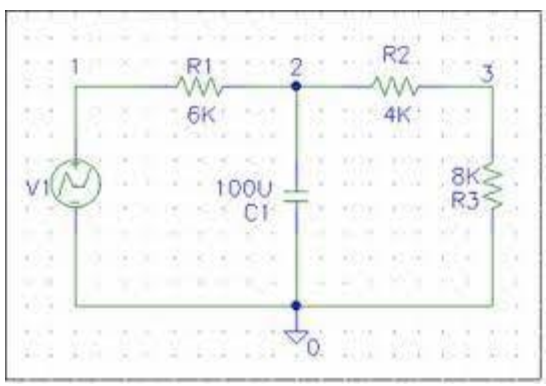
CIRCUITS SIMULATION &ANALYSIS USING PSPICE LABORATORY LAB MANUAL



Department of Electrical and Electronics Engineering

VEMU INSTITUTE OF TECHNOLOGY::P.KOTHAKOTA

NEAR PAKALA, CHITTOOR-517112

(Approved by AICTE, New Delhi & Affiliated to JNTUA, Anantapuramu)

CIRCUITS SIMULATION &ANALYSIS USING PSPICE LABORATORY LAB MANUAL

Name: ____

H.T.No: ____

Year/Semester: ________________________________

Department of Electrical and Electronics Engineering

VEMU INSTITUTE OF TECHNOLOGY::P.KOTHAKOTA

NEAR PAKALA, CHITTOOR-517112

(Approved by AICTE, New Delhi & Affiliated to JNTUA, Anantapuramu)

VEMU INSTITUTE OF TECHNOLOGY

DEPT.OF ELECTRICAL AND ELECTRONICS ENGINEERING

VISION OF THE INSTITUTE

- To be a premier institute for professional education producing dynamic and vibrant force of technocrats with competent skills, innovative ideas and leadership qualities to serve the society with ethical and benevolent approach.

MISSION OF THE INSTITUTE

- To create a learning environment with state-of-the art infrastructure, well equipped laboratories, research facilities and qualified senior faculty to impart high quality technical education.
- To facilitate the learners to foster innovative ideas, inculcate competent research and consultancy skills through Industry-Institute Interaction.
- To develop hard work, honesty, leadership qualities and sense of direction in rural youth by providing value based education.

VISION OF THE DEPARTMENT

- To produce professionally deft and intellectually adept Electrical and Electronics Engineers and equip them with the latest technological skills, research & consultancy competencies along with social responsibility, ethics, Lifelong Learning and leadership qualities.

MISSION OF THE DEPARTMENT

- To produce competent Electrical and Electronics Engineers with strong core knowledge, design experience & exposure to research by providing quality teaching and learning environment.
- To train the students in emerging technologies through state - of - the art laboratories and thus bridge the gap between Industry and academia.
- To inculcate learners with interpersonal skills, team work, social values, leadership qualities and professional ethics for a holistic engineering professional practice through value based education.

PROGRAM EDUCATIONAL OBJECTIVES(PEOs)

Programme Educational Objectives (PEOs) of B.Tech (Electrical and Electronics Engineering) **program are:**

Within few years of graduation, the graduates will

PEO 1: Provide sound foundation in mathematics, science and engineering fundamentals to analyze, formulate and solve complex engineering problems.

PEO 2: Have multi-disciplinary Knowledge and innovative skills to design and develop Electrical & Electronics products and allied systems.

PEO 3: Acquire the latest technological skills and motivation to pursue higher studies leading to research.

PEO 4: Possess good communication skills, team spirit, ethics, modern tools usage and the life-long learning needed for a successful professional career.

PROGRAM OUTCOMES (POs)

PO-1	**Engineering knowledge:** Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
PO-2	**Problem analysis:** Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
PO-3	**Design/development of solutions:** Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
PO-4	**Conduct investigations of complex problems:** Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
PO-5	**Modern tool usage:** Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
PO-6	**The engineer and society:** Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the

	consequent responsibilities relevant to the professional engineering practice.
PO-7	**Environment and sustainability:** Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
PO-8	**Ethics:** Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
PO-9	**Individual and team work:** Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
PO-10	**Communication:** Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
PO-11	**Project management and finance:** Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.
PO-12	**Life-long learning:** Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

PROGRAM SPECIFIC OUTCOMES (PSOs)

On completion of the B.Tech. (Electrical and Electronics Engineering) degree, the graduates will be able to

PSO-1: Higher Education: Apply the fundamental knowledge of Mathematics, Science, Electrical and Electronics Engineering to pursue higher education in the areas of Electrical Circuits, Electrical Machines, Electrical Drives, Power Electronics, Control Systems and Power Systems.

PSO-2: Employment: Get employed in Public/Private sectors by applying the knowledge in the domains of design and operation of Electronic Systems, Microprocessor based control systems, Power systems, Energy auditing etc.

CONTENTS

(20A02404) CIRCUITS SIMULATION &ANALYSIS USING PSPICE LABORATORY

S.NO.	NAME OF THE EXPERIMENT	PAGE NO.
1	**Simulation of DC &AC Circuits**	**01-04**
2	**Mesh Analysis**	**05-10**
3	**Nodal Analysis**	**11-16**
4	**DC Transient Response**	**17-20**
5	**Simulation of Single-phase half wave, Semi and full converters with RLE loads.**	**21-26**
6	**Simulation of Three-phase half wave, Semi and full converters with RLE loads.**	**27-30**
7	**Simulation of Buck, Boost and Buck-Boost Converters**	**31-36**
8	**Simulation of Single-phase AC voltage controller**	**37-39**
9	**Simulation of Single and Three phase Quasi Square wave and PWM Inverters.**	**40-44**
ADDITIONAL EXPERIMENTS		
10	**Frequency response of RLC Series Circuits**	**49-51**
11	**Verification of the maximum power dissipation (plot the power dissipated versus the load).**	**52-55**

JAWAHARLAL NEHRU TECHNOLOGICAL UNIVERSITY, ANANTAPUR

B. Tech II - II SEM (E.E.E)

L	C	P	C
1	0	2	2

(20A02404) CIRCUITS SIMULATION &ANALYSIS USING PSPICE LABORATORY

The objectives of this lab course are to make the student practically learn about

- Simulation of various circuits using PSPICE software.
- Simulation of single-phase half & fully-controlled converters, and inverters
- Simulation of single-phase AC Voltage controllers with different loads.

List of Experiments:

I Simulation of Electrical Circuits

a) DC & AC Circuits

b) Mesh Analysis

c) Nodal Analysis

d) Transient Response

II Simulation of Power Electronic Circuits

a) Single-phase half wave, Semi and full converters with RLE loads.

b) Three-phase half wave, Semi and full converters with RLE loads.

c) Buck, Boost and Buck-Boost Converters

d) Single-phase AC voltage controller

e) Single and Three phase Quasi Square wave and PWM Inverters.

OUTCOMES: At the end of the course the student will be able to

- Simulation of various circuits using PSPICE software.
- Simulation of single-phase half & fully-controlled converters, and inverters
- Simulation of single-phase AC Voltage controllers with different loads.

GENERAL INSTRUCTIONS FOR LABORATORY CLASSES

DO'S

1. Without Prior permission do not enter into the Laboratory.
2. While entering into the LAB students should wear their ID cards.
3. The Students should come with proper uniform.
4. Students should sign in the LOGIN REGISTER before entering into the laboratory.
5. Students should come with observation and record note book to the laboratory.
6. Students should maintain silence inside the laboratory.
7. Circuit connections must be checked by the lab-in charge before switching the supply

DONT'S

1. Students bringing the bags inside the laboratory.
2. Students wearing slippers/shoes insides the laboratory.
3. Students scribbling on the desk and mishandling the chairs.
4. Students using mobile phones inside the laboratory.
5. Students making noise inside the laboratory.
6. Students mishandle the devices.
7. Students write anything on the devices

SCHEME OF EVALUATION

S. NO	EXPERIMENT NAME	DATE	MARKS AWARDED				Total 30 (M)
			Record (10M)	Observation (10M)	Viva voce (5M)	Attendance (5M)	
1	Simulation of DC &AC Circuits						
2	Mesh Analysis						
3	Nodal Analysis						
4	DC Transient Response						
5	Simulation of Single-phase half wave, Semi and full converters with RLE loads.						
6	Simulation of Three-phase half wave, Semi and full converters with RLE loads.						
7	Simulation of Buck, Boost and Buck-Boost Converters						
8	Simulation of Single-phase AC voltage controller						
9	Simulation of Single and Three phase Quasi Square wave and PWM Inverters.						
ADDITIONAL EXPERIMENTS							
11	Frequency response of RLC Series Circuits						
12	Verification of the maximum power dissipation (plot the power dissipated versus the load).						

Signature of Lab In-charge

Exp. No.: 1 **Date:**

SIMULATION OF DC&AC CIRCUITS

AIM:

1) To Simulate the DC Circuit for determining the Theremin's equivalent circuit using PSPICE.

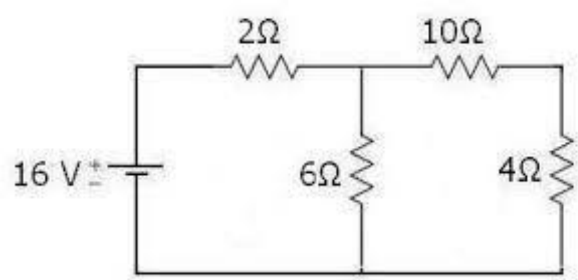
2) To Simulate the AC Circuit analysis by using PSPICE.

APPARATUS REQUIRED:

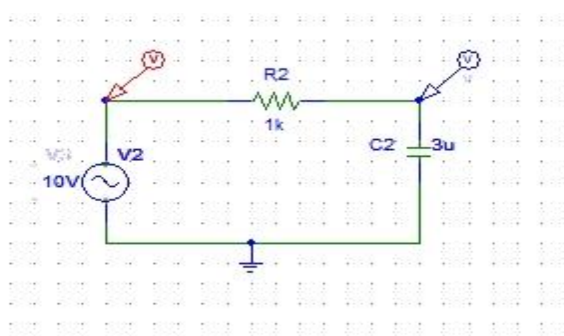
S.No	Name Of The Equipment	Quantity
1	PC	1NO
2	PSPICE software	1NO

CIRCUIT DIAGRAM:

DC CIRCUIT:



AC CIRCUIT:



PROCEDURE:

1) 1. Open PSPICE A/D windows

2. Create a new circuit file

3. Enter the program representing the nodal interconnections of various components

4. Run the program

5. Observe the response through all the elements in the output file

6. Observe the voltage, current graph of any in probe window

2) 1.Open the PSPICE design manager in search bar from the design manager click on the run

Schematic button to open a new blank schematic.

2. Save the file and the search the components and connect the circuit as shown in figure.

3. Place the voltage marker at resistor and capacitor.

4. In analysis we can simulate the program.

PROGRAM

THEVENINS THEOREM:

V_{IN} 1 0 dc 16V
R_1 1 2 2
R_2 2 3 10
R_3 2 0 6
R_4 3 0 4
.TF V (3, 0) V_{IN}
.END

THEORETICAL CALCULATIONS

RESULT:

VIVA QUESTIONS:

1. State Thevenin's theorem?
2. What are active elements and passive elements?
3. What are non linear elements and give examples?
4. Can you find maximum power transferred to the load by using Thevenins theorem?
5. Define KVL and KCL?
6.

