MALLA REDDY COLLEGE OF ENGINEERING & TECHNOLOGY
(Autonomous Institution – UGC, Govt. of India)
Recognized under 2(f) and 12 (B) of UGC ACT 1956
(Affiliated to JNTUH, Hyderabad, Approved by AICTE-Accredited by NBA & NACC-'A' Grade – ISO 9001:2015 Certified)
Maisammaguda, Dhulapally (Post Via. Hakimpet), Secunderabad -500100, Telangana State, India

ELECTRICAL CIRCUITS AND SIMULATION LABORATORY MANUAL

Student Name:……………………………………………………
Roll No:…………………………………………………………
Branch:…………………………..Section……………………
Year …………………………Semester………………………..

FACULTY INCHARGE

PROGRAM OUTCOMES (POs)

Engineering Graduates will be able to:

1. **Engineering knowledge**: Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
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.
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.
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.
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.
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.
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.
8. **Ethics**: Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
9. **Individual and team work**: Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
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.
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 multi disciplinary environments.
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.

ELECTRICAL CIRCUITS AND SIMULATION LABORATORY

COURSE OBJECTIVES:

1. This course introduces the basic concepts of simple DC & AC Circuits.
2. The basic Two Port Network Parameters.
3. Will able to articulate in working of various components of a circuit.
4. To design electrical systems.
5. To analyze a given network by applying various network theorems.
6. To understand the locus diagram.
7. The Emphasis of this course is laid on the PSPICE Simulation of DC & AC Circuits.
8. Ability to measure three phase voltages, current, active and reactive powers.

COURSE OUTCOMES:

1. The student will analyze the characteristics of Electrical circuits & PSpice Simulation.
2. To Perform Laboratory Experiments practically.
3. To carry out laboratory experiments on simulation & Networks.
4. To understand the fundamentals of electrical circuits & PSpice simulation.

MALLA REDDY COLLEGE OF ENGINEERING AND TECHNOLOGY

II Year B.Tech EEE-II Sem	**L**	**T/P/D**	**C**
	-	**-/-3/-**	**1.5**

(R18A0284) ELECTRICAL CIRCUITS AND SIMULATION LAB

COURSE OBJECTIVES:

- To design electrical systems.
- To analyze a given network by applying various Network Theorems.
- To measure three phase Active and Reactive power.

PART A

1) Millmann's Theorem
2) Series and Parallel Resonance
3) Z and Y Parameters
4) Transmission(ABCD) and Hybrid parameters
5) Measurement of Active Power for Star and Delta connected balanced loads
6) Measurement of Reactive Power for Star and Delta connected balanced loads

PART-B: PSPICE SIMULATION

1) Simulation of DC Circuits
2) DC Transient response
3) Mesh Analysis
4) Nodal Analysis

Note: Any 6 Experiments from PART-A, PART-B Is Mandatory

COURSE OUTCOMES:

After successfully studying this course, students will:

- Design electrical systems.
- Analyze a given network by applying various Network Theorems.
- Measure three phase Active and Reactive power.

INSTRUCTIONS TO STUDENTS

- Before entering the lab the student should carry the following things.
  - Identity card issued by the college.
  - Class notes
  - Lab observation book
  - Lab Manual
  - Lab Record
- Student must sign in and sign out in the register provided when attending the lab session without fail.
- Come to the laboratory in time. Students, who are late more than 15 min., will not be allowed to attend the lab.
- Students need to maintain 100% attendance in lab if not a strict action will be taken.
- All students must follow a Dress Code while in the laboratory
- Foods, drinks are NOT allowed.
- All bags must be left at the indicated place.
- The objective of the laboratory is learning. The experiments are designed to illustrate phenomena in different areas of Physics and to expose you to measuring instruments, conduct the experiments with interest and an attitude of learning
- You need to come well prepared for the experiment.
- Work quietly and carefully
- Be honest in recording and representing your data.
- If a particular reading appears wrong repeat the measurement carefully, to get a better fit for a graph
- All presentations of data, tables and graphs calculations should be neatly and carefully done
- Graphs should be neatly drawn with pencil. Always label graphs and the axes and display units.
- If you finish early, spend the remaining time to complete the calculations and drawing graphs. Come equipped with calculator, scales, pencils etc.
- Do not fiddle with apparatus. Handle instruments with care. Report any breakage to the Instructor. Return all the equipment you have signed out for the purpose of your experiment.

SPECIFIC SAFETY RULES FOR ELECTRICAL CIRCUITS AND SIMULATION LABORATORY

- You must not damage or tamper with the equipment or leads.
- You should inspect laboratory equipment for visible damage before using it. If there is a problem with a piece of equipment, report it to the technician or lecturer. DONOT return equipment to a storage area
- You should not work on circuits where the supply voltage exceeds 40 volts without very specific approval from your lab supervisor. If you need to work on such circuits, you should contact your supervisor for approval and instruction on how to do this safely before commencing the work.
- Always use an appropriate stand for holding your soldering iron.
- Turn off your soldering iron if it is unlikely to be used for more than 10 minutes.
- Never leave a hot soldering iron unattended.
- Never touch a soldering iron element or bit unless the iron has been disconnected from the mains and has had adequate time to cool down.
- Never strip insulation from a wire with your teeth or a knife, always use an appropriate wire stripping tool.
- Shield wire with your hands when cutting it with a pliers to prevent bits of wire flying about the bench.

CONTENTS

CYCLE – 1

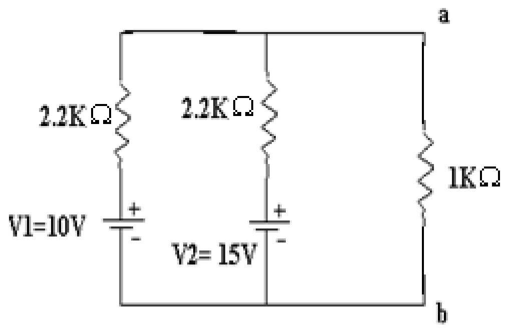
1. VERIFICATION OF MILLMAN'S THEOREM

AIM: To verify Millman‟s theorem for the given circuit.

APPARATUS REQUIRED:

S.NO	NAME	RANGE	QUANTITY
1	Ammeter	(0-200)mA	2
2	Voltmeter	(0-30)V	1
3	Resistors	2.2K Ω	1
		1.1KΩ	1
		100Ω	2
		220Ω	1
		1000Ω	1
4	Bread board	------	1
5	RPS	(0-30)V	1
6	Connecting wires	-----	Required

CIRCUIT DIAGRAM:



PRACTICAL CIRCUITS:

CIRCUIT-1

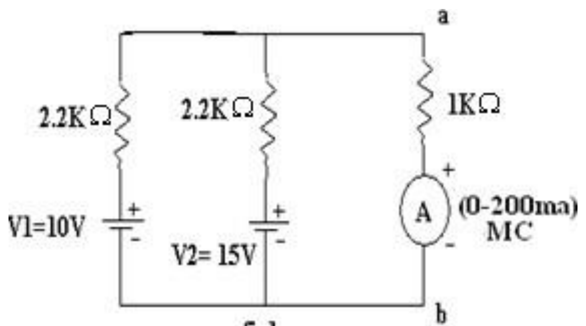


fig1

CIRCUIT-2:

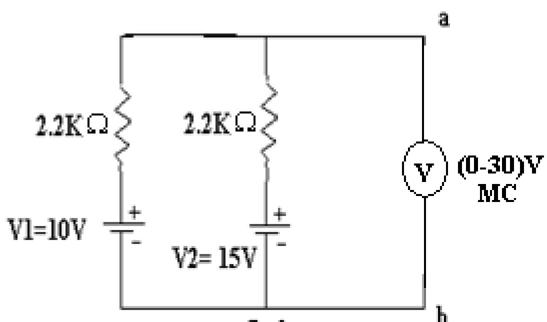


fig 2

CIRCUIT-3:

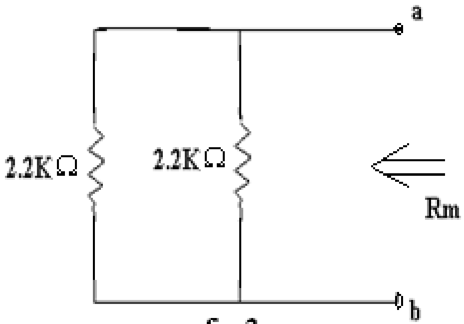


fig 3

CIRCUIT-4:

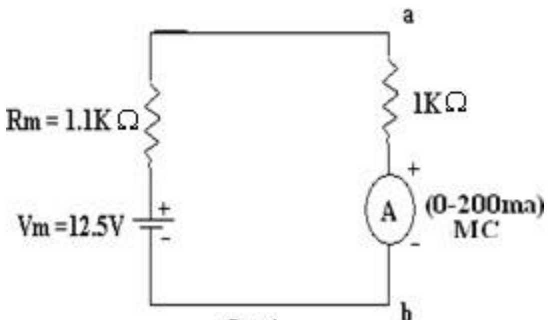
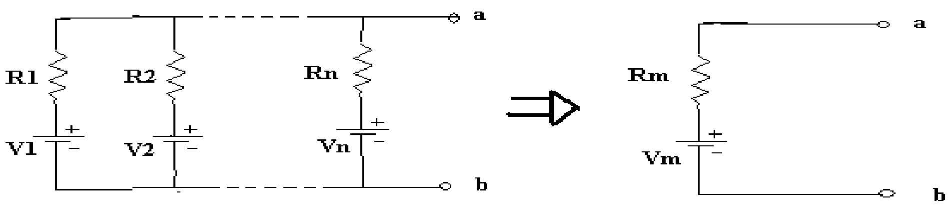


fig 4

THEORY:

STATEMENT: "Millman"s theorem states that in any network, if the voltage sources V1, V2,……..Vn in series with internal resistances R1,R2,............Rn, respectively are in parallel then these sources may be replaced by a single voltage source "Vm" in series with "Rm" as shown.



