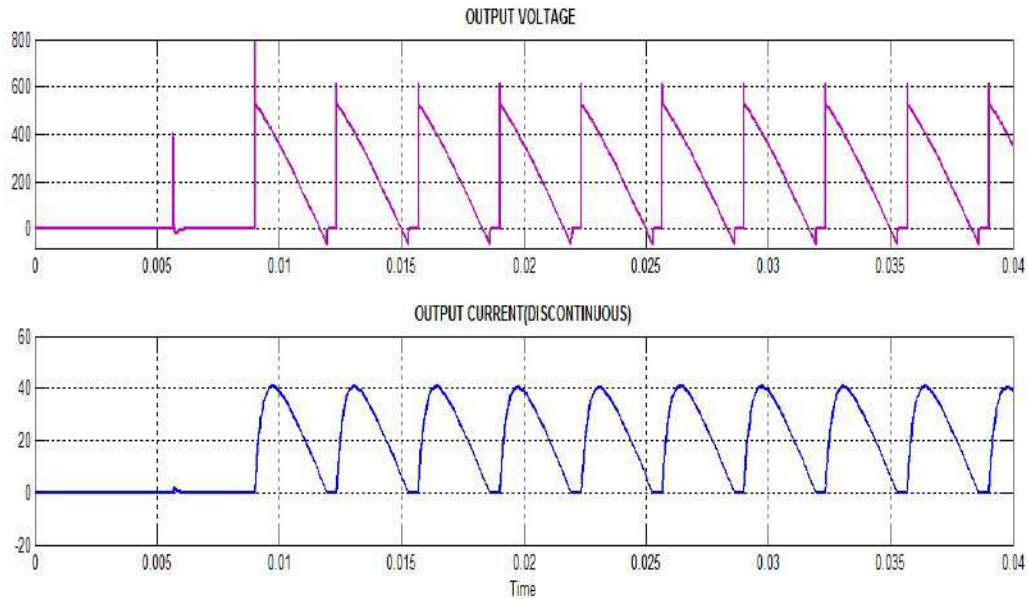
Output Waveforms for RLE- LOAD

   a)   **For firing angle α = 72⁰**
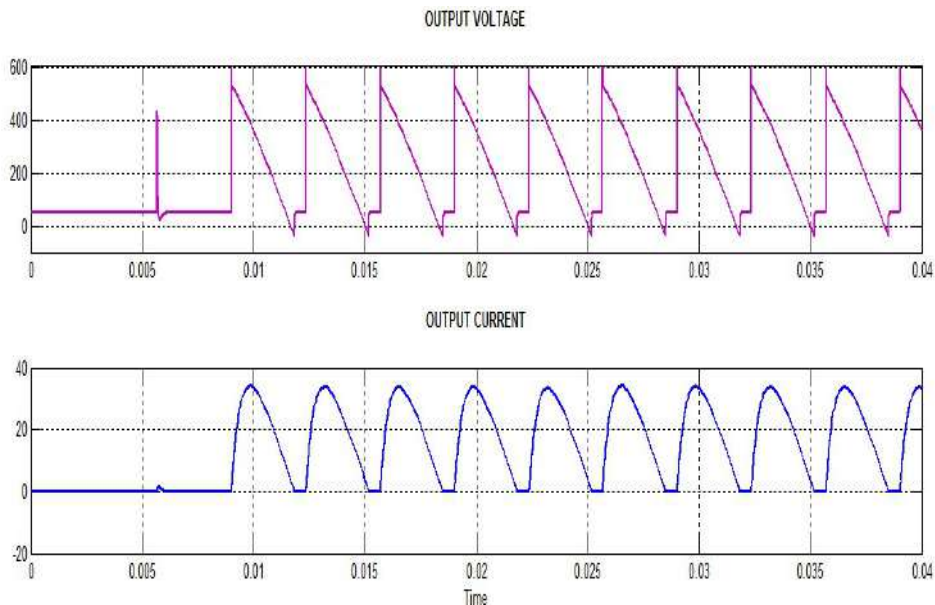
Electrical Systems Simulation Lab   I M.Tech – I Sem

Result : The Three phase Rectifier is simulated using MATLAB/SIMULINK and its   operation
        is studied under various load conditions using output waveforms.

(Dr G.Annapurna)   (Dr N. Malla Reddy)
   Professor. – EEE   HOD - EEE

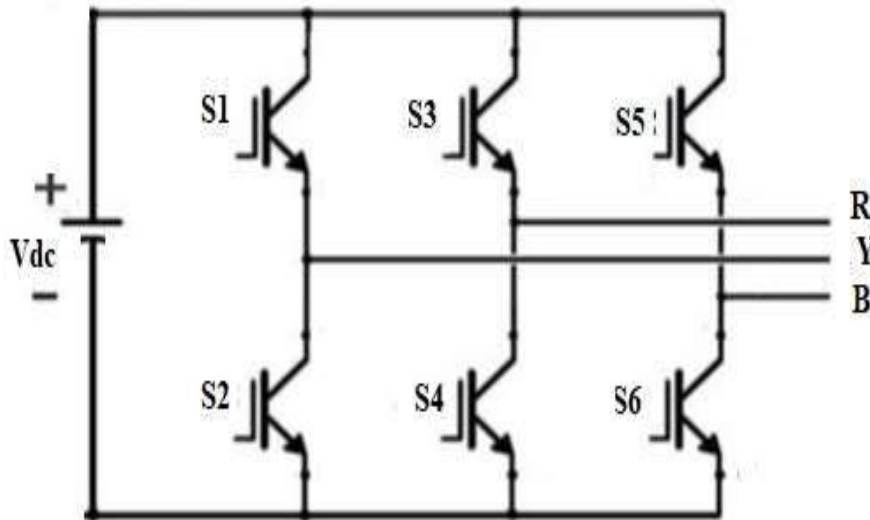Electrical Systems Simulation Lab   I M.Tech – I Sem

THREE PHASE INVERTER WITH SPWM CONTROLLER

AIM: To study the three phase inverter with R-Load using sinusoidal PWM technique.

SOFTWARE: MATLAB / SIMULINK software

CIRCUIT DIAGRAM:

THEORY:

Inverter circuits convert DC power from a DC source to AC of a desired voltage and frequency. Three-phase inverters are widely used in AC motor drive applications, in three-phase grid connection to wind generators and other power system applications. The DC source is usually in the form of a battery, solar cell, rectified wind generator output or a rectified DC output from the fixed AC supply from the utility. The input may have characteristics of a voltage source or a current source. This experiment concerns a three-phase voltage source inverter in which the input to the inverter is from an ideal DC voltage source.

In Sinusoidal PWM, the width of each pulse is varied in proportion to the amplitude of the sine wave evaluated at the centre of the same pulse. The gating signals are generated by

Electrical Systems Simulation Lab   I M.Tech – I Sem

comparing a sinusoidal reference wave with a triangular carrier wave of frequency Fr and Fc respectively. Fr determines the inverter output frequency fo and its peak amplitude Ar controls the Modulation Ratio (Ar/Ac) and hence the rms output voltage Vo. Several pulses per half cycle are used and the pulse width is a sinusoidal function of angular position of pulses in a cycle. A high frequency carrier wave Vc is compared to a reference signal Vr having the desired frequency through a comparator. When the sinusoidal wave has a higher magnitude, output is high otherwise it is low. The comparator output is processed in a trigger pulse generator in such a way that the output voltage wave has a pulse width in agreement with comparator pulse width.

$$\text{Modulation Index (M)} = \frac{V_r}{V_c}$$

For M<1

Phase Voltage $V_p = M \frac{V_{dc}}{2\sqrt{2}}$

Line Voltage $V_L = M \frac{\sqrt{3}V_{dc}}{2\sqrt{2}}$

DESIGN CONSIDERATIONS:

Sampling Time = 1μSec

Supply Frequency = 50 Hz
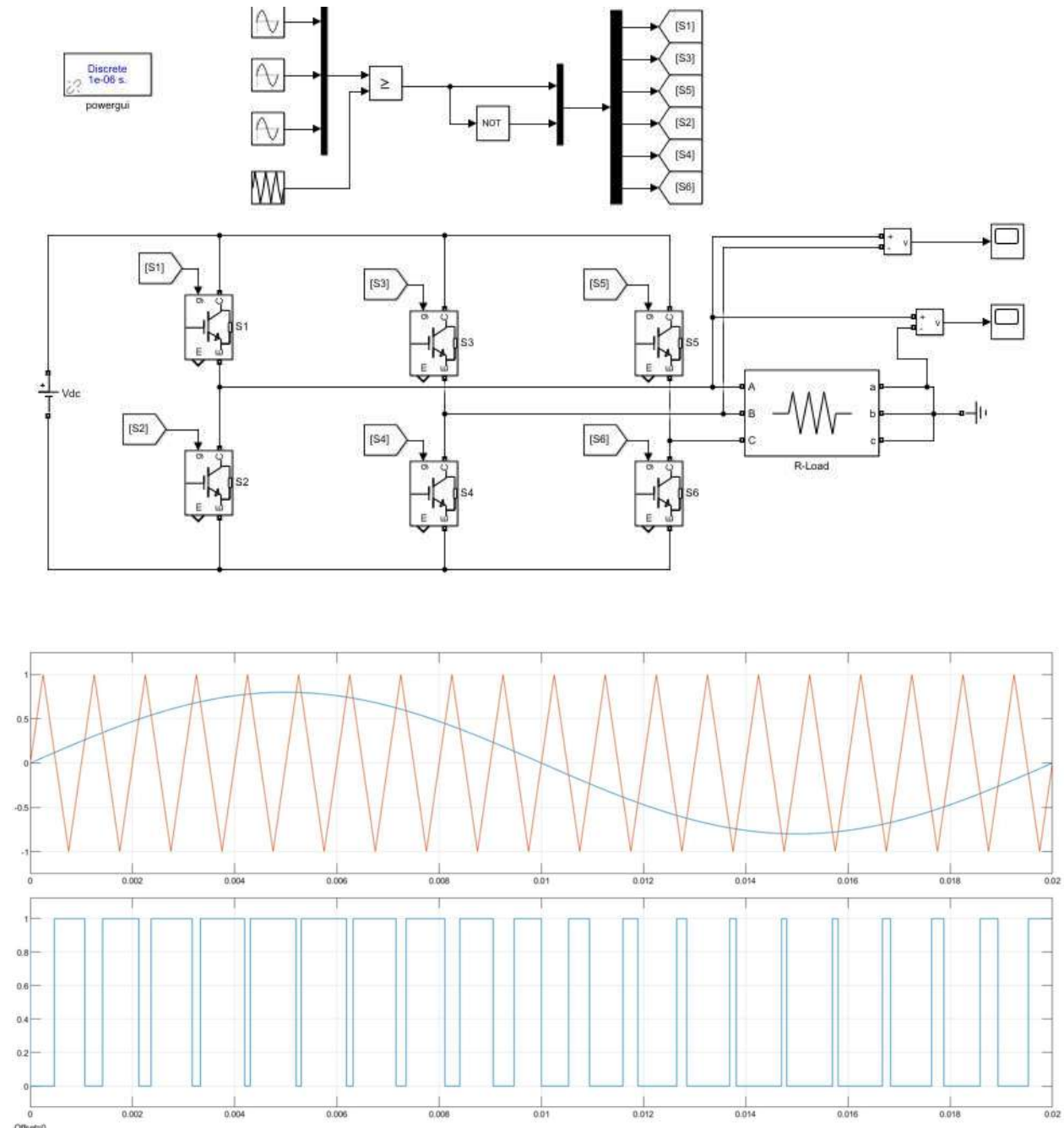
Switching Frequency = 1 kHz

Input Voltage = 100 V

Y-Connected R-Load = 10 ohm/Phase

Procedure:

1. Rig up the circuit as per the circuit diagram in MATLAB/SIMULINK Model platform.
2. Observe the output phase and line voltages.

Electrical Systems Simulation Lab   I M.Tech – I Sem
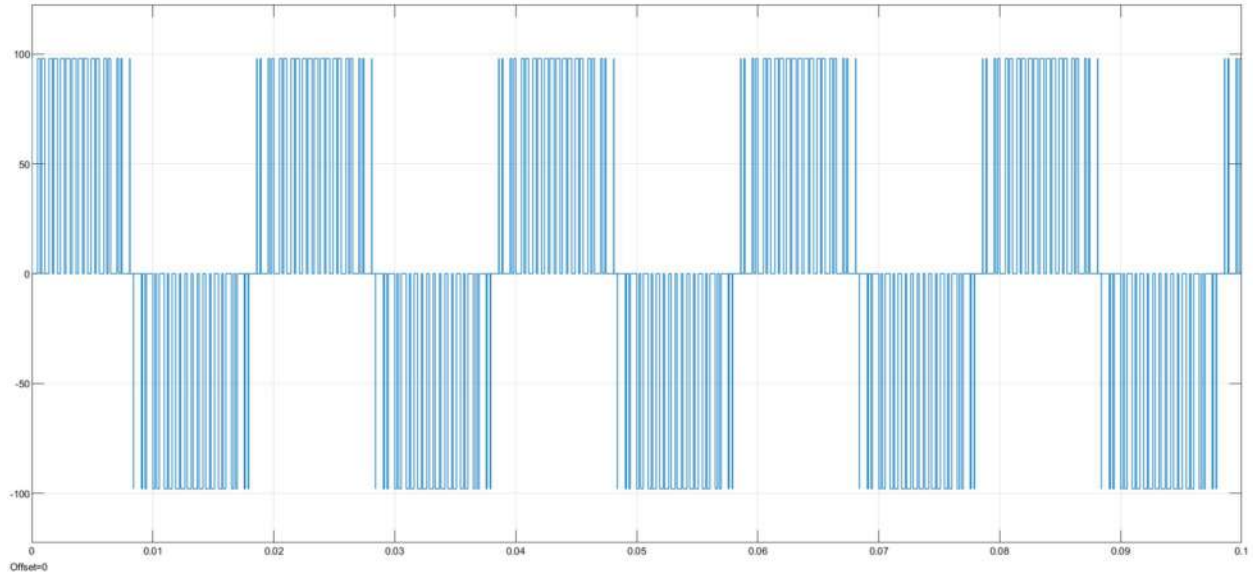
MATLAB/SIMULINK Circuit:

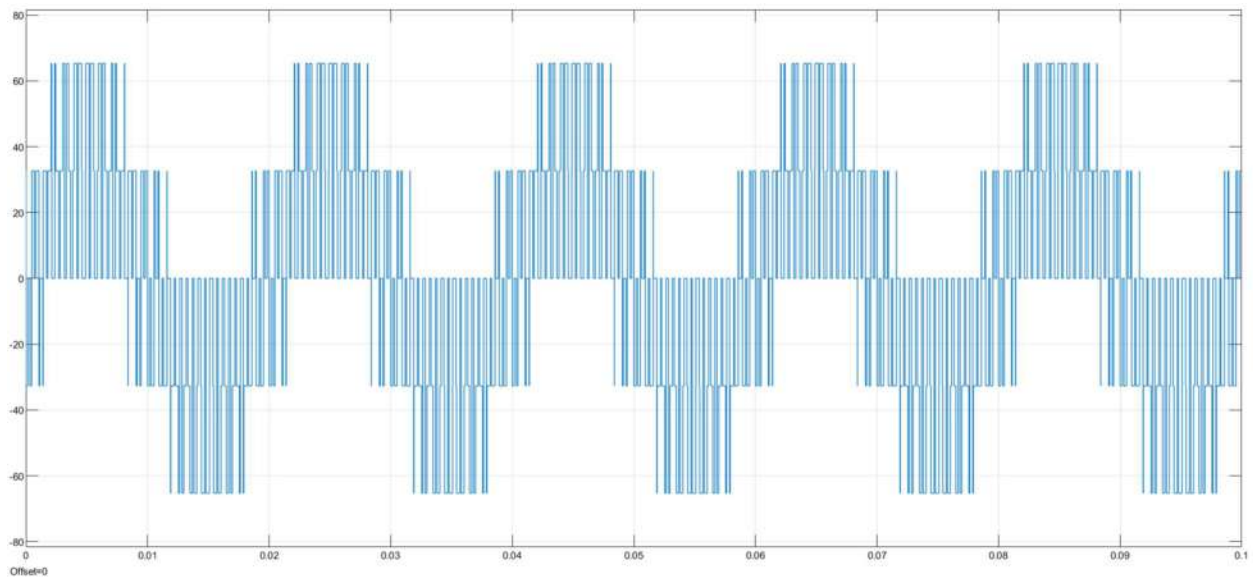**Sinusoidal PWM Technique referred to R-Phase**