Exp. No.: 2 **Date:**

MESH ANALYSIS

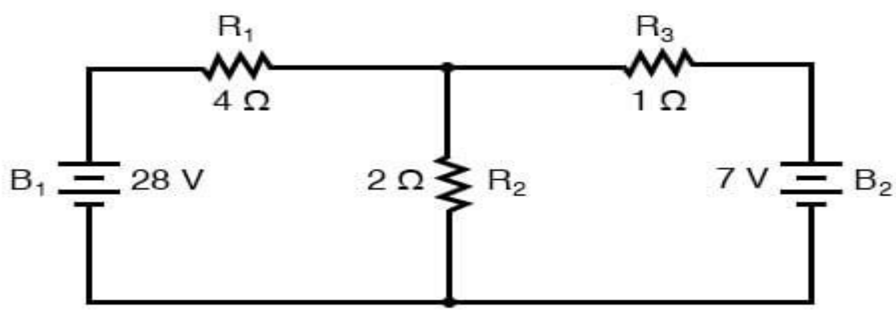
AIM:

To determine currents for the given DC circuit by mesh analysis.

APPARATUS REQUIRED:

S.No	Name Of The Equipment	Quantity
1	PC	1NO
2	PSPICE software	1NO

CIRCUIT DIAGRAM:



PROCEDURE:

1. Open PSPICE A/D windows
2. Create a new circuit file
3. Enter the program representing the nodal interconnections of various components
4. Run the program
5. Observe the response through all the elements in the output file
6. Observe the required outputs (Graphs) in output window.

PROGRAM

.OP

```
V1   1 0 DC 28V
V2   3 0 DC 7V
R1   1 2 4
R2    2 0 2
R3    2 3 1
.OP
.print DC I(R1) I(R2) I(R3)
.end
```

THEORETICAL CALCULATIONS:

RESULT:

VIVA QUESITIONS:

1. What are internal resistance of an ideal voltage source and an ideal current source?

2. What are active elements and passive elements?

3. What are non linear elements and give examples?

4. What is meant by super mesh?

Exp. No.: 3 **Date:**

NODAL ANALYSIS

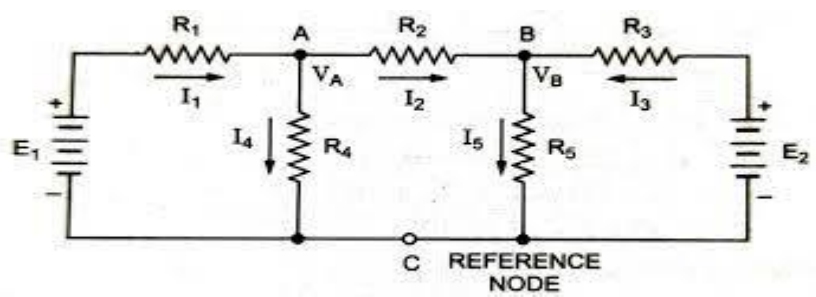
AIM:

To Simulate the DC Circuit for determining the all node voltages using PSPICE.

APPARATUS REQUIRED:

S.No	Name Of The Equipment	Quantity
1	PC	1NO
2	PSPICE software	1NO

C IRCUIT DIAGRAM:



E1=30V, E2=10V
R1=1000Ω, R2=4000Ω, R3=6000Ω, R4=2000Ω, R5=8000Ω

PROCEDURE:

1. Open PSPICE A/D windows
2. Create a new circuit file
3. Enter the program representing the nodal interconnections of various components
4. Run the program
5. Observe the response through all the elements in the output file
6. Observe the required outputs (Graphs) in output window.

PROGRAM

E1 1 0 DC 30V : DC Voltage source of 30V between 1& 0 nodes

E2 4 0 DC 10V : DC Voltage source of 10V between 4& 0 nodes

R1 1 2 1000 : Resistance of 1000ohms between 1 & 2 nodes

R2 2 3 4000 : Resistance of 800ohms between 5 & 2 nodes

R3 3 4 6000 : Resistance of 6000ohms between 2 & 3 nodes

R4 2 0 2000 : Resistance of 200ohms between 4 & 0 nodes

R5 3 0 8000 : Resistance of 200ohms between 3 & 0 nodes

.OP : Directs the bias point to the output file

.END : End of the program.

THEORETICAL CALCULATIONS:

RESULT:

VIVA QUESITIONS:

1. Define Node

2. What are the advantages of nodal analysis over mesh analysis?

3. Which law is applicable for nodal analysis?

4. What is the difference between nodal analysis and super node analysis?

5. Give any two comparisons between nodal analysis and mesh analysis?

Exp. No.:4 **Date:**

DC TRANSIENT RESPONSE

AIM:

To find out the transient response and parametric analysis by simulation of RLC circuits Using Pulse, and Step response.

SOFTWARE REQUIRED:

PSPICE – Personal Computer Simulated Program with Integrated Circuit Emphasis.

a) Simulation of STEP RESPONSE Using PSPICE:

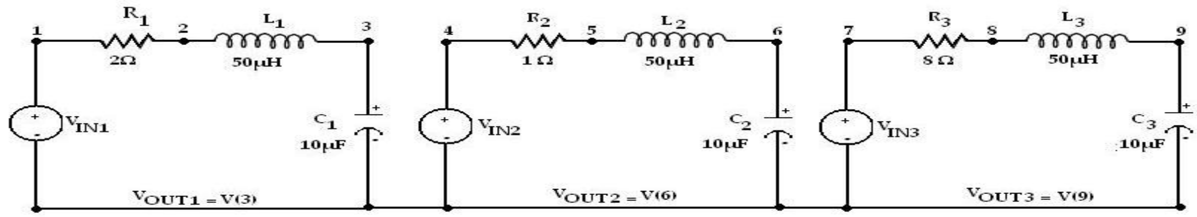
SYNTAX USED:

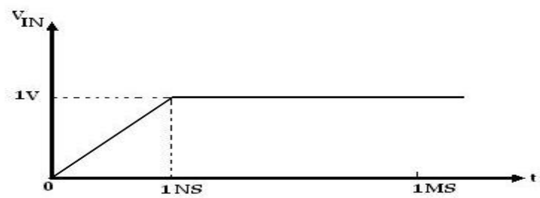
S.NO	TYPE OF SOURCE	REPRESENTATION OF SOURCE	DECLARATION FORMAT
1	STEP RESPONSE	PWL	STEP (Time at a Point) (Voltage at a Point)
2	TRANSIENT ANALYSIS	.TRAN	.TRAN TStep Tstop [TStart TMax] [UIC]
3	PROBE STATEMENT	.PROBE	It is a wave form analyzer
4	PLOT STATEMENT	.PLOT	.PLOT (Output Variables) {(Lower limit

DATA REQUIRED FOR DRAWING THE CIRCUIT DIAGRAM:

For example, Three RLC circuits with R=2Ω, 1 Ω, and 8 Ω respectively, with L having the values of 50μH each, with C having the values of 10μF each. The inputs are identical Step Response. The Step having the Time at points as 1nsec and 1msec respectively and Voltage at a point as 1V respectively. Use PSPICE to plot and calculate the transient response from 0 to 400μseconds with an increment of 1μsecond. Plot the voltages across the capacitors.

CIRCUIT DIAGRAM:





PROCEDURE:

1. Open PSpice A/D windows
2. Create a new circuit file
3. Enter the program representing the nodal interconnections of various components
4. Run the program
5. Observe the response through all the elements in the output file
6. Observe the voltage, current graph of any in probe window.

PROGRAM

```
VIN1 1 0 PWL(0 0 1NS 1V 1MS 1V)
VIN2 4 0 PWL(0 0 1NS 1V 1MS 1V)
VIN3 7 0 PWL(0 0 1NS 1V 1MS 1V)
R1  1 2  2
R2   4 5  1
R3   7 8  8
L1    2 3  50UH
L2    5 6  50UH
L3    8 9 50UH
C1    3 0 10UF
C2    6 0 10UF
C3    9 0 10UF
.TRAN 1US 400US
.PLOT TRAN V(3) V(6) V(9)
.PROBE
.END
```

THEORETICAL CALCULATIONS

RESULT:

VIVA QUESITIONS:

1. Define transient response.

2. Define sinusoidal response.

3. Define time constant.

4. When Transient behavior occur in any circuits ?

Expt. No: 5 **Date:**

PSPICE ANALYSIS OF SINGLE PHASE FULL CONVERTERWITH RL & RLE LOADS

AIM:

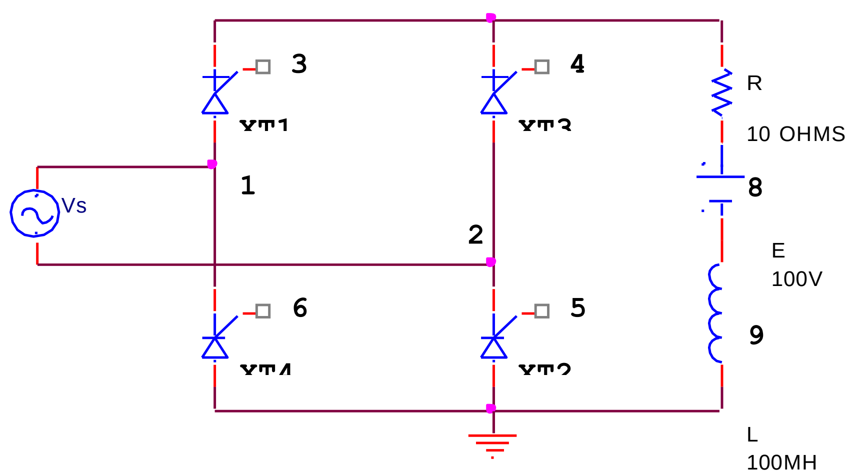
To analyze the single phase full converter with RL and RLE Loads.

SIMULATION TOOLS REQUIRED:

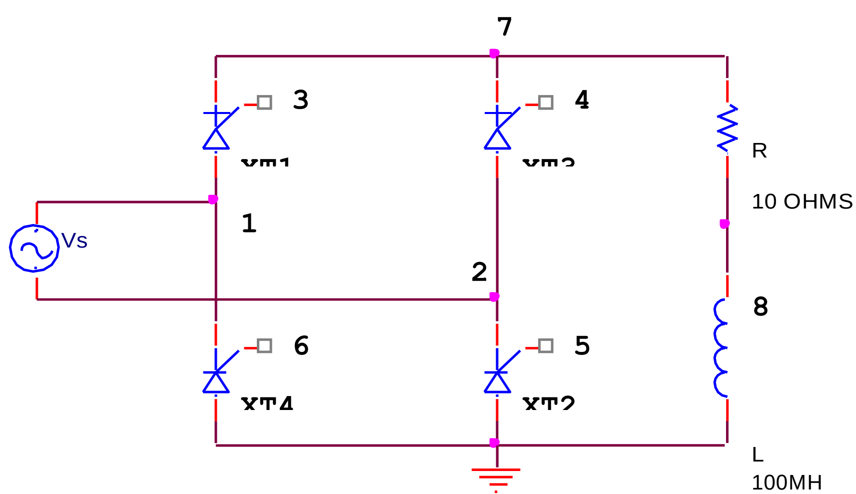
- PC with PSPICE Software

CIRCUIT DIAGRAMS:

Single Phase full converter with RL load



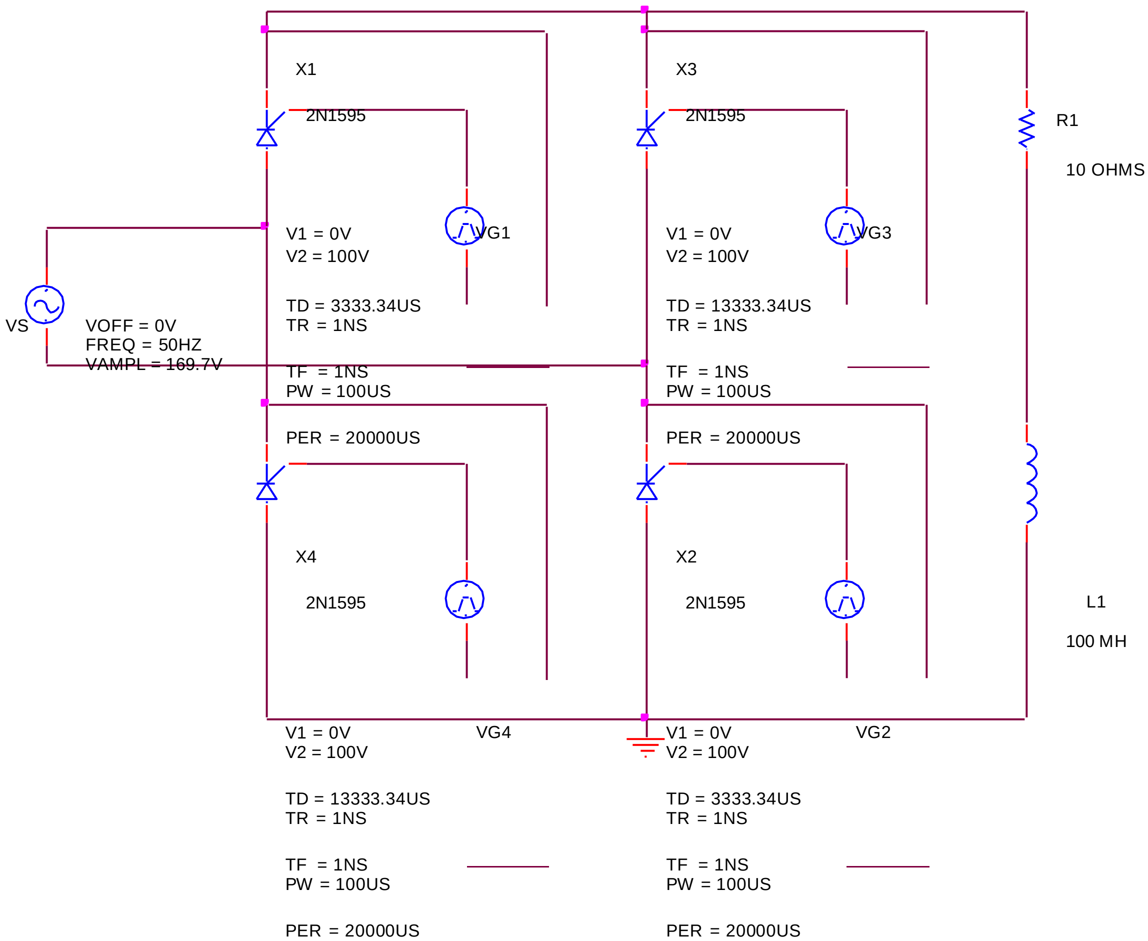
Single Phase full converter with RLE load



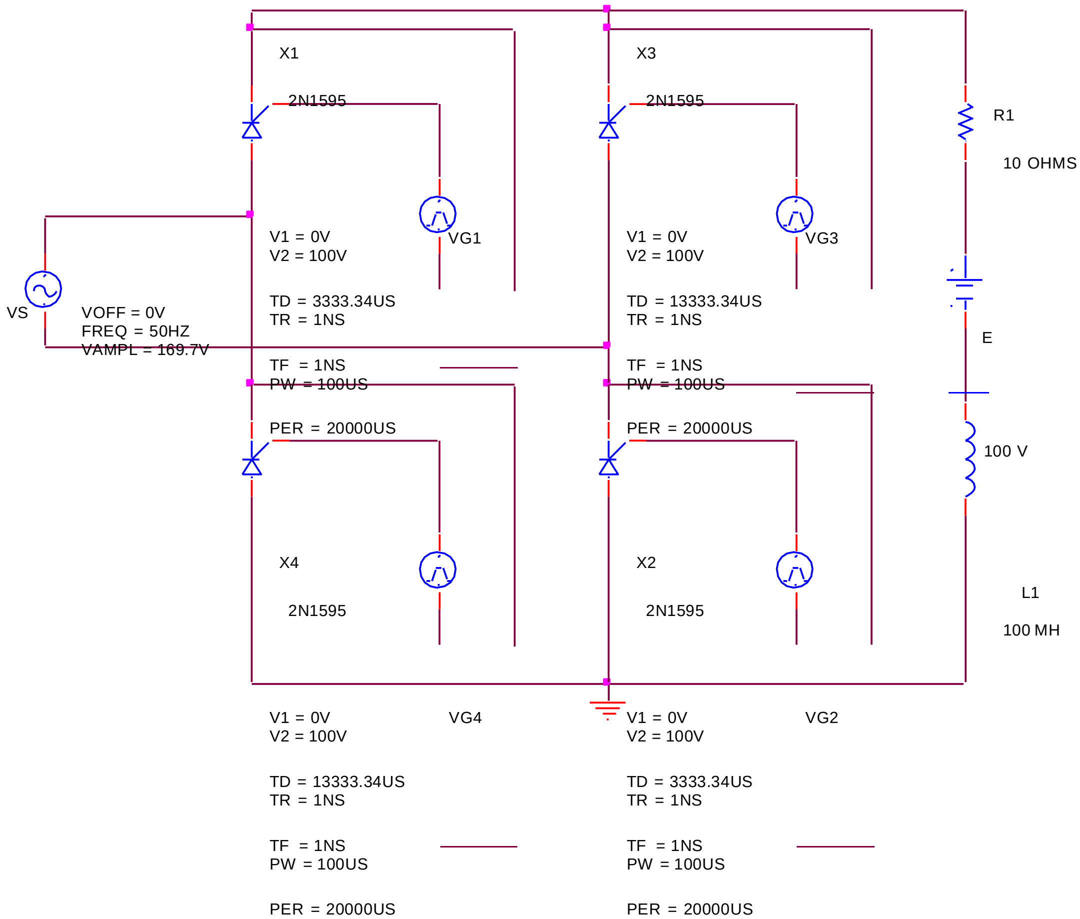
0

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

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

$$V0 = \frac{2V_M}{\Pi} COS(\_\_\_\_\_\_)$$

$= \_\_\_\_\_\_\_\_ V$

B)FOR RLE LOAD

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

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

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

Max value of firing angle

$\theta_2 = 180 - \theta_1 =$

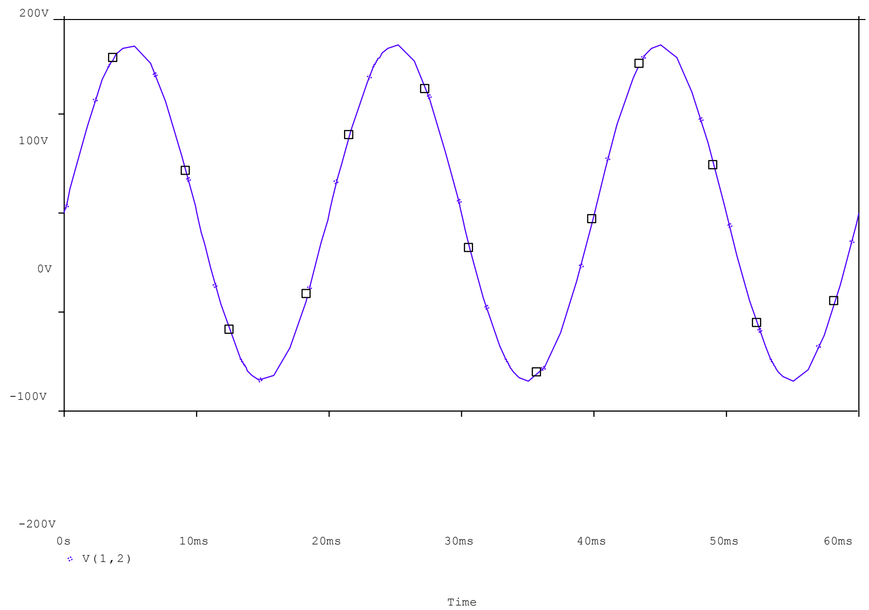
RESULT:

APPLICATIONS:

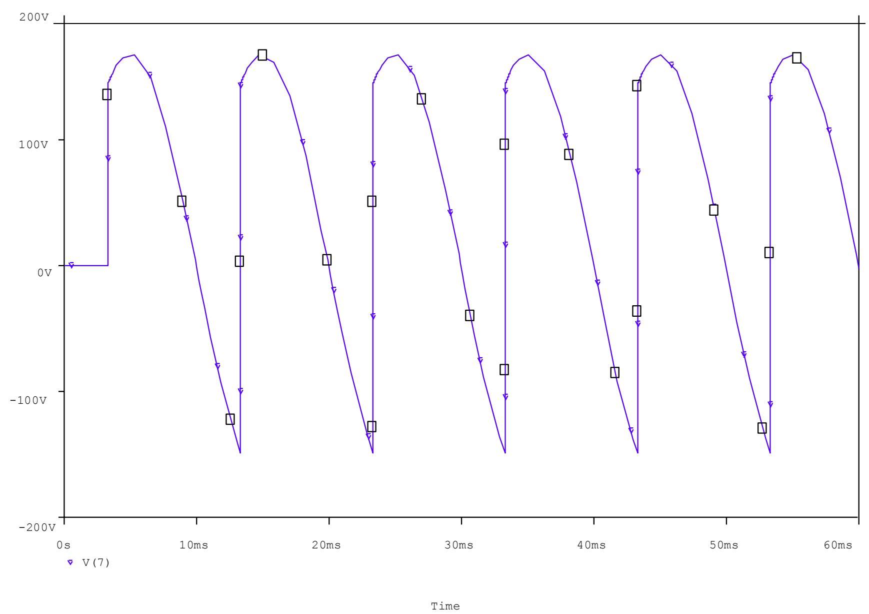
The single-phase full-wave controlled rectifier is used to control power flow in many applications (e.g., power supplies, variable-speed dc motor drives, and input stages of other converters)

MODEL WAVEFORMS FOR FULL CONVERTER:

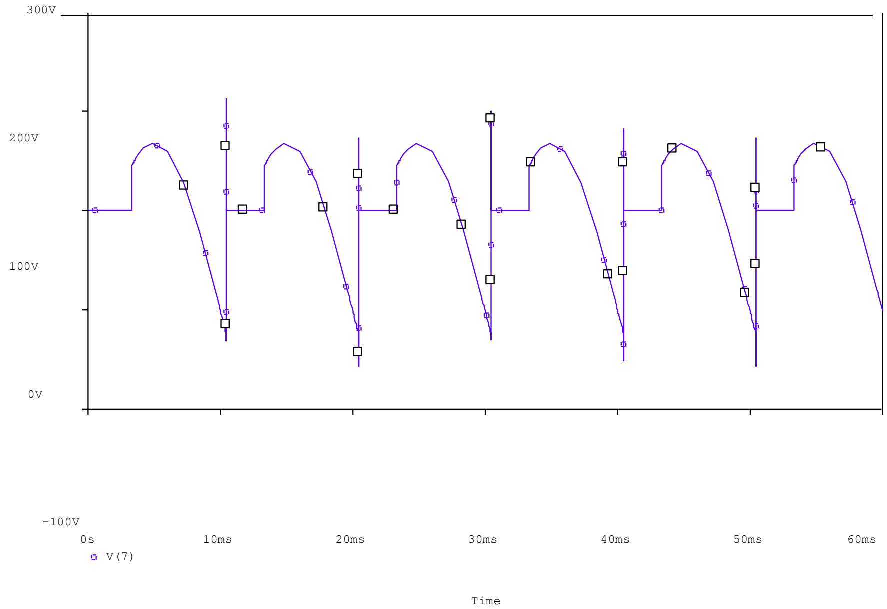
INPUT WAVEFORM



OUTPUT WAVEFORM WITH RL LOAD



OUTPUT WAVEFORM WITH RLE LOAD



Expt. No: 5 **Date:**

PSPICE ANALYSIS OF SINGLE PHASE AC VOLTAGECONTROLLER WITH RL LOAD

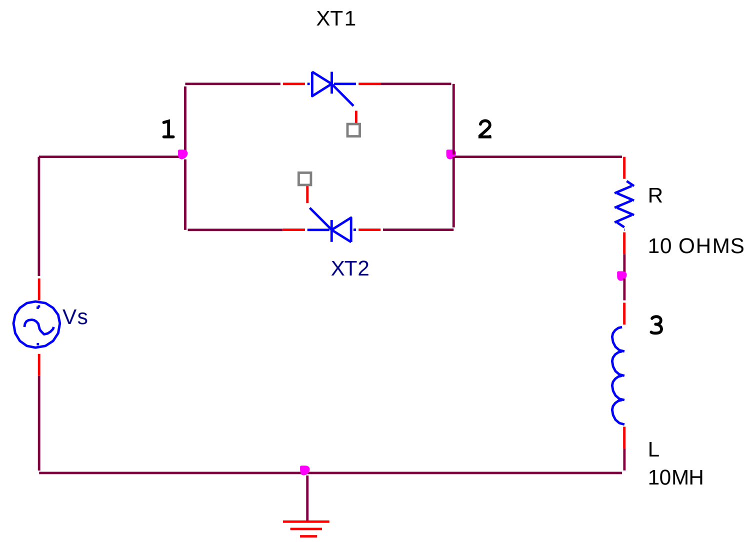
AIM: To analyze the single phase full converter with RL and RLE Loads.