According to Millman's theorem,

$$V_m = \frac{V_1 G_1 + V_2 G_2 + \text{-------} + V_n G_n}{G_1 + G_2 + \text{----------} + G_n}$$

$$R_m = \frac{1}{G_1 + G_2 + \text{----------} + G_n}$$

PROCEDURE:

1. Make the connections as per the circuit diagram shown in fig 1.
2. Switch on the supply, apply the source voltages V_1=10V & V_2=15V.
3. Note down the readings of ammeter and tabulate in table1.
4. Make the connections as per the circuit diagram shown in fig 2.
5. Switch on the supply, apply the source voltages V_1=10V & V_2=15V.
6. Note down the readings of ammeter and tabulate in table2
7. Make the connections as per the circuit diagram shown in fig 3 and determine R_M using Multimeter
8. Make the connections of equivalent Millman's circuit as shown in fig 4.
9. Switch on the supply, apply the millman voltage V_m (calculated) and note down the readings of ammeter and tabulate in table2.
10. Repeat the experiment at different source voltages and compare the readings.

TABULAR COLUMNS:

S.No	V_1 (volts)	V_2 (volts)	I_L (mA)
1			
2			
3			

Table 1

S.No	V_1 (volts)	V_2 (volts)	V_m (volts)	I_L (mA)
1				
2				
3				

Table 2

PRECAUTIONS:
1. Initially keep the RPS output voltage knob in zero volt position.
2. Avoid loose connections.
3. Avoid short circuit of RPS output terminals.

RESULT:

VIVA QUESTIONS:

1. State Millman‟s theorem?
2. Write the advantage of Millman‟s theorem?
3. Draw the equivalent circuit of Millmann‟s theorem?
4. What are the limitations of Millmann‟s theorem?

2. SERIES AND PARALLEL RESONANCE

AIM: To find the resonant frequency, quality factor and band width of a given series and parallel resonant circuits.

APPARATUS REQUIRED:

S.No	Name Of The Equipment	Range	Type	Quantity
1	Bread board	-	-	1 NO
2	Resistor	1k Ω	-	1 NO
3	Inductor	50 mH	-	1 NO
4	Capacitors	0.1 uF	-	1 NO
5	CRO	20MHz.Dual CH	-	1 NO
6	Function generator	100-10MHz	-	1 NO
7	Ammeter	0-20mA	Digital	1 NO
8	Connecting wires			Required number

CIRCUIT DIAGRAM:

SERIES RESONANCE:

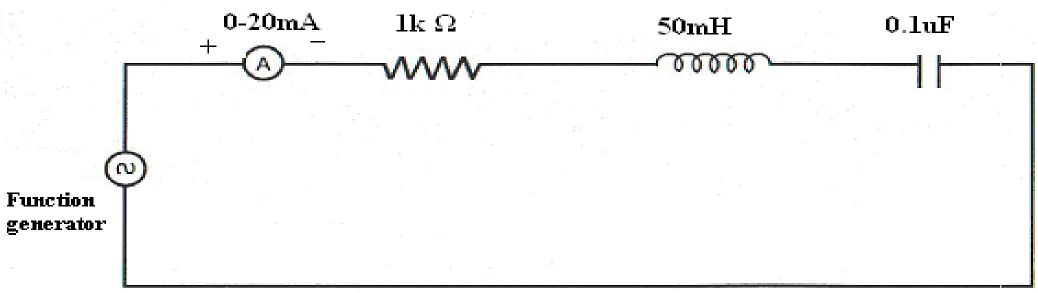


Fig.1

PARALLEL RESONANACE:

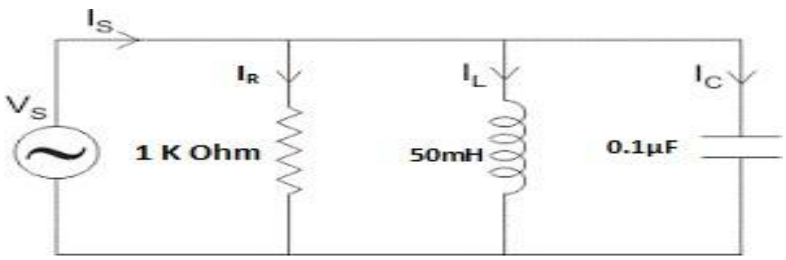


Fig.2

THEORY:

Resonance is a particular type of phenomenon inherently found normally in every kind of system, electrical, mechanical, optical, Acoustical and even atomic. There are several definitions of resonance. But, the most frequently used definition of resonance in electrical system is studied state operation of a circuit or system at that frequency for which the resultant response is in time phase with the forcing function.

SERIES RESONANCE:

A circuit is said to be under resonance, when the applied voltage „V" and current are in phase. Thus a series RLC circuit, under resonance behaves like a pure resistance network and the reactance of the circuit should be zero. Since V & I are in phase, the power factor is unity at resonance.

The frequency at which the resonance will occur is known as resonant frequency. Resonant frequency,

$$f_r = \frac{1}{2\Pi\sqrt{LC}}$$

Thus at resonance the impedance Z is minimum. Since I = V/Z. The current is maximum So that current amplification takes place. Quality factor is the ratio of reactance power inductor (or) capacitor to its resistance.

PARALLEL RESONANCE:

In the circuit (parallel RLC circuit) shown in figure.2, the condition for resonance occurs when the susceptance part is zero. The frequency at which the resonance will occur is known as resonant frequency. Resonant frequency,

$$f_r = \frac{1}{2\Pi\sqrt{LC}}$$

Thus at resonance the admittance(Y) is Minimum and voltage is Maximum. However the performance of such a circuit is of interest in the general subject of resonance. Lower cut-off frequency is above the resonant frequency at which the current is reduced to $\frac{1}{\sqrt{2}}$ times of its minimum value. Upper cut-off frequency is above. Quality factor is the ratio of resistance to reactance of inductor (or) capacitor. Selectivity is the reciprocal of the quality factors.

THEORITICAL CALCULATIONS:

For Series Resonance circuit:

1. Resonant frequency $f_r = \frac{1}{2\pi\sqrt{LC}}$
2. Lower Cut off Frequency $f_1 = f_r - (R/4\pi L)$
3. Upper Cut off Frequency $f_2 = f_r + (R/4\pi L)$
4. Band width = $f_2 - f_1$
5. Quality factor $Q = \frac{w_0 L}{R} = \frac{2\pi fr.L}{R}$

6. Current at Resonance $I_o = V_{Ro}/R$

For Parallel Resonance circuit:

1. Resonant frequency $f_r = \frac{1}{2\pi\sqrt{LC}}$
2. Lower Cut off Frequency $f_1 = \{1/2\pi\}\{(-1/2RC)+((1/2RC)^2+(1/LC))^{0.5}\}$
3. Upper Cut off Frequency $f_2 = \{1/2\pi\}\{(1/2RC)+((1/2RC)^2+(1/LC))^{0.5}\}$
4. Band width = f_2-f_1:
5. Quality factor_Q = $\frac{R}{W_0 L}$
6. Current at resonance $I_o = V_{Ro}/R$

PROCEDURE:

1. Connect the circuit as shown in fig.1 for series resonant circuit & fig.2 for parallel resonant circuit.
2. Set the voltage of the signal from function generator to 5V.
3. Vary the frequency of the signal over a wide range in steps and note down the corresponding ammeter readings.
4. Observe that the current first increases & then decreases in case of series resonant circuit & the value of frequency corresponding to maximum current is equal to resonant frequency.
5. Observe that the current first decreases & then increases in case of parallel resonant circuit & the value of frequency corresponding to minimum current is equal to resonant frequency.
6. Draw a graph between frequency and current & calculate the values of bandwidth & quality factor.