G. Narayanamma Institute of Technology & Science
(For Women)
Shaikpet, Hyderabad – 500 104
Department of Electrical and Electronics Engineering

Electrical Systems Simulation Lab   I M.Tech – I Sem

Phase Voltage (V$_{RN}$)

Line Voltage (V$_{RY}$)

Electrical Systems Simulation Lab   I M.Tech – I Sem

TABULAR COLUMN:

S.No	Modulation Index (M)	Phase Voltage (V)	Line Voltage (V)

Result: The three phase Inverter is simulated using MATLAB/SIMULINK and its operation is studied using various waveforms.
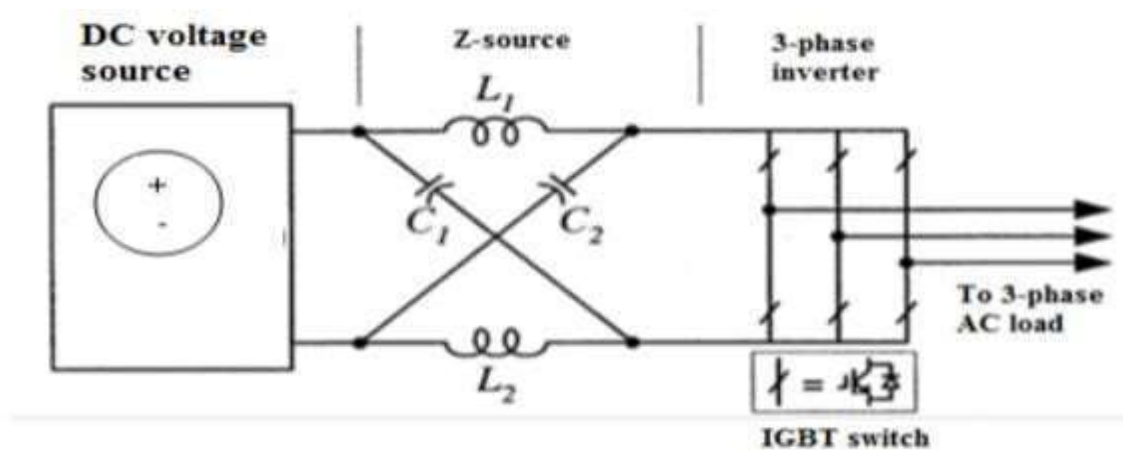
(Mr.P.Siva Prasad)   (Dr N. Malla Reddy)
Asst. Prof. – EEE   HOD - EEE

Electrical Systems Simulation   M. Tech – I Sem

Z-SOURCE INVERTER

Aim: To study the Z-Source Inverter with R-Load using MATLAB / SIMULINK software.

Circuit Diagram:

Fig. 1 Z – Source inverter

Theory:

The Z-source inverter is a buck–boost inverter that has a wide range of obtainable voltage. The traditional V- and I-source inverters cannot provide such feature. To describe the operating principle and control of the Z-source inverter, the three-phase Z-source inverter bridge has nine permissible switching states (vectors) unlike the traditional three-phase V-source inverter that has eight. The traditional three-phase V-source inverter has six active vectors when the dc voltage is impressed across the load and two zero vectors when the load terminals are shorted through either the lower or upper three devices, respectively. However, the three-phase Z-source inverter bridge has one extra zero state(or vector) when the load terminals are shorted through both the upper and lower devices of any one phase leg (i.e., both devices are gated on), any two phase legs, or all three phase legs. This shoot-through zero state (or vector) is forbidden in the traditional V-source inverter, because it would cause a shoot-through.

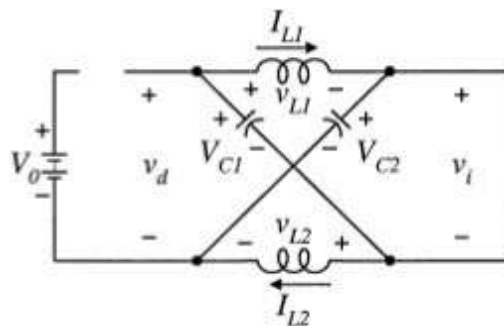Electrical Systems Simulation   M. Tech – I Sem

We call this third zero state (vector) the shoot-through zero state (or vector), which can be generated by seven different ways: shoot-through via any one phase leg, combinations of any two phase legs, and all three phase legs. The Z-source network makes the shoot-through zero state possible. This shoot-through zero state provides the unique buck-boost feature to the inverter.

Mode-1: Equivalent circuit of the Z-source inverter viewed from the dc link.
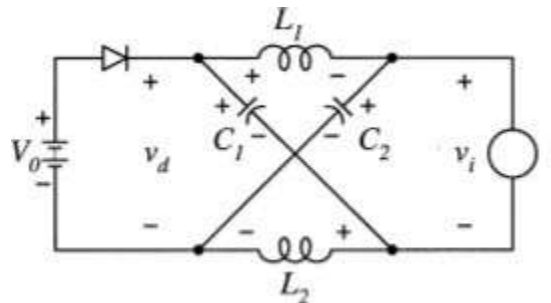
Mode-2: Equivalent circuit of the Z-source inverter viewed from the dc link when the inverter bridge is in the shoot-through zero state.

Electrical Systems Simulation   M. Tech – I Sem

Mode-3: Equivalent circuit of the Z-source inverter viewed from the dc link when the inverter bridge is in the shoot-through zero state.

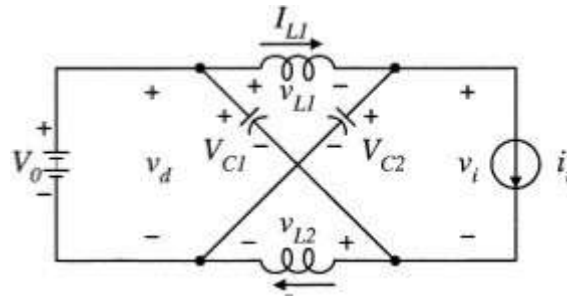**Analysis of Impedance Network:**

For simplicity, assuming that the inductors L1 and L2 and capacitorsC1 and C2 have the same inductance and capacitance respectively, the Z-source network become symmetrical. From the symmetry and the equivalent circuits, we have

$$V_{C1}=V_{C2}= V_C \text{-----------------------------}(1)$$

$$V_{L1}=V_{L2}=V_L \text{---------------------------}(2)$$

Given that the inverter bridge is in the shoot-through zero state for an interval ofT_0, during aswitching cycle, T and from the equivalent circuit, one has

$$V_L=V_C \; ; V_d=2V_c; \; V_i=0 \; ;$$

Now consider that the inverter bridge is in one of the eight non shoot-through states for aninterval of T1, during the switching cycle. From the equivalent circuit, one has

$$V_L=V_0\text{-}V_d \text{----------------------------------}(3)$$

$$V_d=V_0;$$

$$V_i=Vc\text{-}V_L=V_C\text{-}(V_0\text{-}V_C)V_i$$

$$=2V_c\text{-}V_0 \text{-----------------}(4)$$

Where V_O is the DC source voltage and $T=T_0+T_1$----------------- (5)

The average voltage of the inductor over one switching period should be zero in steady state,

G. Narayanamma Institute of Technology & Science
(For Women)
Shaikpet, Hyderabad – 500 104
Department of Electrical and Electronics Engineering

Electrical Systems Simulation   M. Tech – I Sem

thus,

$$V_L=[T_0V_C+T_1(V_0-V_C)]/T=0$$

$$V_L=(T_0V_C+V_0T_1-V_CT_1)/T=0$$

$$V_L=[(T_0-T_1)*V_C/T]+[(T_1V_0)/T]$$

$$(V_C/V_0)=(T_1/T_1-T) \text{-------------}(6)$$

Similarly, the average DC link voltage across the inverter bridge can be found as follows.

From equation 4:

$$V_i=[T_0 * 0+T_1(2V_c-V_0)] /T$$

$$2V_C=V_0 \text{--------------------------------} (7)$$

From equation 6:

$$V_C=V_0*T_1/(T_1-T_0)$$

The peak DC-link voltage across the inverter bridge is

$$V_i=V_c-V_L=2V_C-V_0$$

$$=T/(T1-T0) *V_0=B*V_0 \text{-------------}(8)$$

Where $B=T/(T_1-T_0) \geq 1 \text{--------------------} (9)$

B is a boost factor

The output peak phase voltage from the inverter

$$V_{out}=B*V_0 /2 \text{…..........} (10)$$

In this source

$$V_{out}=MB*V_0 /2 \text{Where M is the modulation index}$$

The output voltage can be stepped up and down by choosing an appropriate buck - boost factor B*

B*= B.M (it varies from 0 to α)   (11)

The capacitor voltage can be expressed as

$$V_{C1}=V_{C2}= V_C =(1-T_0/T)*V_C /(1-2T_0/T)$$

Electrical Systems Simulation   M. Tech – I Sem

The boost factor B is determined by the modulation index M. The boost factor B can becontrolled by duty cycle of the shoot-through zero state over the non-shoot through states of the PWMinverter. The shoot-through zero state does not affect PWM control of the inverter. Because, itequivalently produces the same zero voltage to the load terminal, the available shoot-through period islimited by the modulation index.

Pulse Width Modulation Schemes:

The basic idea to control the ZSI is to turn traditional zero state into shoot-through zero state, while keeping the active vectors unchanged, thus we can maintain the sinusoidal output and at the same time achieve voltage boost from the shoot-through of the dc link. Actually, this control strategy inserts shoot through in all the PWM traditional zero states during one switching period. This maintains the six active states unchanged as in the traditional carrier based PWM. In simple boost control method is illustrated by two straight lines, which are employed to realize the shoot through duty ratio (Do). The first one is equal to the peak value of the three-phase sinusoidal reference Since $Do = T_0/T$, thus voltages while the other one is the negative of the first one.

Whenever the triangular carrier signal is higher than the positive straight line or lower than the negative straight line, the inverter will operate in shoot-through. Otherwise it works as a traditional PWM inverter. Since the value of the positive straight line equals to the maximum of the sinusoidal reference signals and the value of the negative straight line equals to the minimum of the sinusoidal reference signal, then the modulation index (M) and the shoot-through duty ratio (Do) are interdependent each other. It is also observed that shoot-through duty ratio (Do) decreases with increasing modulation index (M).

Electrical Systems Simulation   M. Tech – I Sem
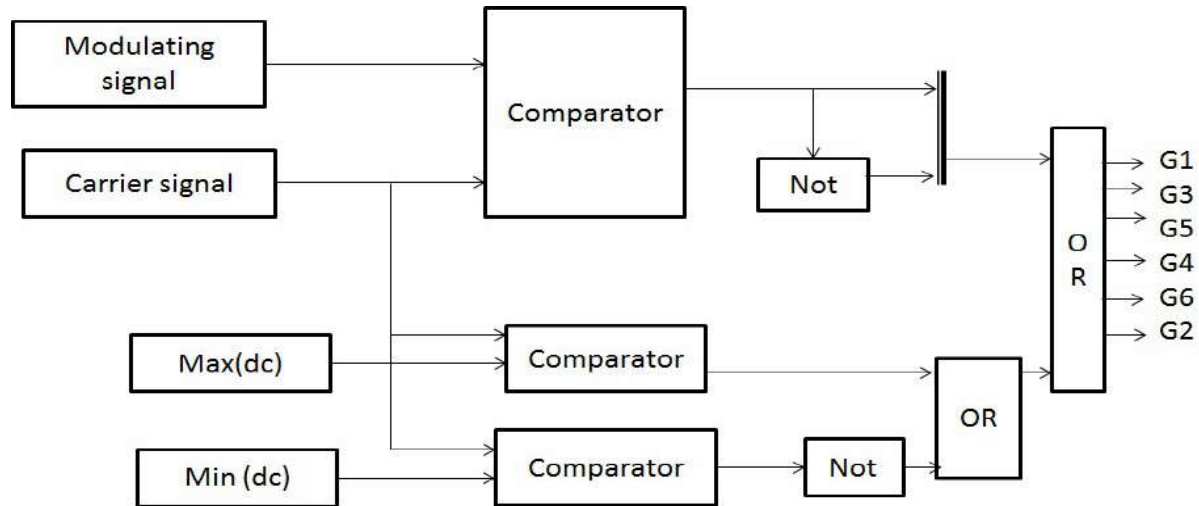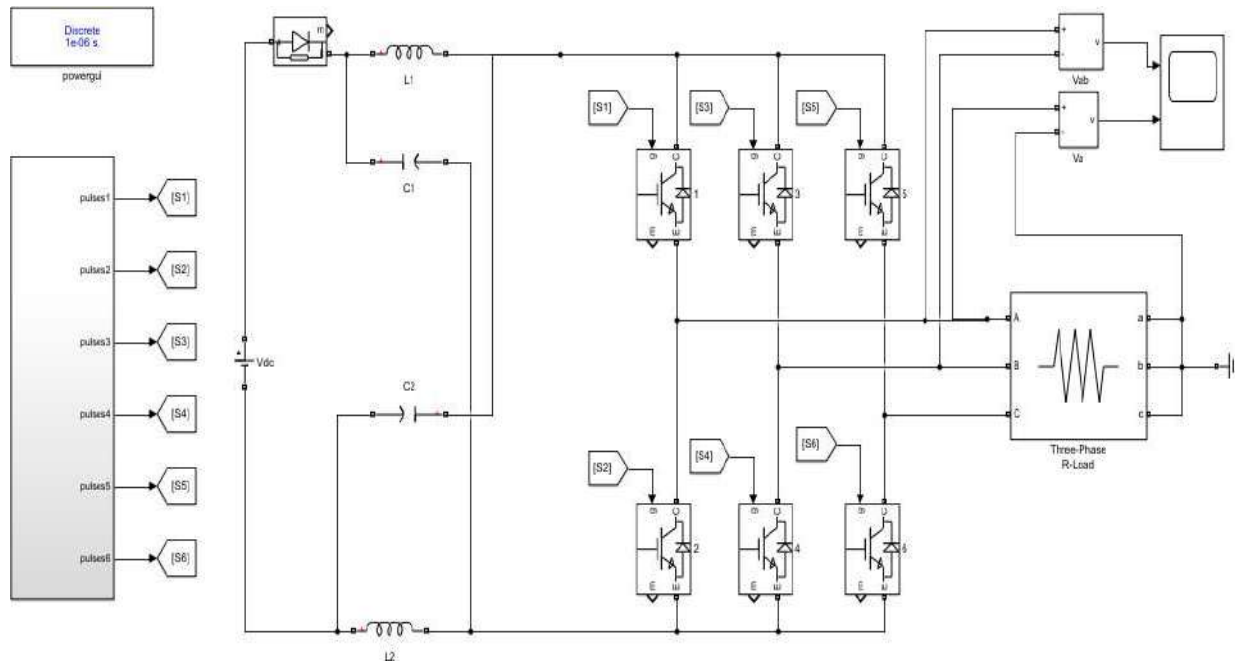

Implementation Diagram of SBC

Shoot-through pulses are inserted into the switching waveforms by logical OR gate. To produce switching pulses, three phase reference waveforms having peak value with modulation index (M) are compared with the same high frequency triangular signal. Comparator compares these two signals and produces pulses (when Vsin>Vtri, on and Vsin<Vtri, off). These pulses are then sent to gates of the power IGBT's through isolation and gate drive circuit. The pulse generation of the three phase leg switches (S1, S3 and S5-positive group/upper switches and S2, S4 and S6-negative group/lower switches).This method is much uncomplicated; however, the resulting voltage stress across the device is relatively high because some traditional zero states are not utilized either partially or fully. This characteristic will restrict the obtainable voltage gain because of the limitation of device voltage rating. For a complete switching period, Tis total switching period, T0is the zero state time period and Do is the shoot-through duty ratio. The control of ZSI is done by this control technique (SBC).