SIMULATION TOOLS REQUIRED:

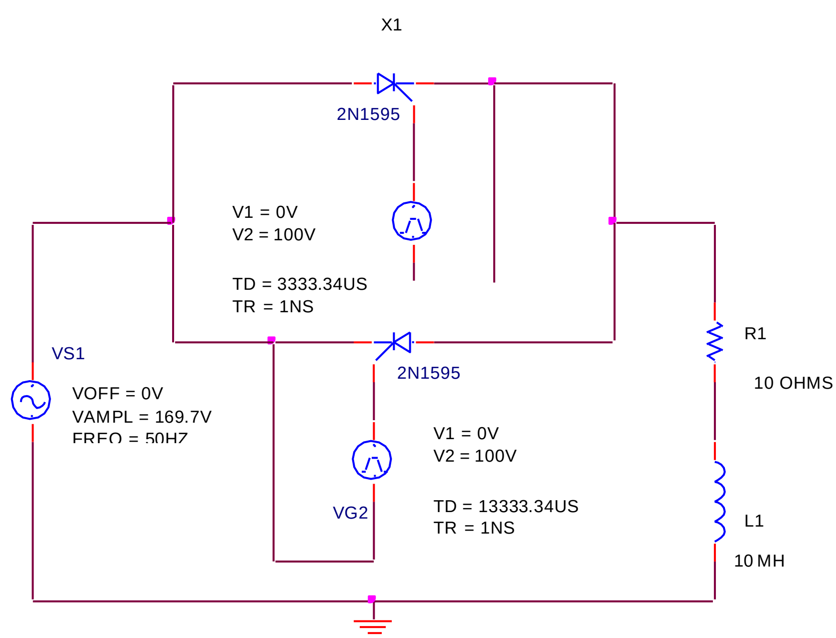
- PC with PSPICE Software

CIRCUIT DIAGRAM:

Single Phase AC VOLTAGE CONTROLLER with RL load



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

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

$$V0 = \frac{V_M}{\Pi}[COS(\_\_\_\_\_) + 1]$$

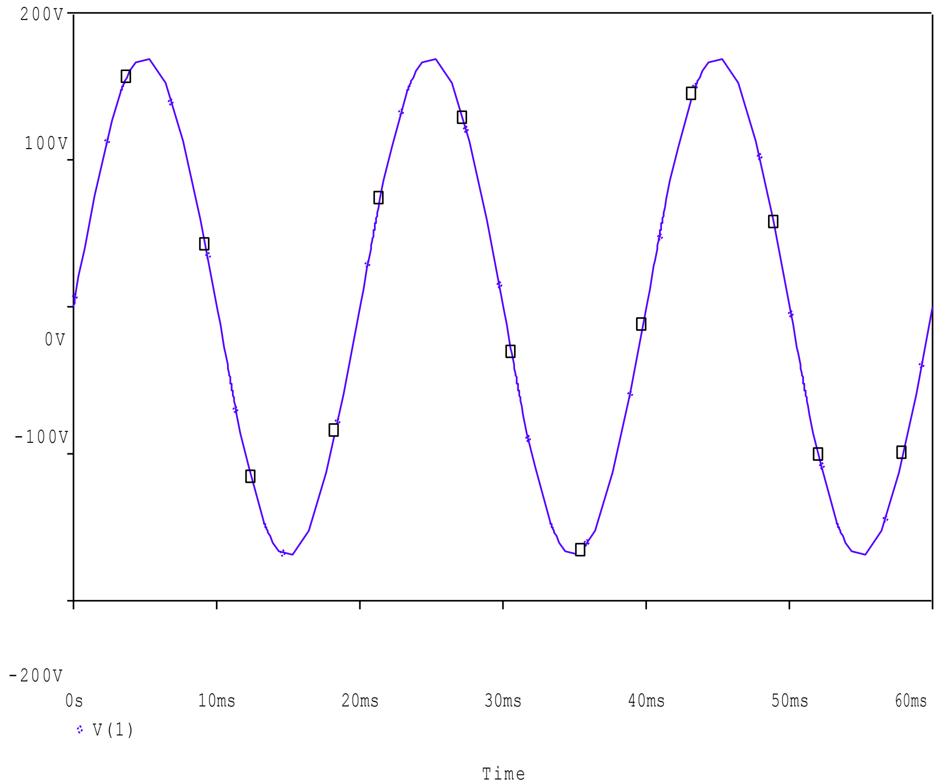
$$= \_\_\_\_\_\_\_ V$$

RESULT:

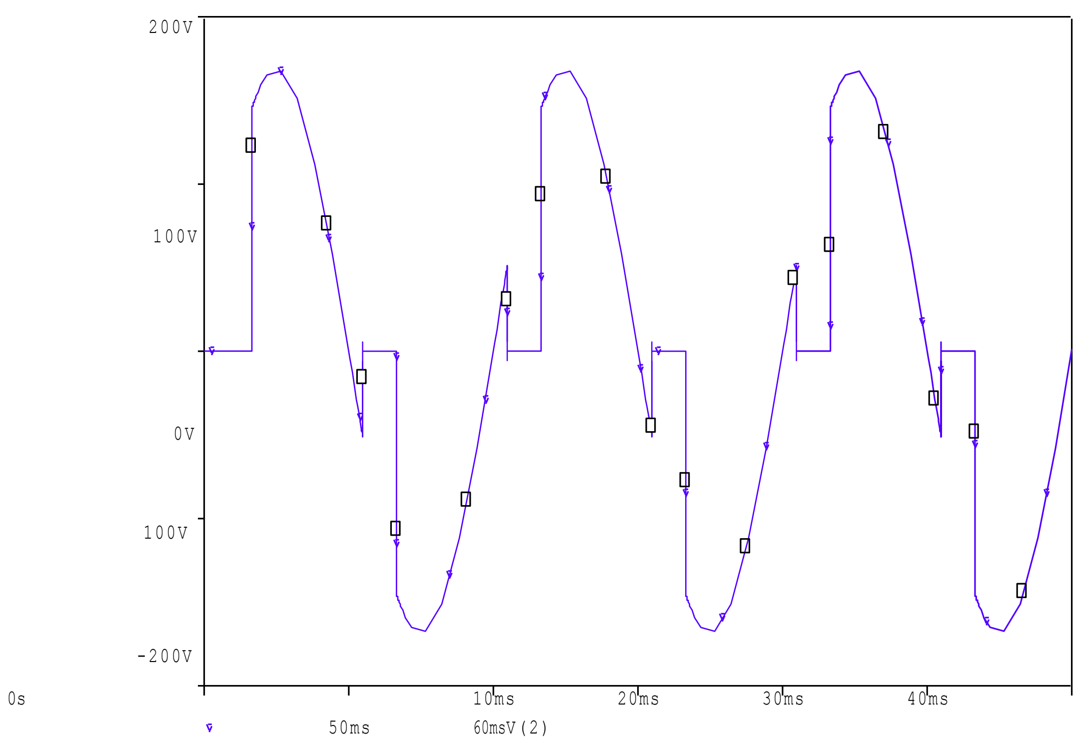
APPLICATIONS:

MODEL WAVEFORMS:

INPUT WAVEFORM



OUTPUT WAVEFORM



EXPERIMENT – 6 DATE:

THREE PHASE HALF CONTROLLED BRIDGE CONVERTER WITH R LOAD

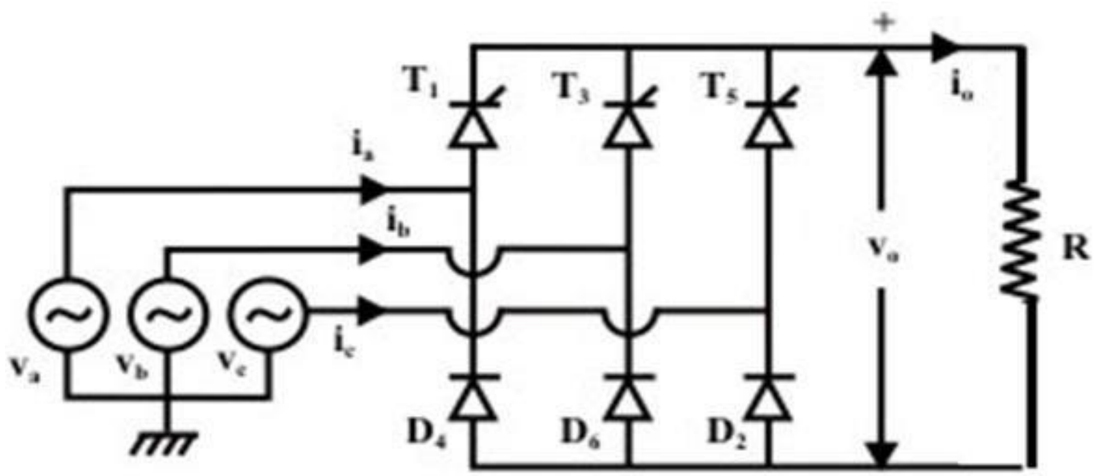
AIM:

To study the three phase half controlled bridge converter with R load.

APPARATUS:

S. No	Equipment	Range	Type	Quantity
1	Three phase half controlled bridge converter power circuit and firing circuit			
2	CRO with deferential module			
3	Patch chords and probes			
4	Three phase transformer			
5	Rheostat			
6	DC Voltmeter			
7	DC Ammeter			

CIRCUIT DIAGRAM:



Half Controlled bridge converter with R load

PROCEDURE:

1. Make all connections as per the circuit diagram.
2. Connect firstly 3 phase AC supply from three phase transformer to circuit.
3. Connect firing pulses from firing circuit to Thyristors as indication in circuit.

4. Connect resistive load 200Ω / 5A to load terminals and switch ON the MCB and IRS switch and trigger output ON switch.
5. Connect CRO probes and observe waveforms in CRO across load and device in three phase half controlled bridge converter.
6. By varying firing angle gradually up to 180^0 and observe related waveforms.
7. Measure output voltage and current by connecting DC voltmeter & Ammeter.
8. Tabulate all readings for various firing angles.
9. Calculate the output voltage and current by theoretically and compare with it practically obtained values.