OBSERVATION TABLE:

Series Resonance:

S. No.	**Frequency** (Hz)	**Current** (mA)

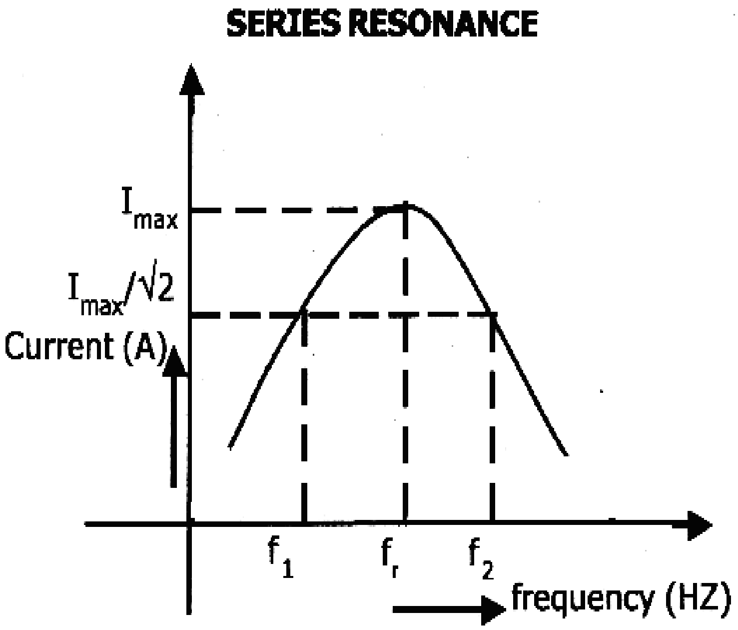
OBSERVATION TABLE:
Parallel Resonance:

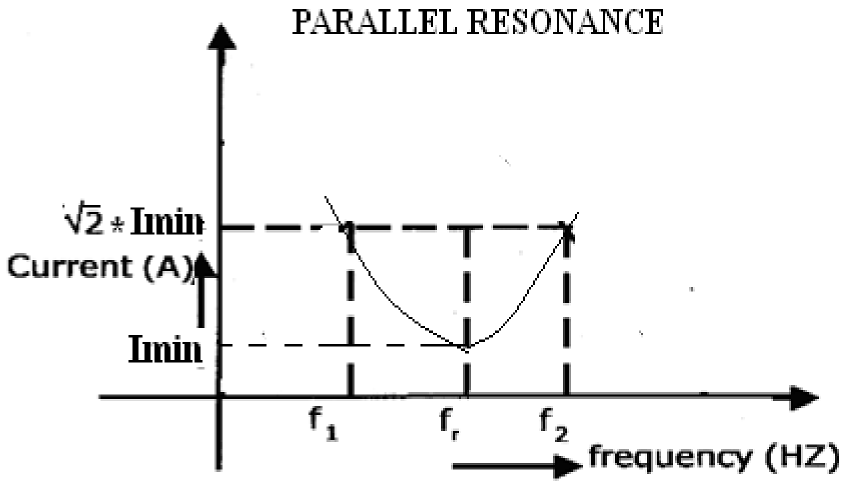
S. No.	Frequency (Hz)	Current (mA)

TABULAR COLUMN:

S.NO	PARAMETER	Series resonant circuit		Parallel resonant circuit	
		Theoretical	Practical	Theoretical	Practical
1	Resonant Frequency(f_r)				
2	Band width				
3	Quality factor				

MODEL GRAPHS:





f_1= lower cutoff frequency
f_2 = upper cutoff frequency
f_r= Resonant Frequency

PRECAUTIONS:
1. Initially keep the RPS output voltage knob in zero volt position.
2. Avoid loose connections.
3. Avoid short circuit of RPS output terminals.

RESULT:

VIVA QUESTIONS:

1) What is resonance of circuit?
2) What is series and parallel resonance?
3) What is cut-off frequency?
4) Define bandwidth and Quality factor?

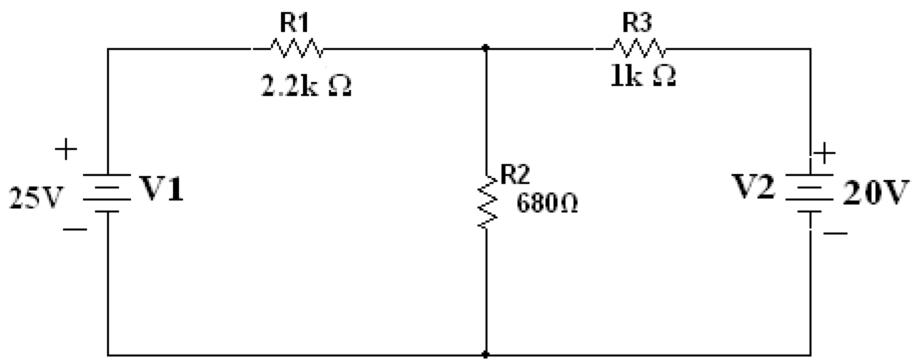
3. DETERMINATION OF Z AND Y PARAMETERS OF A TWO- PORT NETWORK

AIM: To determine the Impedance (Z) and admittance (Y) parameters of a two port network.

APPARATUS REQUIRED:

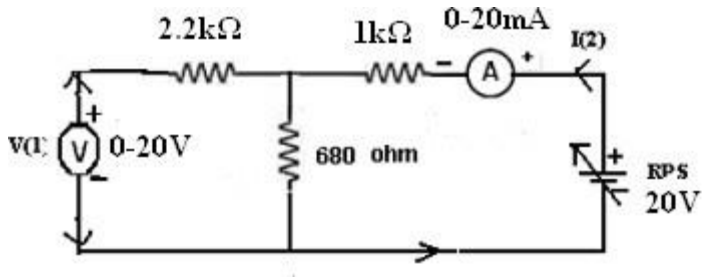
S.No	Name Of The Equipment	Range	Type	Quantity
1	Voltmeter	(0-20)V	Digital	1 NO
2	Ammeter	(0-20)mA	Digital	1 NO
3	RPS	0-30V	Digital	1 NO
4	Resistors	2.2k Ω	-	1 NO
		1k Ω	-	1 NO
		680 Ω	-	1 NO

CIRCUIT DIAGRAMS:
1. GIVEN CIRCUIT:

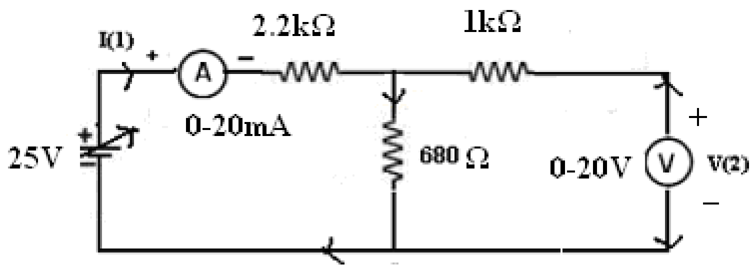


PRACTICAL CIRCUITS:

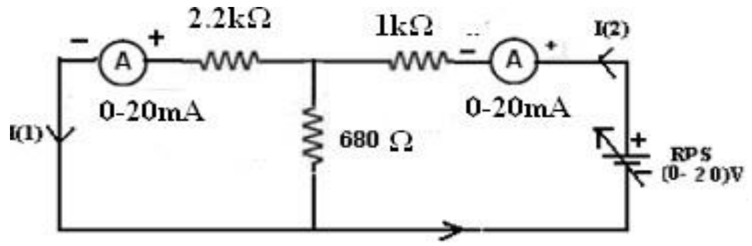
2. When $I_1 = 0$:



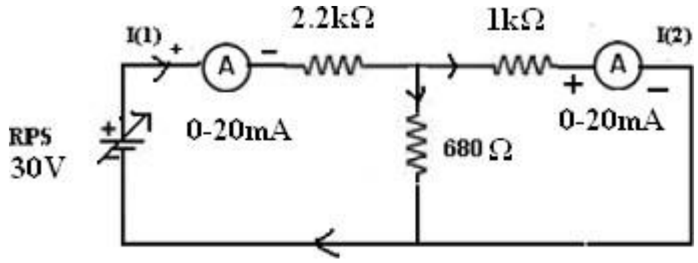
3. When $I_2 = 0$:



4. When $V_1 = 0$:



5. When $V_2 = 0$:



THEORY:

A pair of terminals between which a signal may enter or leave the network is known as port. If a network has one such type pair of terminals it is known as One-Port Network and that have two such type of ports is known as Two-Port Network.

If we relate the voltage of one port to the current of the same port, we get driving point admittance. On the other hand, if we relate the voltage of one port to the current at another port, we get transfer admittance. Admittance is a general term used to represent either the impedance or the admittance of a network. We will consider a general two-port network composed of linear, bilateral elements and no independent sources. The voltage and current at port -1 are V_1 and I_1 and at port -2 are V_2 and I_2. The position of V_1 and V_2 and the directions of I_1 and I_2 are customarily selected. Out of four variables only two are independent. The other two are expressed in terms of the independent variable of network parameters. The relation between the voltages and currents in terms of Z and Y parameters are as follows.

$$V_1 = Z_{11} (I_1) + Z_{12} (I_2)$$
$$V_2 = Z_{21} (I_1) + Z_{22} (I_2)$$

$$I_1 = Y_{11} (V_1) + Y_{12} (V_2)$$
$$I_2 = Y_{21} (V_1) + Y_{22} (V_2)$$

Z-PARAMETERS:

$Z11 = \frac{V1}{I1} / \; I_2 = 0$

$Z12 = \frac{V1}{I2} / \; I_1 = 0$

$Z21 = \frac{V2}{I1} / \; I_2 = 0$

$Z22 = \frac{V2}{I2} / \; I_1 = 0$

Y-PARAMETERS:

$$Y11 = \frac{I1}{V1} / \, V2 = 0$$
$$Y12 = \frac{I2}{V1} / \, V1 = 0$$
$$Y21 = \frac{I2}{V1} / \, V2 = 0$$
$$Y22 = \frac{I2}{V2} / \, V1 = 0$$

PROCEDURE:

1. Connections are made as per the circuit diagram.
2. Open circuit the port – 1 i.e., I_1=0, find the values ofV_1, I2 and V_2.
3. Short circuit the port-1 i.e. V_1 =0, find the values of V_2, $I_1$and I_2.
4. Open circuit the port – 2 i.e., I_2=0, find the values of V_1, I1 and V_2.
5. Short circuit the port-2 i.e. V_2 =0, find the values of V_1, $I_1$and I_2.
5. Find the Z and Y parameters of the given two port network.

