u

LIST OF EXPERIMENTS

Sl. No.	NAME OF THE EXPERIMENT	Page No.
1	Simulation of Transient response of RLC Circuit To an input (i) step (ii) pulse and(iii) Sinusoidal signals	2
2	Analysis of Three Phase Circuit representing the generator transmission line and load. plot three phase currents and neutral current.	7
3	Simulation of Single Phase full converter using RLE loads and single phase AC Voltage Controller using RL loads.	12
4	Plotting of Bode plots, Root Locus and Nyquist plots for the transfer functions of systems up to 5^{th} order.	23
5	Power System load flow using Newton-Raphson Technique.	28
6	Modeling of Transformer and simulation of lossy transmission line	34
7	Integrator and Differentiator circuits using OP-AMP.	38
8	Simulation of D.C separately excited motor using Transfer function approach.	43
9	Simulation of Buck & Boost converters.	45
10	Simulation of single Phase Inverter with PWM control.	51

Expt. No: 1

PSPICE SIMULATION OF SERIES RLC CIRCUITS FOR STEP, PULSE & SINUSOIDAL INPUTS

Date:

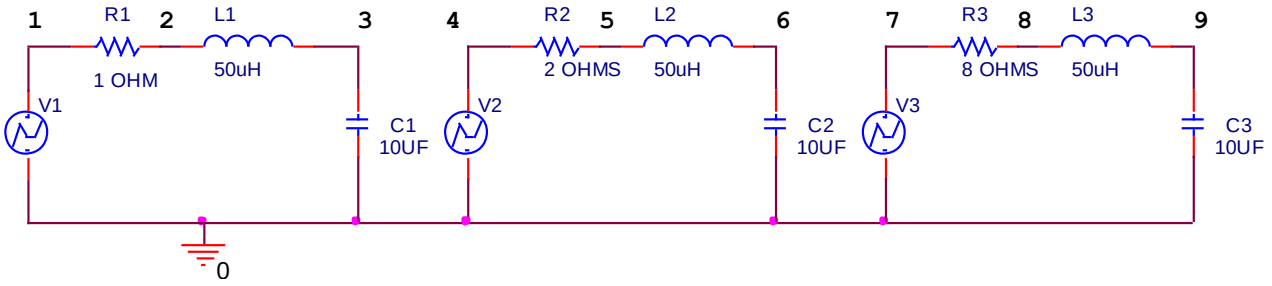
AIM: To study the responses of series RLC circuits for a given step, pulse & sinusoidal inputs.

SIMULATION TOOLS REQUIRED:

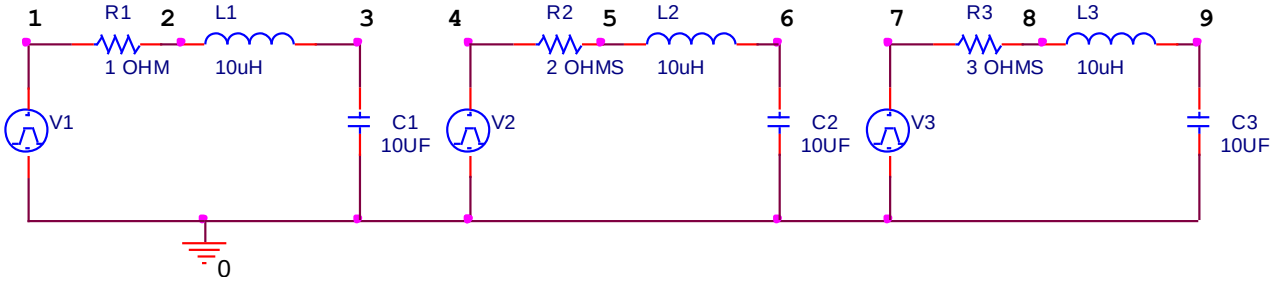
- PC with PSPICE Software

CIRCUIT DIAGRAMS:

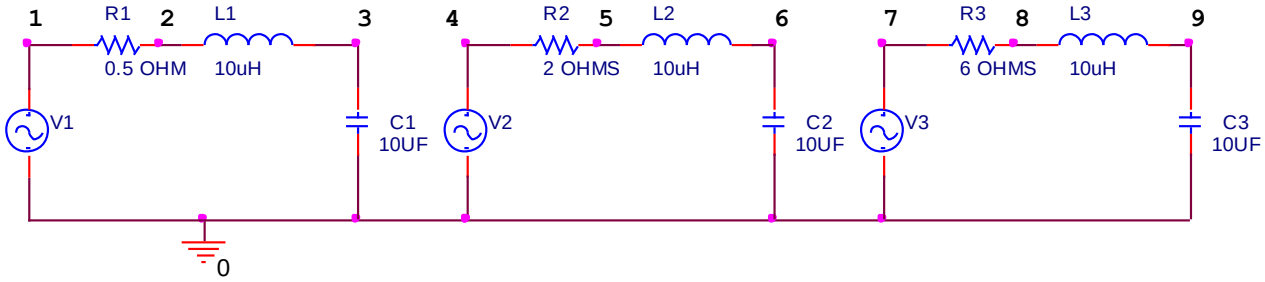
Series RLC circuit for STEP input



Series RLC circuit for SQUARE input



Series RLC circuit for SINUSOIDAL input



SPECIFICATIONS:

THEORY:

PROGRAMS:

STEP INPUT

PULSE INPUT

SINUSOIDAL INPUT

PROCEDURE:

1. Write the program in a new text file in PSpice AD.
2. Save the file using the notation filename.cir.
3. Activate the file by opening it.
4. Run the simulation process using blue button.
5. By clicking Add Trace icon, get the required waveform.

MODEL CALCULATIONS:

$$\alpha == \frac{R}{2*L}$$

$$\omega_0 == \frac{1}{\sqrt{L*C}}$$

A).OVER DAMPED ($\alpha < \omega_0$)

$\alpha =$

$\omega_0 =$

B) UNDER DAMAPED ($\alpha < \omega_0$)

$\alpha =$

$\omega_0 =$

C) CRITICALLY DAMPED ($\alpha > \omega_0$)

α

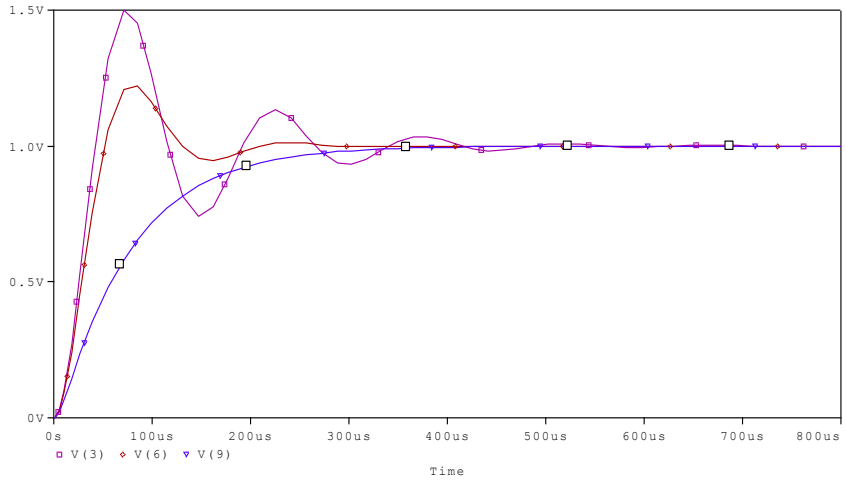
$\omega_0 =$

RESULT:

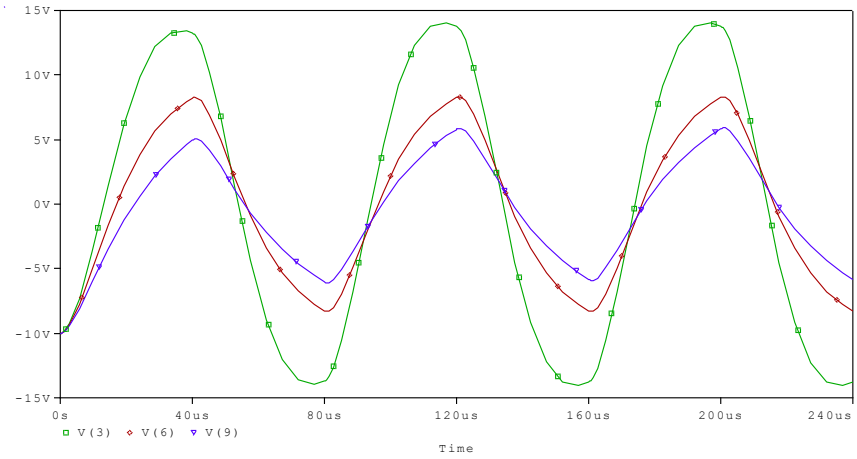
APPLICATIONS:

MODEL GRAPH:

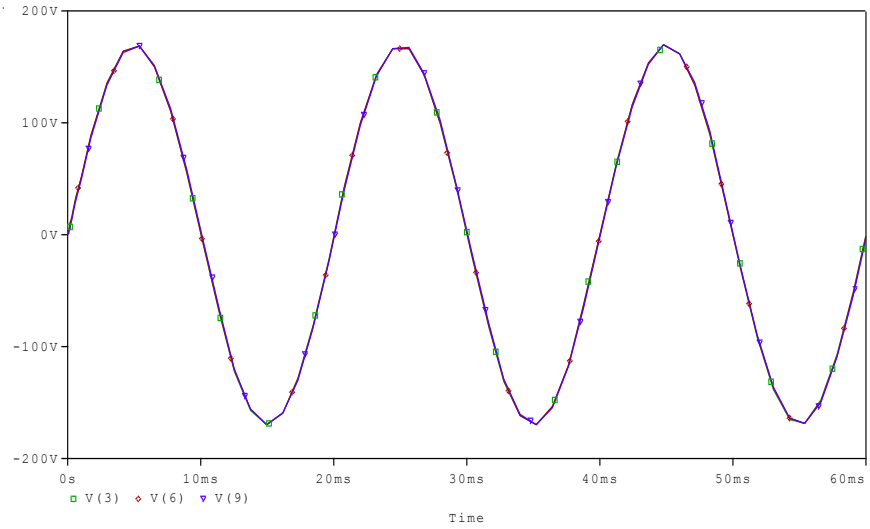
SERIES RLC CIRCUIT WITH STEP INPUT



SERIES RLC CIRCUIT WITH SQUARE INPUT



SERIES RLC CIRCUIT WITH SINUSOIDAL INPUT



Expt. No: 2

PSPICE ANALYSIS OF THREE PHASE CIRCUIT

Date:

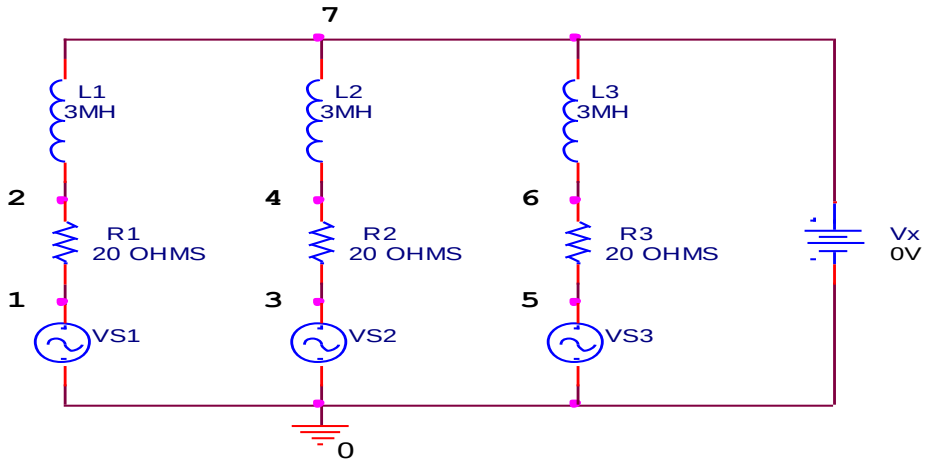
AIM: To study the analysis of simple three phase circuit for balanced and unbalanced loads.

SIMULATION TOOLS REQUIRED:

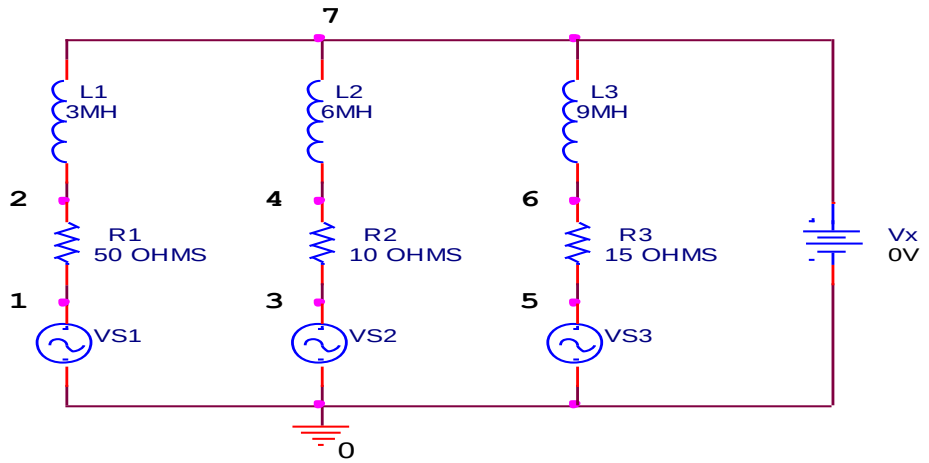
- PC with PSPICE Software

CIRCUIT DIAGRAMS:

Three Phase circuit with Balanced load



Three Phase circuit with Unbalanced load



SPECIFICATIONS:

THEORY:

PROGRAMS:

BALANCED LOAD CONDITION

UNBALANCED LOAD CONDITION

PROCEDURE:

1. Write the program in a new text file in PSpice AD.
2. Save the file using the notation filename.cir.
3. Activate the file by opening it.
4. Run the simulation process using blue button.
5. By clicking Add Trace icon, get the required waveform.