TABULAR COLUMN:

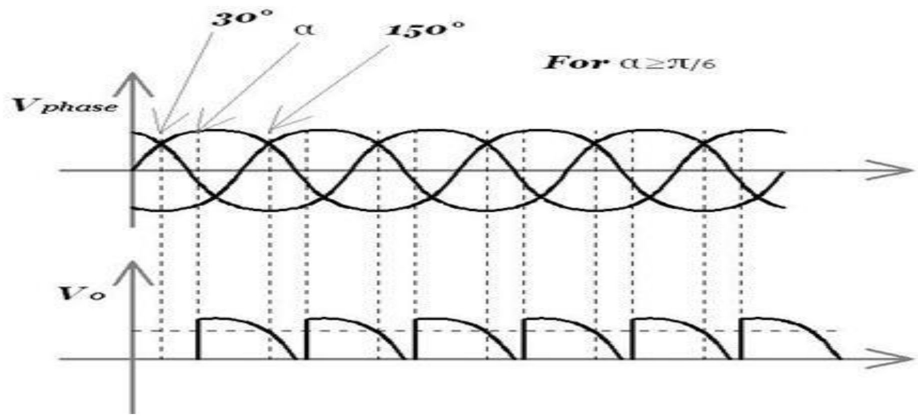
S. No	Input Voltage (V_{in})	Firing Angle in Degrees	Output voltage (V_0)		Output Current (I_0)	
			Theoretical	Practical	Theoretical	Practical
1						
2						
3						
4						
5						
6						

MODEL CALCULATIONS:

Vo = 3
Vml*(1+cosα)/2π Io = 3
Vml*(1+cosα)/2πRα=
firing angle

Vml = line to line voltage

MODEL GRAPHS:



Input and output wave forms of a three phase halfcontrolled bridge converter

RESULT:

PRE LAB VIVA QUESTIONS

1. A converter which can operate in both 3 pulse and six pulse modes is?
2. What is the interval for SCRs triggering in three phase semi converter?
3. What is the interval for SCRs triggering in three phase full converter?
4. What is the function of freewheeling diode in three phase converters?
5. What are the advantages of three phase half controlled converters?
6. Which quadrant operation is possible with three phase half controlled converter?

Expt. No: 7

SIMULATION OF BOOST CONVERTERS

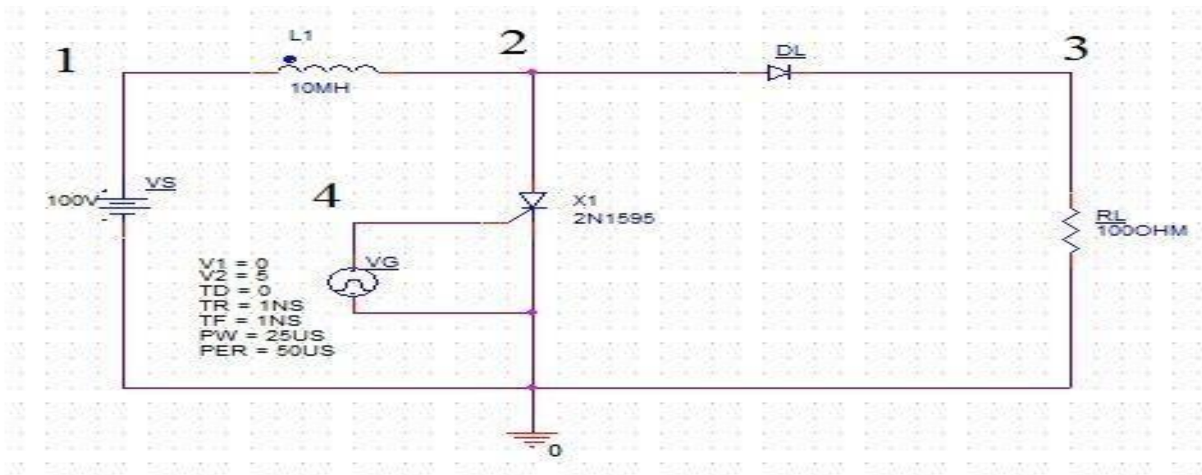
AIM:

To simulate boost converters using Pspice.

SIMULATION TOOLS REQUIRED:

- PC with PSPICE Software

CIRCUIT DIAGRAMS:



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:

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

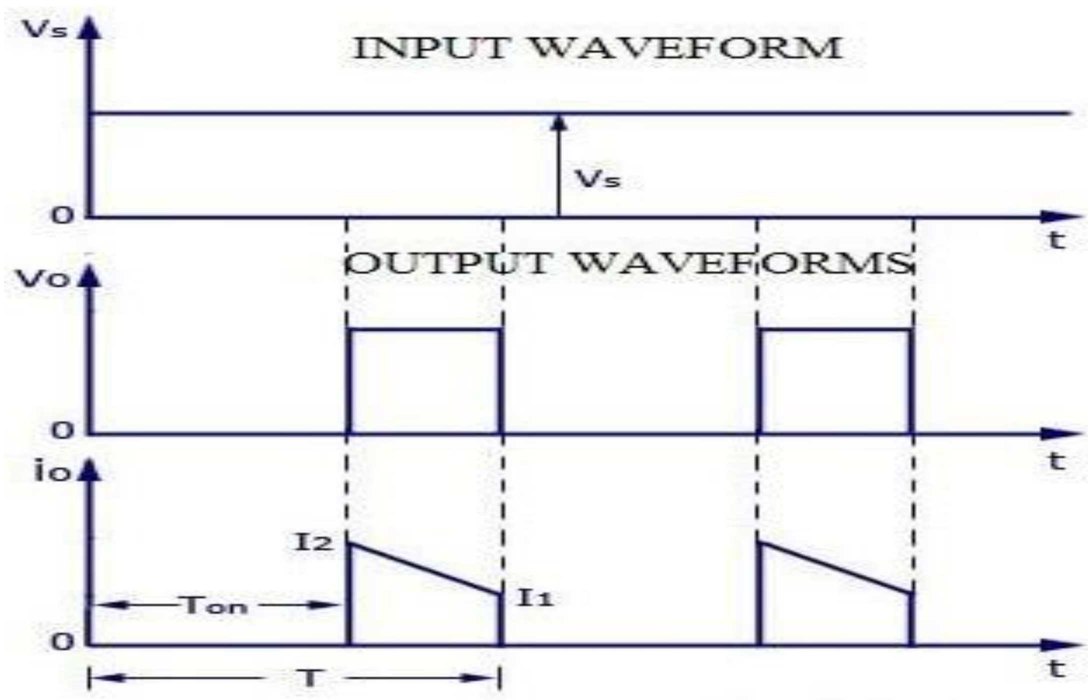
=

=

RESULT:

APPLICATIONS:

MODEL WAVEFORMS:



Experiment No:7 **Date:**

SIMULATION OF BUCK CONVERTERS

AIM:

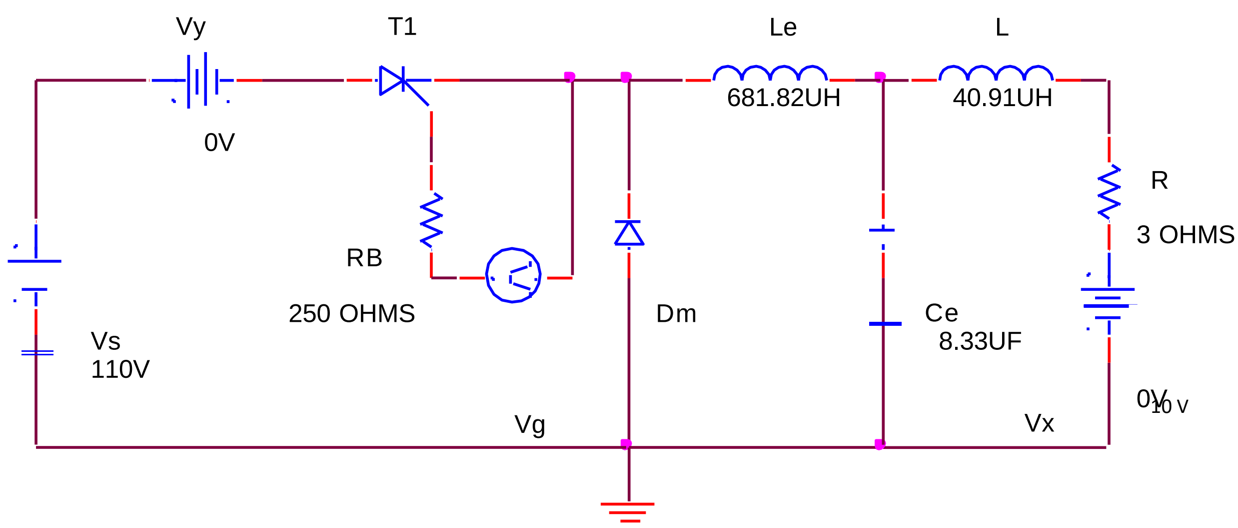
To analyze Buck chopper using Pspice.

SIMULATION TOOLS REQUIRED:

- PC with PSPICE Software

CIRCUIT DIAGRAM:

<u>Buck chopper</u>



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 :

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

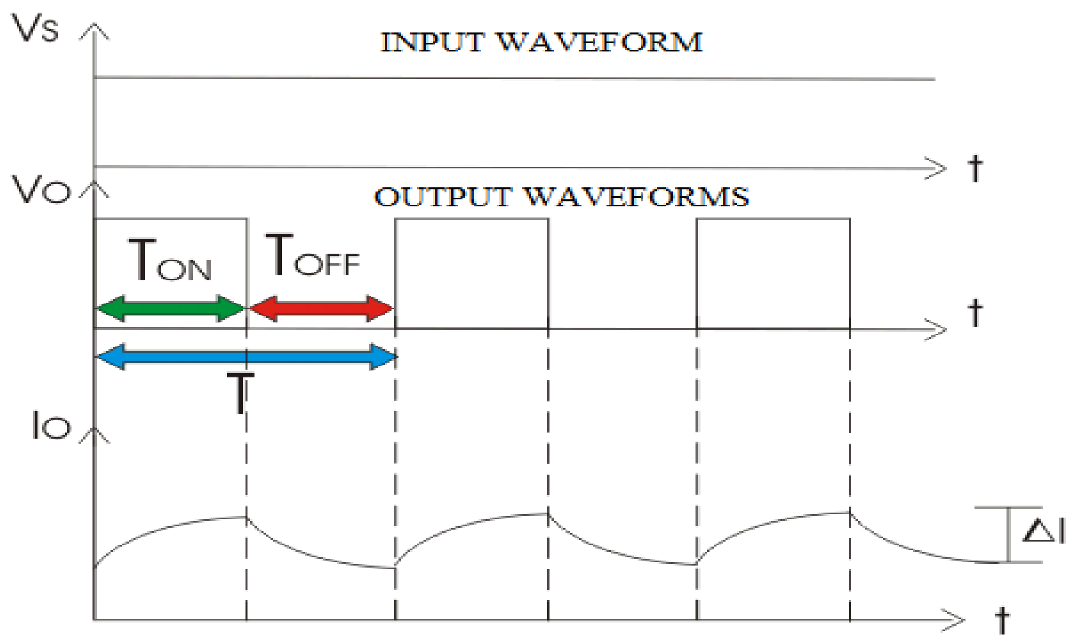
$=$

$=$

RESULT:

APPLICATIONS:

MODEL WAVEFORMS:



EXPERIMENT – 8

SINGLE PHASE A.C. VOLTAGE CONTROLLER

AIM:

To study the single phase AC voltage controller with R and RL Load

APPARATUS:

S. No	Equipment	Range	Type	Quantity
1	Single phase AC voltage controller power circuit and firing circuit			
2	CRO with deferential module			
3	Patch chords and probes			
4	Isolation Transformer			
5	Variable Rheostat			
6	Inductor			
7	AC Voltmeter			
8	AC Ammeter			

CIRCUIT DIAGRAM:

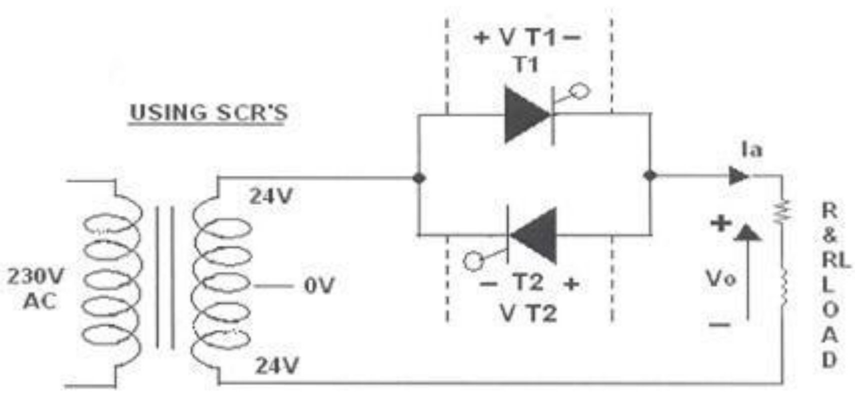


Fig - 8.1 Single Phase AC Voltage Controller with Thyristors

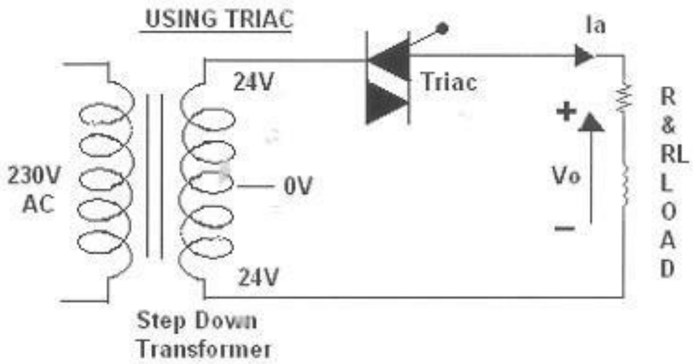


Fig - 8.2 Single Phase AC Voltage Controller with Traic

PROCEDURE:

AC VOLTAGE CONTROLLER WITH TWO THYRISTORS:

1. Make all connections as per the circuit diagram.
2. Connect firstly 30V AC supply from Isolation Transformer to circuit.
3. Connect firing pulses from firing circuit to Thyristors as indication in circuit.
4. Connect resistive load 200Ω / 5A to load terminals and switch ON the MCB and IRS switch and trigger output ON switch.
5. Observe waveforms in CRO, across load by varying firing angle gradually up to 180^0.
6. Measure output voltage and current by connecting AC voltmeter & Ammeter.
7. Tabulate all readings for various firing angles.
8. For RL Load connect a large inductance load in series with Resistance and observe all waveforms and readings as same as above.
9. Observe the various waveforms at different points in circuit by varying the Resistive Load and Inductive Load.
10. Calculate the output voltage and current by theoretically and compare with it practically obtainedvalues.

A.C. VOLTAGE CONTROLLER WITH TRIAC:

1. Make all connections as per the circuit diagram.
2. Connect firstly 30V AC supply from Isolation Transformer to circuit.
3. Connect firing pulse from firing circuit to TRIAC as indication in circuit.
4. Connect resistive load 200Ω / 5A to load terminals and switch ON the MCB and IRS switch and trigger output ON switch.

6. Observe waveforms in CRO, across load by varying firing angle gradually up to 180^0.
7. Measure output voltage and current by connecting AC voltmeter & Ammeter.
8. Tabulate all readings for various firing angles.
9. For RL Load connect a large inductance load in series with Resistance and observe all waveforms and readings as same as above.
10. Observe the various waveforms at different points in circuit by varying the Resistive Load and Inductive Load.

Calculate the output voltage and current by theoretically and compare with it practically obtained values.

TABULAR COLUMN:

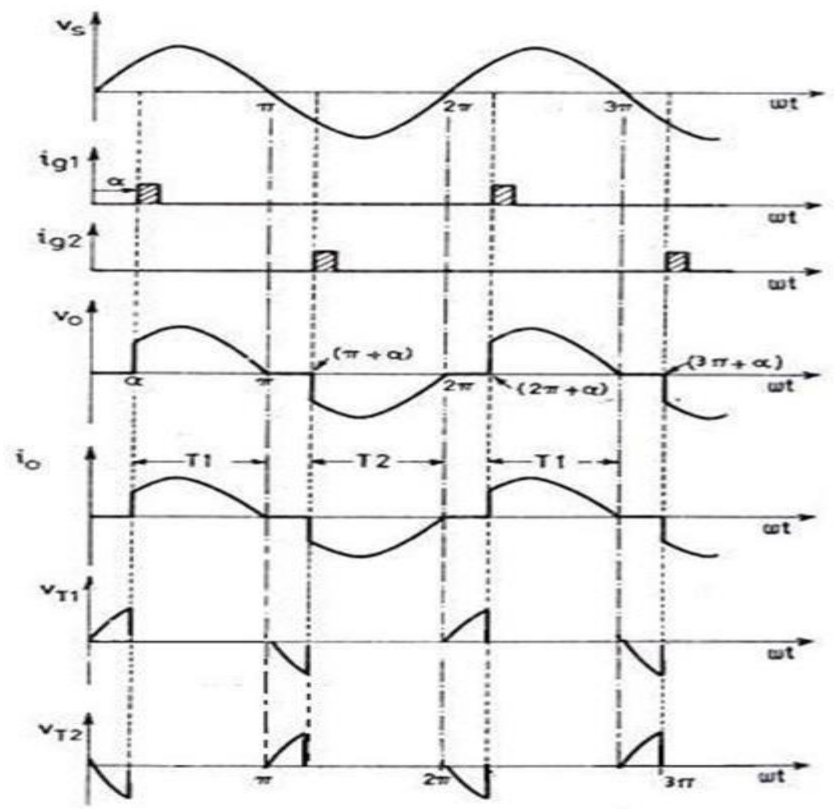
S. No.	Input Voltage (V_{in})	Firing angle in Degrees	Output voltage (V_{0r})		Output Current (I_{0r})	
			Theoretical	Practical	Theoretical	Practical
1						
2						
3						
4						
5						
6						

MODULE CALCULATIONS:

$I_{0r} = V_{0r} / R$

α = Firing Angle

MODEL GRAPH:



Single Phase AC Voltage controller with R-Load

VIVA QUESTIONS:

What type of commutation is used in this circuit?

What are the effects of load inductance on the performance of AC voltage controllers?

What is extinction angle?

What are the disadvantages of unidirectional controllers?

What are the advantages of ON-OFF control?

Expt. No:9

SINGLE PHASE INVERTER WITH PWM CONTROL

AIM:

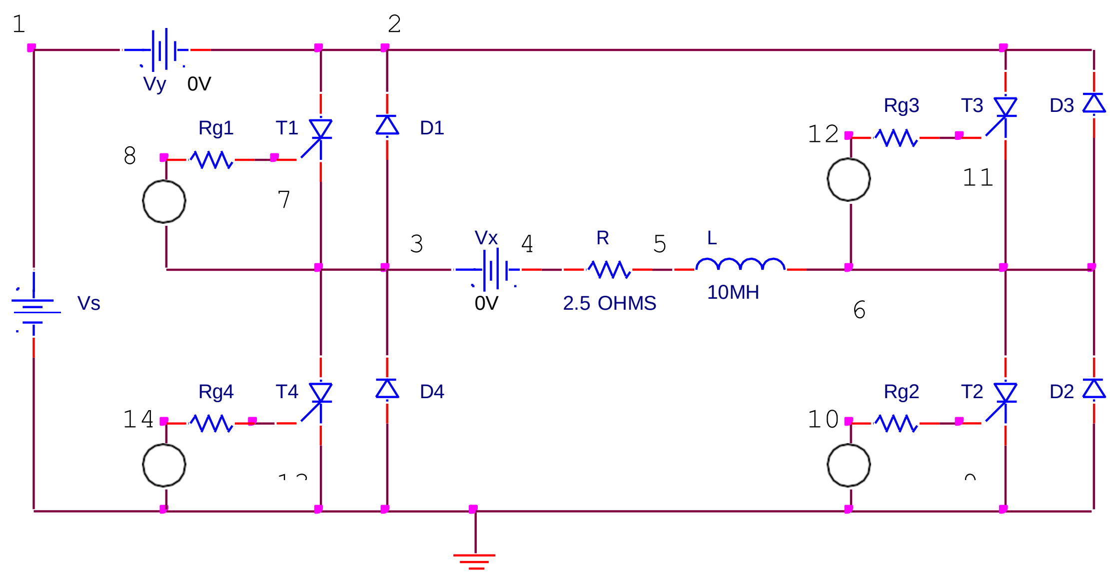
PSpice analysis of single phase inverter with PWM control.

SIMULATION TOOLS REQUIRED:

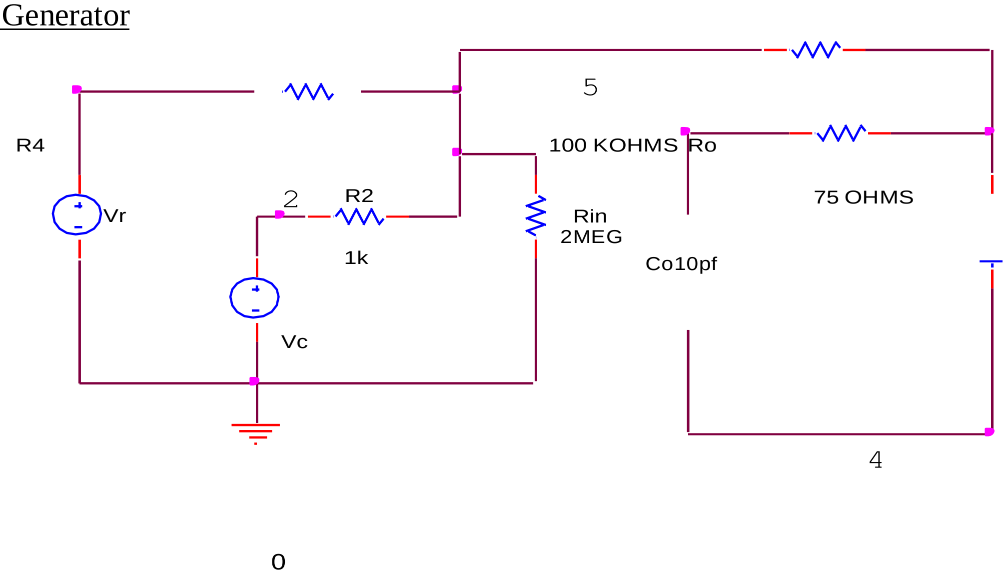
- PC with PSPICE Software

CIRCUIT DIAGRAM:

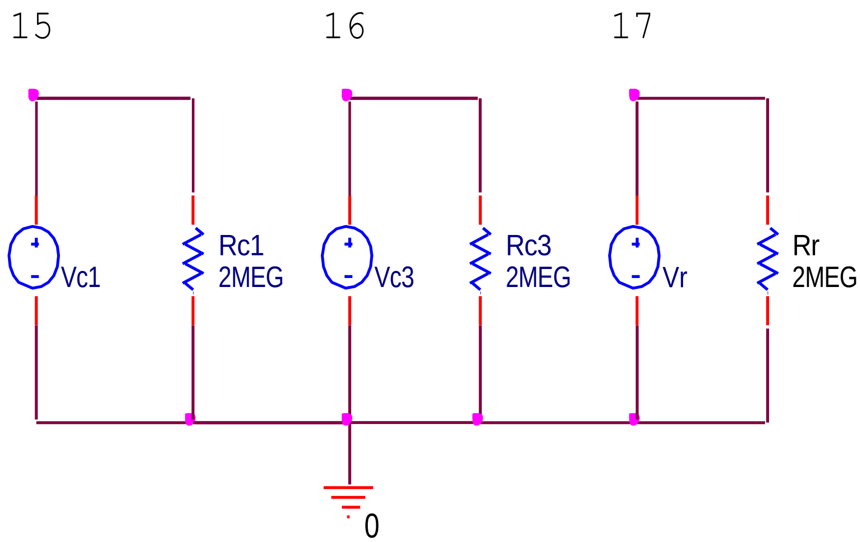
Single phase inverter with PWM control



PWM Generator



Carrier and Reference signals

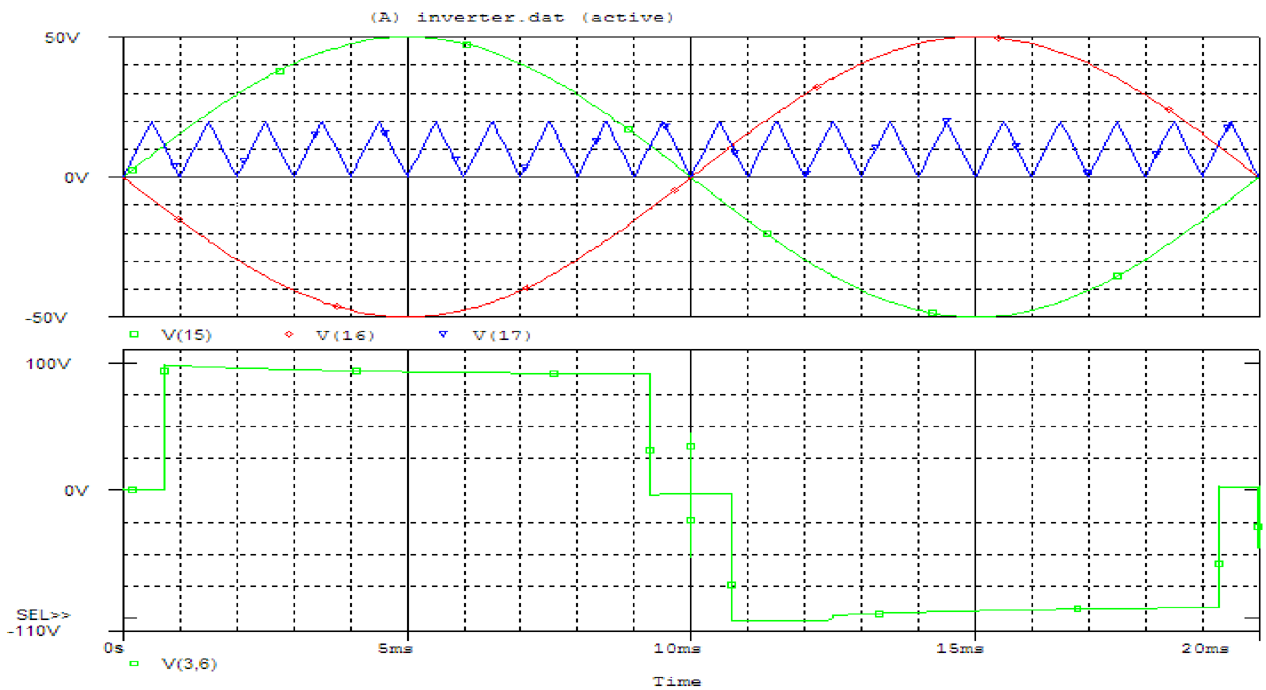


PROGRAM:

PROCEDURE:

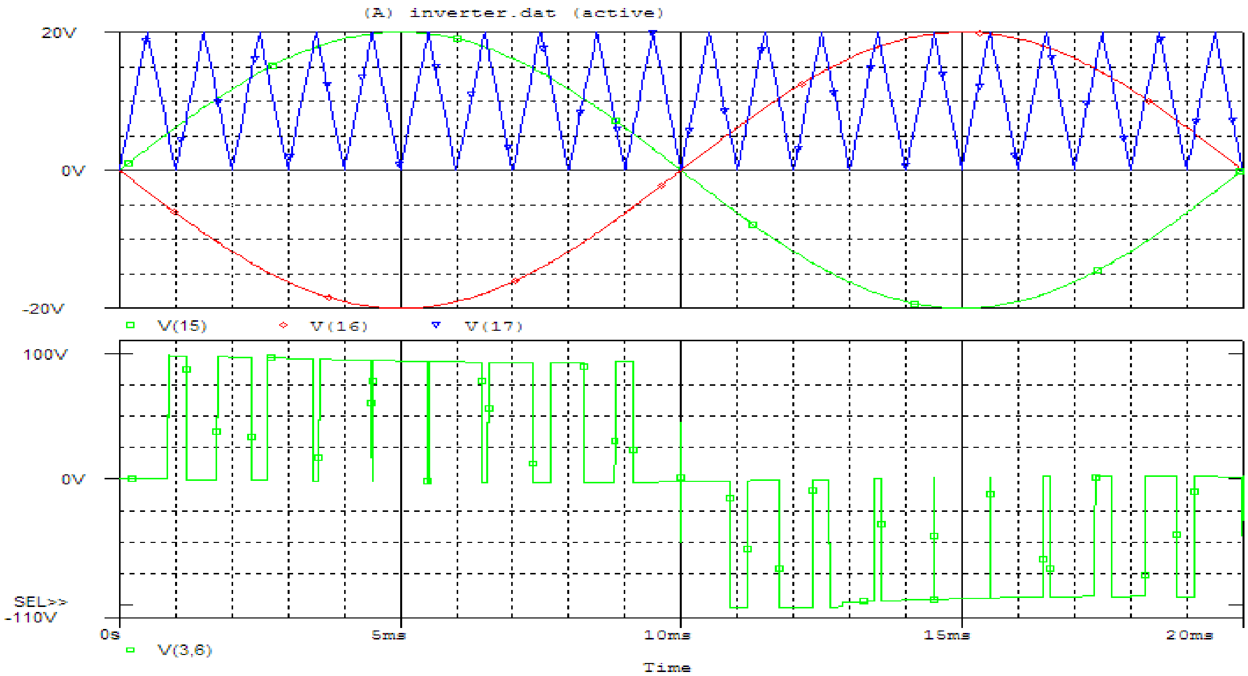
1. Write the program in a new text file in PSpice AD.
2. Save the file using the notation filename.cir.
3. Activate the file by opening it.
4. Run the simulation process using blue button.
5. By clicking Add Trace icon, get the required waveform.

MODEL WAVEFORM:



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

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


(A) inverter.dat (active)
20V
0V
-20V
V(15)
V(16)
V(17)
100V
0V
SEL>>
-110V
0s
5ms
10ms
15ms
20ms
V(3,6)
Time

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:

EXPERIMENT – 10

SIMULATION OF THREE PHASE FULL CONVERTER AND PWM INVERTER

AIM:

Simulation of three phase full converter and PWM inverter with R and RL loads by using MATLAB.

APPARATUS:

S. No	Equipment	Quantity
1	Desktop With MATLAB	1

THEORY:

Three phase full converter:

Three phase bridge controlled rectifier consist of upper group (T_1, T_3, T_5) and lower group (T_2,T_4,T_5) of thyristors. Thyristor T1 is forward biased ad can be triggered for conduction only when V_A is greater than both V_B and V_C. From figure this condition occurs at w_t=30^0 . Hence T_1 can be triggered only at w_t=30^0.If firing angle is α, then T_1 starts conduction at w_t=30^0 + α and conducts for 120^0 where it get commutated by turning on of next thyristor ie,T_3.Similarly triggering instant for T_3 and T_5 are determined when considering V_B and V_C respectively. For lower group T_4,T_6 and T_2, negative voltages,-V_A,-V_B and -V_C respectively are considered.

Average Value of output voltage is given by

$$V_{avg} = \frac{3\sqrt{3}}{\Pi} V_m \text{Cos } \alpha$$

Three Phase PWM Inverter:

Three phase inverter consists of on and off controlled switches such as MOSFET or IGBT. Sine PWM pulses are used to gate the switches. Upper switches are gated with signals obtained by comparing three reference sine waves each are phase shifted with 1200 with a high frequency triangular carrier wave. Thus, switches are ON when amplitude of corresponding reference sine wave is greater than amplitude of triangular carrier wave. Lower switches are gated with a gate signal which is complement of upper switches of same leg.

Rms Value of phase to neutral output voltage is given by

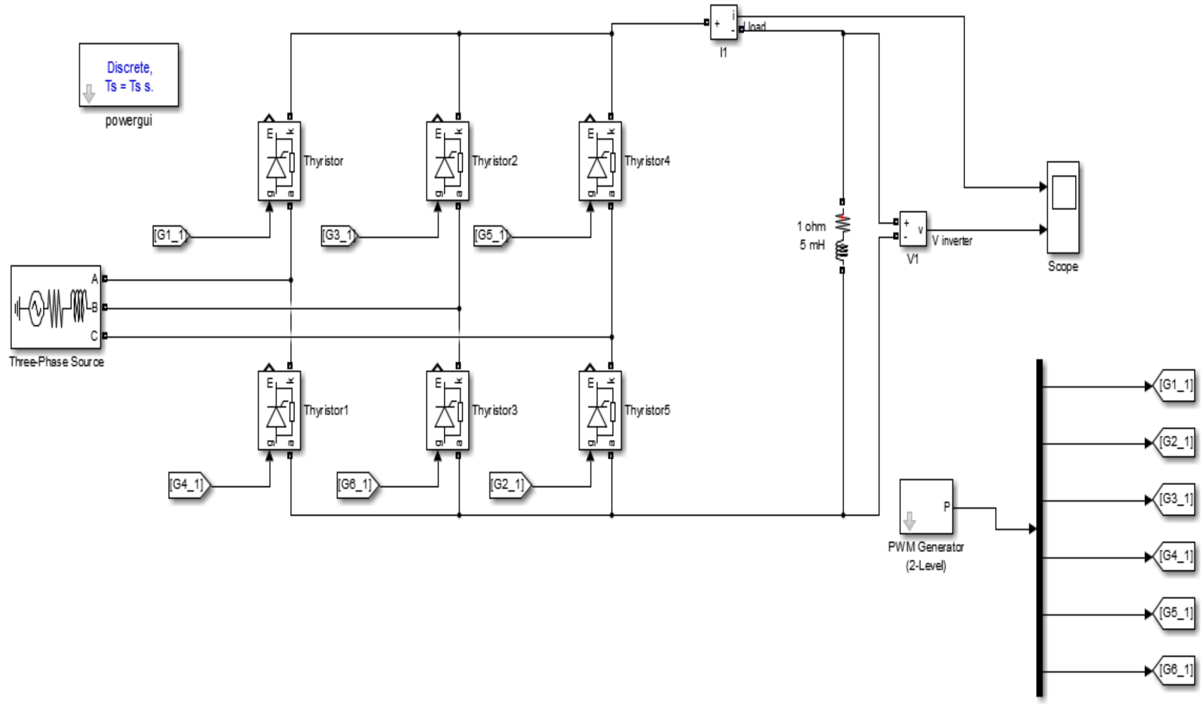
$$V_{Ph\,rms} = \frac{mV_{dc}}{2\sqrt{2}}$$

Rms Value of line to line output voltage is given by

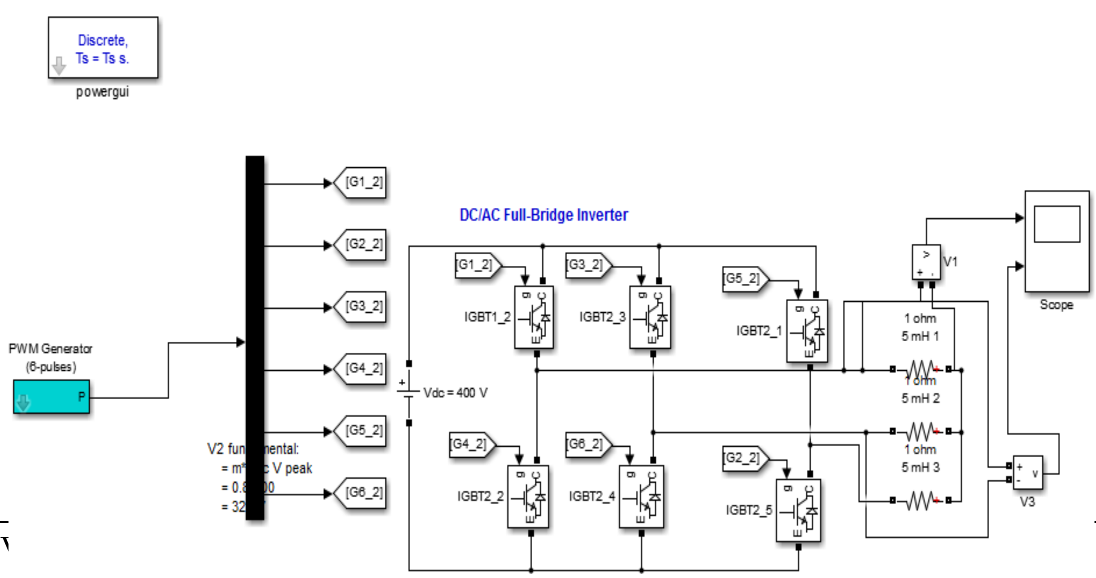
$$V_{Ph\,rms} = \frac{\sqrt{3}mV_{dc}}{2\sqrt{2}}$$