THEORITICAL VALUES:

$V_1 = 0$	$V_{2=}$	$I_{1=}$	$I_{2=}$
$V_2 = 0$	$V_{1=}$	$I_{1=}$	$I_{2=}$
$I_1 = 0$	$V_{1=}$	$V_{2=}$	$I_{2=}$
$I_2 = 0$	$V_{1=}$	$V_{2=}$	$I_{1=}$

PRACTICAL VALUES:

$V_1 = 0$	$V_{2=}$	$I_{1=}$	$I_{2=}$
$V_2 = 0$	$V_{1=}$	$I_{1=}$	$I_{2=}$
$I_1 = 0$	$V_{1=}$	$V_{2=}$	$I_{2=}$
$I_2 = 0$	$V_{1=}$	$V_{2=}$	$I_{1=}$

Z-PARAMETERS:

Z-parameters	Theoretical	Practical
$Z11 = \frac{V1}{I1}$ / I_2 =0		
$Z12 = \frac{V1}{I2}$ / I_1 =0		
$Z21 = \frac{V2}{I1}$ / I_2 =0		
$Z22 = \frac{V2}{I2}$ / I_1 =0		

Y-PARAMETERS:

Y-Parameters	Theoretical	Practical
$Y11 = \frac{I1}{V1} / V2 = 0$		
$Y12 = \frac{I2}{V1} / V1 = 0$		
$Y21 = \frac{I2}{V1} / V2 = 0$		
$Y22 = \frac{I2}{V2} / V1 = 0$		

PRECAUTIONS:

1. Initially keep the RPS output voltage knob in zero volt position.
2. Avoid loose connections.
3. Avoid short circuit of RPS output terminals.

RESULT:

VIVA QUESTIONS:

1. Define Port?
2. Define Z & Y parameters?
3. What is the condition for symmetry in case Z & Y parameters?
4. Define characteristic impedance?
5. What is the condition for reciprocity in case Z & Y parameters?

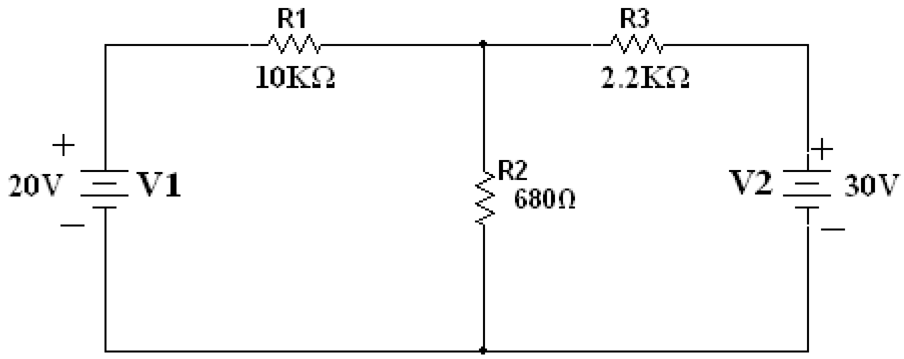
4. DETERMINATION OF TRANSMISSION AND HYBRID PARAMETERS OF A TWO-PORT NETWORK

AIM: To determine the Transmission and Hybrid parameters of a two port network.

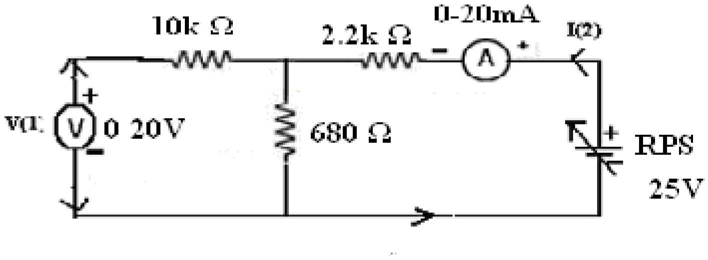
APPARATUS REQUIRED:

S.No	Name Of The Equipment	Range	Type	Quantity
1	Voltmeter	(0-20)V	Digital	1 NO
2	Ammeter	(0-20)mA	Digital	1 NO
3	RPS	0-30V	Digital	1 NO
4	Resistors	10K Ω		1 NO
		2.2Ω		1 NO
		680 Ω		1 NO
5	Breadboard	-	-	1 NO
6	Connecting wires			Required number

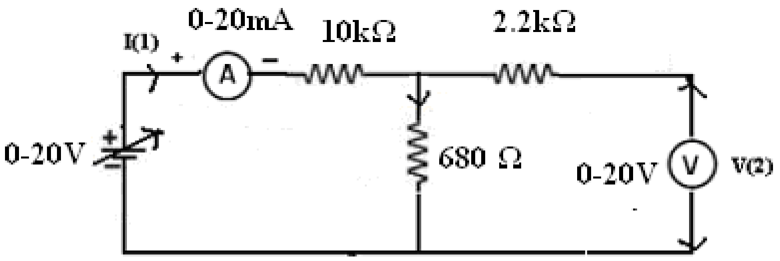
CIRCUIT DIAGRAMS:
GIVEN CIRCUIT:



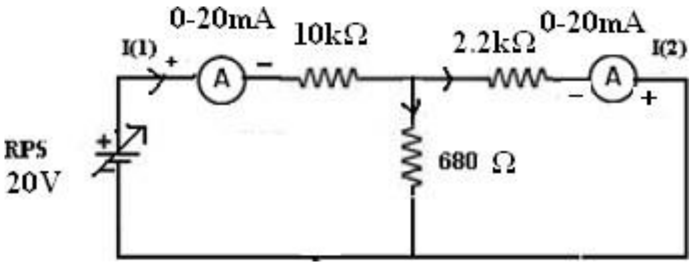
PRACTICAL CIRCUITS:
1. When $I_1 = 0$:



2. When $I_2 = 0$:



3. When $V_2 = 0$:



THEORY:

The relation between the voltages and currents of a two port network in terms of ABCD and h–parameters is given as follows.

ABCD PARAMETERS:

$V_1 = AV_2 - BI_2$

$I_1 = CV_2 - DI_2$

H-PARAMETERS

$V_1 = h_{11}I_1 + h_{12}V_2$

$I_2 = h_{21}I_1 + h_{22}V_2$

ABCD PARAMETERS:

$A = \frac{V_1}{V_2} | I_2 = 0$

$B = \frac{V_1}{I_2} | V_2 = 0$

$C = \frac{I_1}{V_2} | I_2 = 0$

$D = \frac{I_1}{I_2} | V_2 = 0$

H-PARAMETERS:

h11 = $\frac{v_1}{I_1}|V_2 = 0$

h12 = $\frac{v_1}{V_2}|I_1 = 0$

h21 = $\frac{I_2}{I_1}|V_2 = 0$

h22 = $\frac{I_2}{V_2}|I_1 = 0$

PROCEDURE:

1. Connections are made as per the circuit diagram.
2. Open circuit the port – 1 i.e., I_1=0 find the values ofV_1, I2 and V_2.
3. Short circuit the port-1 V_1 =0 find the values of V_2, $I_1$and I_2.
4. Open circuit the port – 2 i.e., I_2=0 find the values of V_1, I1 and V_2.
5. Short circuit the port-2 i.e. V_2 =0 find the values of V_1, $I_1$and I_2
5. Find the ABCD and h-parameters of the given two port network from the above data.

THEORITICAL VALUES:

$V_2 = 0$	$V_{1=}$	$I_{1=}$	$I_{2=}$
$I_1 = 0$	$V_{1=}$	$V_{2=}$	$I_{2=}$
$I_2 = 0$	$V_{1=}$	$V_{2=}$	$I_{1=}$

PRACTICAL VALUES:

$V_2 = 0$	$V_{1=}$	$I_{1=}$	$I_{2=}$
$I_1 = 0$	$V_{1=}$	$V_{2=}$	$I_{2=}$
$I_2 = 0$	$V_{1=}$	$V_{2=}$	$I_{1=}$

ABCD-PARAMETERS:

T-Parameters	Theoretical	Practical
A = $\frac{V_1}{V_2}|I_2 = 0$		
B = $\frac{V_1}{I_2}|V_2 = 0$		
C = $\frac{I_1}{V_2}|I_2 = 0$		
D = $\frac{I_1}{I_2}|V_2 = 0$		

H- PARAMETERS:

H-Parameters	Theoretical	Practical
h11 = $\frac{v_1}{I_1}|V_2 = 0$		
h12 = $\frac{v_1}{V_2}|I_1 = 0$		
h21 = $\frac{I_2}{I_1}|V_2 = 0$		
h22 = $\frac{I_2}{V_2}|I_1 = 0$		

PRECAUTIONS:

1. Initially keep the RPS output voltage knob in zero volt position.
2. Avoid loose connections.
3. Avoid short circuit of RPS output terminals.