RESULT:

MODEL CALCULATIONS:

A) FOR BALANCED LOAD

$Ir =$

$Iy =$

$Ib =$

$In = Ir + Iy + Ib =$

B) FOR UNBALANCED LOAD

$Ir =$

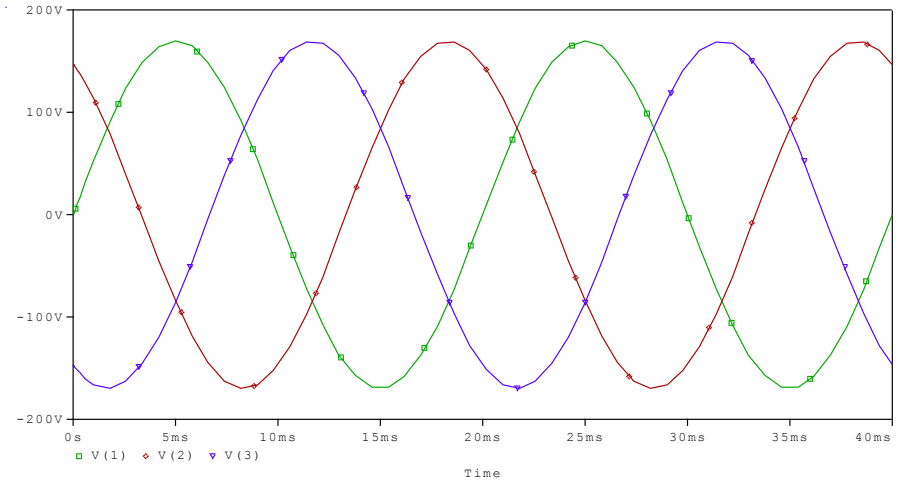
$Iy =$

$Ib =$ **s**

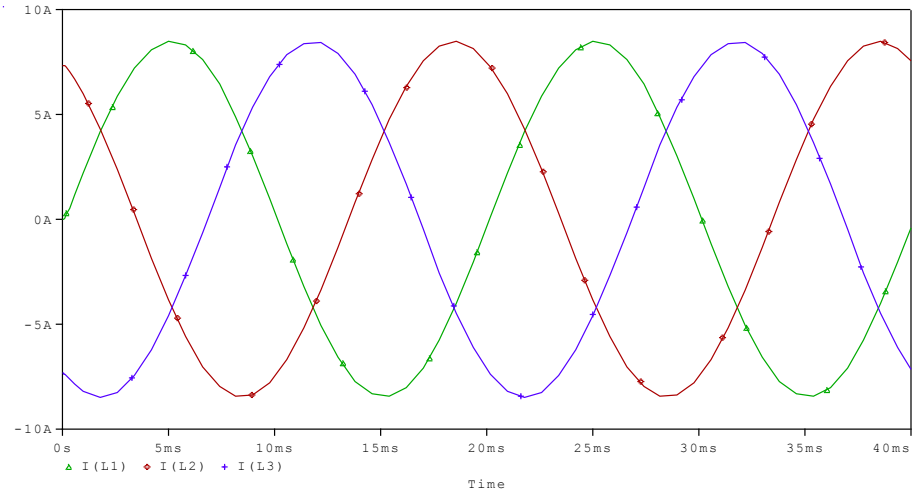
$In = Ir\angle 0^0 + Iy\angle -120^0 + Ib\angle -240^0 =$

MODEL WAVEFORMS:

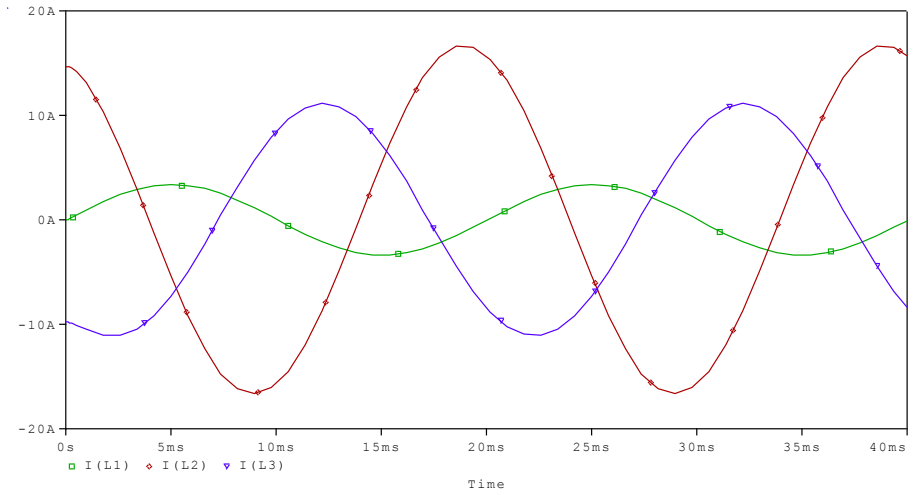
INPUT WAVEFORM



BALANCED LOAD CONDITION



UNBALANCED LOAD CONDITION



Expt. No: 3(a)

PSPICE ANALYSIS OF SINGLE PHASE FULL CONVERTER WITH RL & RLE LOADS

Date:

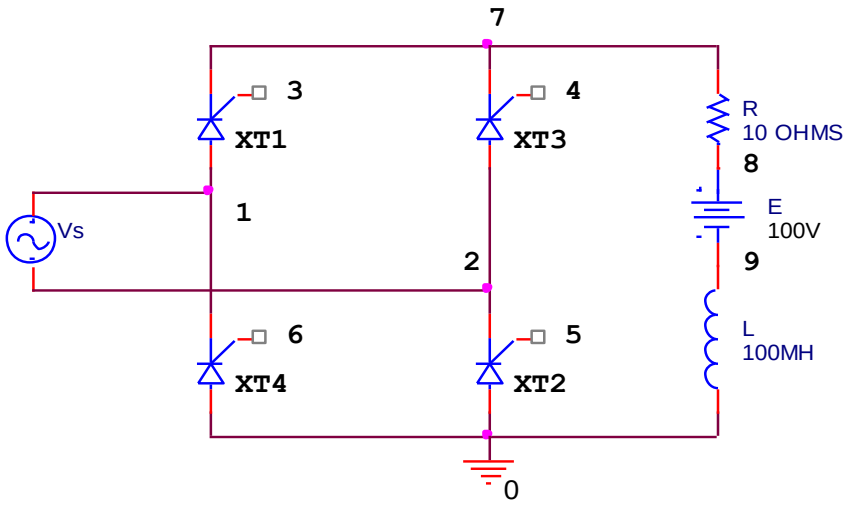
AIM: To analyze the single phase full converter with RL and RLE Loads.

SIMULATION TOOLS REQUIRED:

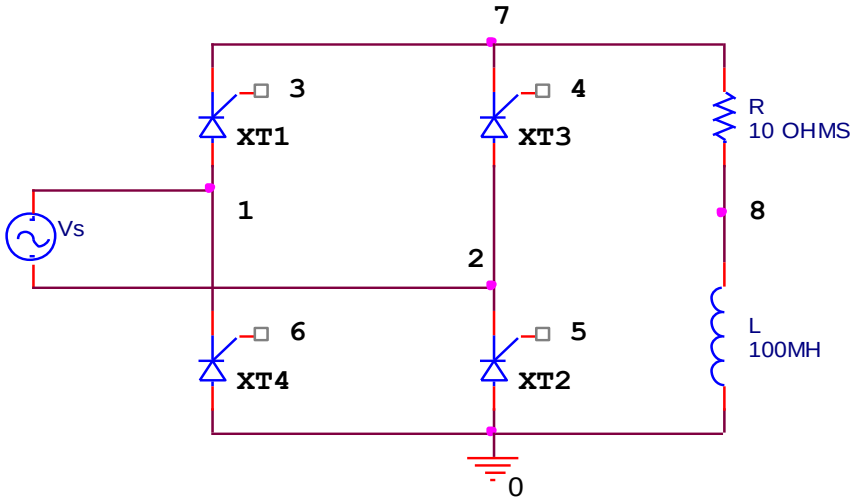
- PC with PSPICE Software

CIRCUIT DIAGRAMS:

Single Phase full converter with RL load



Single Phase full converter with RLE load



SPECIFICATIONS:

THEORY:

PROGRAMS:

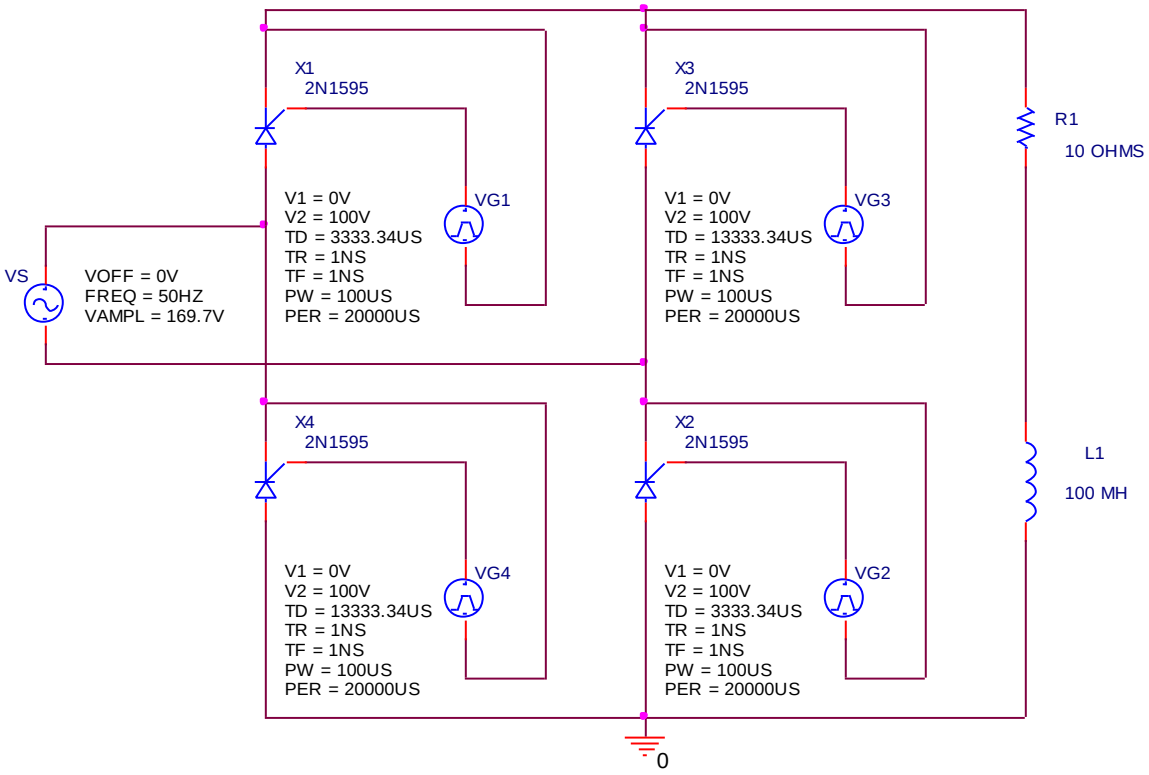
WITH RL LOAD

WITH RLE LOAD

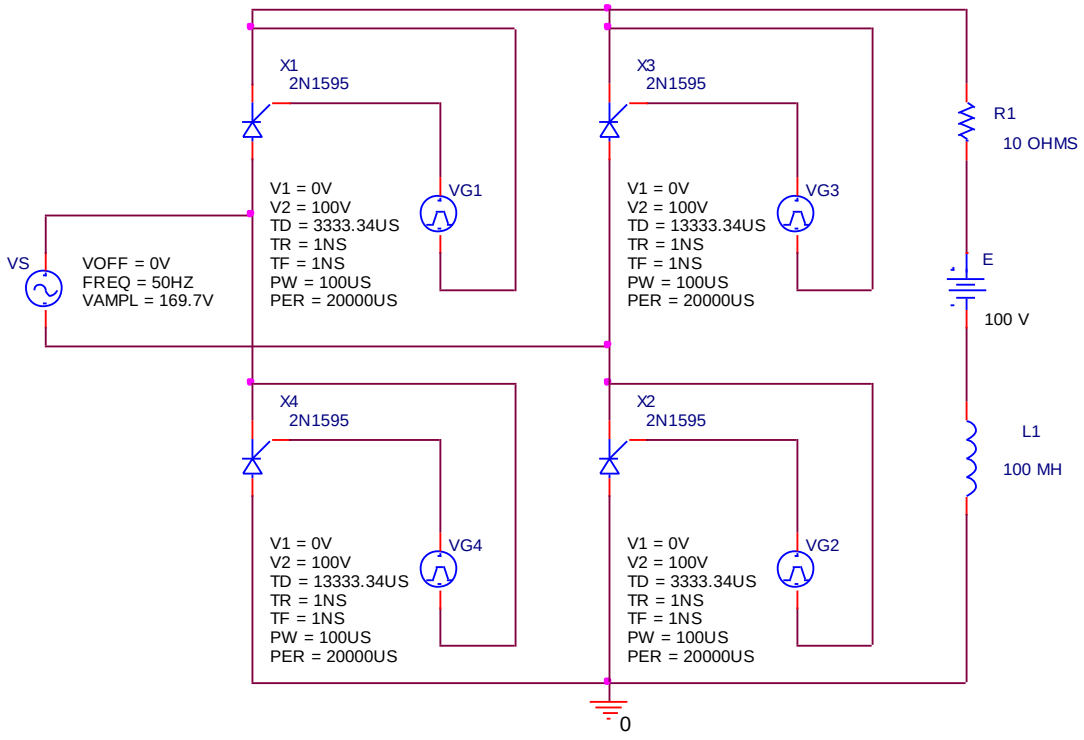
SIGLE-PHASE FULL CONVERTER CIRCUIT WITH RLE LOAD

CIRCUIT DIAGRAMS FOR ANALYSIS USING CIRCUIT:

Single Phase full converter with RL load



Single Phase full converter with RLE load



PROCEDURE:

1. Write the program in a new text file in PSpice AD.
2. Save the file using the notation filename.cir.
3. Activate the file by opening it.
4. Run the simulation process using blue button.
5. By clicking Add Trace icon, get the required waveform.