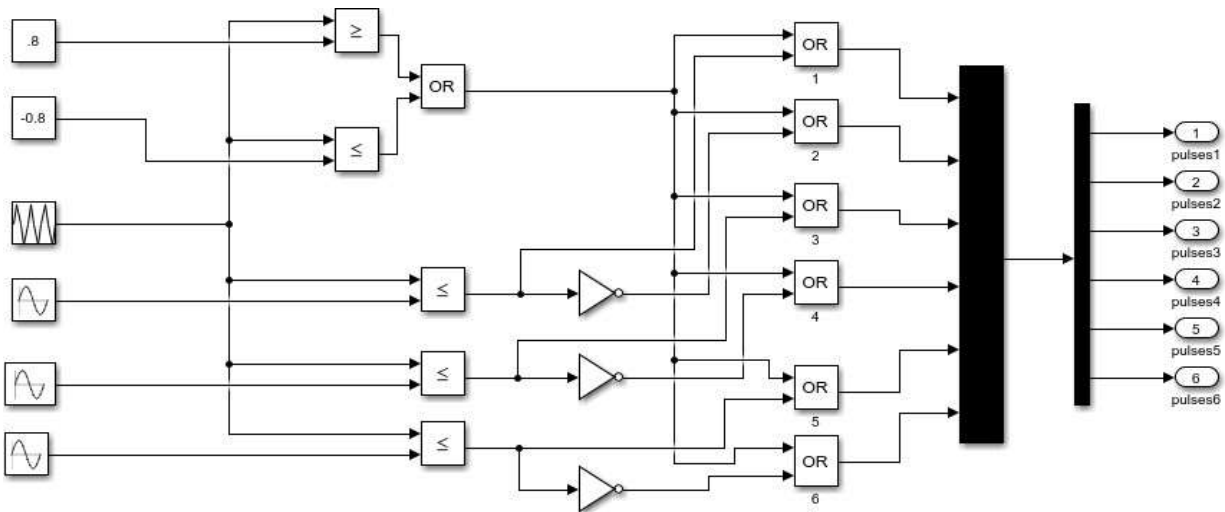Electrical Systems Simulation   M. Tech – I Sem

Theoretical Waveforms:
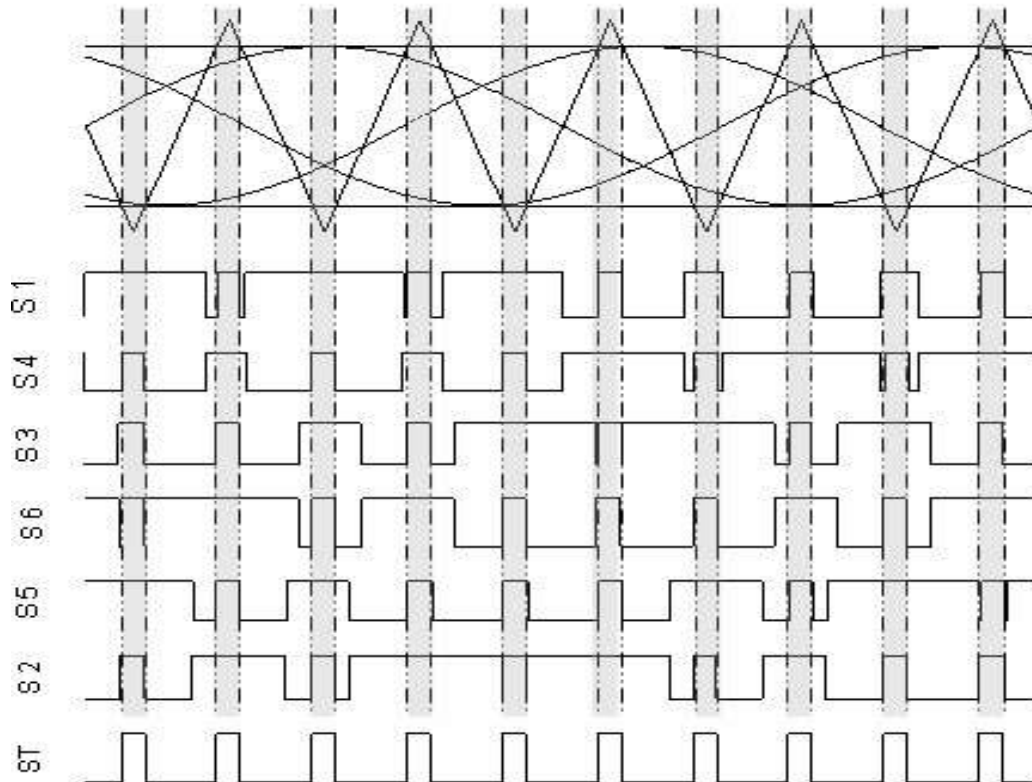
PWM Signals from Simple Boost Control

Electrical Systems Simulation   M. Tech – I Sem

MATLAB/SIMULINK Circuit:

Z-Source Inverter

Z-Source Inverter PWM Generation

Electrical Systems Simulation   M. Tech – I Sem

Design Specifications:

Considering the Parameters for the given models followed as:

Input DC Voltage(Vdc)=250V

Supply Frequency (fs) = 50 Hz ,

Switching Frequency (fsw)= 10 KHz ,

Three Phase Resistive Load = 10 ohm per phase

In a Z-source Network: $L_1=L_2=160\mu H$ and $C_1=C_2=1000\mu F$

Modulation Index (M) = 0.8

Time Period (T)= 10^-5 Sec

Shoot Through Period (T_0) = 0.2*10^-5 Sec

Boost factor= $\dfrac{1}{1-2\frac{T_0}{T}}$ =1.66

Output Voltage across Z-Source= M * B*Vdc= 0.8*1.66*250= 332V

Output across inverter= M*B* $\dfrac{V_{dc}}{2}$ =0.8*1.66*125=166V
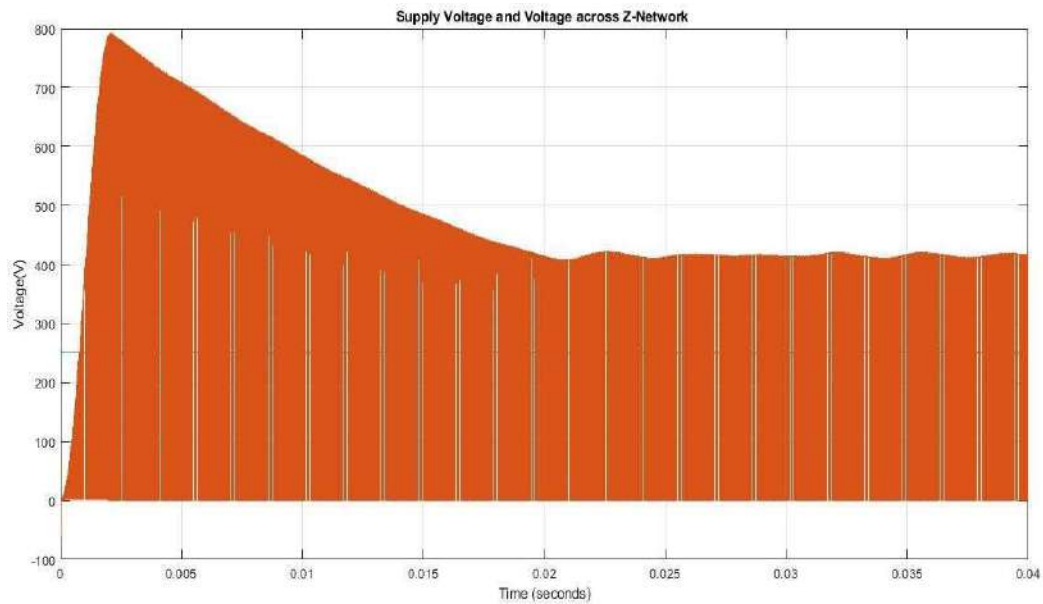
Procedure:

1. Rig up the circuit as per the circuit diagram in MATLAB/SIMULINK.
2. Observe the waveforms across the Z-source network, line voltage and phase voltages.
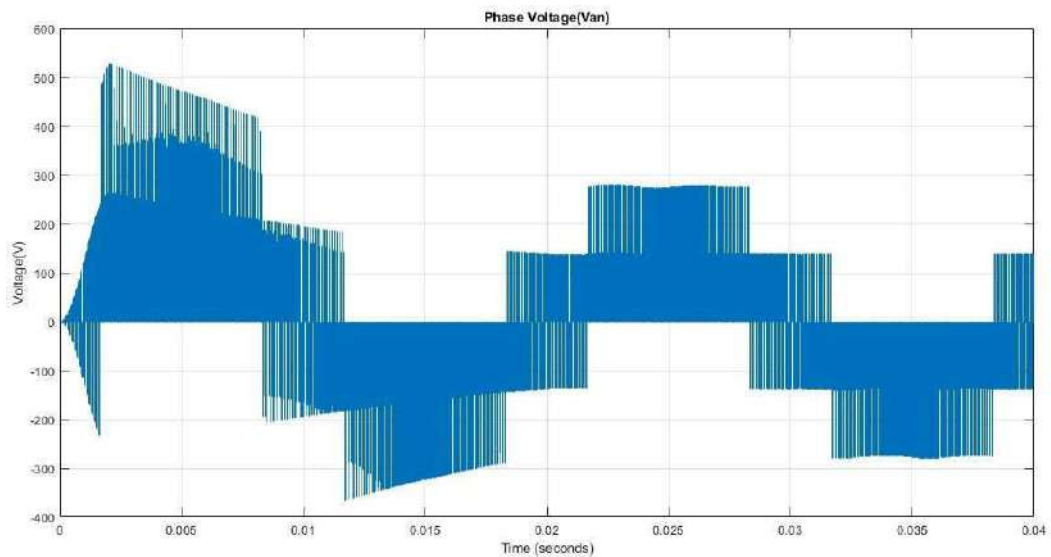
Electrical Systems Simulation   M. Tech – I Sem

Simulation Waveforms:

Supply Voltage and Voltage across the Z-Source netwok

Phase Voltage (Van)

G. Narayanamma Institute of Technology & Science
(For Women)
Shaikpet, Hyderabad – 500 104
Department of Electrical and Electronics Engineering

Electrical Systems Simulation   M. Tech – I Sem
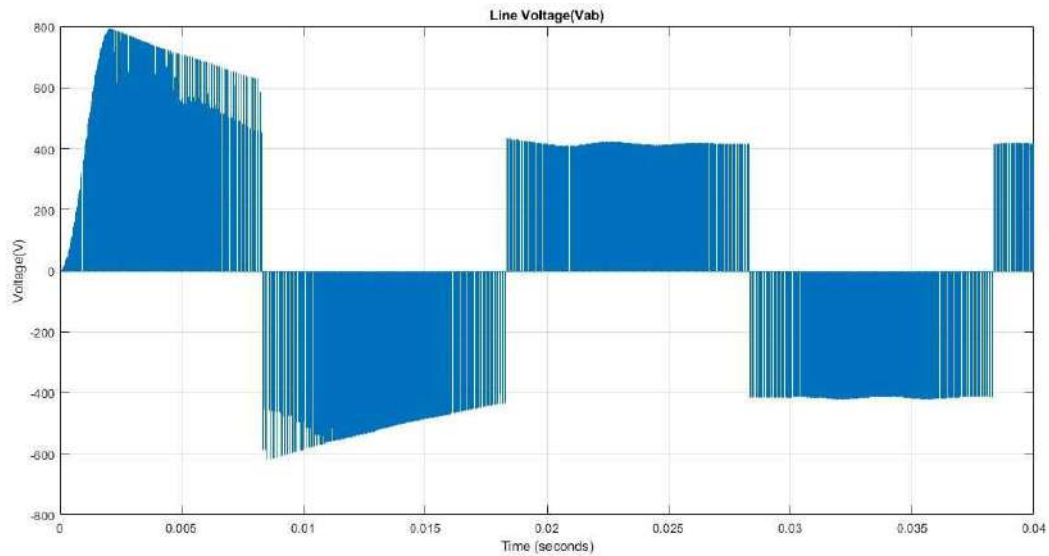
Line Voltage (Vab)

Result: Z-Source Inverter is simulated using MATLAB/SIMULINK and its operation is studied using various waveforms.

(Dr.G.Annapurna)   (Dr N. Malla Reddy)
Professor. – EEE   HOD - EEE

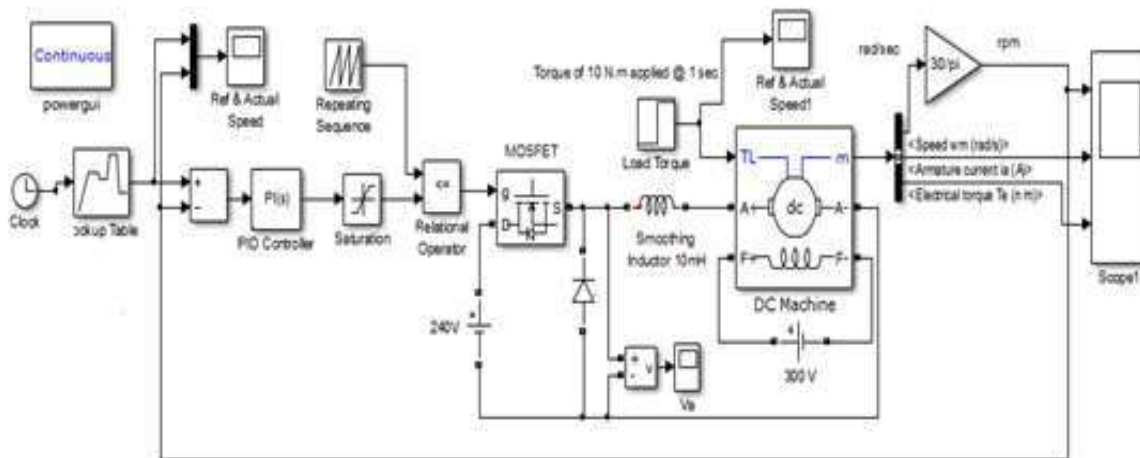Electrical Systems Simulation Lab   I M.Tech – I Sem

MODELLING AND SIMULATION OF SEPARATELY EXCITED D.C MOTOR TO STUDY THE DYNAMIC BEHAVIOUR OF THE MACHINE FOR CHANGE IN LOAD TORQUE

Aim: Simulation of Separately excited DC motor for controlling the speed using armature voltage control method.