RESULT:

VIVA QUESTIONS

1. Define Port?
2. What is the condition for symmetry in case h-parameters & ABCD (T) parameters?
3. Define characteristic impedance?
4. What is the condition for reciprocity in case Hybrid (h) & ABCD (T) parameters?

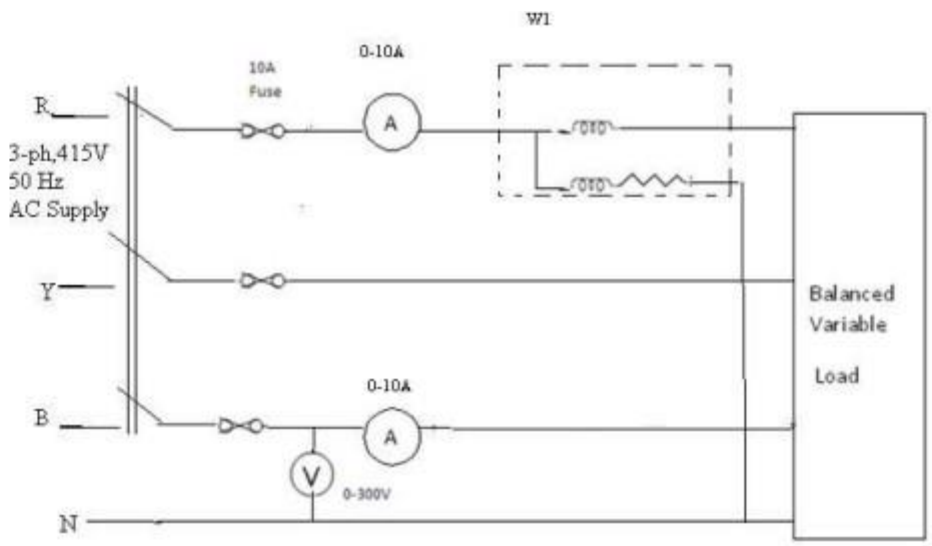
5. MEASUREMENT OF ACTIVE POWER FOR STAR AND DELTA CONNECTED BALANCED LOAD

AIM: To measure the active power for the given star and delta networks.

APPARATUS REQUIRED:

Sl. No.	Name of the Equipment	Range	Type	Quantity
01	Auto Transformer	415V/(0-440), (0-20)A	3–Φ	1 NO
02	U.P.F. Wattmeter	(150/300/600) (0-5/10)A	Dynamometer Type	1 NO
03	L.P.F. Wattmeter	(150/300/600)V(0-5/10)A	Dynamometer Type	1 NO
04	Ammeter	(0-10)A	MI	1 NO
05	Voltmeter	(0-600)V	MI	1 NO
06	Connecting Wires	------	------	As required

CIRCUIT DIAGRAMS:



THEORY:

A three phase balanced voltage is applied on a balanced three phase load when the current in each of the phase lags by an angle Φ behind corresponding phase voltages. Current through current coil of $w_1=I_r$, current through current coil of $W_2=I_B$, while potential difference across voltage coil of $W_1=V_{RN}-V_{YN}=V_{RY}$(line voltage), and the potential difference across voltage coilof $W_2= V_{RN}-V_{YN}=V_{BY}$.Also, phase difference between I_R and V_{RY} is (300+ Φ).While that between I_B and V_{BY} is (300- Φ).Thus reading on wattmeter W_1 is given by $W_1=V_{RY}\ I_Y Cos(300+ \Phi)$While reading on wattmeter W_2 is given by $W_2=V_{BY}I_B Cos(300-$

Φ)Since the load is balanced, $|I_R|=|I_Y|=|I_B|=I$ and $|V_{RY}|=|V_{BY}|=V_L W_1=V_L I Cos(300+ \Phi) W_2=V_L I Cos(300- \Phi)$.

Thus total power P is given by

$$W= W_1 +W_2 = V_L I Cos(300+ \Phi) + V_L I Cos(300- \Phi)$$
$$= V_L I[Cos(300+ \Phi) + Cos(300- \Phi)]$$
$$= [\sqrt{3}/2 *2Cos \Phi]V_L I = \sqrt{3} V_L I Cos \Phi$$

PROCEDURE:

(Star connection):

1) Connect the circuit as shown in the figure.
2) Ammeter is connected in series with wattmeter whose other end is connected to one of the loads of the balanced loads.
3) The Y-phase is directly connected to one of the nodes of the 3-ph supply.
4) A wattmeter is connected across R-phase & Y-phase as shown in fig. The extreme of B phase is connected to the third terminal of the balanced 3-ph load.
5) Another wattmeter is connected across Y & B phase, the extreme of B-phase is connected to the third terminal of the balanced three phases load.
6) Verify the connections before switching on the 3-ph power supply.

(Delta connection):

1) Connect the circuit as shown in the figure.
2) Ammeter is connected in series with wattmeter whose other end is connected to one of the loads of the balanced loads.
3) The Y-phase is directly connected to one of the nodes of the 3-ph supply.
4) A wattmeter is connected across Y & B phase, the extreme of B-phase is connected to the third terminal of the balanced 3-ph load.
5) Another wattmeter is connected across R & Y phase, the extreme of R-phase is connected to the third terminal of the balanced three phases load.
6) Verify the connections before switching on the 3-ph power supply.

TABULAR COLUMN:

S.No	Voltage V (Volts)	Line Current I_L (Amps) I	W_1 (Watts)	W_2 (Watts)	$W= W_1 + W_2$

THEORITICAL CALCULATIONS :

For a star connected load

Line voltage(V_L) = VL/$\mathbf{3}^{1/2}$

Line current(I_L) = I_L

$Ø$ =tan-1 $3^{1/2}(W_1-W_2)/(W_1+W_2)$

$P = 3^{1/2} V_L I_L COSØ$

$P = W_1+W_2$

For a delta connected load

Line voltage(VL) = VL

Line current(IL) = $I_L/3^{1/2}$

$Ø$ =tan-1 $3^{1/2}$ (W1-W2)/(W1+W2)

$P = 3^{1/2} V_L I_L COSØ$

$P = W_1+W_2$

PRECAUTIONS:

1. Avoid making loose connections.
2. Readings should be taken carefully without parallax error.

RESULT:

VIVA QUESTIONS

1. Define active power, reactive power & apparent power.
2. Define power factor?
3. What are the different types of loads?
4. Write the equations of active power, reactive power & apparent power?

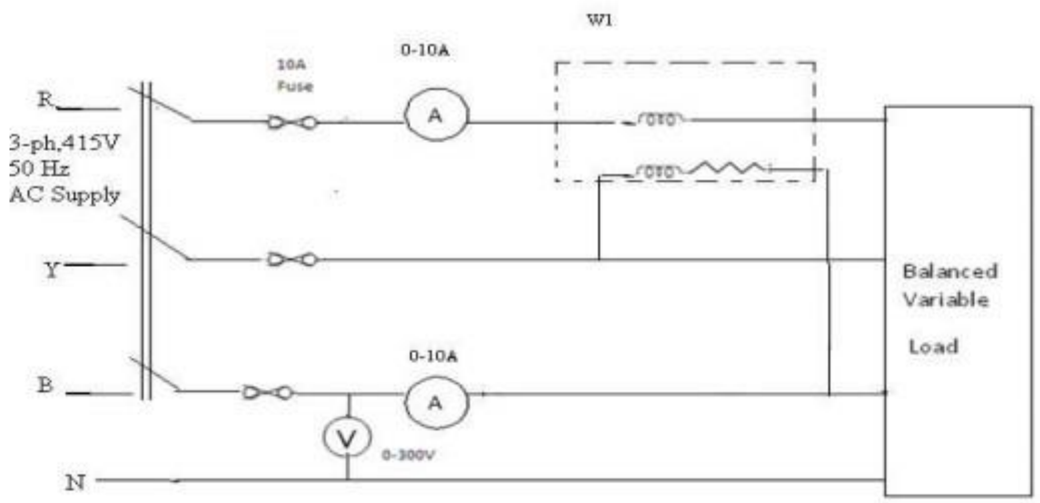
6. MEASUREMENT OF REACTIVE POWER FOR STAR AND DELTA CONNECTED BALANCED LOAD

AIM: To measure the reactive power for the given star and delta network.