THERITICAL CALCULATIONS:

A)FOR RL LOAD

$$V0 = \frac{2V_M}{\prod} COS(\alpha)$$

$$V0 = \frac{2V_M}{\prod} COS(____)$$

$$= _____ V$$

B)FOR RLE LOAD

At $\omega t = \alpha$ i.e. at $t = \alpha/\omega$. $i_o = 0$

We know $\omega t = \alpha => V_m \sin(\theta) = E$

Min value of firing angle $\theta = \sin^{-1}(\frac{E}{V_m}) = \sin^{-1}(-) = 3$

Max value of firing angle $\theta_2 = 180 - \theta_1 =$

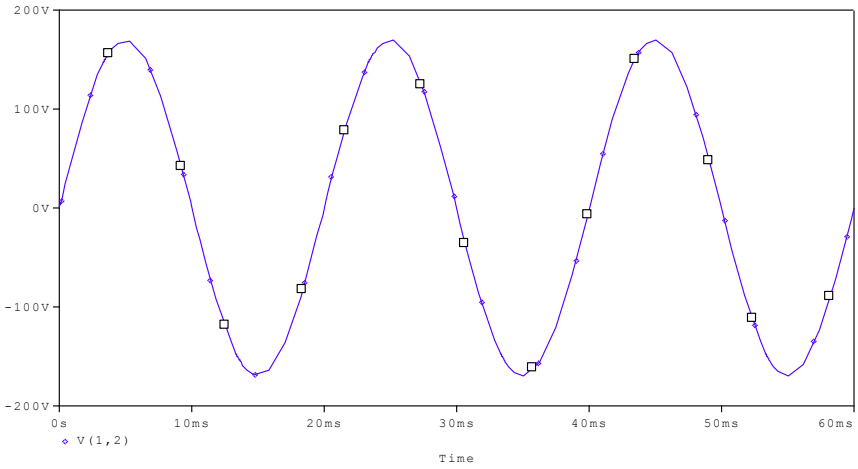
RESULT:

APPLICATIONS:

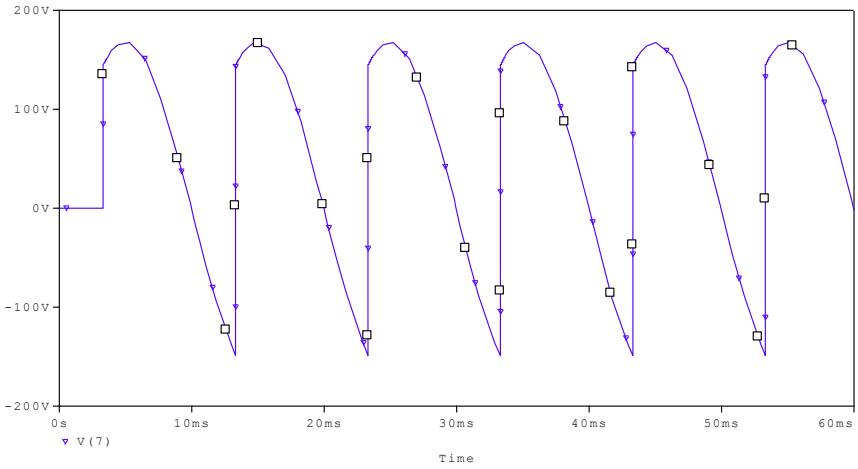
The single-phase full-wave controlled rectifier is used to control power flow in many applications (e.g., power supplies, variable-speed dc motor drives, and input stages of other converters)

MODEL WAVEFORMS FOR FULL CONVERTER:

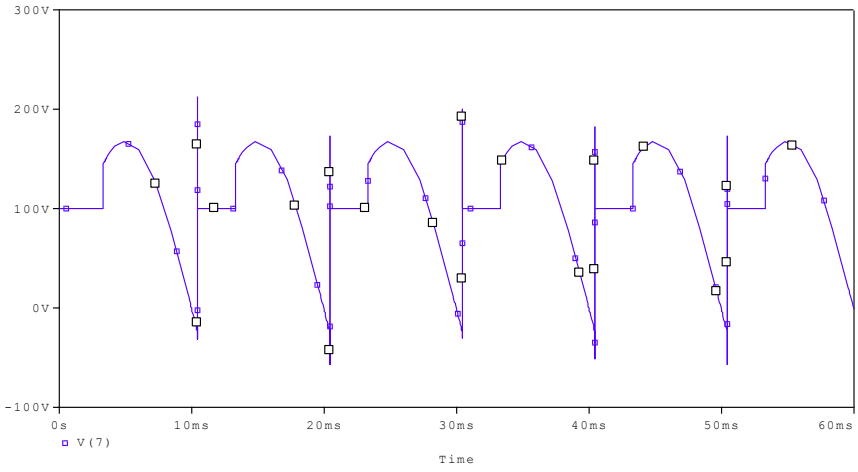
INPUT WAVEFORM



OUTPUT WAVEFORM WITH RL LOAD



OUTPUT WAVEFORM WITH RLE LOAD



Expt. No: 3(b)

PSPICE ANALYSIS OF SINGLE PHASE AC VOLTAGE CONTROLLER WITH RL LOAD

Date:

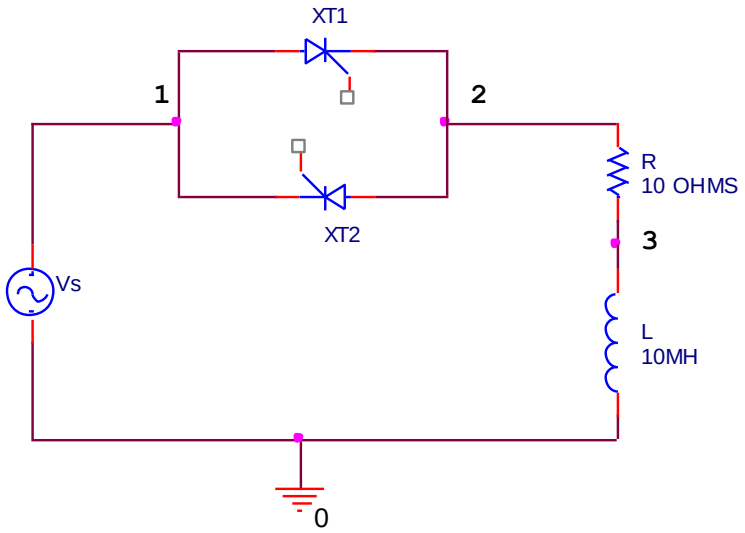
AIM: To analyze the single phase full converter with RL and RLE Loads.

SIMULATION TOOLS REQUIRED:

- PC with PSPICE Software

CIRCUIT DIAGRAM:

Single Phase AC VOLTAGE CONTROLLER with RL load



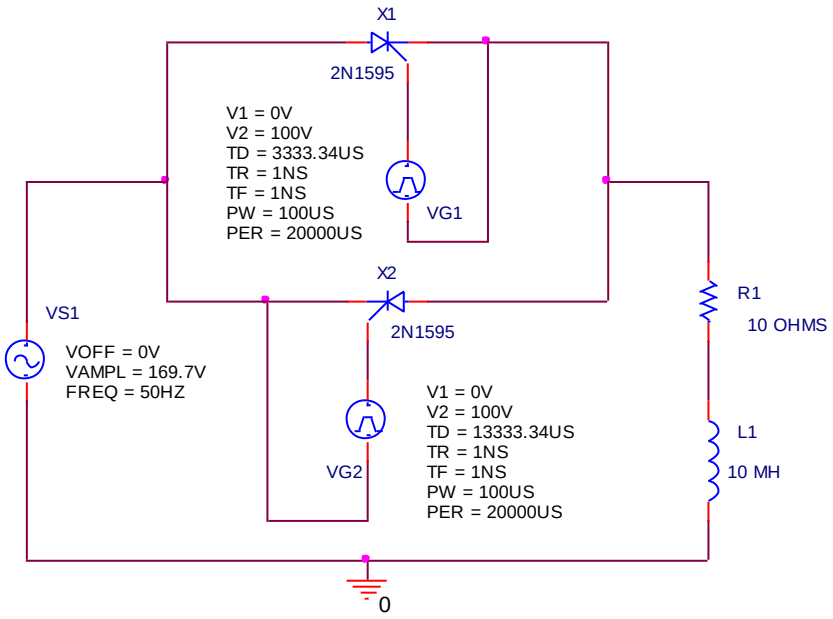
SPECIFICATIONS:

THEORY:

PROGRAM:

SIGLE-PHASE AC VOLTAGE CONTROLLER CIRCUIT WITH RL LOAD

CIRCUIT DIAGRAM FOR ANALYSIS USING CIRCUIT:



PROCEDURE:

1. Write the program in a new text file in PSpice AD.
2. Save the file using the notation filename.cir.
3. Activate the file by opening it.
4. Run the simulation process using blue button.
5. By clicking Add Trace icon, get the required waveform.

THERITICAL CALCULATIONS(FOR RL LOAD):

Wt=_____=_____ms

$$V0 = \frac{V_M}{\prod}[COS(\alpha)+1]$$

$$V0 = \frac{V_M}{\prod}[COS(____)+1]$$

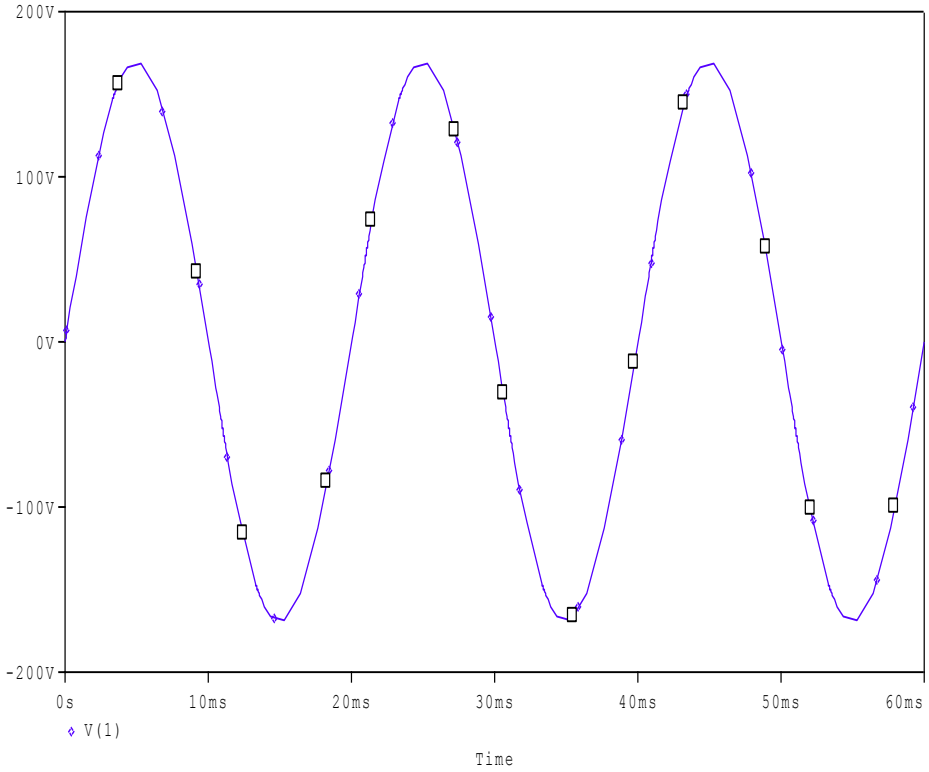
$$= _____ V$$

RESULT:

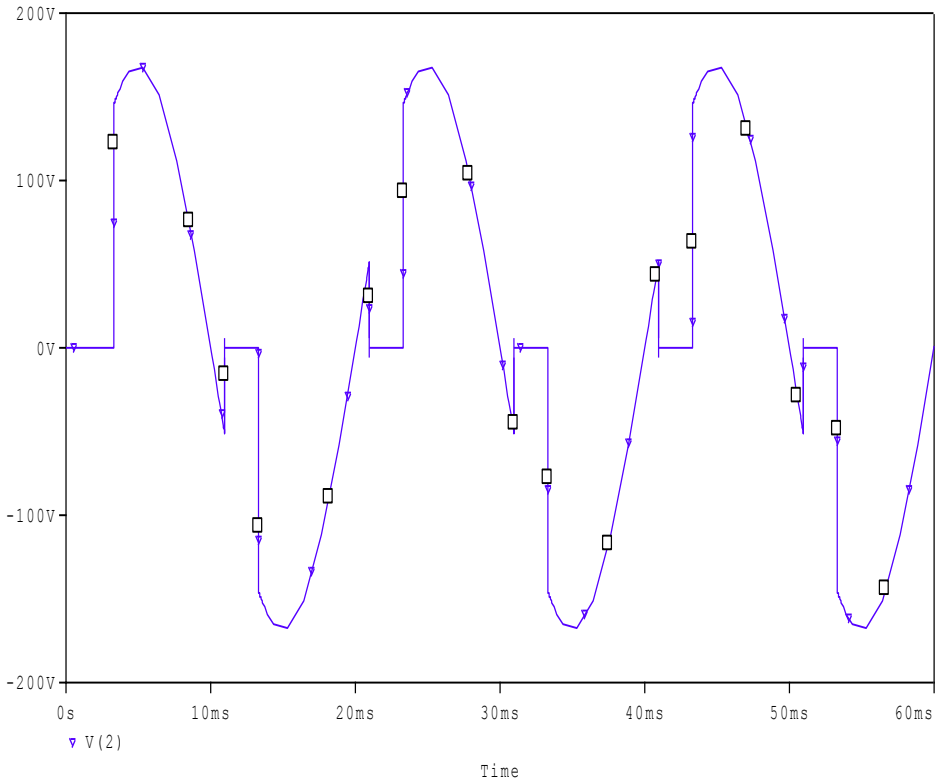
APPLICATIONS:

MODEL WAVEFORMS:

INPUT WAVEFORM



OUTPUT WAVEFORM



Expt. No: 4

STABILITY ANALYSIS OF LINEAR TIME INVARIANT SYSTEMS

(Bode, Root Locus, Nyquist plots using MATLAB)

Date:

AIM: To analyze the stability of given linear time invariant systems using MATLAB.