Software : MATLAB/ SIMULINK

Circuit Diagram:

Fig.1. Simulink diagram for speed control of DC motor

Theory & Operation:

     Chopper drives are widely used in traction applications all over the world. A DC chopper is used to convert the fixed DC voltage to variable DC voltage and it is connected between a fixed voltage DC source and a DC motor to vary the armature voltage. The chopper in this application acts as a switch and the switch can be BJT, MOSFET, IGBT , SCR tec. A DC chopper can be used to step up or step down a fixed DC voltage.

Electrical Systems Simulation Lab   I M.Tech – I Sem

Separately excited DC motor consists of field and armature winding with separate supply. When a separately excited DC motor is excited by a field current , an armature current flows in the armature circuit. The back emf and torque will be developed in the motor and that torque is used to balance the load torque at a particular speed. Both armature current and field current are independent in separately excited DC motor. Hence, the speed of the separately excited DC motor can be varied by a) controlling the armature voltage and b) controlling the field current. The speed which corresponds to the rated armature voltage, rated field current and rated armature current is called as base speed. For a speed of the less than the base speed, the armature current and field current are kept constant to the torque demand and the armature voltage is varied to control the speed. Whereas for a speed of the motor higher than the base speed, the armature voltage is maintained at the rated value and the field current is varied to control the speed.

The speed of the DC motor changes with the load torque. In order to maintain constant speed, the armature voltage should be varied continuously by varying duty cycle of the chopper. As practical drive system is required to be operated at constant torque or constant power, the drive must have a closed loop feed back system. A closed loop control system has the advantage of improved accuracy, fast response and reduced effects of load disturbances.
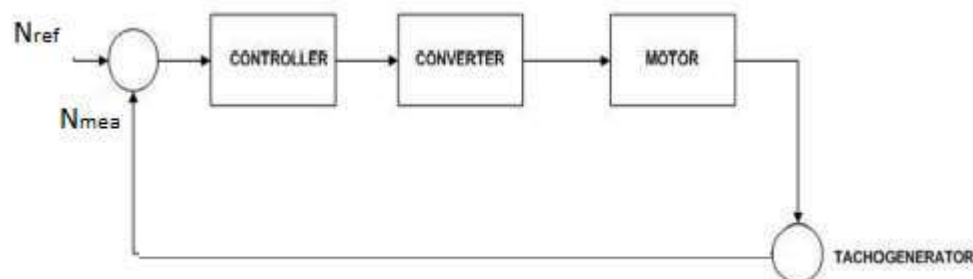
Fig. 2 Block diagram of closed loop speed control of DC motor

Electrical Systems Simulation Lab   I M.Tech – I Sem

The basic principle of dc motor speed control is that keeping field voltage constant, the output speed of the dc motor can be varied below and up to rated speed by controlling armature voltage. The output speed (measured speed) of the motor is compared with the reference speed, and the error signal is fed to speed controller. The output of the speed controller is fed to the converter pulse generator circuit and thus duty cycle of the converter will be varied . Thus the converter generates the required voltage to bring motor speed to the specified value.

Sometimes it is required to maintain constant speed irrespective of the load condition. For such condition, the reference speed is set at a constant value while the load torque is varied step wise. The feed back loop operates as explained above and the motor runs at constant set speed.

Procedure:

1. Rig up the circuit as given in the diagram.
2. Set the specifications of the motor as per requirement.
3. Set the ramp generator specifications suitably.
4. Variable speeds are set using look up table.
5. Run the simulation and trace the output waveforms.

Simulation parameters:

D.C. Motor specifications:

a) **Constant load torque**

Switching frequency = 2000 Hz

Speed data:

Time	0	0.5	1	1.1	1.5	1.6	2	2.1	2.5
Speed	1	1000	1000	850	850	1500	1500	500	500

PI : Proportional – 0.001

Integral   - 0.015

Saturation : Upper limit – 1   Lower limit – 0

Load torque : 10 N-m at 0.75

Electrical Systems Simulation Lab   I M.Tech – I Sem

b)Variable load torque:

Speed data:

Input value	0	0.5	1	1.1	1.5	1.6	2	2.1	2.5
Table data	1000	1000	1000	1000	1000	1000	1000	1000	1000

Load torque:

Time	0	0.5	1.1	1.6	2.1
Amplitude	0	2	4	6	10

Simulation Results:

a) Actual and Reference speed curves for constant load and variable speed condition

4

Electrical Systems Simulation Lab   I M.Tech – I Sem

b) Output waveforms for variable speed and constant torque

c) Reference and actual speeds for constant speed and variable load condition

G. Narayanamma Institute of Technology & Science
(For Women)
Shaikpet, Hyderabad – 500 104
Department of Electrical and Electronics Engineering

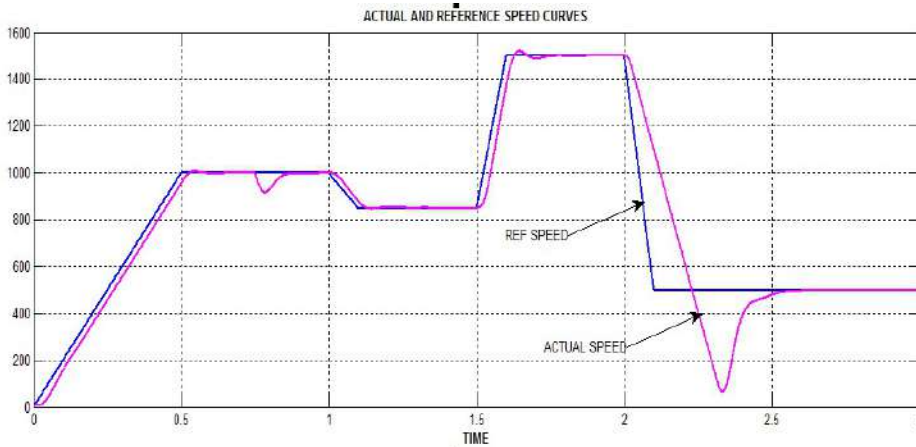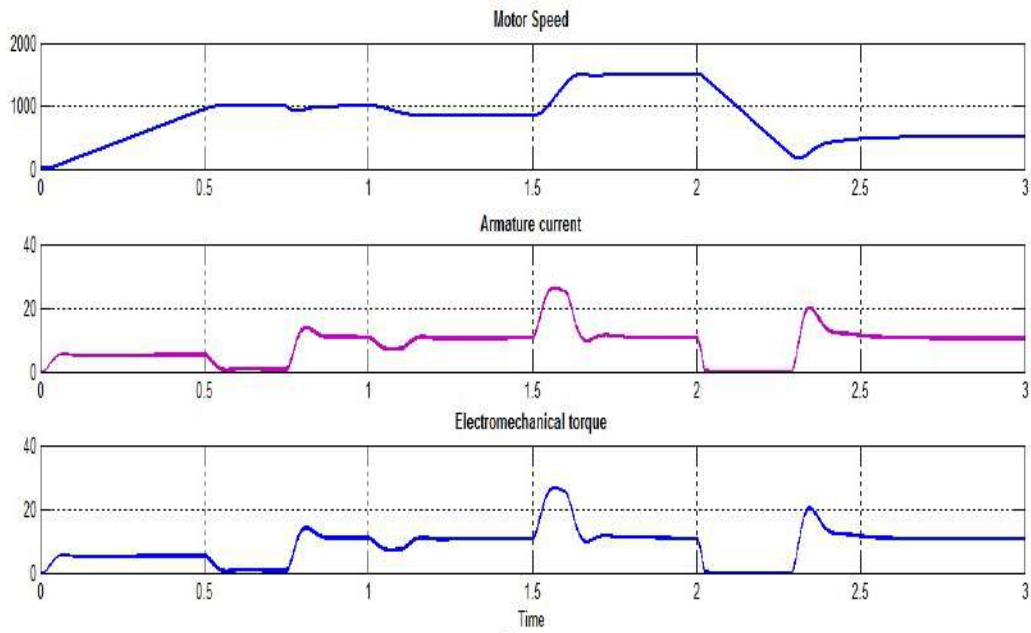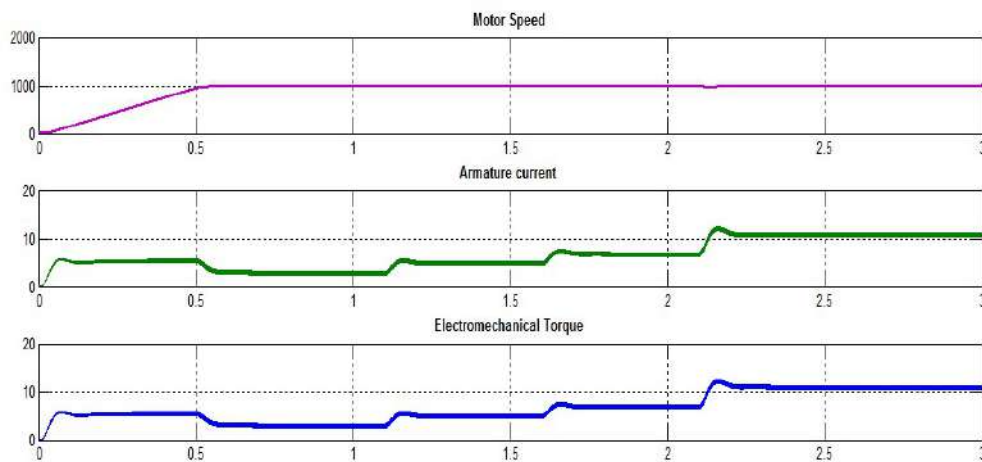Electrical Systems Simulation Lab   I M.Tech – I Sem
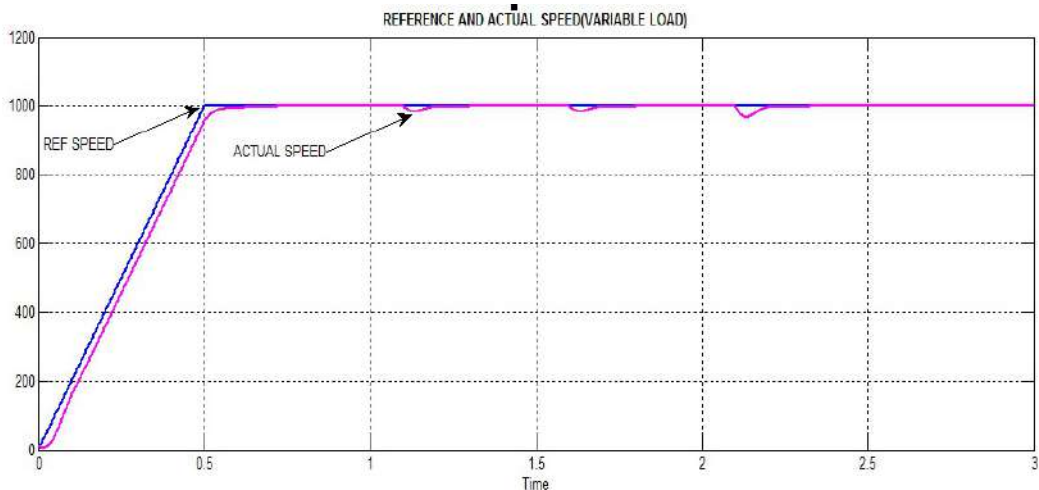
d) Output waveforms for constant speed and variable torque

Result : The separately excited DC motor is simulated using MATLAB/SIMULINK and its operation is studied for control of speed under various load conditions.

(Dr G.Annapurna)   (Dr N. Malla Reddy)

Professor. – EEE   HOD - EEE

Electrical Systems Simulation   M. Tech – I Sem

Aim : To mathematically modelled Separately excited DC Motor
SOFTWARE: MATLAB/SIMULINK
Theory:
The speed of DC motor is directly proportional to armature voltage and inversely proportional to flux in field winding. In armature controlled DC motor the desired speed is obtained by varying the armature voltage. This speed control system is an electro-mechanical control system.
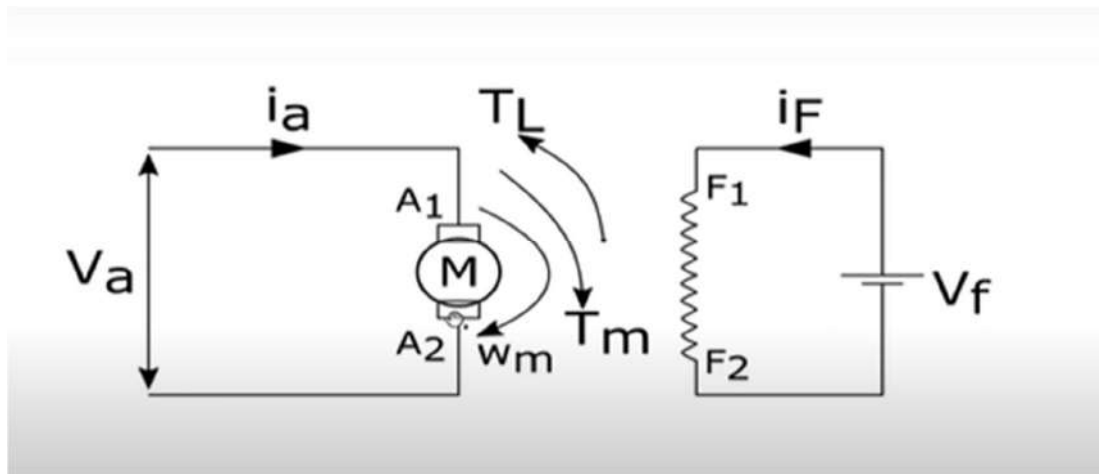
        The electrical system consists of the armature and the field circuit but for analysis purpose, only the armature circuit is considered because the field is excited by a constant voltage.

The mechanical system consists of the rotating part of the motor and load connected to the shaft of the motor.
The armature controlled DC motor speed control system is shown in the below figure.

Let R_a = Armature resistance,

           L_a = Armature inductance

           I_a = Armature current

           V_a = Armature voltage

           e_b = Back emf

           K_e = Torque constant

Electrical Systems Simulation   M. Tech – I Sem
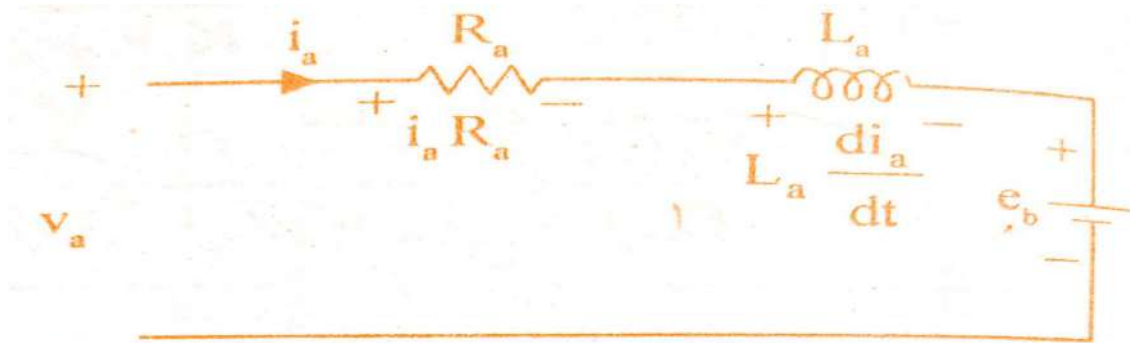
T_m = Torque developed by motor

θ = Angular displacement of shaft

J = Moment of inertia of motor and load

B = Frictional coefficient of motor and load

K_e = Back emf constant

The equivalent circuit of armature is shown in the below figure.