APPARATUS REQUIRED:

Sl. No.	Name of the Equipment	Range	Type	Quantity
01	Capacitive Load	440V, 1.5KVA	3–Φ	1 NO
02	Auto Transformer	415V/(0-440), (0-20)A	3–Φ	1 NO
03	U.P.F. Wattmeter	(150/300/600) (0-5/10)A	Dynamometer Type	1 NO
04	L.P.F. Wattmeter	(150/300/600)V(0-5/10)A	Dynamometer Type	1 NO
05	Ammeter	(0-10)A	MI	1 NO
06	Voltmeter	(0-600)V	MI	1 NO
07	Connecting Wires	------	------	As required

CIRCUIT DIAGRAMS:



THEORY:

A three phase balanced voltage is applied on a balanced three phase load when the current in each of the phase lags by an angle Φ behind corresponding phase voltages. Current through current coil of w_1=I_r, current through current coil of W_2=I_B, while potential difference across voltage coil of W_1=V_{RN}-V_{YN}=V_{RY}(line voltage), and the potential difference across voltage coilof W_2= V_{RN}-V_{YN}=V_{BY}.Also, phase difference between I_R and V_{RY} is (300+ Φ).While that between I_B and V_{BY} is (300- Φ).Thus reading on wattmeter W_1 is given by W_1=V_{RY} I_YCos(300+ Φ)While reading on wattmeter W_2 is given by W_2=$V_{BY}$$I_B$Cos(300-

Φ)Since the load is balanced, $|I_R|=|I_Y|=|I_B|=I$ and $|V_{RY}|=|V_{BY}|=V_L W_1=V_L I Cos(300+ \Phi) W_2=V_L I Cos(300- \Phi)$.

Thus total power P is given by

$$W= W_1 +W_2 = V_L ICos(300+ \Phi) + V_L ICos(300- \Phi)$$

$$= V_L I[Cos(300+ \Phi) + Cos(300- \Phi)]$$

$$= [\sqrt{3}/2 *2Cos \Phi]V_L I = \sqrt{3} V_L ICos \Phi$$

PROCEDURE:

1. Make the Connections as per circuit diagram.
2. Keep the 3-Phase Autotransformer is in minimum output position.
3. Switch on the supply and by slowly varying the autotransformer, rated value is applied to motor.
4. Note down the readings of Ammeter, Voltmeter, Wattmeter‟s readings (Wr & Wa)
5. After noting the values slowly decrease the Auto Transformer till Volt meter comes to zero voltage position, and switch of the supply.

TABULAR COLUMN:

S.No	Voltage V (Volts)	Line Current I_L (Amps) I	W_1 (Watts)	W_2 (Watts)	$W= W_1 +W_2$

THEORITICAL CALCULATIONS :

Ammeter reading = I_{ph} =

Voltmeter reading = V_{ph} =

Wattmeter reading (W_a) = Active power / Phase

Wattmeter reading (W_a) =

i.e. total active power = 3×Wa Total active power = 3VIcosϕ = $3W_a$

Cosϕ = W_a / VI

$Sin^2\phi = 1- Cos2\phi$

Total calculated reactive power = W_{RC} = 3VISin ϕ

Total measured reactive power = 3Wr

PRECAUTIONS:

1. There should not be any loose connections.
2. Meter readings should not be exceeded beyond their ratings
3. Readings of the meters must be taking without parallax error.
4. Ensure that setting of the Auto Transformer at zero output voltage during starting.

RESULT:

VIVA QUESTIONS

1. Define active power, reactive power & apparent power.
2. Define power factor?
3. What are the different types of loads?
4. Write the equations of active power, reactive power & apparent power.

CYCLE – 2

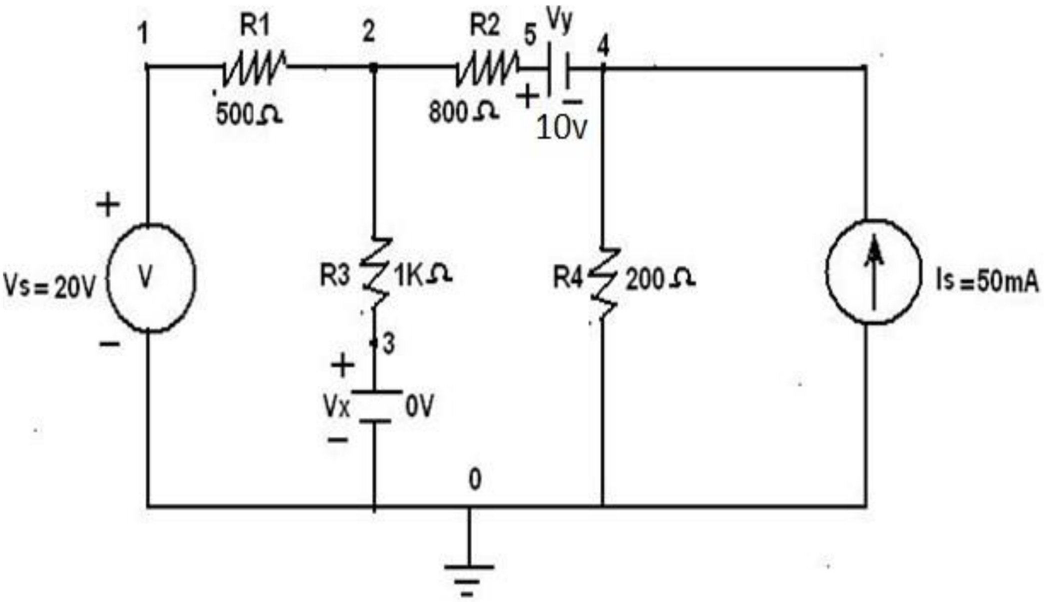
1. SIMULATION OF DC CIRCUIT

AIM: To obtain the node voltages, branch currents, power of all voltage sources of a given DC circuit by using PSPICE programming.

APPARATUS REQUIRED:

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

CIRCUIT DIAGRAM:



THEORY:

PSPICE is a general-purpose circuit program that simulates electronic circuits. PSPICE can perform various types of analysis of electronic circuits, the operating points of transistors, time domain response, small signal frequency response, etc…

PSPICE contains models for common circuit elements, active as well as passive, and it is capable of simulating most electronic circuits. It is a versatile program and is widely used in industries and universities.

The acronym PSPICE stands for **Personal Simulation Program with Integrated Circuit Emphasis**.

Until recently, PSPICE was available only on mainframe computers. In addition to the initial cost of the computer system, such a machine can be unwieldy and inconvenient for class room use. In 1984, MICROSIM introduced the PSPICE simulator, which is similar to Berkeley PSPICE and runs on an IBM-PC or compatible. It was available at no cost to students for classroom use. PSPICE, therefore widened the scope of the integrated computer aided circuit analysis into electronic circuit courses at the under graduate level. Other versions of PSPICE that will run on computers such as the Macintosh-II, VAX, SUN, and NEC are also available.

PSPICE allows the various types of analysis as follows:

1. DC Analysis:- Calculation of node voltages and branch currents and their quiescent values are the outputs.

Eg:- DC sweep voltage (.DC),
Small-Signal transfer function (Thevenin's equivalent) (.TF)
DC Small-Signal sensitivities (.SENS)

2. Transient Analysis:- Responses of time-invariant systems, DC transient analysis and Fourier analysis

Eg:- Transient responses______(.TRAN)
Fourier Analysis ______(.FOUR)

3. AC Analysis:- (.AC) & (.NOISE) etc.

PSPICE PROGRAM :-

V_S 1 0 DC 20V	: DC Voltage source of 20V between 1& 0 nodes
I_S 0 4 DC 50mA	: DC Current source of 50mA between 4 & 0 nodes
R_1 1 2 500	: Resistance of 500ohms between 1 & 2 nodes
R_2 2 5 800	: Resistance of 800ohms between 5 & 2 nodes
R_3 2 3 1000	: Resistance of 1000ohms between 2 & 3 nodes
R_4 4 0 200	: Resistance of 200ohms between 4 & 0 nodes
V_X 3 0 DC 0V	: DC Voltage source of 0V between 3& 0 nodes
V_Y 5 4 DC 10V	: DC Voltage source of 10V between 5& 4 nodes
.OP	: Directs the bias point to the output file.
.END	: End of the program.

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.

RESULT:

VIVA QUESTIONS:

1. How simulation can be used for network analysis?
2. Define network and circuit?
3. What is the difference between unilateral and bilateral network?
4. What is the difference between active elements and passive elements?

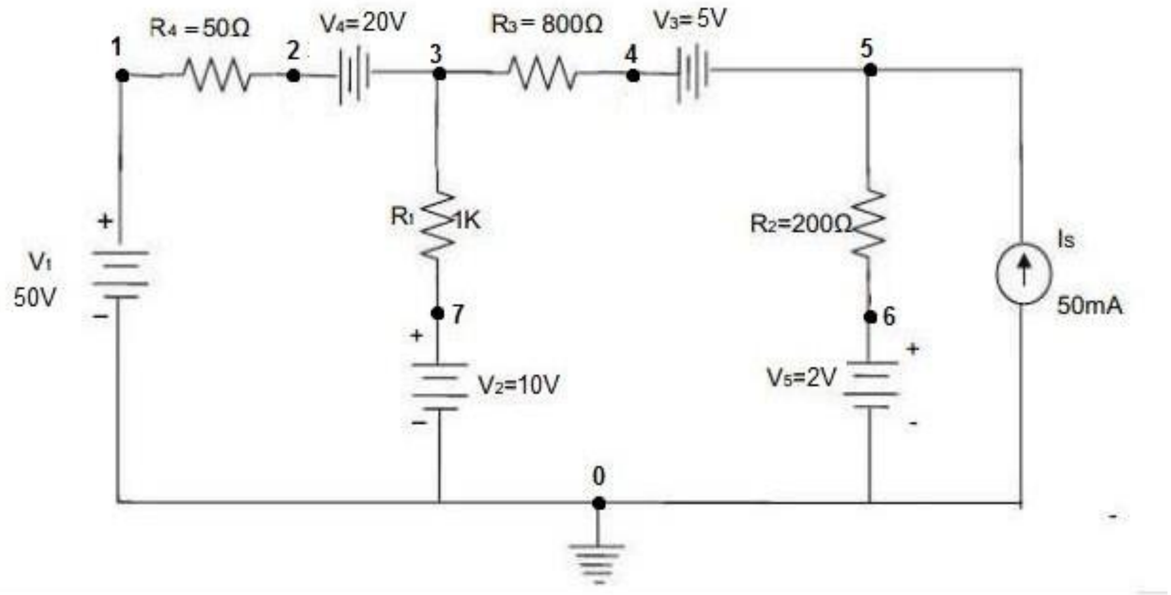
2. SIMULATION OF MESH ANALYSIS

AIM: To find the voltage across each resistor, branch currents of a given circuit using mesh analysis by PSPICE Software..