SIMULATION TOOLS REQUIRED:

- PC with MATLAB Software

THEORY:

TRANSFER FUNCTION:

The given system is represented by a transfer function as follows

$$G(S) = \frac{242(S+5)}{S(S+1)(S^2+5S+121)}$$

THEORITICAL CALCULATIONS:

Draw Bode, Nyquist and Root locus plots for the given transfer function.

PROGRAMS:

BODE PLOT:

NYQUIST PLOT:

ROOT LOCUS PLOT:

PROCEDURE:

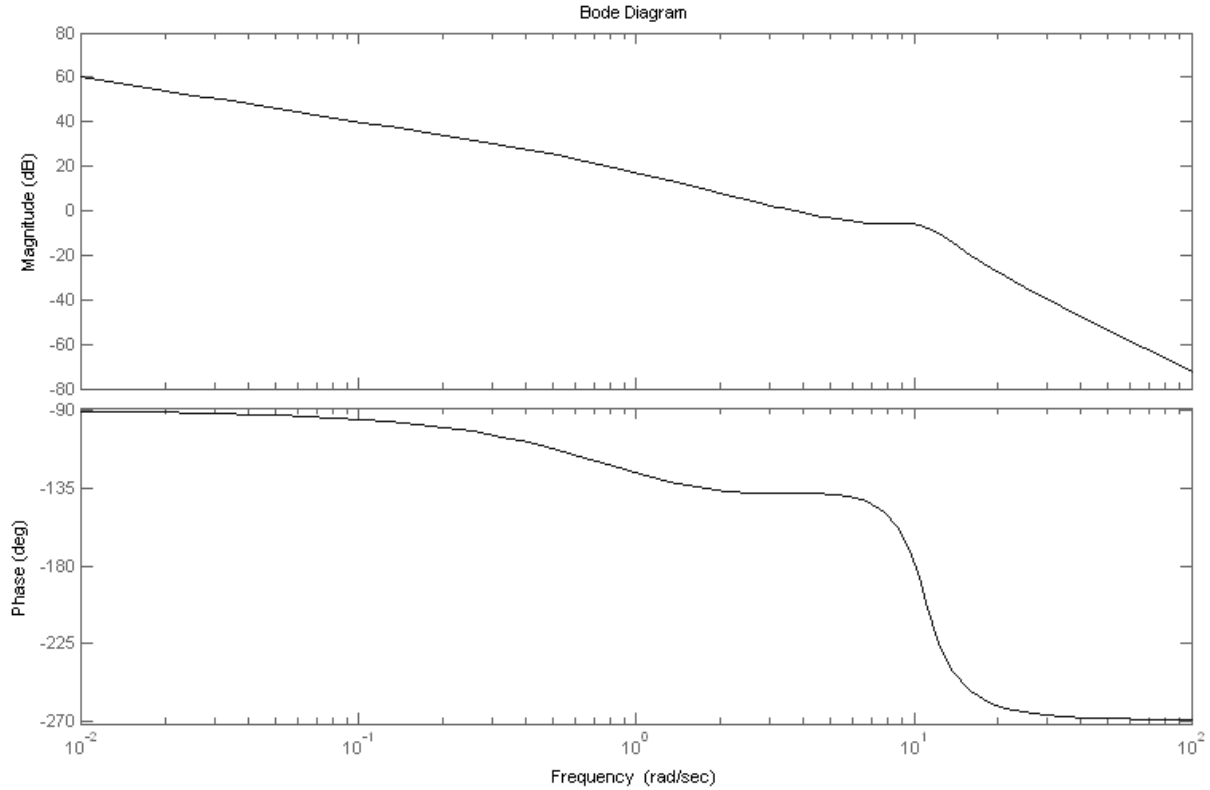
1. Open the MATLAB command window clicking on the MATLAB icon.
2. Click on file menu and open new M file.
3. Enter the MATLAB code.
4. Click on the debug menu and run the code.
5. Then copy the obtained plot.

RESULT:

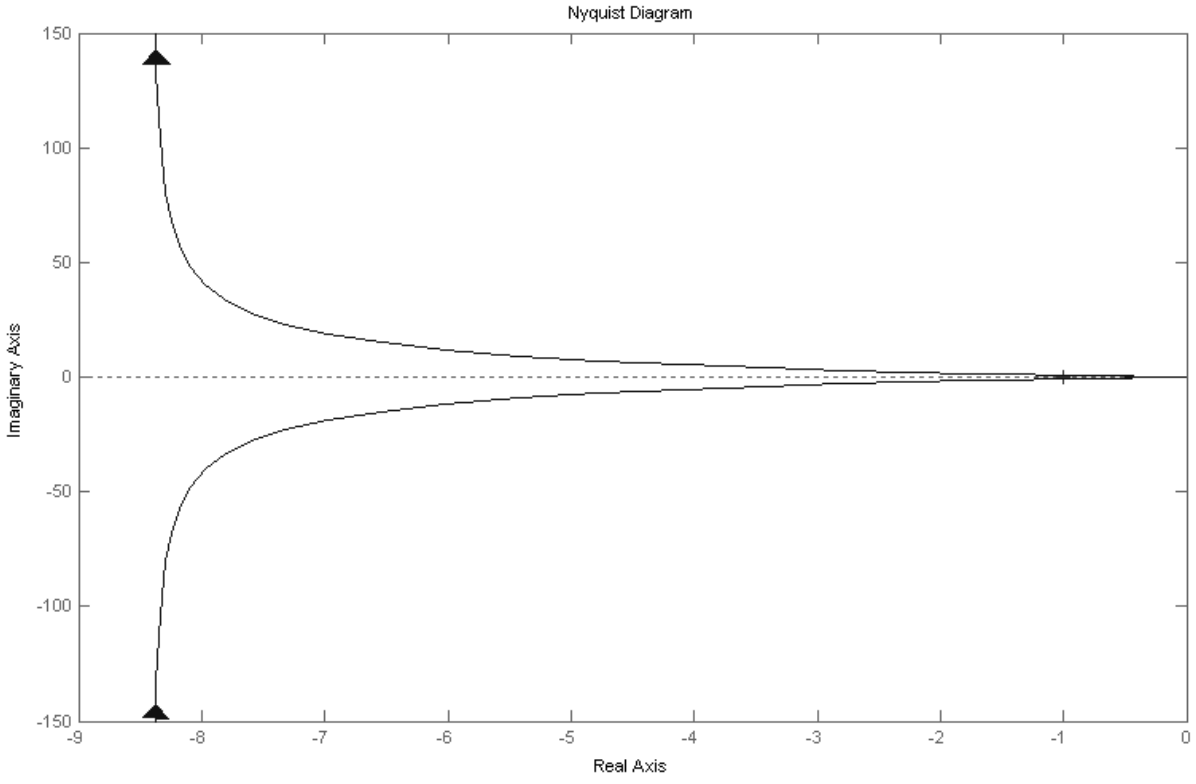
APPLICATIONS:

MODEL GRAPHS:

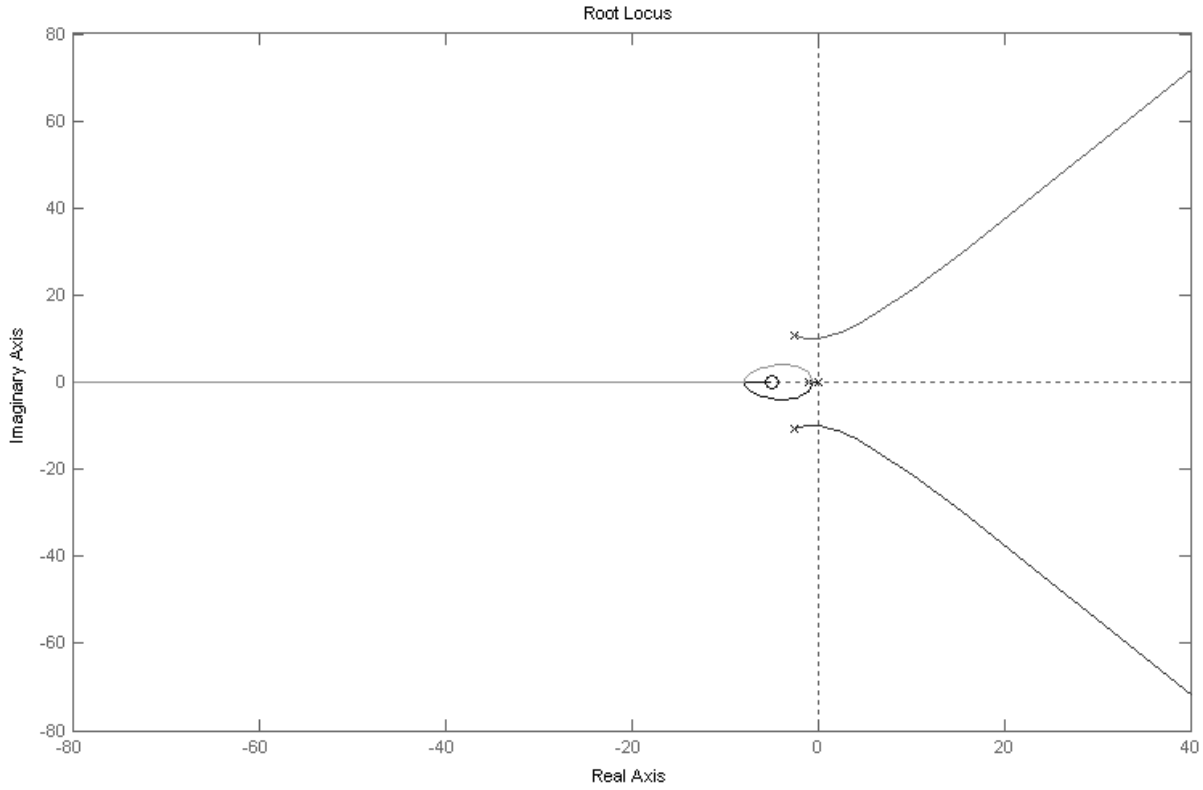
BODE PLOT



NYQUIST PLOT



ROOT LOCUS PLOT



Expt. No: 5

POWER SYSTEM LOAD FLOW USING GAUSS-SEIDEL AND NEWTON-RAPHSON TECHNIQUE

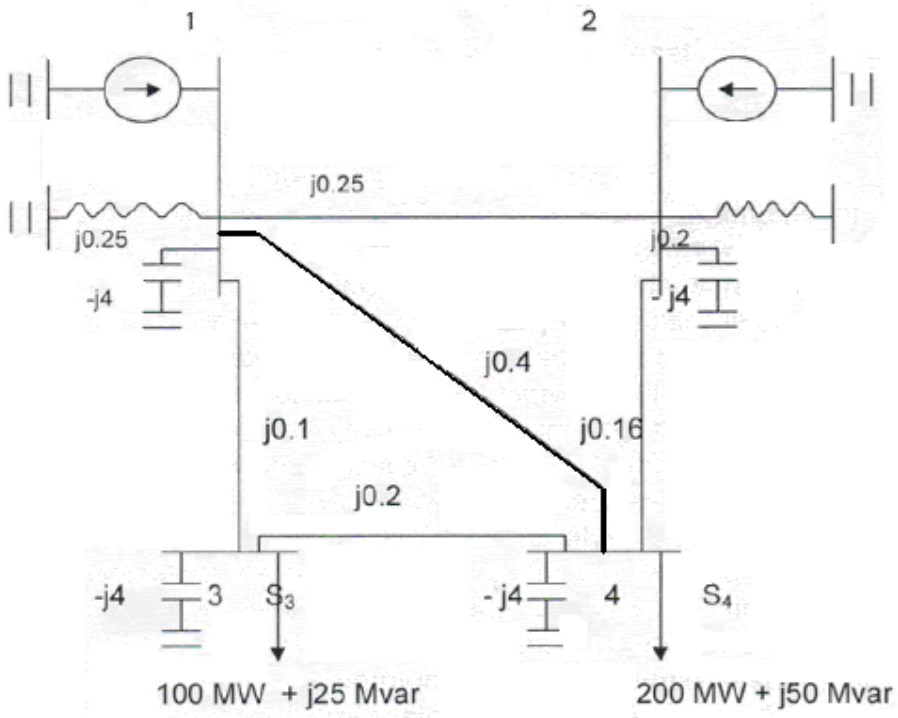
Date:

AIM: To find the power flow solution of a given power system using Gauss-Seidel and Newton Raphson methods.