CIRCUIT DIAGRAM:

Three phase full converter:



Circuit diagram for three phase full converterThree Phase PWM Inverter:



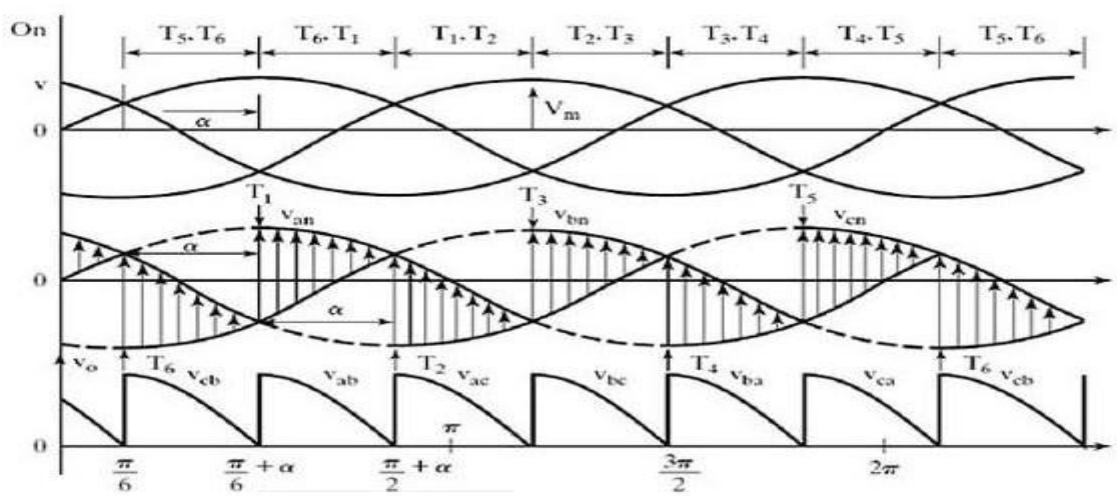
circuit diagram for Three phase PWM Inverter

PROCEDURE:

1. Make the connections as shown in the figures 13.1 and 13.2 by using MATLAB Simulink.
2. Set the parameters in PWM generator for firing the switches, set the values for load and inputvoltage.
3. Check the scope wave forms in each circuit.

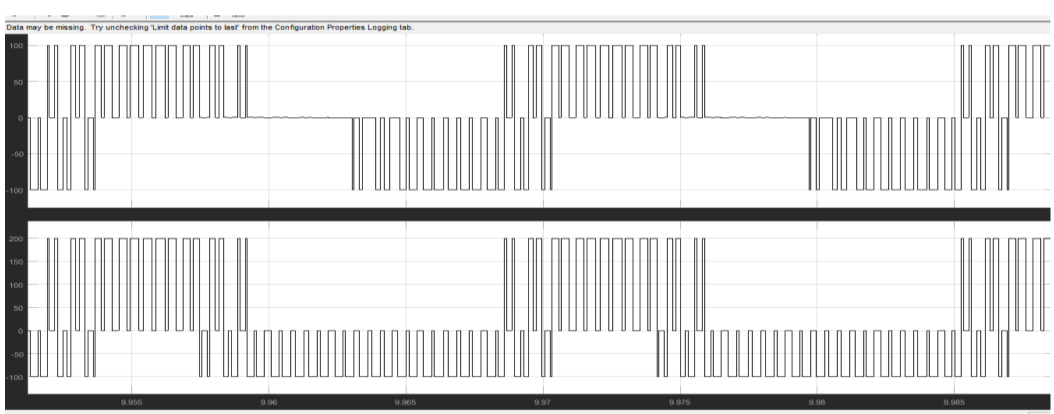
EXPECTED GRAPH

Three phase full converter:



output voltage and current waveforms of Three Phase Full ConverterThree

Phase PWM Inverter:



output voltage and current waveforms of Three Phase PWM Inverter

RESULT:

PRE-LAB VIVA QUESTIONS:

1. What is PWM?
2. What is Duty cycle?
3. What is three phase converter?
4. What is an inverter?

POST LAB VIVA QUESTIONS:

1. What are the advantages of PWM inverters?
2. What is the difference between three phase and single phase inverters?
3. What is the time delay for each thyristor conduction in three phase full converter?

Exp. No.: 11 **Date:**

FREQUENCY RESPONSE OF SERIES RLC CIRCUIT

AIM: To find out the transient response and parametric analysis by simulation of RLC circuits Using Sinusoidal Responses.

SOFTWARE REQUIRED: PSPICE – Personal Computer Simulated Program with Integrated Circuit Emphasis.

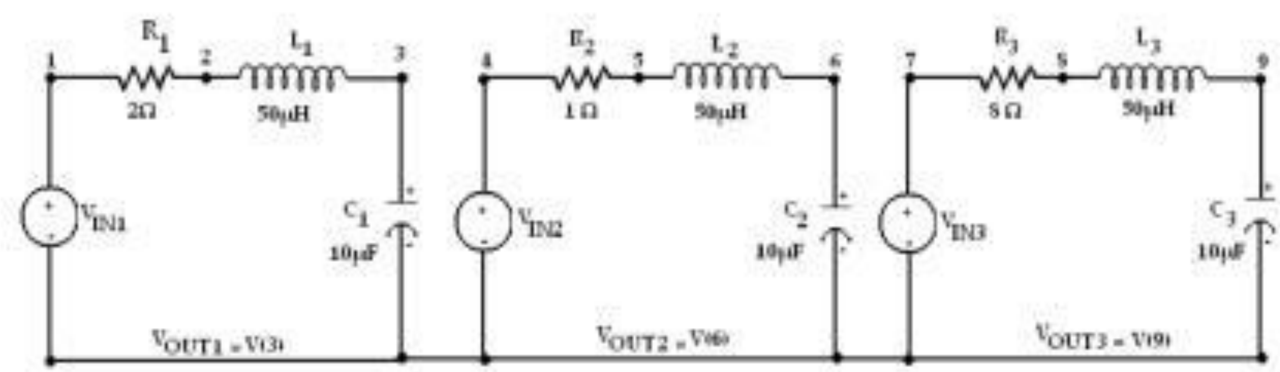
SYNTAX USED:

S.NO	TYPE OF SOURCE	REPRESENTATION OF SOURCE	DECLARATION FORMAT
1	SINUSOIDAL RESPONSE	SIN	(Frequency)(Delay Time) (Damping Factor) (Phase Delay)
2	TRANSIENT ANALYSIS	.TRAN	.TRAN TStep Tstop [TStart TMax]
3	PROBE STATEMENT	.PROBE	It is a wave form analyzer
4	PLOT STATEMENT	.PLOT	.PLOT (Output Variables) {(Lower limit Value), (Upper Limit Value)}

DATA REQUIRED FOR DRAWING THE CIRCUIT DIAGRAM:

For example, Three RLC circuits with R=2Ω, 1 Ω, and 8 Ω respectively, with L having the values of 50μH each, with C having the values of 10μF each. The inputs are identical Sinusoidal Response. The Sinusoidal response having the offset voltage as 0V, RMS voltage as 120V and the frequency as 50Hz. Use PSPICE to plot and calculate the transient response from 0 to 60mseconds with an increment of 1μsecond. Plot the voltages across the capacitors.

CIRCUIT DIAGRAM:



PROCEDURE:

1. Open PSpice A/D windows
2. Create a new circuit file .

3. Enter the program representing the nodal interconnections of various components.
4. Run the program
5. Observe the response through all the elements in the output file.
6. Observe the voltage, current graph of any in probe window.

PROGRAM:

```
VIN1 1 0 ac 1V
VIN2 4 0 ac 1V
VIN3 7 0 ac 1V
R1 1 2 2
R2 4 5 1
R3 7 8 8
L1 2 3 50UH
L2 5 6 50UH
L3 8 9 50UH
C1 3 0 10UF
C2 6 0 10UF
C3 9 0 10UF
.ac dec 100 100Hz 100KHz
.PLOT ac vm(3) vp(3)
.PROBE
.END
```

RESULT:

VIVA QUESITIONS:

1. Write equations for voltage across R, L & C
2. Define Transient
3. The transient response occurs in L & C
4. Define steady state
5. The time constant of series RL network _______
6. Transient circuit in RLC circuit is oscillation when R< $2\sqrt{L}/C$

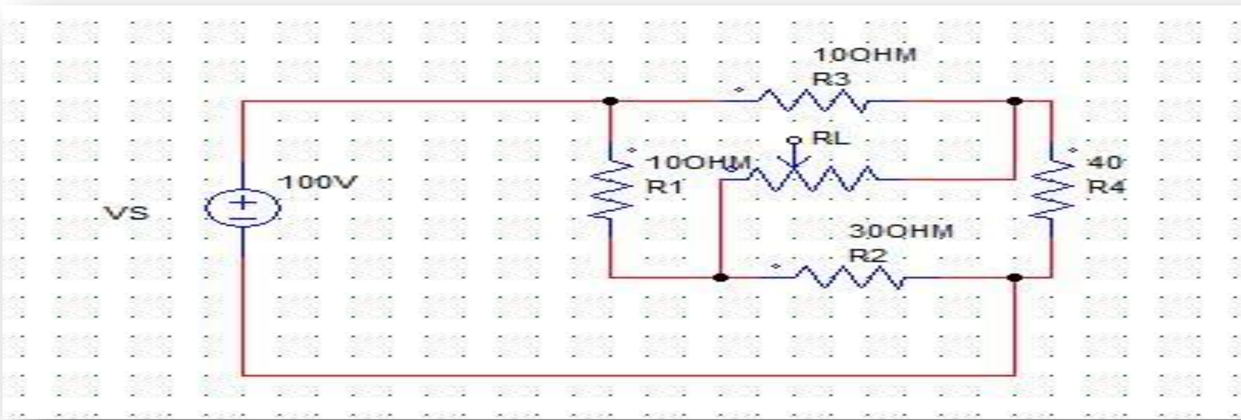
Exp. No.: 12 **Date:**

VERIFICATION OF THE MAXIMUM POWER DISSIPATION

AIM: To find out the unknown resistance and maximum power for dc circuits

SOFTWARE REQUIRED: **PSPICE – P**ersonal Computer **S**imulated **P**rogram with **I**ntegrated **C**ircuit **E**mphasis.

CIRCUIT DIAGRAM:



PROCEDURE:

1. Open PSPICE A/D windows.
2. Create a new circuit file.
3. Enter the program representing the nodal interconnections of various components.
4. Run the program.
5. Observe the response through all the elements in the output file.
6. Observe the voltage, current graph of any in probe window.

PROGRAM:

```
VS 1 0 DC 100
R1 1 2 10
R2 2 0 30
R3 1 3 20
R4 3 0 40
RL 2 3 RLOAD 1
.MODEL RLOAD RES(R=25)
.DC  RES  RLOAD(R)  0.001  40  0.01
.TF V(2,3) VS
.PROBE
.END
```

RESULT:

VIVA QUESITIONS:

1. Define Statement of maximum power transfer theorem .
2. Write the expression for maximum power ?
3. What is the condition for maximum power in DC circuit?
4. What is the condition for maximum power in AC circuit?
5. Give any two applications of maximum power transfer theorem.