APPARATUS REQUIRED:

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

CIRCUIT DIAGRAM:



THEORY:

PSPICE is a general-purpose circuit program that simulates electronic circuits. PSPICE can perform various types of analysis of electronic circuits, the operating points of transistors, time domain response, small signal frequency response, etc…

PSPICE contains models for common circuit elements, active as well as passive, and it is capable of simulating most electronic circuits. It is a versatile program and is widely used in industries and universities.

The acronym PSPICE stands for **Personal Simulation Program with Integrated Circuit Emphasis**.

Until recently, PSPICE was available only on mainframe computers. In addition to the initial cost of the computer system, such a machine can be unwieldy and inconvenient for class room use. In 1984, MICROSIM introduced the PSPICE simulator, which is similar to Berkeley PSPICE and runs on an IBM-PC or compatible. It was available at no cost to students for classroom use. PSPICE, therefore widened the scope of the integrated computer aided circuit analysis into electronic circuit courses at the under graduate level. Other versions of PSPICE that will run on computers such as the Macintosh-II, VAX, SUN, and NEC are also available.

PSPICE allows the various types of analysis as follows:

1. DC Analysis:- Calculation of node voltages and branch currents and their quiescent values are the outputs.

Eg:- DC sweep voltage (.DC),
Small-Signal transfer function (Thevenin‟s equivalent) (.TF)
DC Small-Signal sensitivities (.SENS)

2. Transient Analysis:- Responses of time-invariant systems, DC transient analysis and Fourier analysis

Eg:- Transient responses______(.TRAN)
Fourier Analysis ______(.FOUR)

3. AC Analysis:- (.AC) & (.NOISE) etc.

PSPICE PROGRAM :-

V_1 1 0 DC 50V	: DC Voltage source of 50V between 1& 0 nodes
V_2 7 0 DC 10V	: DC Voltage source of 10V between 7& 0 nodes
V_3 4 5 DC 5V	: DC Voltage source of 5V between 4& 5 nodes
V_4 2 3 DC 20V	: DC Voltage source of 20V between 2& 3 nodes
V_5 6 0 DC 2V	: DC Voltage source of 2V between 6& 0 nodes
I_s 5 0 50mA	: DC Current source of 50mA between 5& 0 nodes
R_1 3 7 1k	: Resistance of 1000ohms between 3 & 7 nodes
R_2 5 6 200	: Resistance of 200ohms between 5 & 6 nodes
R_3 3 4 800	: Resistance of 800ohms between 3 & 4 nodes
R_4 1 2 50	: Resistance of 50ohms between 1 & 2 nodes
.OP	: Directs the bias point to the output file.
.END	: End of the program.

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.

RESULT:

VIVA QUESTIONS:

1. Define Kirchhoff‟s Voltage law?
2. For which type of circuits Mesh Analysis can be used ?
3. What is the difference between planar and non-planar?
4. When there is a current source between two loops which method is preferred?
5. Kirchhoff‟s laws can not applied at_____?

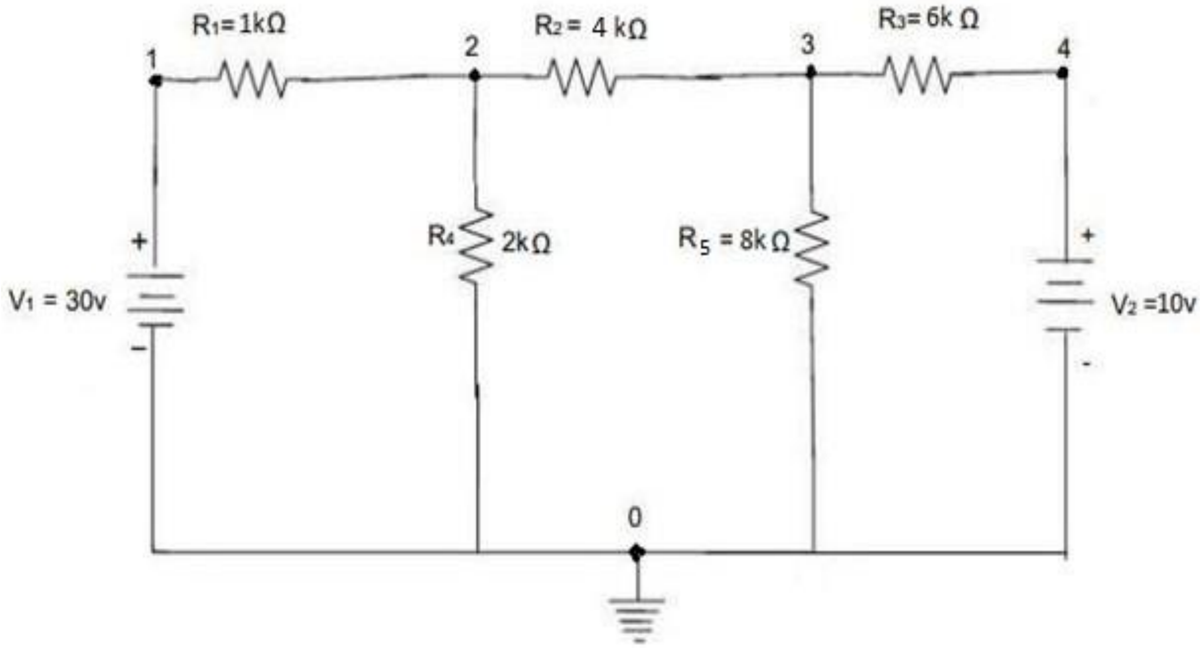
3. SIMULATION OF NODAL ANALYSIS

AIM: To find the node voltages, branch currents of a given circuit using nodal analysis by PSPICE Software.

APPARATUS REQUIRED:

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

CIRCUIT DIAGRAM:



THEORY:

PSPICE is a general-purpose circuit program that simulates electronic circuits. PSPICE can perform various types of analysis of electronic circuits, the operating points of transistors, time domain response, small signal frequency response, etc…

PSPICE contains models for common circuit elements, active as well as passive, and it is capable of simulating most electronic circuits. It is a versatile program and is widely used in industries and universities.

The acronym PSPICE stands for **Personal Simulation Program with Integrated Circuit Emphasis**.

Until recently, PSPICE was available only on mainframe computers. In addition to the initial cost of the computer system, such a machine can be unwieldy and inconvenient for class room use. In 1984, MICROSIM introduced the PSPICE simulator, which is similar to Berkeley PSPICE and runs on an IBM-PC or compatible. It was available at no cost to students for classroom use. PSPICE, therefore widened the scope of the integrated computer aided circuit analysis into electronic circuit courses at the under graduate level. Other versions of PSPICE that will run on computers such as the Macintosh-II, VAX, SUN, and NEC are also available.

PSPICE allows the various types of analysis as follows:

1. DC Analysis:- Calculation of node voltages and branch currents and their quiescent values are the outputs.

Eg:- DC sweep voltage (.DC),
Small-Signal transfer function (Thevenin‟s equivalent) (.TF)
DC Small-Signal sensitivities (.SENS)

2. Transient Analysis:- Responses of time-invariant systems, DC transient analysis and Fourier analysis

Eg:- Transient responses______(.TRAN)
Fourier Analysis ______(.FOUR)

3. AC Analysis:- (.AC) & (.NOISE) etc.

PSPICE PROGRAM :-

V_1 1 0 DC 30V	: DC Voltage source of 30V between 1& 0 nodes
V_2 4 0 DC 10V	: DC Voltage source of 10V between 4& 0 nodes
R_1 1 2 1000	: Resistance of 1000ohms between 1 & 2 nodes
R_2 2 3 4000	: Resistance of 800ohms between 5 & 2 nodes
R_3 3 4 6000	: Resistance of 6000ohms between 2 & 3 nodes
R_4 2 0 2000	: Resistance of 200ohms between 4 & 0 nodes
R_5 3 0 8000	: Resistance of 200ohms between 3 & 0 nodes
.OP	: Directs the bias point to the output file.
.END	: End of the program.

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.

RESULT:

VIVA QUESTIONS:

1. Define Kirchhoff‟s Current law?
2. For which type of circuits Nodal Analysis can be used?
3. How many nodes are taken as reference nodes in a nodal analysis?
4. When there is only voltage source between two nodes which method is preferred?