SIMULATION TOOLS REQUIRED:

- PC with MATLAB Software

SINGLE LINE DIAGRAM:



THEORY:

PROGRAM:

PROCEDURE:

NEWTON – RAPHSON METHOD:

1. Open the MA TLAB Command window by clicking on the **MA TLAB.exe** icon.

2. Enter the programs LF-Bus, LF-Newton busout and lineflow in the MATLAB Text Editor.

3. Prepare the line, transformer parameters and transformer tap settings data in a matrix named **line data.**

4. Run the programs LF-Y-Bus, LF-Newton, busout and line flow in MATLAB Command Window to get the power flow solution using Newton-Raphson.

RESULT:

APPLICATIONS:

Expt. No: 6(a)

MODELING OF TRANSFORMER

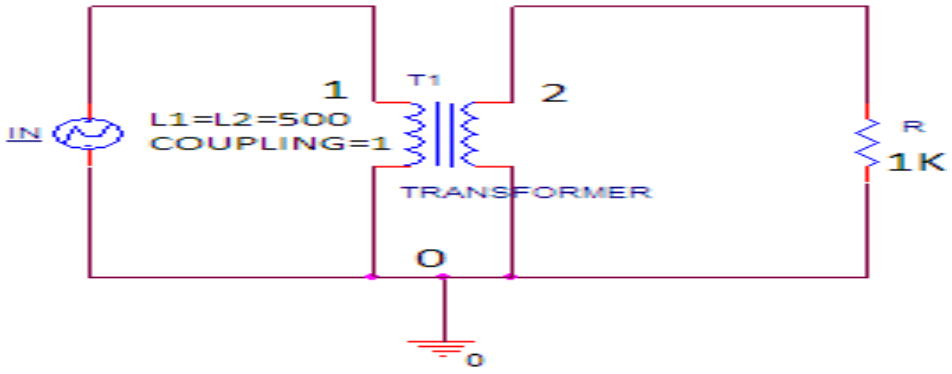
Date:

AIM: To simulate an Op-amp based Integrator & Differentiator circuits using Pspice.

SIMULATION TOOLS REQUIRED:

- PC with PSPICE Software

CIRCUIT DIAGRAMS:



THEORY:

SPECIFICATIONS:

PROGRAMS:

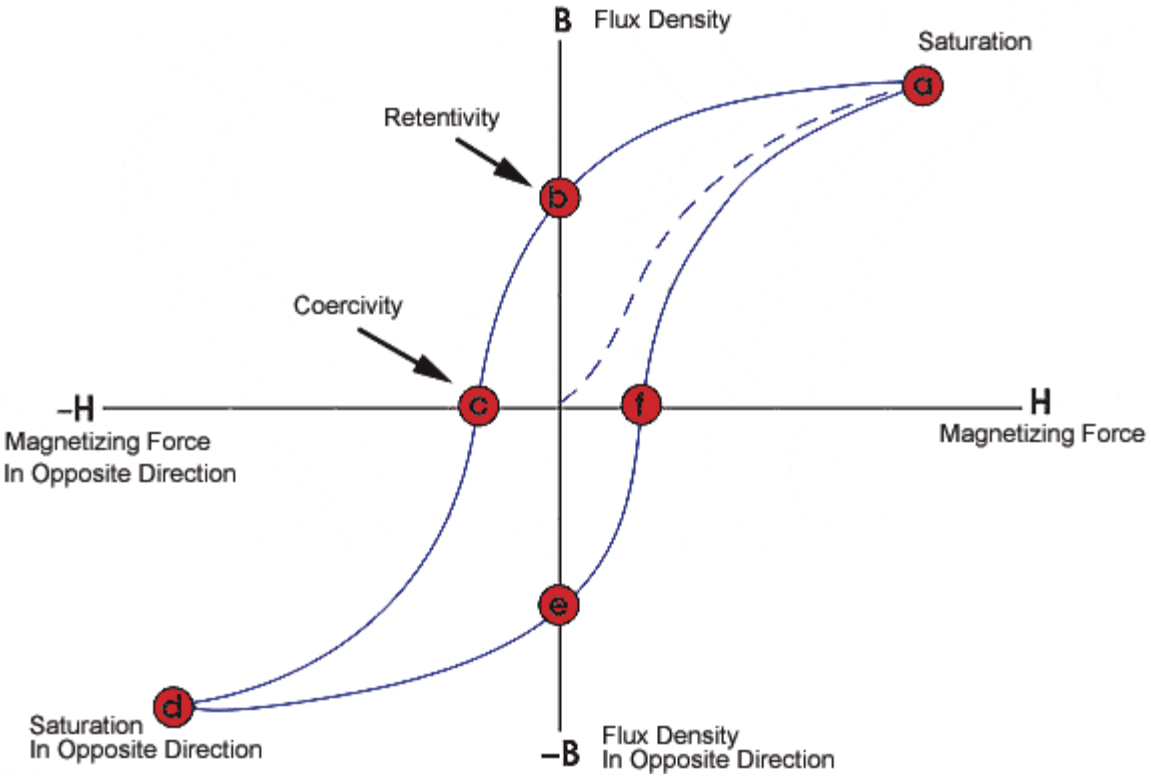
PROCEDURE:

1. Write the program in a new text file in PSpice AD.
2. Save the file using the notation filename.cir.
3. Activate the file by opening it.
4. Run the simulation process using blue button.
5. By clicking Add Trace icon, get the required waveform.

RESULT:

APPLICATIONS:

MODEL GRAPTH


B Flux Density
Saturation
a
Retentivity
b
Coercivity
c
f
–H
Magnetizing Force
In Opposite Direction
H
Magnetizing Force
e
d
Saturation
In Opposite Direction
–B
Flux Density
In Opposite Direction

Expt. No: 6(b)

PSPICE Simulation of a Lossy Transmission Line

Date:

Aim: Calculation of lossy transmission line parameters by applying PSPICE simulation.

Software tools required: PSPICE

THEORY:

Procedure:

1. Open PSPICE and create 5 sections of a lossless transmission line by placing 5 inductors (use values of L=10H) in series, then connect each node between the inductors (and the node after the last inductor) to ground through a capacitor (use C=1nF) as shown in Figure 1. This is the lumped element model for a lossless transmission line.

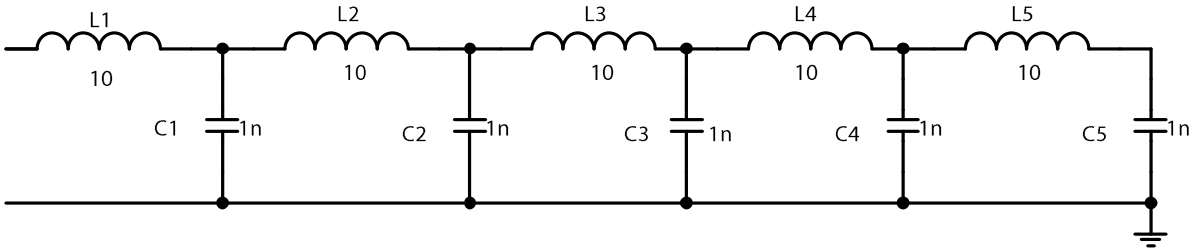


Figure 1

2. Calculate the characteristic impedance for this lossless transmission line model.

3. Place a resistor having a value of the characteristic impedance of your line at the end of your line in parallel with the final capacitor. This is the load.

4. Place a VSIN source on the left side of your circuit and connect it to the line by means of the first inductor. Adjust VSIN such that it has amplitude of 10 V, phase of 0 degrees, and a frequency of 1 kHz. Note that VSIN also requires that you set a "Voff" value. Set this to zero.

5. Determine the phase velocity of the transmission line model.

A. The lumped element model is in terms of segments rather than position, so the phase velocity will be in units of segments/seconds.

B. Place a marker at the node between L2 and L3, between L3 and L4, and between L4 and L5.

C. Go to the Analysis menu and open the setup window. Find the transient analysis section and then open it. Set the following parameters: Print Step (.01m), final Time (5m), No-Print Delay (0), and Step Ceiling (.01m). Run the simulation and examine the trace plots associated with each marker.

D. Divide the number of segment by the time difference between the peak for each marker.

E. Determine the phase velocity for following three cases (1) between L2-L3 and L3-L4, (2) between L3-L4 and L4-L5, and (3) between L2-L3 and L4-L5. The phase velocity for these cases should be approximately the same. (See Figure 2)

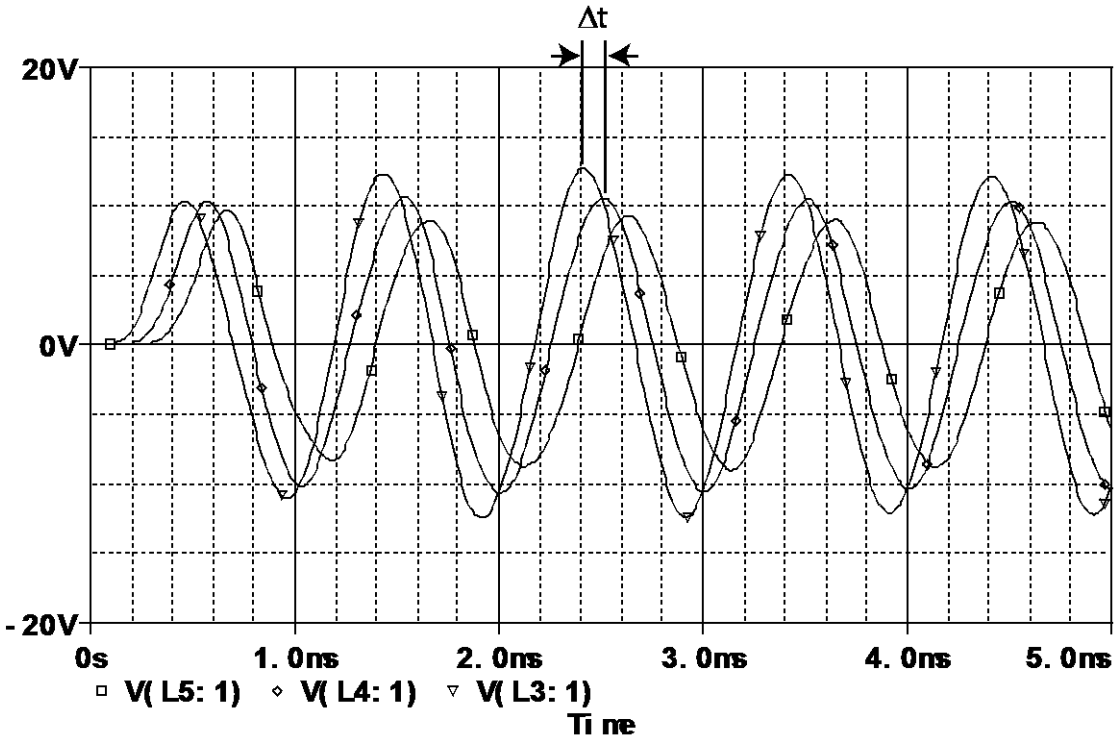


Figure 2. The phase velocity between segments 3 and 4 is 1/Δt.

1. Determine the propagation constant, β, for this lumped element model.

a. The propagation constant is the position variable quantity.

b. It is easier to determine the wavelength, λ, and then calculate the propagation constant as given by $\beta = 2\pi/\lambda$.

c. To find the wavelength you need to place a voltage monitor between each segment and then measure the voltage on each marker at the same time. (see Figure 3)

d. Add more segments until you can map out more than one entire period of the sinusoidal wave.

e. Plot the voltage as a function of segment number.

f. Measure the period in terms of segments.