By Kirchoff's voltage law, we can write,

$$V_a = i_a R_a + L_a \frac{d}{dt} i_a + E_b$$
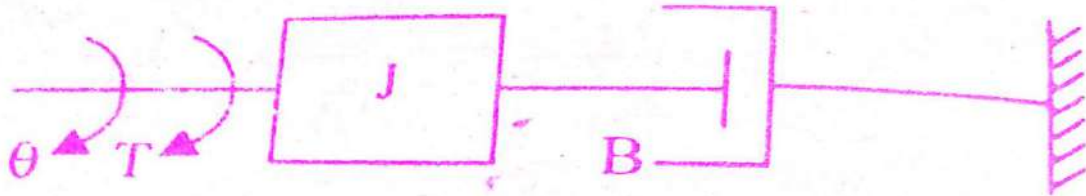
Torque of DC motor is proportional to the product of flux and current. Since flux is constant in this system, the torque is proportional to i_a alone.

$$T\alpha i_a$$

Torque $T = k_e i_a$

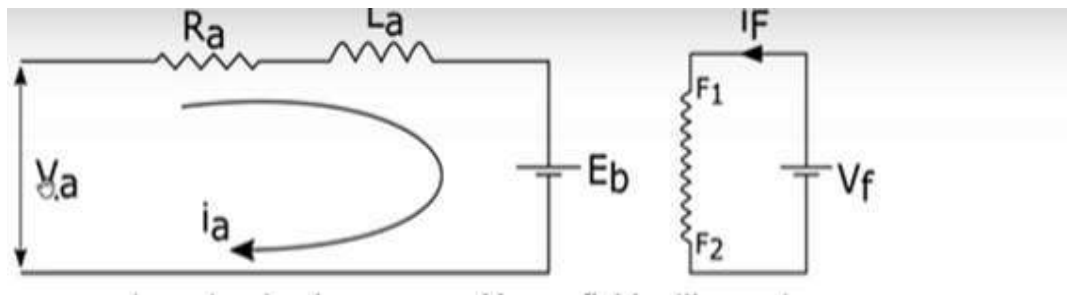Electrical Systems Simulation   **M. Tech – I Sem**

The mechanical system of the motor is shown in the below figure.

The back emf of DC machine is proportional to speed (angular velocity) of shaft.

$$\therefore e_b \propto \frac{d\Theta}{dt} \quad \text{or} \quad \text{Back emf, } e_b = K_b \frac{d\Theta}{dt}$$

Field Voltage is maintained constant. Hence field will remain constant.

Mathematically, $(K_e \, \phi) = $ Constant

Governing Equations are as follows:

$$V_a = i_a R_a + L_a \frac{d}{dt} i_a + E_b \qquad\qquad (1)$$

$$E_b = (K_e \, \phi) \, \omega_m \qquad\qquad (2)$$

Electrical Systems Simulation   M. Tech – I Sem

$$T_m ==(K_e\ \boldsymbol{\phi})\ I_a ==T_L +J\frac{d}{dt}\omega_m + B\omega_m \qquad (3)$$

Taking Laplace Transforms

$$V_a(s) - E_b(s) = I_a(s)[R_a + sL_a] \qquad (4)$$

$$I_a(s) = \frac{V_a(s) - E_b(s)}{R_a + sL_a}$$

$$I_a(s) = \frac{V_a(s) - E_b(s)}{R_a(1 + s\frac{L_a}{R_a})}$$

$\frac{L_a}{R_a} = \tau_a$, where τ_a is the time constant of armature circuit

Hence, the armature current is

$$I_a\ (s) = \frac{V_a(s) - E_b\ (s)}{R_a\ (1+s\tau_a)} = \frac{V_a(s) -(K_e\ \boldsymbol{\phi})\ \omega_m(s)}{R_a\ (1+s\tau_a)} \qquad (5)$$
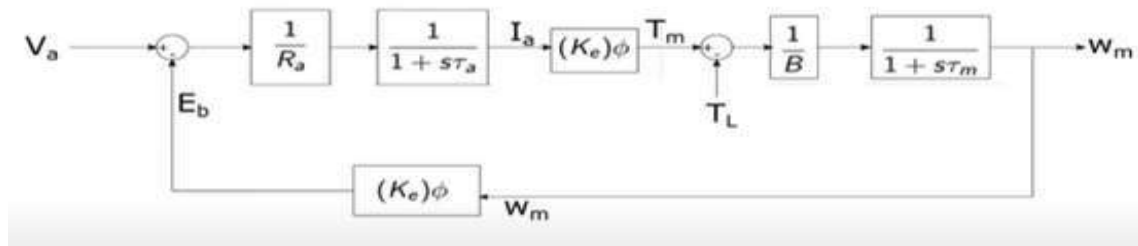
$$T_m(s) - T_L(s) = \omega_m(s)[B + sJ] \qquad (6)$$

$$\omega_m(s) = \frac{T_m(s) - T_L(s)}{B + sJ}$$

$$\omega_m(s) = \frac{T_m(s) - T_L(s)}{B(1 + s\frac{J}{B})}$$

$\frac{J}{B} = \tau_m$, where, τ_m is the mechanical time constant of motor

Hence, the motor speed is

$$\omega_m(s) = \frac{T_m(s) - T_L(s)}{B(1 + s\tau_m)} = \frac{(K_e\phi)\ I_a(s) - T_L(s)}{B(1 + s\tau_m)} \qquad (7)$$

Electrical Systems Simulation   M. Tech – I Sem

Specifications:
V_a =220V
R_a= 0.5Ω
L_a=0.003H
K_e ϕ = 0.8
B=0.01
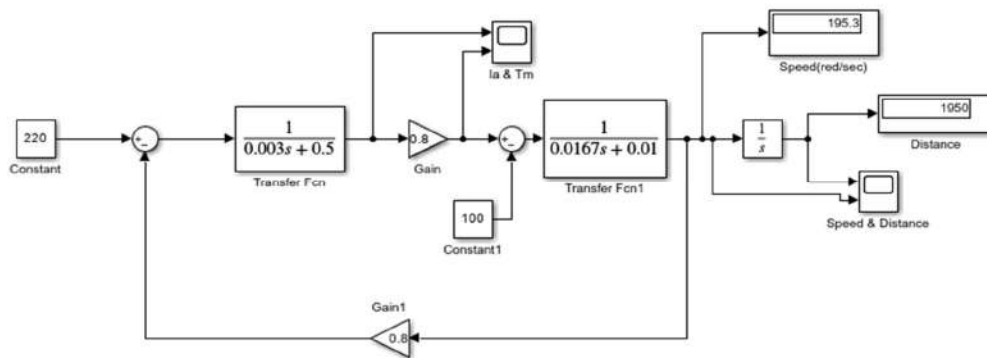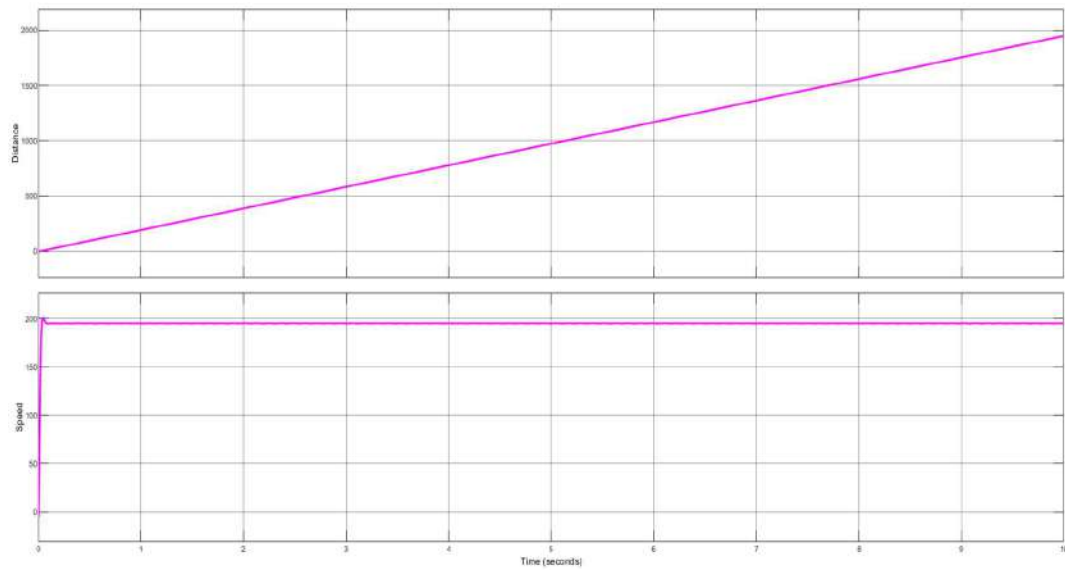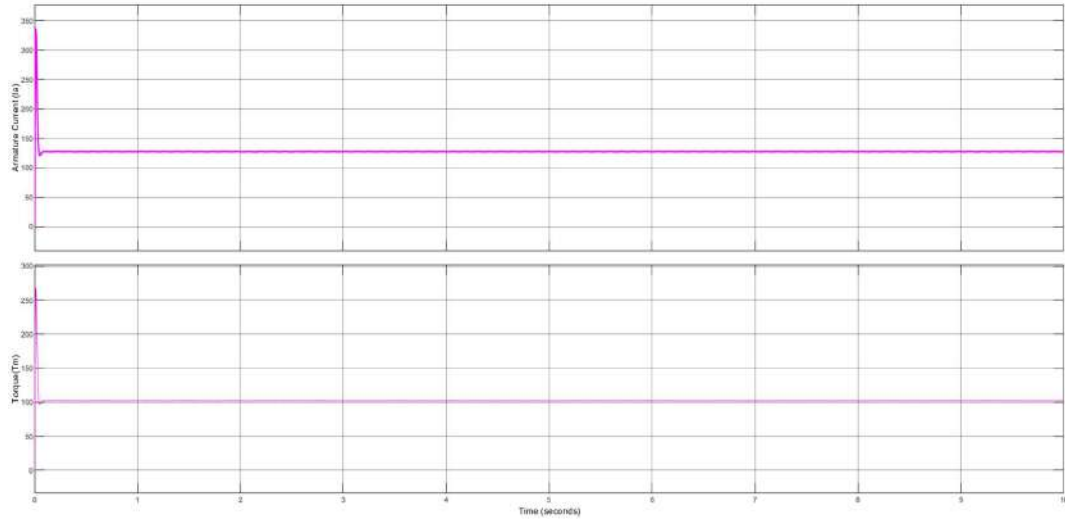J=0.0167
T_L= 100 N-m

Procedure:
1. Rig up the circuit as per the circuit diagram in MATLAB/SIMULINK model Platform.
2. Observe the waveforms of speed (rad/sec) and distance in scope
3. Note down the values of speed and distance in display

Simulink Model of Separately Excited DC Motor

Electrical Systems Simulation   M. Tech – I Sem

Simulation Results

Electrical Systems Simulation   M. Tech – I Sem

Results: Mathematically modelling of Separately excited Dc Motor is done and the waveforms are observed

(Mrs.Y.Priyanka)   (Dr N. Malla Reddy)
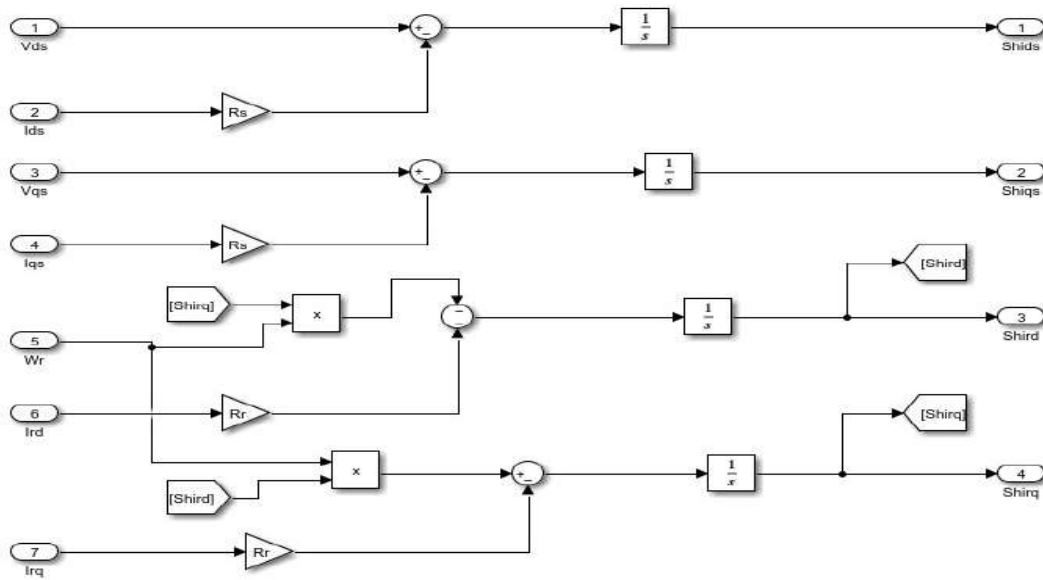
Asst. Prof. – EEE   HOD - EEE

Electrical Systems Simulation Lab   I M.Tech – I Sem

Sub System-1

Sub System-2

Electrical Systems Simulation   M. Tech – I Sem

Electrical Systems Simulation   M. Tech – I Sem
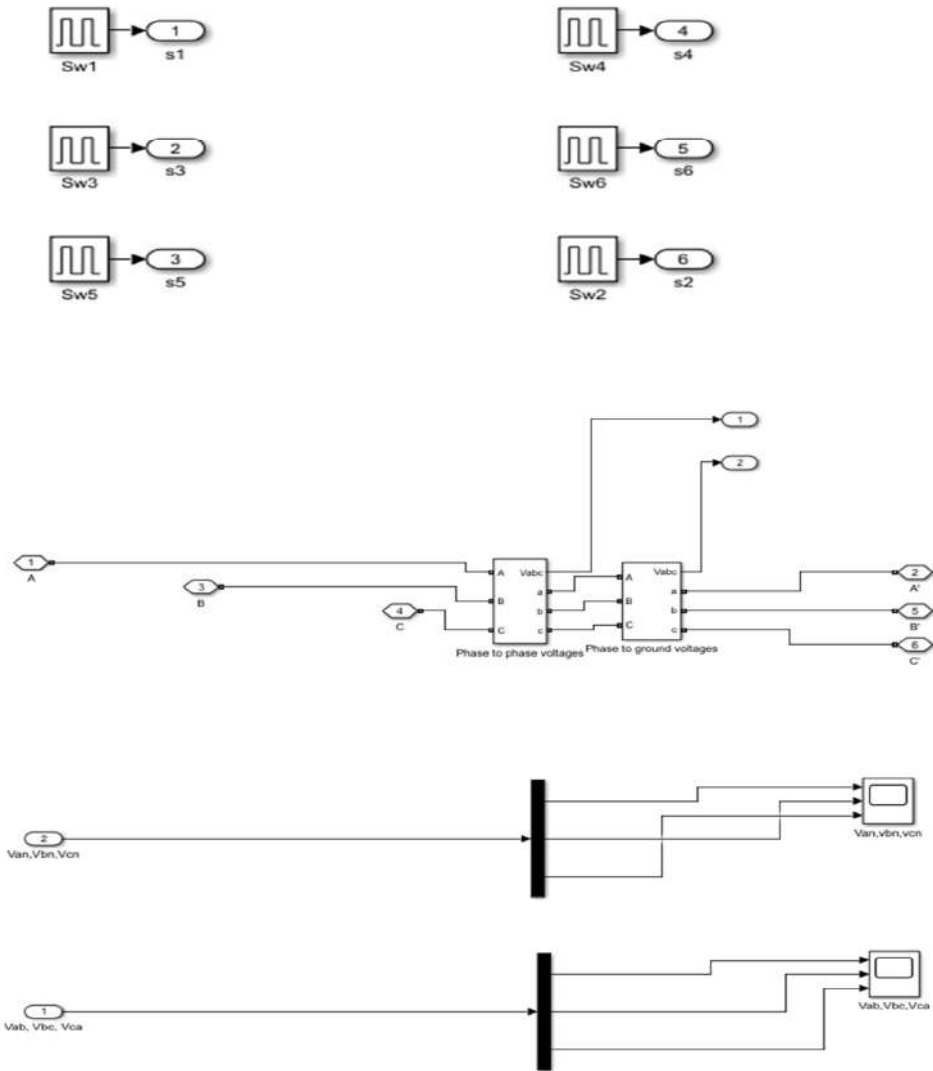
ROOT LOCUS, NYQUIST AND BODE PLOTS

ROOT LOCUS:

Aim:

To obtain the root locus of the system whose transfer function is given by

$$G(s) = \frac{s+5}{s^2 + 7s + 25}$$

Theory:

The root locus technique is used for analysis of control system. It is a powerful tool for adjusting location if closed loop poles to achieve the desired system performance by varying one or more system parameters.

The path taken by the roots of characteristic equation when open loop gain k is varied from o to ∞ is called root locus.

For the open loop transfer function with unity feedback is given by

$$\frac{G(s)}{1 + G(s)} = \frac{k}{s(s + p_1)(s + p_2) + k}$$

Its characteristics equation is S(S+P1) (S+P2) +K = 0. It is clear that the roots of characteristics equation are different.

Procedure:

1. Input the numerator and denominator co-efficient.
2. Formulate the transfer function using the numerator and denominators co-efficient with the help of function T = tf (num, den)
3. Plot the root locus of the above transfer function using rlocus(t).

G. Narayanamma Institute of Technology & Science(For Women)
Shaikpet, Hyderabad – 500 104
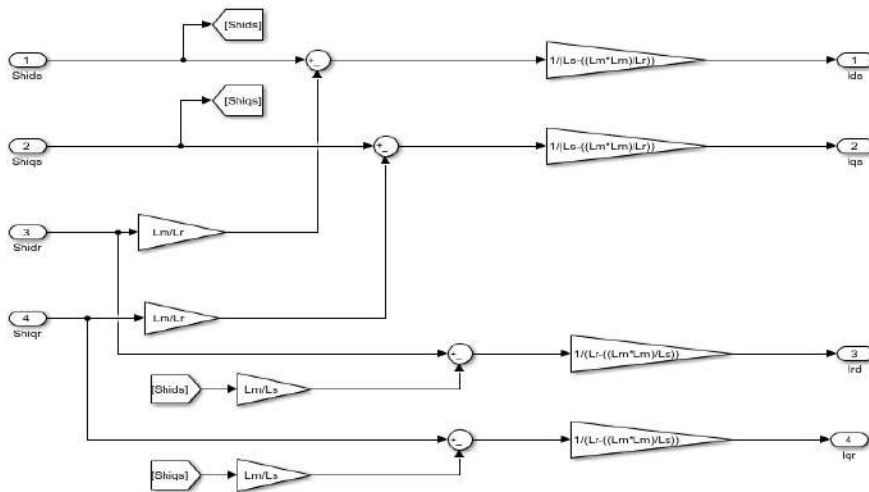### Department of Electrical and Electronics Engineering

Electrical Systems Simulation   M. Tech – I Sem

%Program to find the root locus of transfer function%

    s+5

% ....

% s^2+7s+25

clc;

clear all;

close all;

% initializations

num=input('enter the numerator coefficients---->');

den=input('enter the denominator coefficients --- >');

%Transfer function

sys=tf(num,den);

rlocus(sys);

Result:

enter the numerator coefficients--- >[1 5]

enter the denominator coefficients --- >[1 7 25]

Electrical Systems Simulation   M. Tech – I Sem
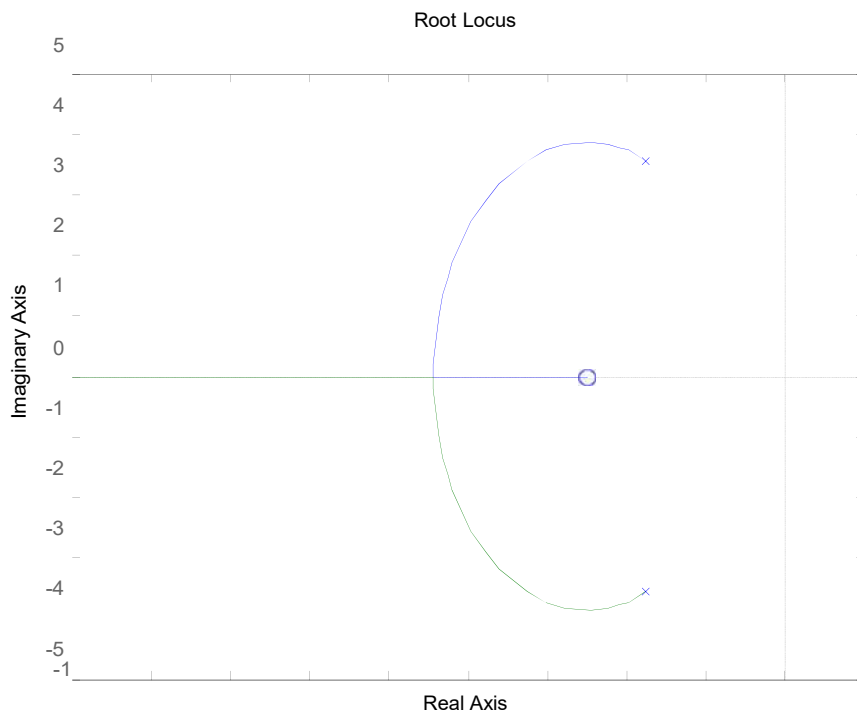
BODE PLOT:

Aim:

To obtain the bode plot and to calculate the phase margin, gain margin, phase cross over and gain cross over frequency for the systems whose open loop transfer function is given as follows.

$$G(s) = \frac{25(s+1)(s+7)}{s(s+2)(s+4)(s+8)}$$

Theory:

The bode plot is a frequency response plot of transfer function of a system. A bode plot consists of two graphs. One is a plot of magnitude of a sinusoidal transfer

Electrical Systems Simulation   M. Tech – I Sem

function versus logω, phase cross over frequency ω_{PC}. It is the frequency at which phase of a $(j\omega)$ is -180^0.

Gain Margin:

The gain margin in db is given by negative of db. Magnitude of $(j\omega)$ at $\omega_{PC.}$

Gain Cross Over Frequency:

It is the frequency at which db magnitude of a $(j\omega)$ is zero.

Phase Margin:

It is given by $v= 180^0+ \phi_{gc}$. Where ϕ_{gc} is the phase angle a $(j\omega)$ at gain cross over frequency.

Procedure:

1. Input the zeroes, poles and gain of the given system.
2. Formulate the transfer function from zeroes, poles and gain of the system.
3. Plot the bode plot using function bode (t).
4. Estimate Pm,Gm, W_{PC}, and W gc. Using function margin (λ).

```
%Program to find Bode Plot
%  25(s+1)(s+7)
% ........................
% s(s+2)(s+4)(s+8)
clc;
clear all;
close all;
% initializations
k=input('enter the gain--- >');
z=input('enter the zeros--- >');
p=input('enter the ploes---->');
t=zpk(z,p,k);
bode(t);
```

Electrical Systems Simulation   M. Tech – I Sem

```
[Gm,Pm,Wcg,Wcp]=margin(t);
disp(Gm);
disp(Pm);
disp(Wcg);
disp(Wcp);
```

Result:

enter the gain --->25

enter the zeros---->[-1 -7]

enter the ploes---->[0 -2 -4 -8]
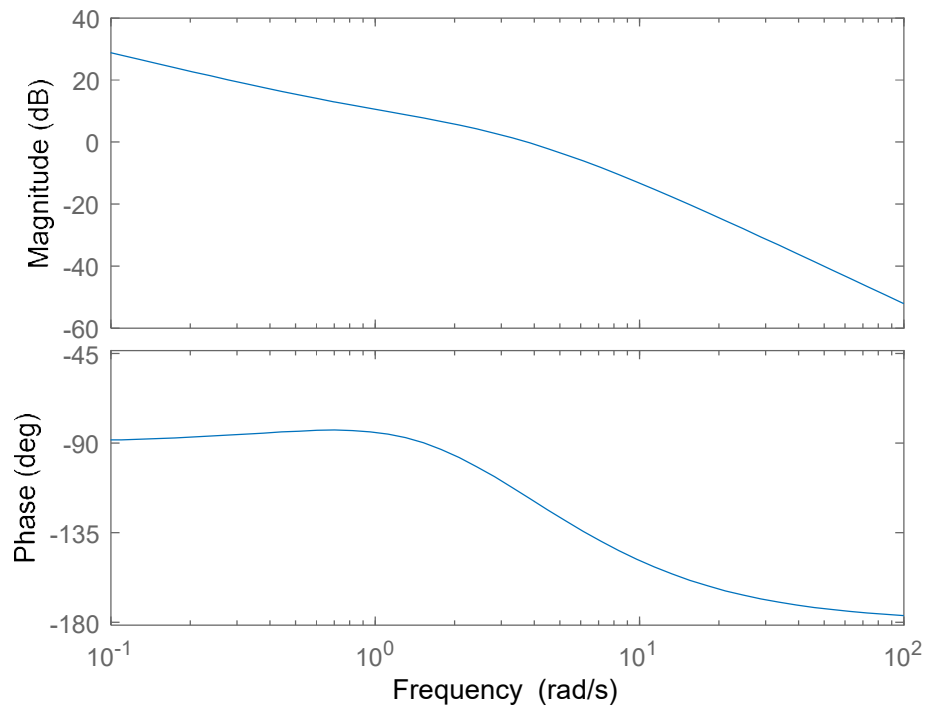
Gm= Inf

 Pm= 63.1105

 Wcg= Inf

Wcp= 3.7440

Bode Diagram

G. Narayanamma Institute of Technology & Science(For Women)
Shaikpet, Hyderabad – 500 104
Department of Electrical and Electronics Engineering

Electrical Systems Simulation   M. Tech – I Sem

NYQUIST PLOT:

AIM:

        To obtain the Nyquist plot and to calculate the phase margin, gain margin, phase cross over and gain cross over frequency for the systems whose open loop transfer function is given as follows.

$$G(s) = \frac{50(s+1)}{s(s+3)(s+5)}$$

THEORY:

        The Nyquist plot is a transfer function G (jω) is a plot of the magnitude of G (jω) versus the phase angle of G (jω) on polar coordinates as w is varied from zero to infinity. Thus Nyquist plot is the locus of vectors |G (jω)|<_G (jω) as w is varied from zero to infinity.

Procedure:

1. Input the zeroes, poles and gain of the given system.
2. Formulate the transfer function from zeroes, poles and gain of the system.
3. Plot the bode plot using function bode (t).
4. Estimate Pm,Gm, W $_{PC}$, and W gc. Using function margin (λ).

%Program to find the Nyquist Plot

%   50(s+1)

% ....

% s(s+3)(s+5)

clc;

clear all;

close all;

% initializations

num=input('enter the numerator coefficients---->');

den=input('enter the denominator coefficients --- >');

sys=tf(num,den);

Electrical Systems Simulation   M. Tech – I Sem

```
nyquist(sys);
title('system1');
[Gm,Pm,Wcg,Wcp]=margin(sys);
disp(Gm);
disp(Pm);
disp(Wcg);
disp(Wcp);
```

Result

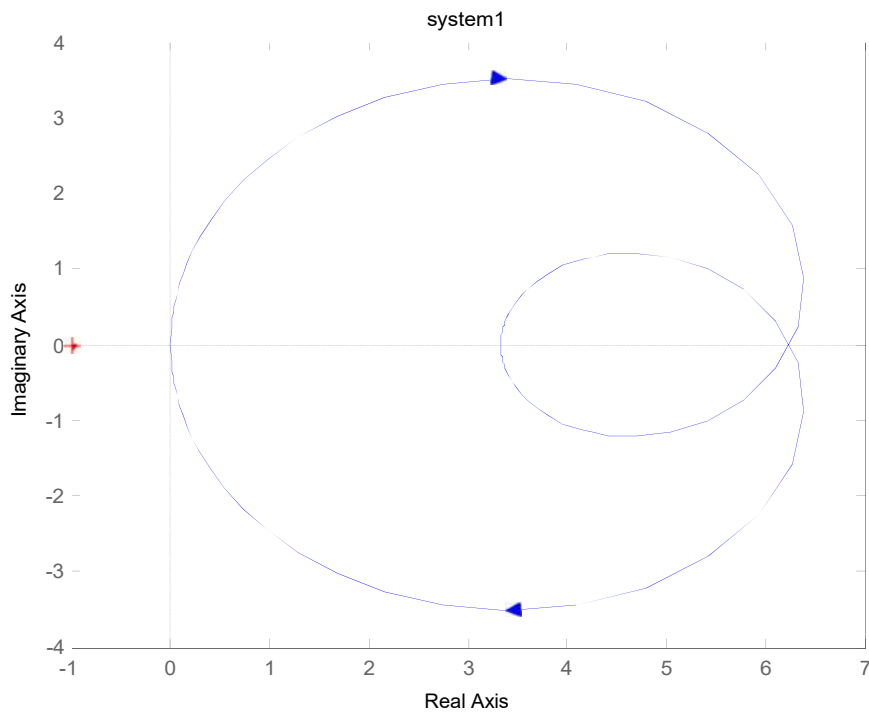enter the numerator coefficients --- >[50 50]

enter the denominator coefficients --- >[1 8 15]

Gm= Inf

Pm= 98.0516

Wcg=Inf

Wcp=49.6681

Electrical Systems Simulation   M. Tech – I Sem

RESULTS & DISCUSSIONS:
Root Locus, Bode plot and Nyquist plot determined using the built-in functions of MATLAB.
• They are a powerful tool to design systems to required performance.
• In order to determine the stability of the system using the root locus technique we find the range of values of k for which the complete performance of the system will be satisfactory and the operation is stable.
 • Bode plots provides relative stability in terms of gain margin and phase margin
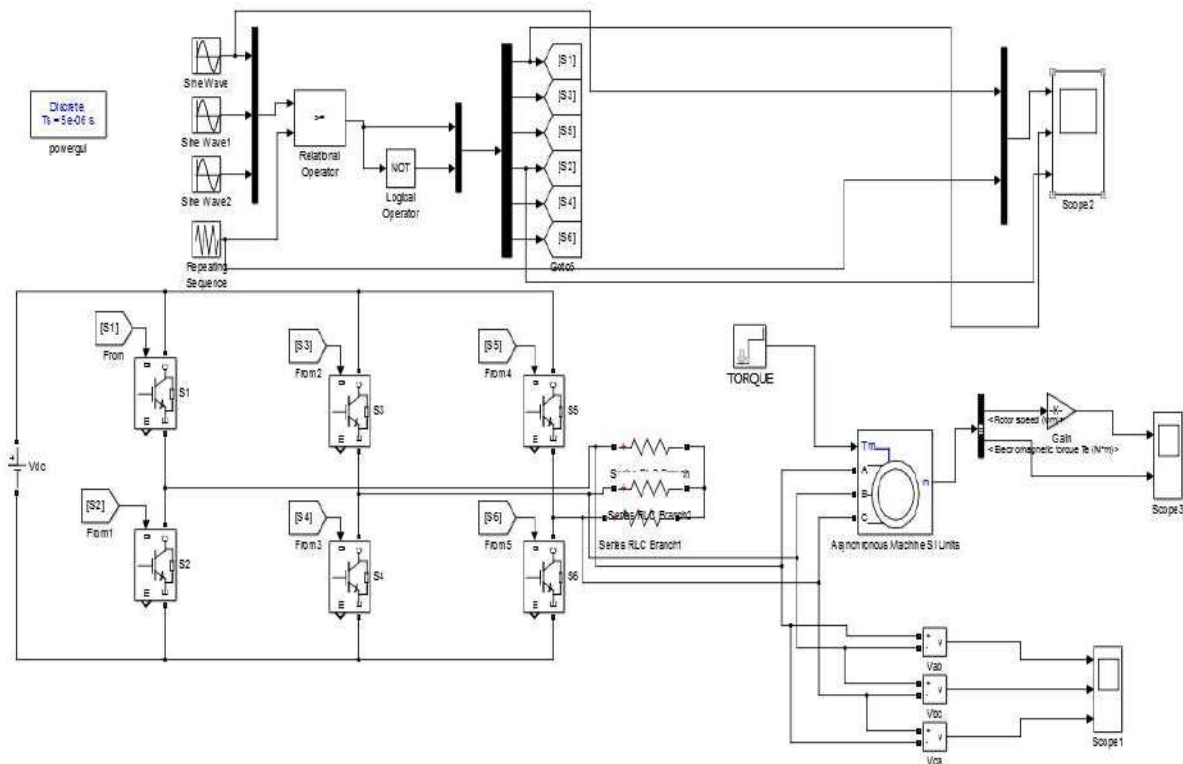

            (Mrs.Y.Priyanka)   (Dr N. Malla Reddy)
             Asst. Prof. – EEE   HOD - EEE

Electrical Systems Simulation Lab   I M.Tech – I Sem

SIMULATION AND ANALYSIS OF 3-Φ PWM INVERTER FED INDUCTION MOTOR

Aim: Simulation of three phase inverter fed Induction Motor for controlling the speed using variable supply voltage method.
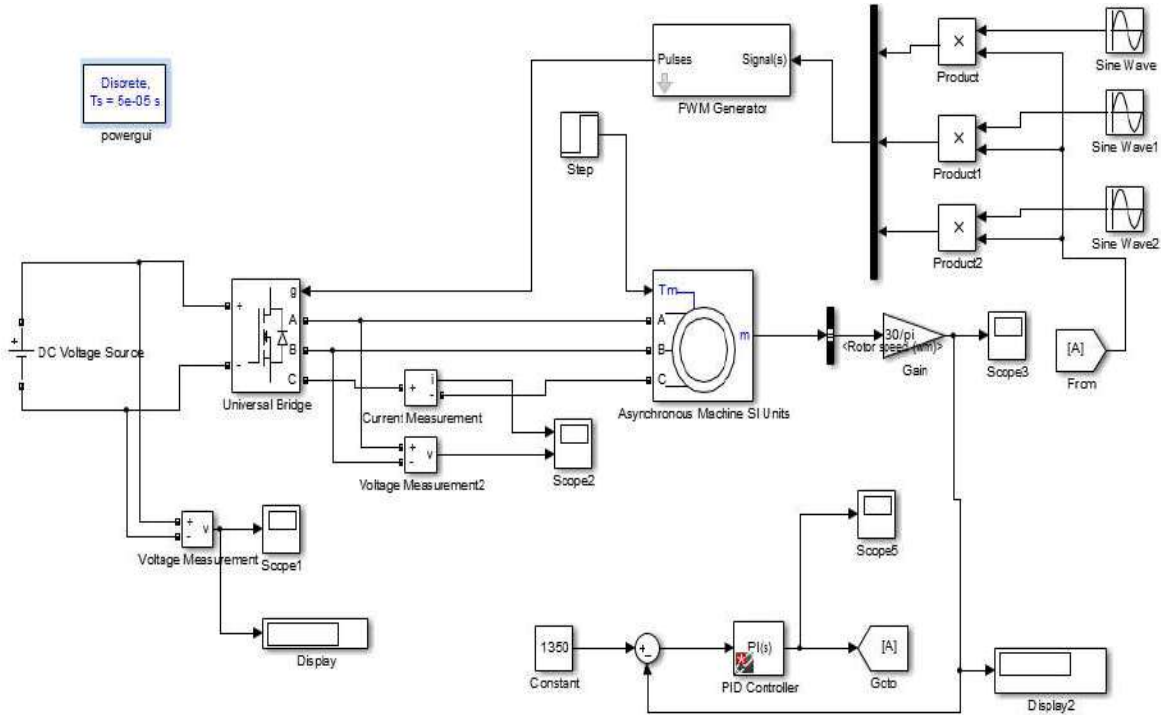
Software : MATLAB/ SIMULINK

Circuit Diagram:

Fig.1. Simulink diagram for speed control of Induction Motor using open loop control method

G. Narayanamma Institute of Technology & Science
(For Women)
Shaikpet, Hyderabad – 500 104
Department of Electrical and Electronics Engineering

Electrical Systems Simulation Lab   I M.Tech – I Sem

Fig.2. Simulink diagram for speed control of Induction Motor using closed loop control method

Theory & Operation:

DC motors have been used during the last century in industries for variable speed applications due to its easy controllability. Now-a- days, induction motor has been the workhorse of industry due to its robustness, low cost, high efficiency and less maintenance. Earlier, induction motors were mainly used for essentially constant speed applications because of the unavailability of the proper speed control techniques. The advancement of power electronics has made it possible to vary the speed of induction motor by varying supply voltage, supply frequency or both. By the Torque-Speed characteristics, we can analyze the performance of the induction motor.

Electrical Systems Simulation Lab   I M.Tech – I Sem

The Induction motor can be simulated using Open loop control and Closed loop control method. Desired speed range can be achieved by processing the error between the actual speed and set speed using a PI controller for the closed loop control. Whereas in open loop control, based on the load on the motor, the speed of the motor will vary.

Three-phase inverters are widely used in AC motor drive applications, in three-phase grid connection to wind generators and other power system applications. The DC source is usually in the form of a battery, solar cell, rectified wind generator output or a rectified DC output from the fixed AC supply from the utility. The input may have characteristics of a voltage source or a current source.

In Sinusoidal PWM, the width of each pulse is varied in proportion to the amplitude of the sine wave evaluated at the centre of the same pulse. The gating signals are generated by comparing a sinusoidal reference wave with a triangular carrier wave of frequency Fr and Fc respectively. Fr determines the inverter output frequency fo and its peak amplitude Ar controls the Modulation Ratio (Ar/Ac) and hence the rms output voltage Vo. A high frequency carrier wave Vc is compared to a reference signal Vr having the desired frequency through a comparator. When the sinusoidal wave has a higher magnitude, output is high otherwise it is low. The comparator output is processed in a trigger pulse generator in such a way that the output voltage wave has a pulse width in agreement with comparator pulse width.

$$\text{Modulation Index (M)} = \frac{V_r}{V_c}$$

For M<1   Phase Voltage $V_p = M \frac{V_{dc}}{2\sqrt{2}}$,   Line Voltage $V_L = M \frac{\sqrt{3}V_{dc}}{2\sqrt{2}}$

Procedure:

1. Rig up the circuit as given in the diagram.
2. Set the specifications of the motor as per requirement.
3. Variable torque is set using staircase block in open loop control and speed is monitored.
4. Variable speed is applied in closed loop system while the load can be constant or variable.
5. Run the simulation and trace the output waveforms.

Electrical Systems Simulation Lab   I M.Tech – I Sem

Simulation Results:

a) Open loop control:
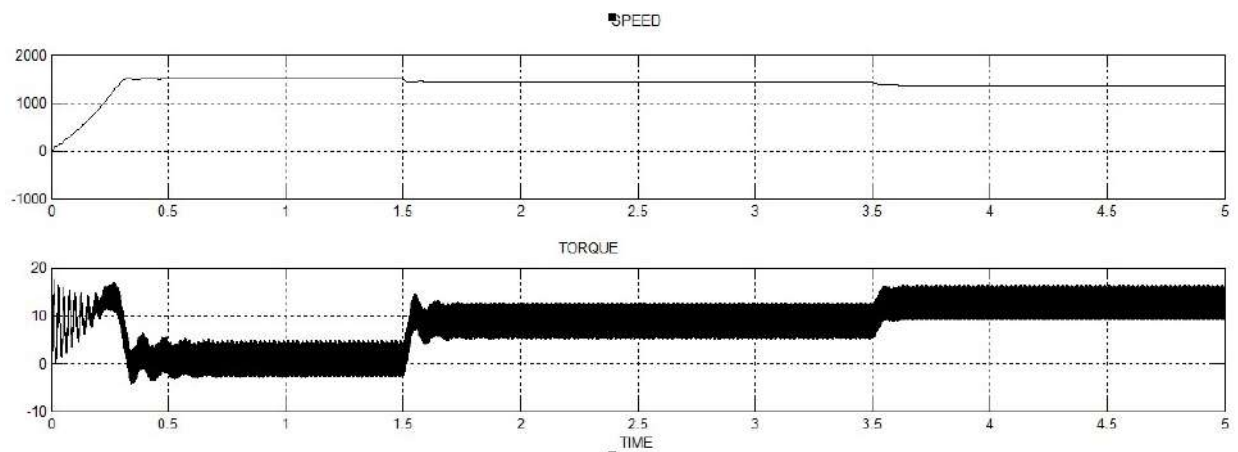
b) Closed loop control:
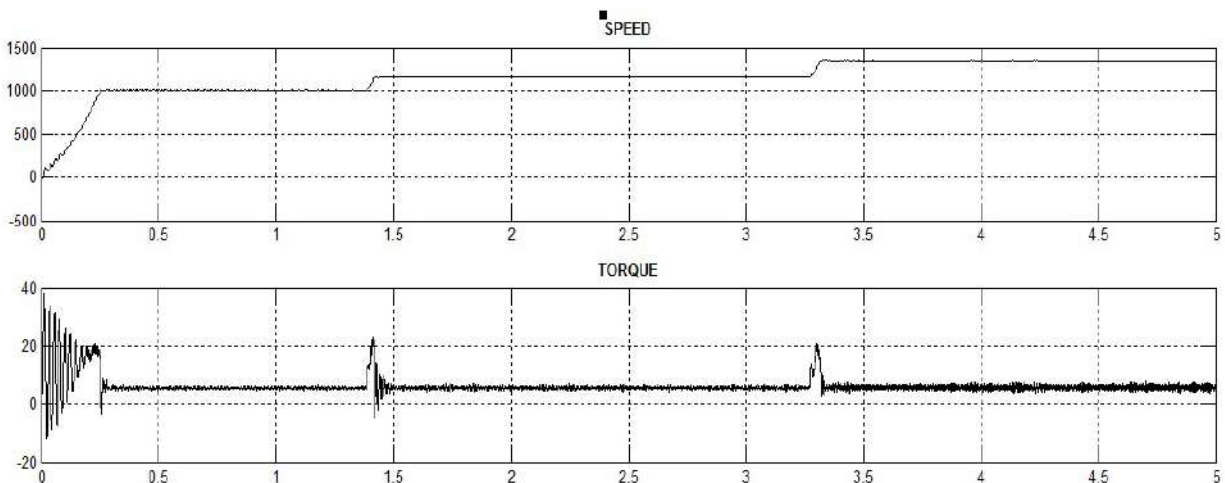
Result : The three phase Inverter fed Induction Motor is simulated using MATLAB/SIMULINK

and its   operation is analyzed under various load conditions.

(Dr G.Annapurna)   (Dr N. Malla Reddy)
Professor. – EEE   HOD – EEE

Electrical Systems Simulation Lab   M. Tech – I Sem

AIM: Simulation of ZCS and ZVS Resonant Converter operation

SOFTWARE: MATLAB/SIMULINK

Theory & Operation :

Resonant Converters :The devices that are designed to provide zero-voltage or zero-current switching are known as Resonant Converters. These converters are basically electric power converters that contain a network of inductors and capacitors that is specifically tuned to resonate at a particular frequency.

In an electric circuit, resonant converters are incorporated to somewhat lessen the chances of switching losses in devices like MOSFET or IGBT.

Zero-Current Switching Resonant Converters

The zero-current switching converters are of two types namely L-type and M-type. In both the circuit configurations, L and C are the necessary components that form the series resonant circuit. The presence of the inductor limits the rate of change of switching current in the circuit.

L-type ZCS Resonant Converters

Here the switching device used in the circuit configuration can be GTO, thyristor, BJT, MOSFET, IGBT, etc. The figure below represents the circuit for the L-type ZCS resonant converter:
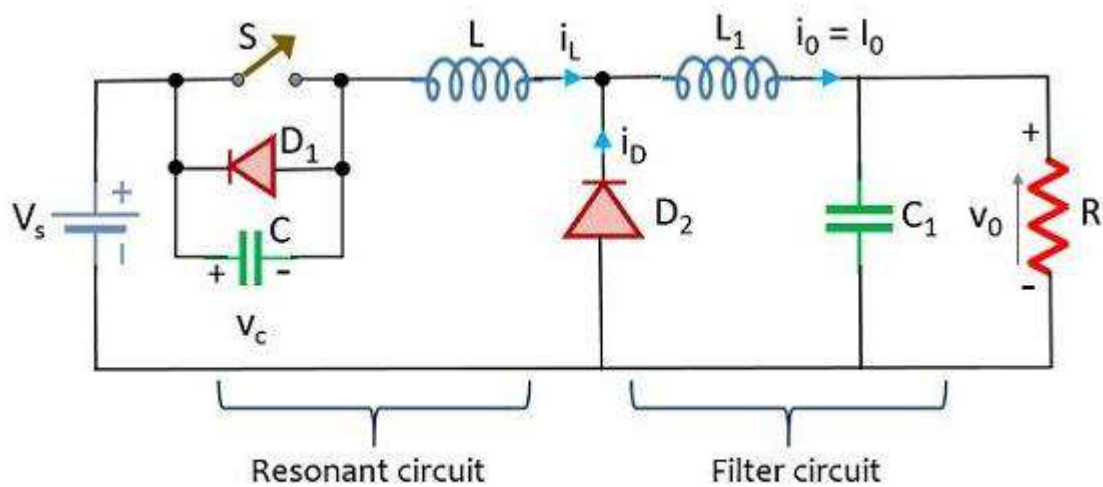

L-type zero-current switching resonant converter

Electrical Systems Simulation Lab   M. Tech – I Sem
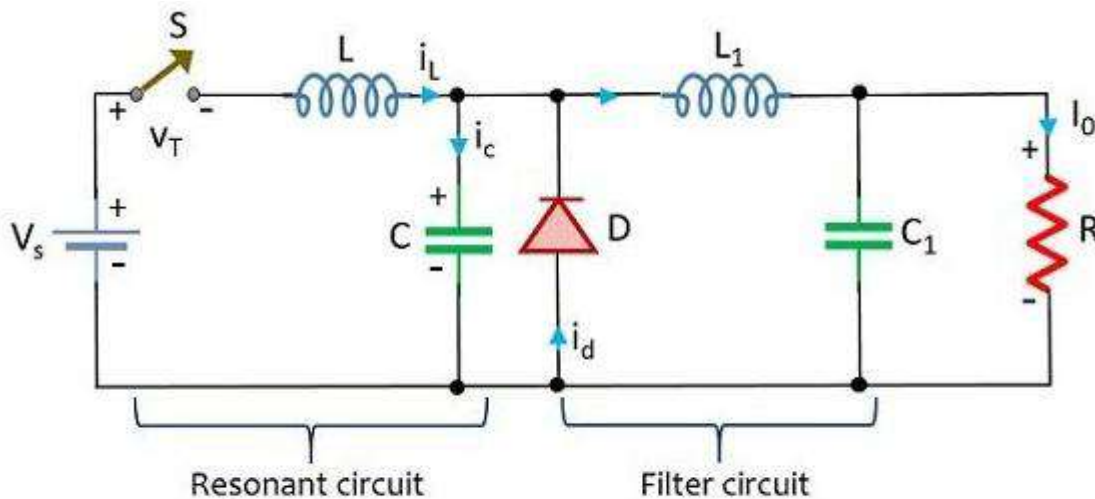
The selection of switching devices depends on the frequency range of operation. Basically, for the low-frequency range of operation, thyristor, transistor, or IGBT can be used. While in the megahertz range, power MOSFETs are used. The resonant circuit is the one consisting of L and C connected across the dc source V_{dc} whereas the filter circuit over here is the one consisting of L_1 and C_1 connected across the load.