4. SIMULATION OF DC TRANSIENT RESPONSE

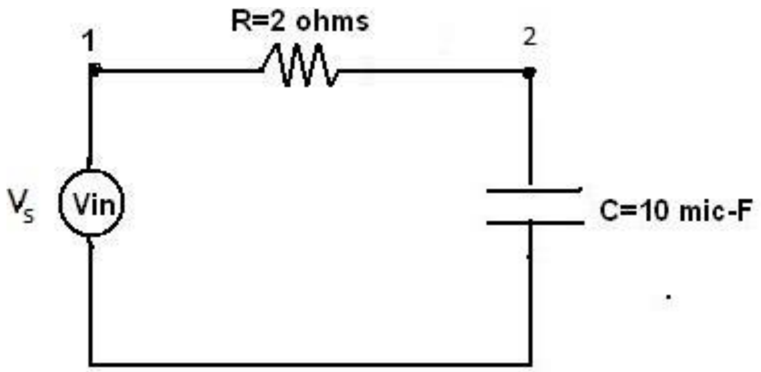
I. SERIES RC CIRCUIT

AIM: To obtain the simulation result of a given series RC circuit with different inputs using PSPICE programming.

APPARATUS REQUIRED:

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

CIRCUIT DIAGRAM:



THEORY:

PSPICE is a general-purpose circuit program that simulates electronic circuits. PSPICE can perform various types of analysis of electronic circuits, the operating points of transistors, time domain response, small signal frequency response, etc…

PSPICE contains models for common circuit elements, active as well as passive, and it is capable of simulating most electronic circuits. It is a versatile program and is widely used in industries and universities.

The acronym PSPICE stands for **Personal Simulation Program with Integrated Circuit Emphasis**.

Until recently, PSPICE was available only on mainframe computers. In addition to the initial cost of the computer system, such a machine can be unwieldy and inconvenient for class room use. In 1984, MICROSIM introduced the PSPICE simulator, which is similar to Berkeley PSPICE and runs on an IBM-PC or compatible. It was available at no cost to students for classroom use. PSPICE, therefore widened the scope of the integrated computer aided circuit analysis into electronic circuit courses at the under graduate level. Other versions of PSPICE that will run on computers such as the Macintosh-II, VAX, SUN, and NEC are also available.

PSPICE allows the various types of analysis as follows:

1. DC Analysis:- Calculation of node voltages and branch currents and their quiescent values are the outputs.

Eg:- DC sweep voltage (.DC),
Small-Signal transfer function (Thevenin's equivalent) (.TF)
DC Small-Signal sensitivities (.SENS)

2.Transient Analysis:- Responses of time-invariant systems, DC transient analysis and Fourier analysis

Eg:- Transient responses______(.TRAN)
Fourier Analysis ______(.FOUR)

3.AC Analysis:- (.AC) & (.NOISE) etc.

PSPICE PROGRAM:

a) **Pulse Input**:-

```
VS 1 0 pulse (-5 5 IN IN 1M   2M)   : Pulse input with specifications
R 1 2 2                              : Resistance of 2ohms between 1 & 2 points
C 2 0 10U                            : Capacitance of 10 micro-F between 2 & 0 points
.TRAN IN 4M                          : Transient response of RC circuit
.PROBE                               : Representation in graphs
.END                                 : End of the program
```

b) **Step Input**:-

```
VS 1 0 PWL(0 0 100N 1)               : Step input with specifications
R 1 2 2                              : Resistance of 2ohms between 1 & 2 points
C 2 0 10U                            : Capacitance of 10micro-F between 2 & 0 points
.TRAN IN 4M                          : Transient response of RC circuit
.PROBE                               : Representation in graphs
.End                                 : End of the program
```

c)**Sinusoidal Input**:-

```
VS 1 0 SIN(0 10 1K)                  : Sinusoidal input with specifications
R 1 2 2                              : Resistance of 2 ohms between 1 & 2 points
C 2 0 10U                            : Capacitance of 10 micro-F between 2 & 0 points
.TRAN IN 4M                          : Transient response of RC circuit
.END                                 :   End of the program
```

d)**Exponential Input**:-

```
VS 1 0 EXP(0.5 1 0.1N 1 1.5N)        : Exponential input with specifications
R 1 2 2                              : Resistance of 2ohms between 1 & 2 points
C 3 0 50U                            : Capacitance of 10 micro-F between 2 & 0 points
```

.TRAN IN 4M	: Transient response of RC circuit
.PROBE	: Representation in graphs
.END	: End of the program

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.

RESULT:

VIVA QUESTIONS:

1. Define transient state response?
2. What are the methods of analysis in DC transients?
3. A square wave is fed to an RC circuit then output voltage across resistance is?
4. A square wave is fed to an RC circuit then output voltage across capacitance is?
5. In a series RC circuit at steady state capacitor C acts as?

4. SIMULATION OF DC TRANSIENT RESPONSE

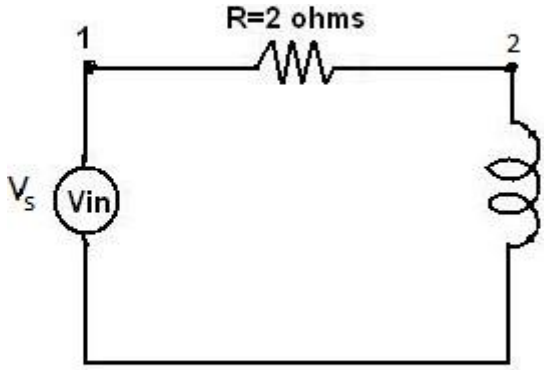
II. SERIES RL CIRCUIT

AIM: To obtain the simulation result of a given series RL circuit with different inputs using PSPICE programming.

APPARATUS REQUIRED:

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

CIRCUIT DIAGRAM:



THEORY:

PSPICE is a general-purpose circuit program that simulates electronic circuits. PSPICE can perform various types of analysis of electronic circuits, the operating points of transistors, time domain response, small signal frequency response, etc…

PSPICE contains models for common circuit elements, active as well as passive, and it is capable of simulating most electronic circuits. It is a versatile program and is widely used in industries and universities.

The acronym PSPICE stands for **Personal Simulation Program with Integrated Circuit Emphasis**.

Until recently, PSPICE was available only on mainframe computers. In addition to the initial cost of the computer system, such a machine can be unwieldy and inconvenient for class room use. In 1984, MICROSIM introduced the PSPICE simulator, which is similar to Berkeley PSPICE and runs on an IBM-PC or compatible. It was available at no cost to students for classroom use. PSPICE, therefore widened the scope of the integrated computer aided circuit analysis into electronic circuit courses at the under graduate level. Other versions of PSPICE that will run on computers such as the Macintosh-II, VAX, SUN, and NEC are also available.

PSPICE allows the various types of analysis as follows:

1.DC Analysis:- Calculation of node voltages and branch currents and their quiescent values are the outputs.

Eg:- DC sweep voltage (.DC),
Small-Signal transfer function (Thevenin's equivalent) (.TF)
DC Small-Signal sensitivities (.SENS)

2. Transient Analysis:- Responses of time-invariant systems, DC transient analysis and Fourier analysis

Eg:- Transient responses______(.TRAN)
Fourier Analysis ______(.FOUR)

3. AC Analysis:- (.AC) & (.NOISE) etc.

PSPICE PROGRAM:

a) **Pulse Input**:-

```
Vs 1 0 pulse (-5 5 IN IN 1M   2M)   : Pulse input with specifications
R 1 2 2                             : Resistance of 2ohms between 1 & 2 points
L 2 0 50U                           : Inductance of 50 micro-F between 2 & 0 points
.TRAN IN 4M                         : Transient response of RL circuit
.PROBE                              : Representation in graphs
.END                                : End of the program
```

b) **Step Input**:-

```
Vs 1 0 PWL(0 0 100N 1)              : Step input with specifications
R 1 2 2                             : Resistance of 2ohms between 1 & 2 points
L 2 0 50U                           : Inductance of 50 micro-F between 2 & 0 points
.TRAN IN 4M                         : Transient response of RL circuit
.PROBE                              : Representation in graphs
.End                                : End of the program
```

c) **Sinusoidal Input**:-

```
Vs 1 0 SIN(0 10 1K)                 : Sinusoidal input with specifications
R 1 2 2                             : Resistance of 2 ohms between 1 & 2 points
L 2 0 50U                           : Inductance of 50 micro-F between 2 & 0 points
.TRAN IN 4M                         : Transient response of RL circuit
.END                                :   End of the program
```

d) **Exponential Input**:-

```
Vs 1 0 EXP(0.5 1 0.1N 1 1.5N)       : Exponential input with specifications
R 1 2 2                             : Resistance of 2ohms between 1 & 2 points
```

L 2 0 50U	: Inductance of 50 micro-F between 2 & 0 points
.TRAN IN 4M	: Transient response of RL circuit
.PROBE	: Representation in graphs
.END	: End of the program

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.

RESULT:

VIVA QUESTIONS:

1. Define steady state response?
2. In a series RL circuit at steady state inductor L acts as?
3. In a series RL circuit at $t=0^+$ inductor L acts as?
4. A square wave is fed to an RL circuit then output voltage across inductance is?