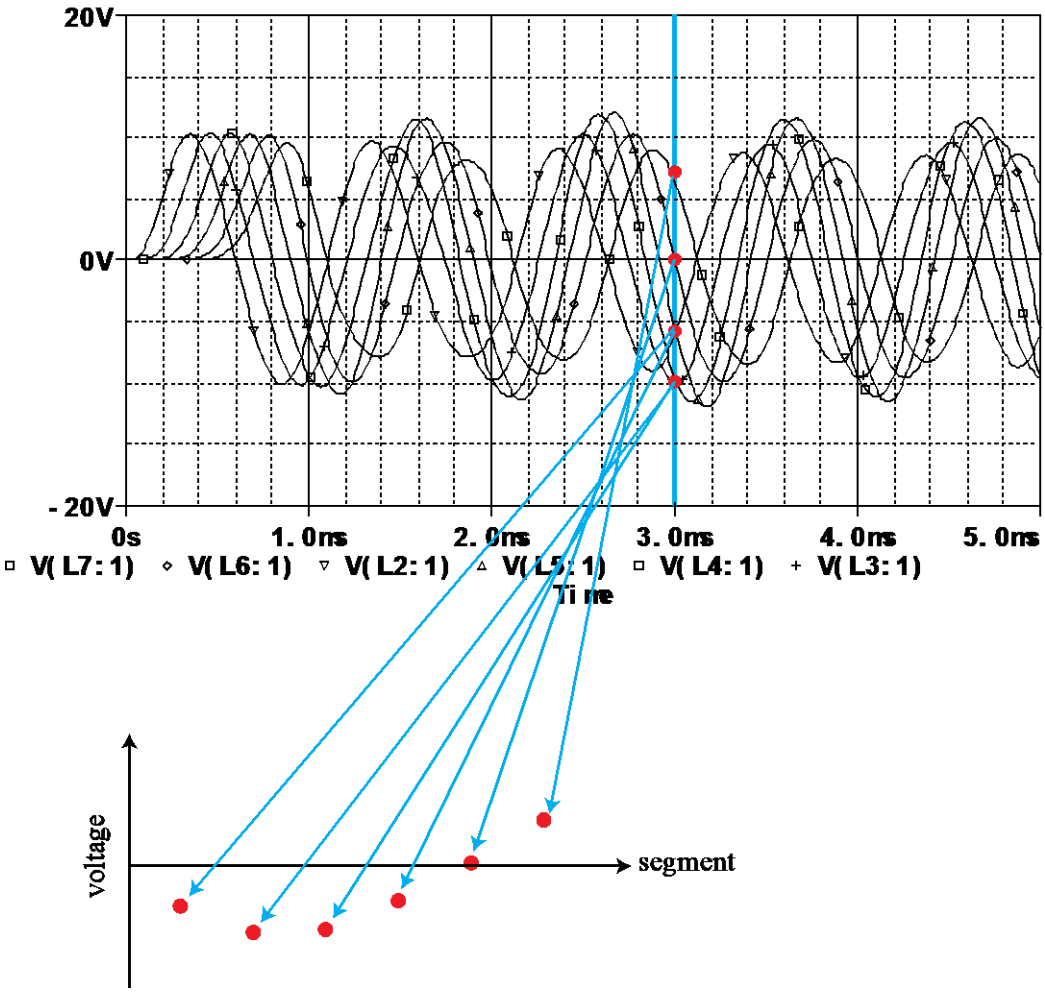


Figure 3

2. Compare the relationship of your calculated phase velocity and wavelength to that derived in the book (Eq. 2.53).

3. Add resistors in series with each inductor that causes the sinusoid to be visibly damped while still showing oscillation. This lumped element models the finite resistance of the metal lines of the transmission line. **Describe your new circuit and its values for decay constant. Include a printout with your description.**

4. Add resistors in parallel with each capacitor that causes the sinusoid to be visibly damped while still showing oscillation. This lumped element models the small conductivity of the insulator (between the two metal lines of the transmission line.

Describe your new circuit and its values for decay constant. Include a printout with your description.

THEROTICAL CALCULATIONS:

CHARACTERISTIC IMPEDENCE $Z_0 = \sqrt{\frac{L}{C}} =$

RESULT:

APPLICATION:

Expt. No: 7

PSPICE SIMULATION OF INTEGRATOR & DIFFERENTIATOR

Date:

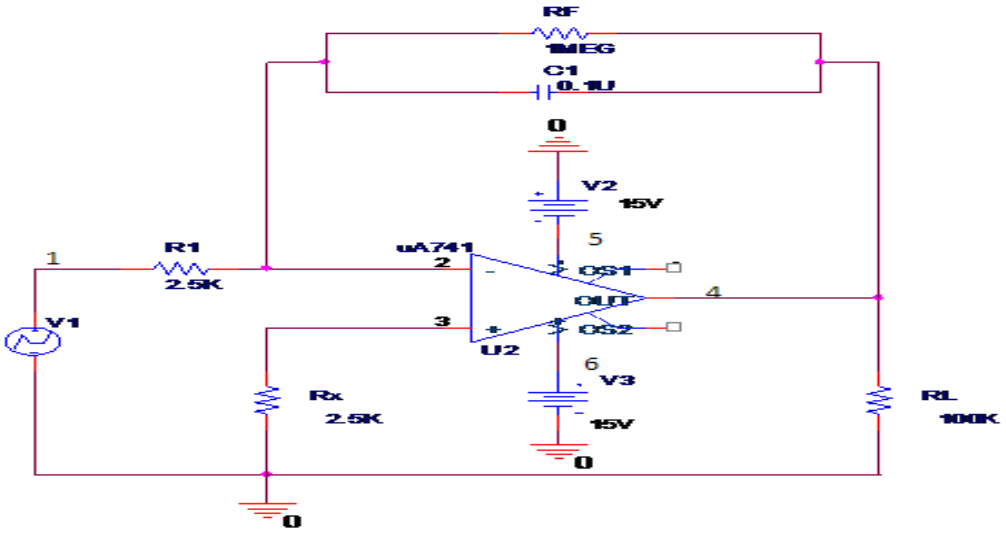
AIM: To simulate an Op-amp based Integrator & Differentiator circuits using Pspice.

SIMULATION TOOLS REQUIRED:

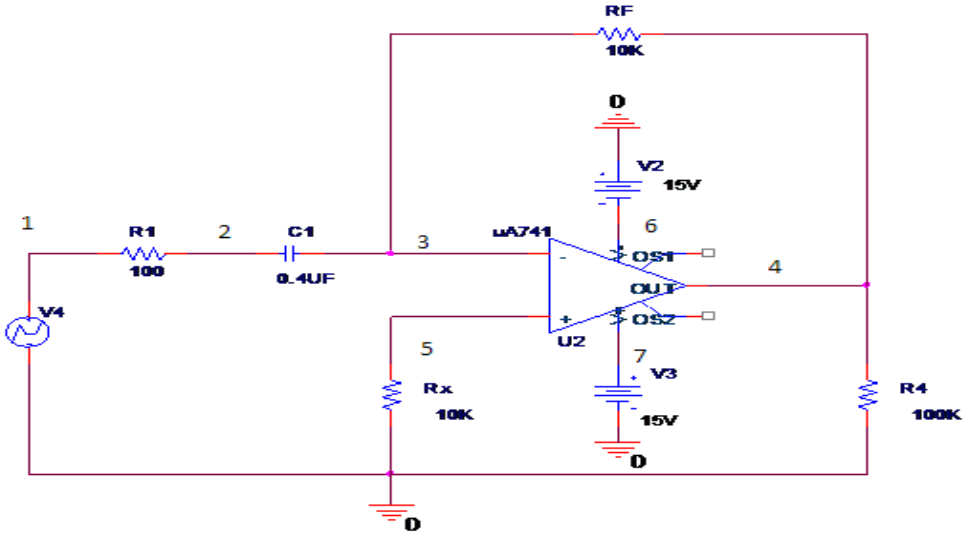
- PC with PSPICE Software

CIRCUIT DIAGRAMS:

INTEGRATOR CIRCUIT



DIFFERENTIATOR CIRCUIT



THEORY:

PROGRAMS:

INTEGRATOR:

DIFFERENTIATOR:

RESPENSE OF DIFFERENTIATOR

PROCEDURE:

1. Write the program in a new text file in PSpice AD.
2. Save the file using the notation filename.cir.
3. Activate the file by opening it.
4. Run the simulation process using blue button.
5. By clicking Add Trace icon, get the required waveform.

MODEL CALCULATION:

A).FOR DIFFERENTATIOR

$$V0 = RF * C * \frac{dV}{dt}$$

$=$

$=$

B).FOR INTEGRATOR

$$V0 = \frac{1}{R1 * C_F} \int_0^T Vdt$$

$=$

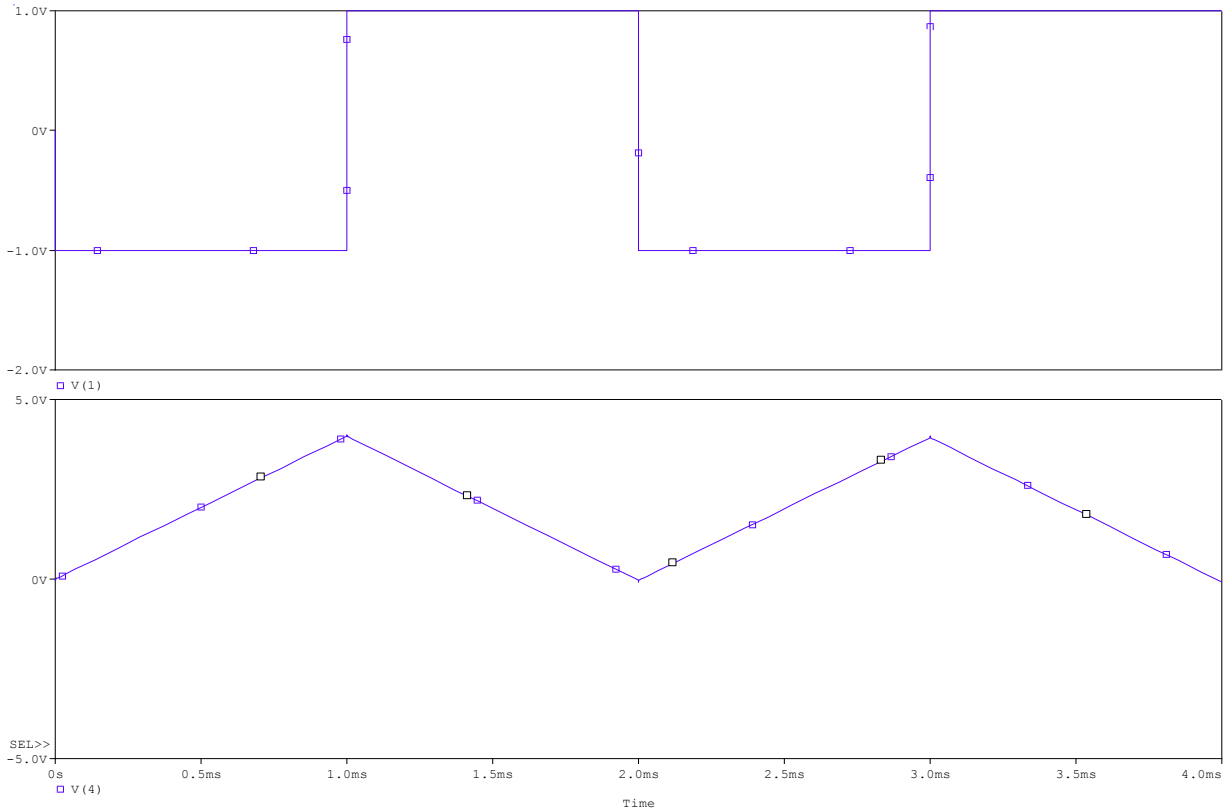
$=$

RESULT:

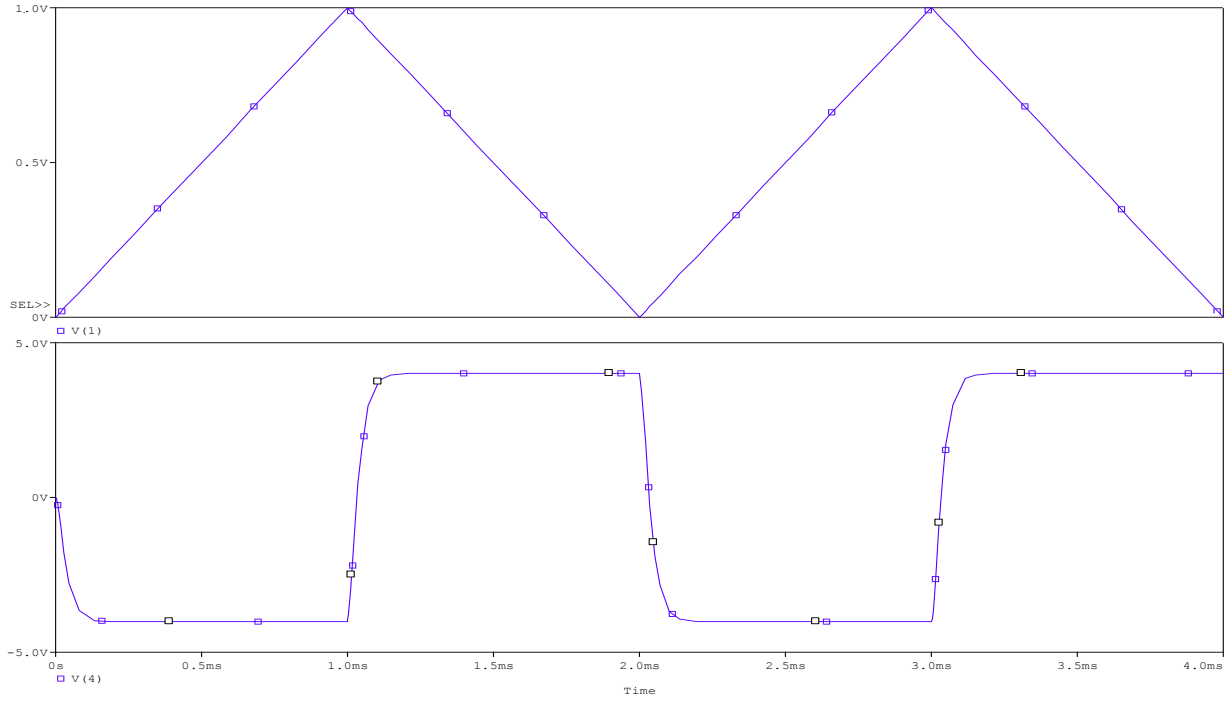
APPLICATIONS:

MODEL WAVEFORMS:

INTEGRATOR INPUT & OUTPUT WAVEFORMS



DIFFERENTIATOR INPUT & OUTPUT WAVEFORMS



Expt. No: 8

SIMULATION OF D.C SEPARATELY EXCITED MOTOR USING TRANSFER FUNCTION APPROACH.