Initially, in the steady-state condition constant current I_0 flows through the load. Also, switch S in the circuit is open, so the resonant circuit parameters will be $i_L = 0$ across the inductor and v_c will be 0 across the capacitor and the load current I_0 easily flows through the diode D.
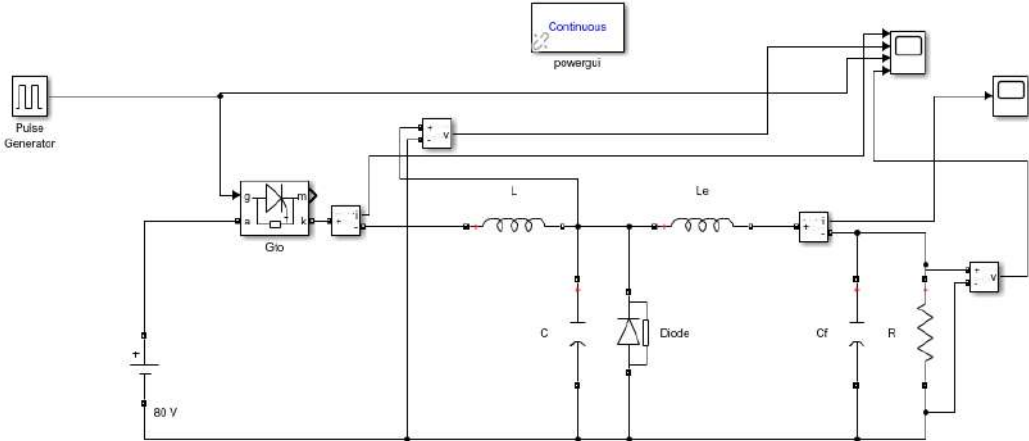
Zero-Voltage-Switching Resonant Converters

This type of resonant converter is one that consists of a diode and capacitor that form a parallel connection with the switch S. Similar to the previously discussed case of zero-current-switching resonant converter here also the components L and C constitute the resonant circuit while L_1 and C_1 form the filter circuit. Here the resonant capacitor is incorporated in the circuit so as to have zero voltage across the switch.
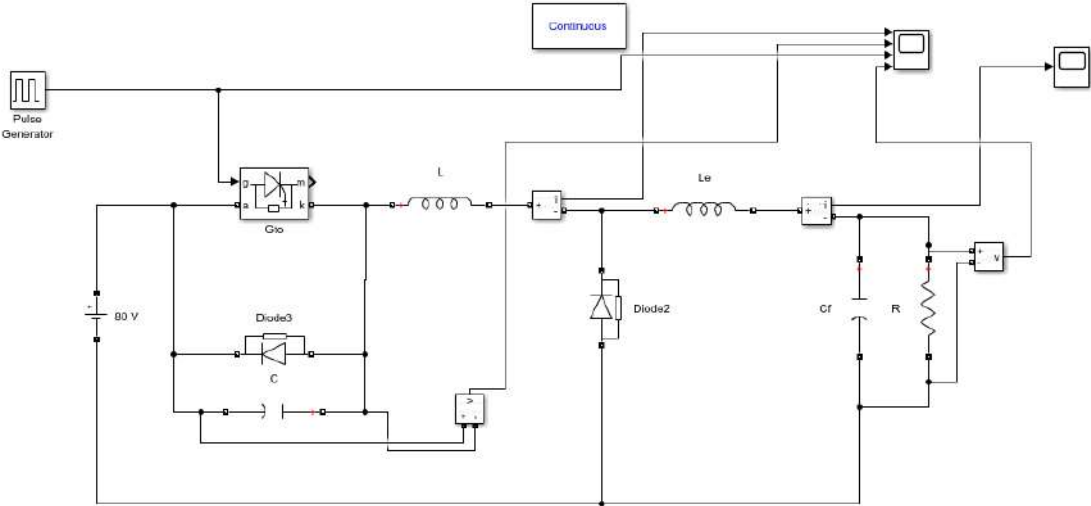

Zero-voltage-switching Resonant Converter

In this circuit, the path through which the load current I_0 flows is provided by the freewheeling diode. The operation of this circuit is concerned with the fact that the switch will get turned on and off when there will be zero voltage across the switch.

G. Narayanamma Institute of Technology & Science(For Women)
Shaikpet, Hyderabad – 500 104
Department of Electrical and Electronics Engineering

Electrical Systems Simulation Lab   M. Tech – I Sem

Simulink Model of L –type ZCS Resonant Converter

Simulink Model of L –type ZVS Resonant Converter

G. Narayanamma Institute of Technology & Science(For Women)
Shaikpet, Hyderabad – 500 104
Department of Electrical and Electronics Engineering

Electrical Systems Simulation Lab   M. Tech – I Sem

ZCS (L-type switch) Quasi Resonant Converter Specifications

Gain, M = 0.5

DC Input Voltage= 80 V

C = 0.261uF

L = 0.261uH

L_e=0.1mH

C_f=1 uF

R= 1 ohms

GTO Switch: forward Voltage V_f= 0.01V

Period =1/250000, Pulse width=50%

ZVS Buck (L-type switch) Quasi Resonant Converter Specifications

Gain, M = 0.375=0.4

DC Input Voltage= 80 V

C = 114 nF

L = 1.272 uF

L_e=0.1mH

C_f =1 uF

R= 1 ohms

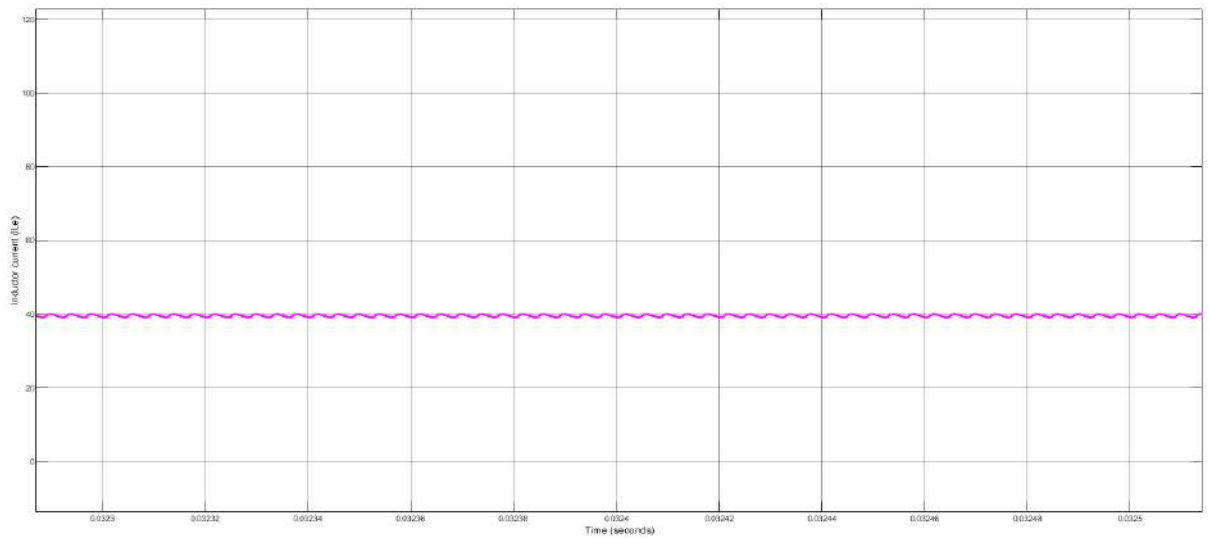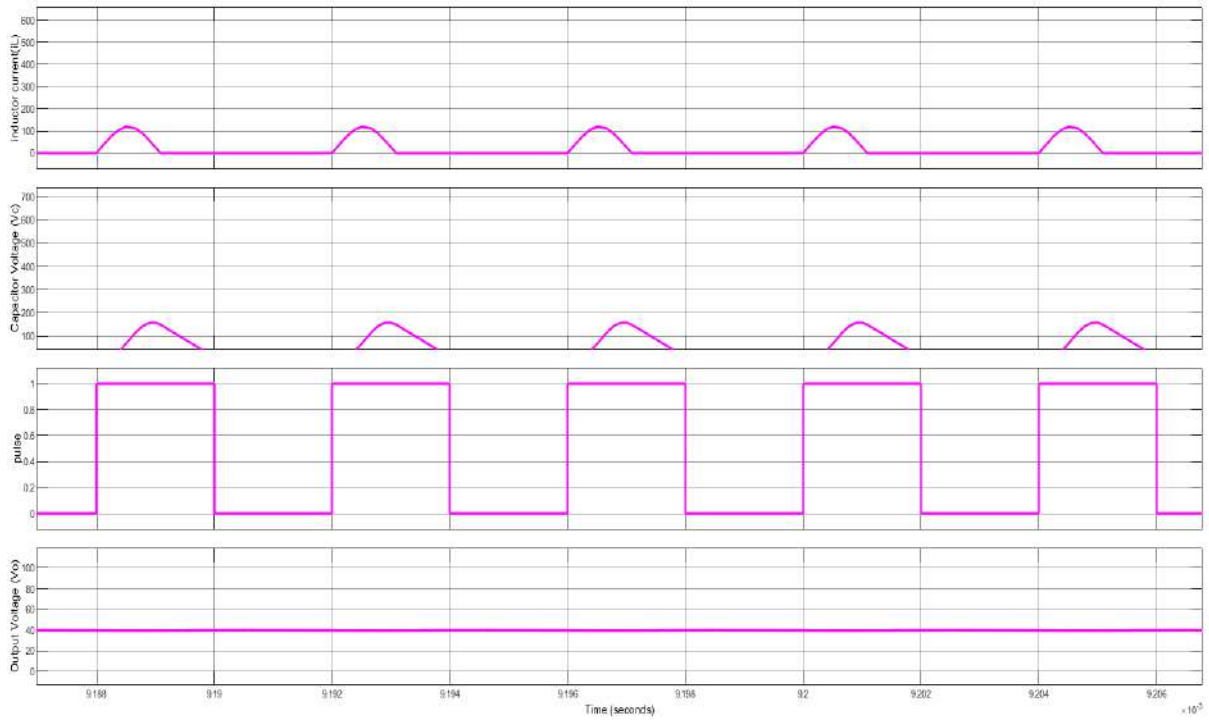GTO Switch: forward Voltage V_f= 0.01V

 Period =1/250000, Pulse width=50%

Procedure:

1. Rig up the circuit as per the circuit diagram in MATLAB/SIMULINK model Platform.
2. Observe the waveforms.

G. Narayanamma Institute of Technology & Science(For Women)
Shaikpet, Hyderabad – 500 104
Department of Electrical and Electronics Engineering

Electrical Systems Simulation Lab   M. Tech – I Sem

Simulation Results for ZCS Resonant Converter

Electrical Systems Simulation Lab   M. Tech – I Sem

Simulation Results for ZVS Resonant Converter
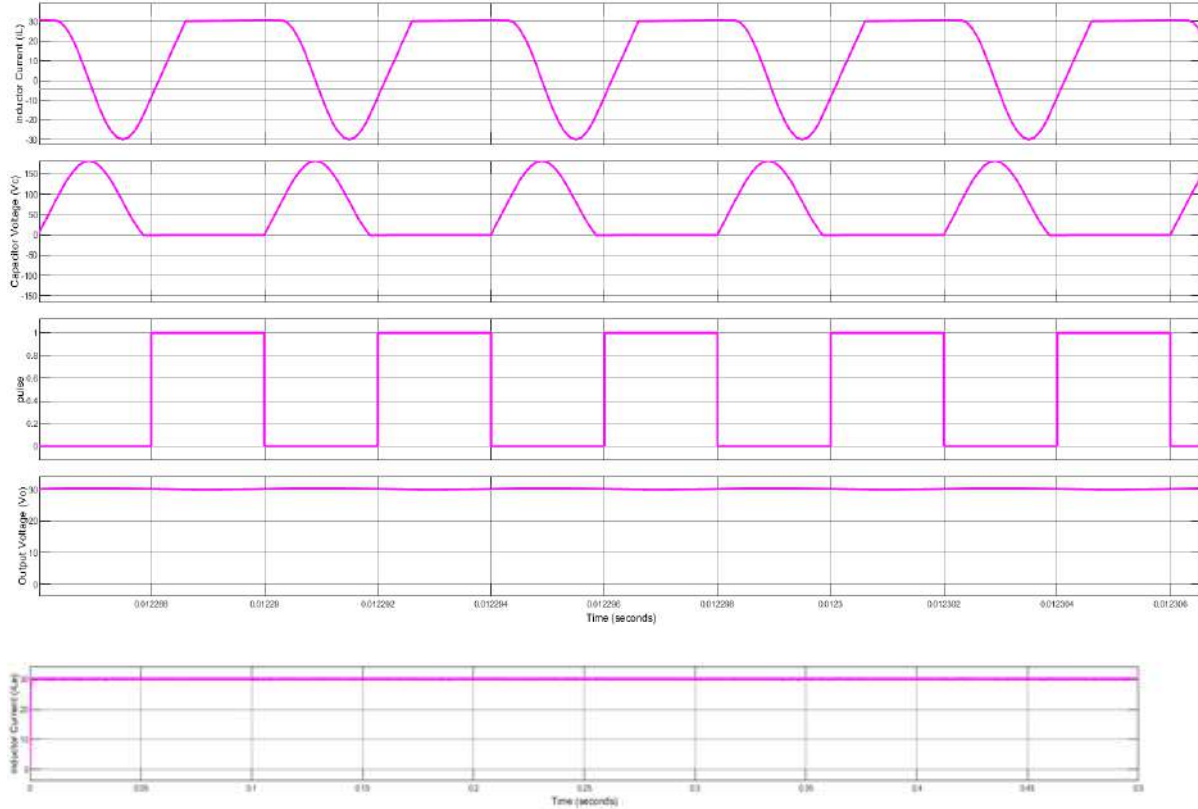
Result: The ZCS and ZVS Resonant Converter is Simulated using MATLAB/SIMULINK and output waveforms are observed.

(Mrs.Y.Priyanka)   (Dr N. Malla Reddy)
Asst. Prof. – EEE   HOD - EEE