Date:

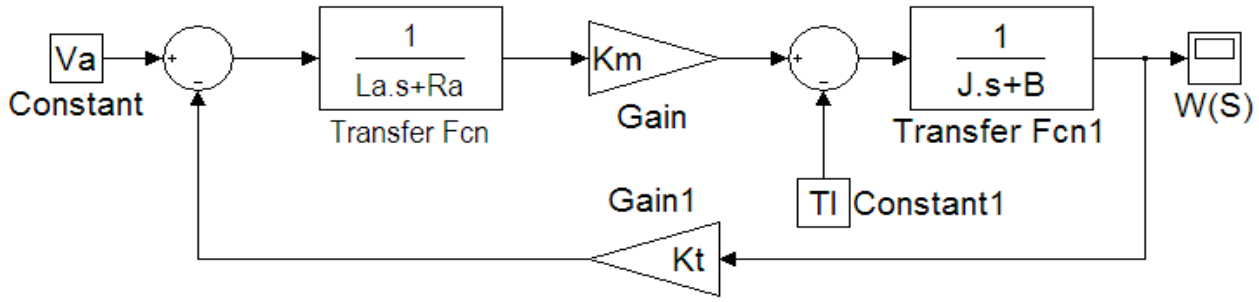
AIM: To Simulation of D.C separately excited motor using Transfer function approach using MATLAB.

SIMULATION TOOLS REQUIRED:

- PC with MATLAB Software

SPECIFICATIONS:

CIRCUIT DIAGRAMS:



THEORY:

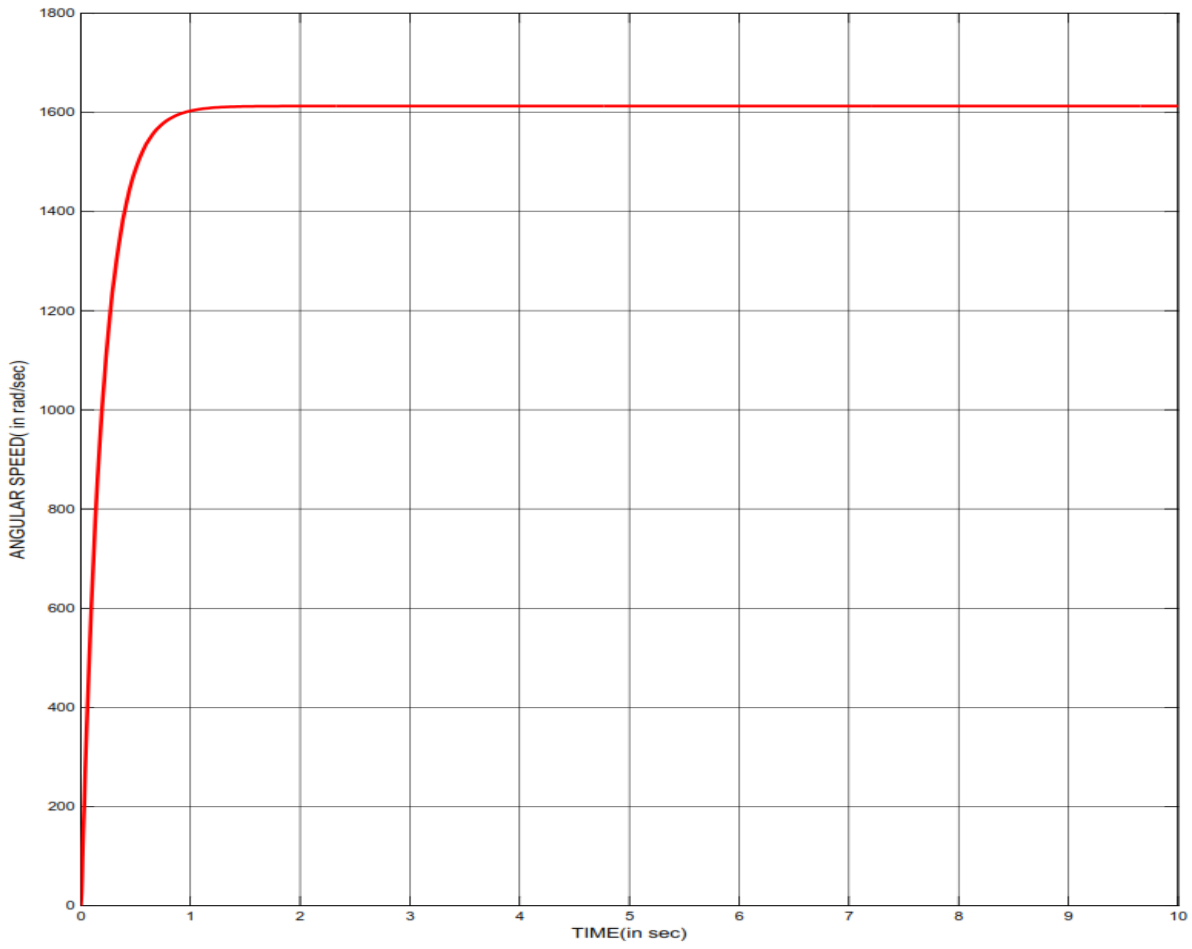
PROCEDURE:

1. Open the MATLAB software by clicking on the MATLAB icon.
2. Click on file menu and open new Model file.
3. Add the required blocks from simulink library browser to model file.
4. Click on the start button to simulate.
5. Draw the plot.

RESULT:

APPLICATIONS:

MODEL WAVEFORMS:



Expt. No: 9(A)

SIMULATION OF BOOST CONVERTERS

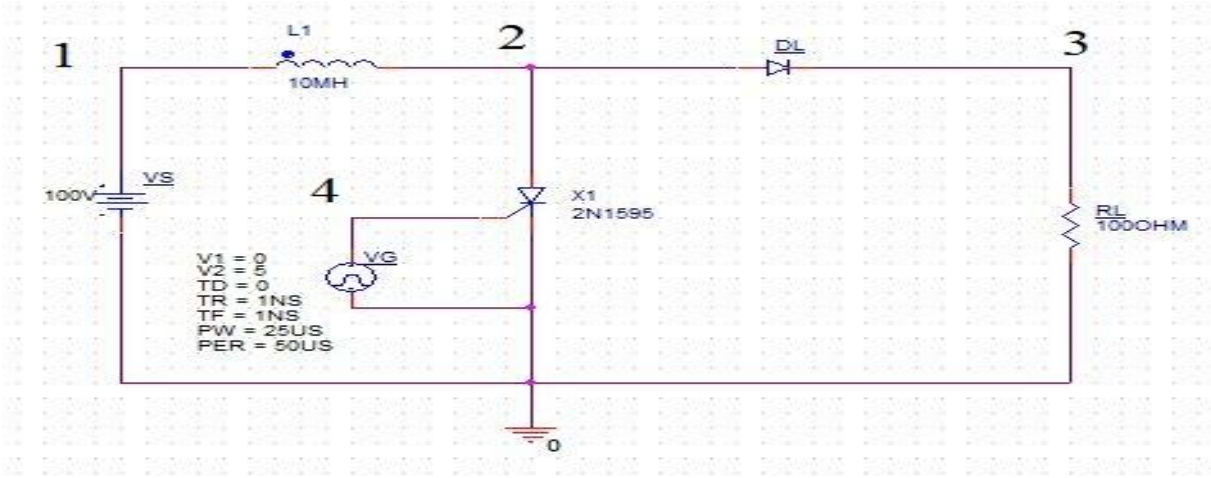
Date:

AIM: To simulate boost converters using Pspice.

SIMULATION TOOLS REQUIRED:

- PC with PSPICE Software

CIRCUIT DIAGRAMS:



SPECIFICATIONS:

THEORY:

BOOST CHOPPER

PROCEDURE:

1. Write the program in a new text file in PSpice AD.
2. Save the file using the notation filename.cir.
3. Activate the file by opening it.
4. Run the simulation process using blue button.
5. By clicking Add Trace icon, get the required waveform.

MODEL CALCULATIONS:

$$V0 = \frac{VS}{1-\alpha} where \alpha = \frac{Ton}{T}$$

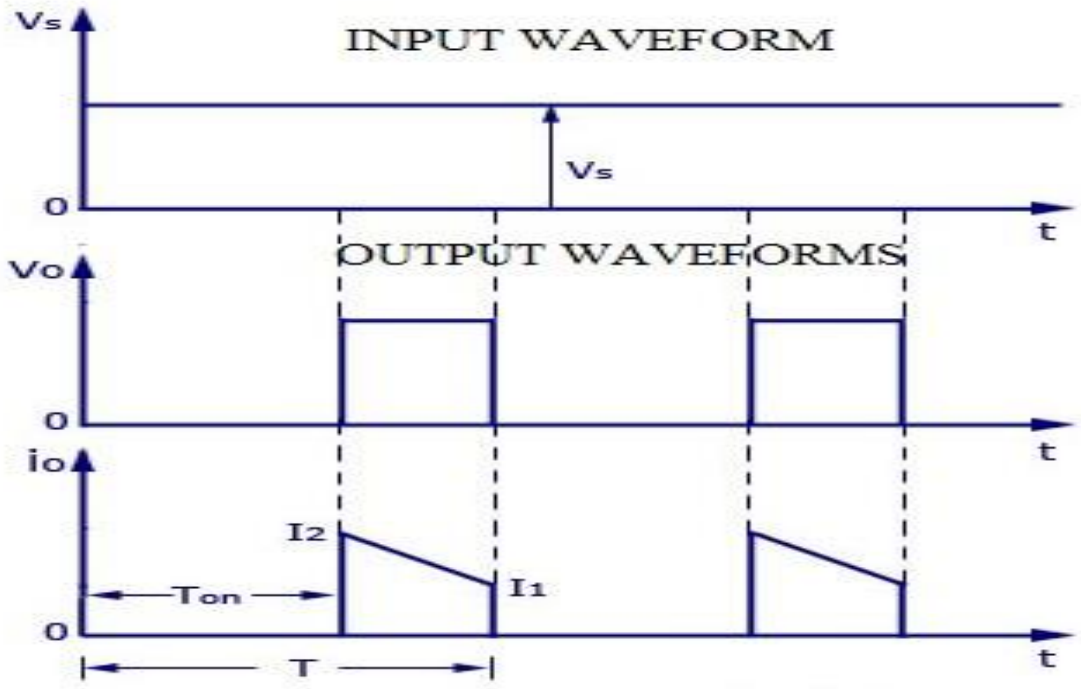
=

=

RESULT:

APPLICATIONS:

MODEL WAVEFORMS:



Expt. No: 9(B)

SIMULATION OF BUCK CONVERTERS

Date:

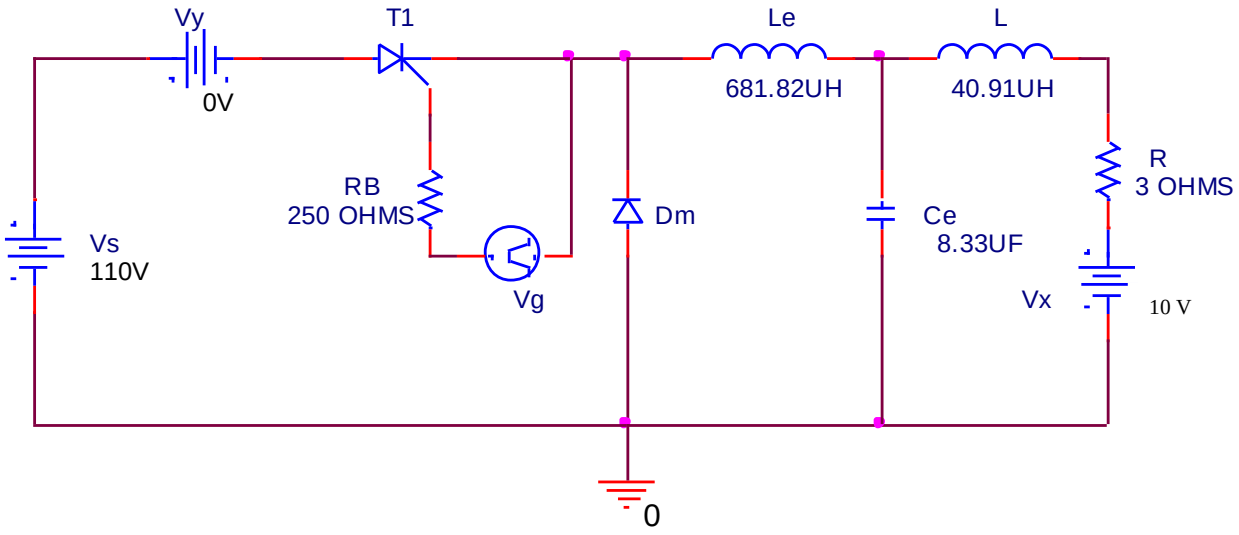
AIM: To analyze Buck chopper using Pspice.

SIMULATION TOOLS REQUIRED:

- PC with PSPICE Software

CIRCUIT DIAGRAM:

Buck chopper



SPECIFICATIONS:

THEORY:

PROGRAM:

PROCEDURE:

1. Write the program in a new text file in PSpice AD.
2. Save the file using the notation filename.cir.
3. Activate the file by opening it.
4. Run the simulation process using blue button.
5. By clicking Add Trace icon, get the required waveform.

MODEL CALCULATIONS :

$$V0 = \alpha * VS = \frac{Ton}{T} VS$$

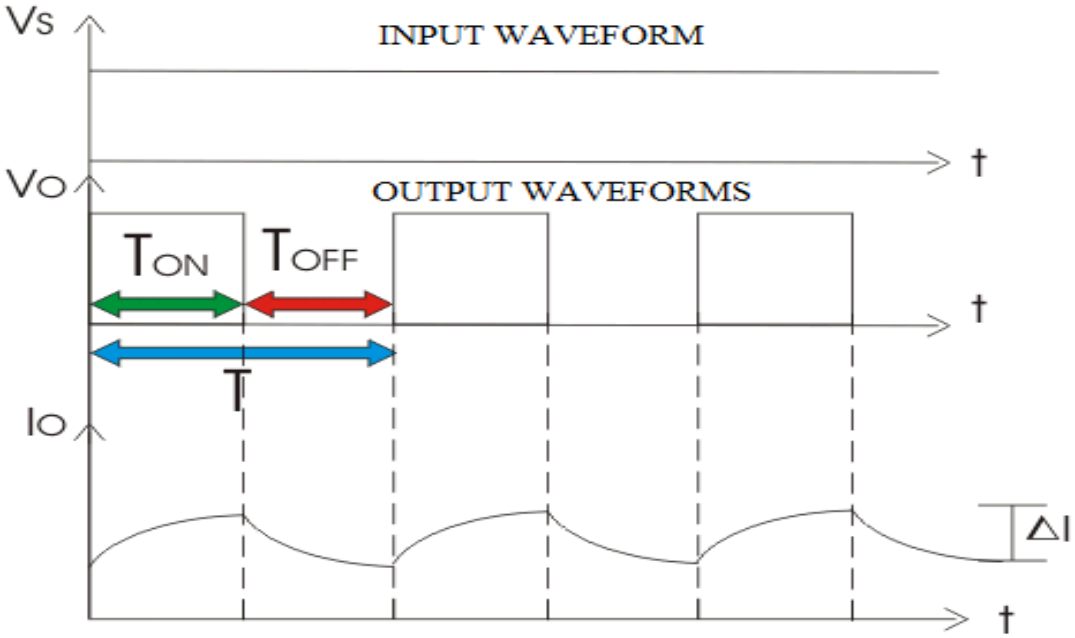
=

=

RESULT:

APPLICATIONS:

MODEL WAVEFORMS:



Expt. No: 10

SINGLE PHASE INVERTER WITH PWM CONTROL

Date:

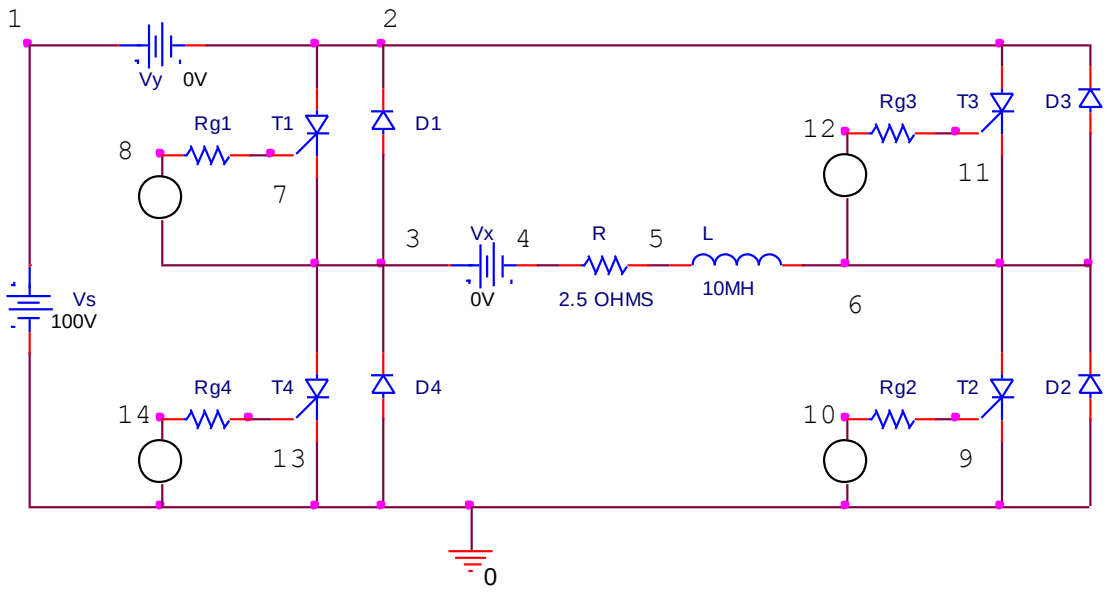
AIM: PSpice analysis of single phase inverter with PWM control.

SIMULATION TOOLS REQUIRED:

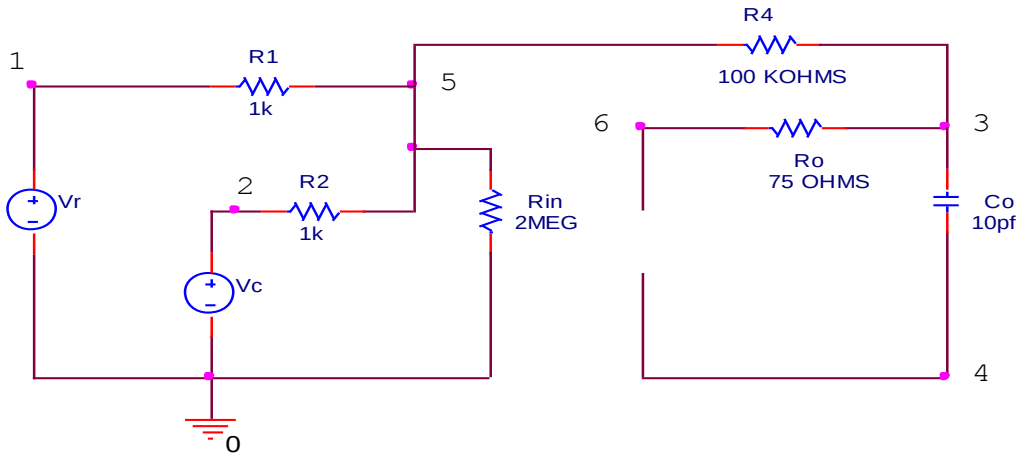
- PC with PSPICE Software

CIRCUIT DIAGRAM:

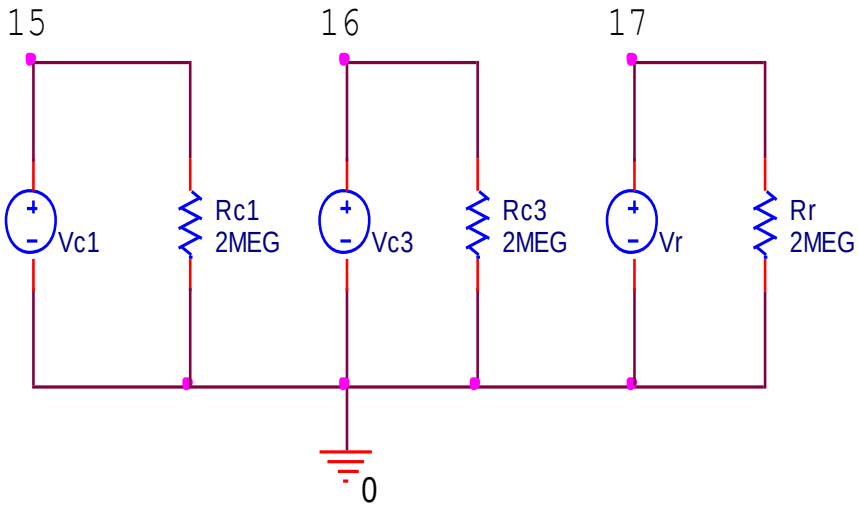
Single phase inverter with PWM control



PWM Generator



Carrier and Reference signals



SPECIFICATIONS:

THEORY:

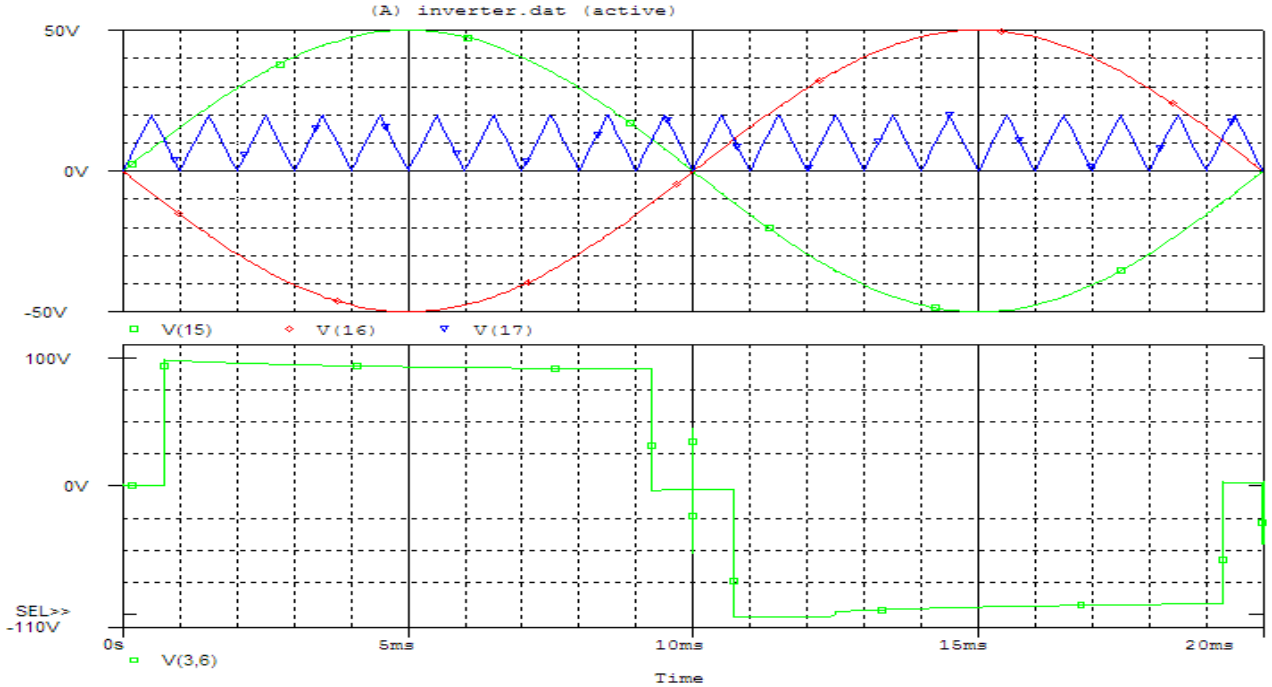
PROGRAM:

PROCEDURE:

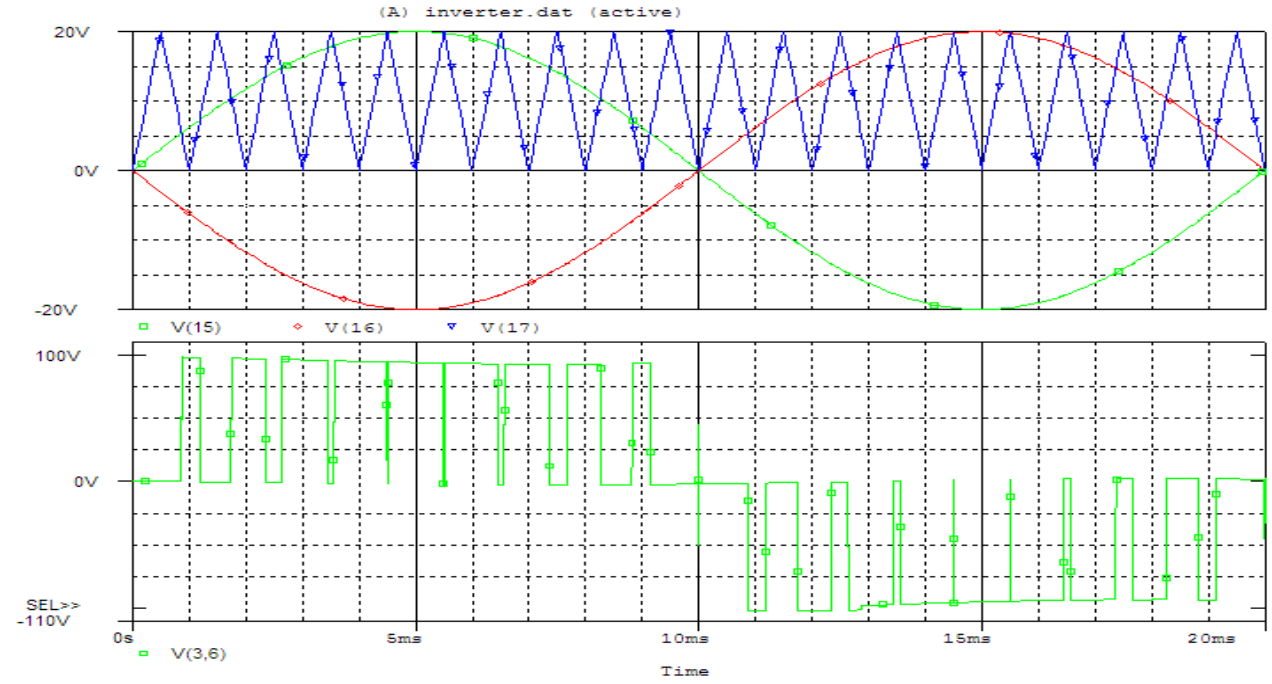
1. Write the program in a new text file in PSpice AD.
2. Save the file using the notation filename.cir.
3. Activate the file by opening it.
4. Run the simulation process using blue button.
5. By clicking Add Trace icon, get the required waveform.

MODEL WAVEFORM:

AMPLITUDE OF 'VC1&VC2'>AMPLITUE OF 'VR'



AMPLITUDE OF 'VC1&VC2'=AMPLITUE OF 'VR'



MODEL CALCULATIONS:

CASDE 'A':OVER MADULATION (Ar >Ac)

==5>1

>Mr>1

=>Ar/Ac

=

N=number of pulses per half cycle

N=1

CASDE 'A':UNDER MADULATION (Ar <Ac)

=>Mr<1

=>Ar/Ac

=

=

N-1=number of pulses per half cycle

$N = fc/(2fr) =$

$N - 1 =$

RESULT:

APPLICATIONS: