ELECTRICAL CIRCUITS AND SIMULATION LAB

B.Tech II Year – II SEM

NAME: ______________________________

ROLL NO: ____________________________

BRANCH: _____________________________

DEPARTMENT OF

ELECTRICAL AND ELECTRONICS ENGINEERING

Aurora's Technological and Research Institute

Parvathapur, Uppal, Hyderabad-500 039.

LABORATORY PRACTICE

I HEAR, I FORGET

I SEE, I REMEMBER

I DO, I UNDERSTAND

PRACTCAL APPROACH IS PROBABLY THE BEST APPROACH TO GAIN A CLEAR INSIGHT

CODE OF CONDUCT FOR THE LABORATORIES

- All students must observe the Dress Code while in the laboratory.
- Sandals or open-toed shoes are NOT allowed.
- Foods, drinks and smoking are NOT allowed.
- All bags must be left at the indicated place.
- The lab timetable must be strictly followed.
- Be PUNCTUAL for your laboratory session.
- Experiment must be completed within the given time.
- Noise must be kept to a minimum.
- Workspace must be kept clean and tidy at all time.
- Handle all apparatus with care.
- All students are liable for any damage to equipment due to their own negligence.
- All equipment, apparatus, tools and components must be RETURNED to their original place after use.
- Students are strictly PROHIBITED from taking out any items from the laboratory.
- Students are NOT allowed to work alone in the laboratory without the Lab Supervisor
- Report immediately to the Lab Supervisor if any injury occurred.
- Report immediately to the Lab Supervisor any damages to equipment.

Before leaving the lab

- Place the stools under the lab bench.
- Turn off the power to all instruments.
- Turn off the main power switch to the lab bench.
- Please check the laboratory notice board regularly for updates

GENERAL LABORATORY INSTRUCTIONS

- You should be punctual for your laboratory session and should not leave the lab without the permission of the teacher.
- Each student is expected to have his/her own lab book where they will take notes on the experiments as they are completed.
- The lab books will be checked at the end of each lab session. Lab notes are a primary source from which you will write your lab reports.
- You and your batch mates will work closely on the experiments together. One partner doing all the work will not be tolerated. All the Batch mates should be able to explain the purpose of the experiment and the underlying concepts.
- Please report immediately to the member of staff or lab assistant present in the laboratory; if any equipment is faulty.

Organization of the Laboratory

- It is important that the experiments are done according to the timetable and completed within the scheduled time.
- You should complete the prelab work in advance and utilize the laboratory time for verification only.
- The aim of these exercises is to develop your ability to understand, analyze and test them in the laboratory.
- A member of staff and a Lab assistant will be available during scheduled laboratory sessions to provide assistance.
- Always attempt experiments; first without seeking help.
- When you get into difficulty; ask for assistance.

Assessment

- The laboratory work of a student will be evaluated continuously during the semester for 25 marks. Of the 25 marks, 15 marks will be awarded for day-to-day work.
- For each experiment marks are awarded under three heads:
  _ Prelab preparation – 5 marks
  _ Practical work – 5marks, and
  _ Record of the Experiment – 5marks
- Internal lab test(s) conducted during the semester carries 10 marks.
- End semester lab examination, conducted as per the JNTU regulations, carries 50 marks.

- At the end of each laboratory session you must obtain the signature of the teacher along with the marks for the session out of 10 on the lab notebook.

Lab Reports

- Note that, although students are encouraged to collaborate during lab, each must individually prepare a report and submit.
- They must be organized, neat and legible.
- Your report should be complete, thorough, understandable and literate.
- You should include a well-drawn and labeled engineering schematic for each circuit
- Investigated.
- Your reports should follow the prescribed format, to give your report structure and to make sure that you address all of the important points.
- Graphics requiring- drawn straight lines should be done with a straight edge. Well drawn freehand sketches are permissible for schematics.
- Space must be provided in the flow of your discussion for any tables or figures. Do not collect figures and drawings in a single appendix at the end of the report.
- Reports should be submitted within one week after completing a scheduled lab session.

Presentation

- Experimental facts should always be given in the past tense.
- Discussions or remarks about the presentation of data should mainly be in the present tense.
- Discussion of results can be in both the present and past tenses, shifting back and forth from experimental facts to the presentation.
- Any specific conclusions or deductions should be expressed in the past tense.

Report Format

Lab write ups should consist of the following sections:

- **Aim**: A concise statement describing the experiment and the results. This is usually not more than 3 sentences. Since the abstract is a summary of what you have done, it's a good idea to write this last.

- **Apparatus**: Describe what equipment and components you used to conduct the experiment.
- **Theory**: Several paragraphs that explain the motivation of the experiment. Usually in this statement you state what you intent to accomplish as well as the expected results of the experiment.
- **Procedure**: Describe how you conducted the experiment
- **Results and Analysis**: This is the main body of the report. Graphs, tables, schematics, diagrams should all be included and explained. Results of any calculations should be explained and shown. State the results of the experiment. Include any problems encountered.
- **Conclusion**: Explain how the experiment went, and whether you were able to achieve the expected results stated in the introduction.

Name of the student: ______________________________

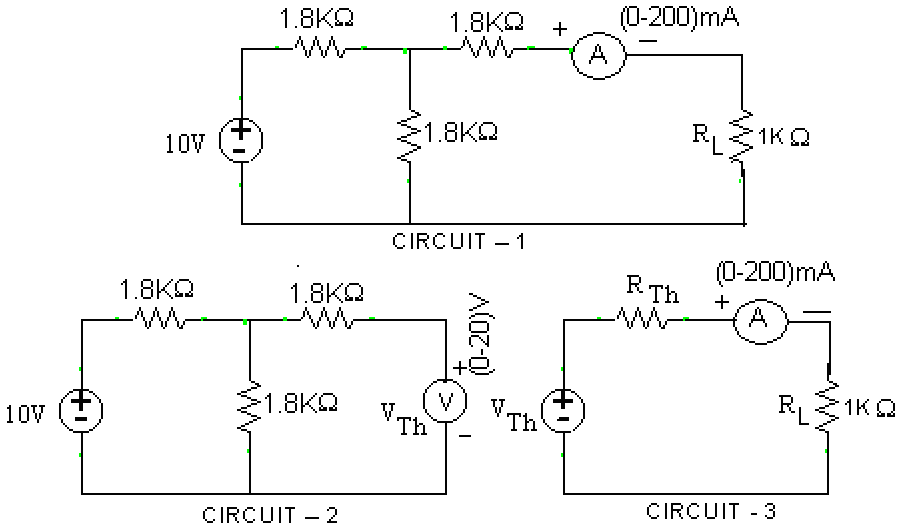
Roll Number: ______________________

INDEX

Cycle - 1

Cycle II

CIRCUIT DIAGRAM:



OBSERVATIONS & CALCULATIONS :

Theoretical:

Open circuit voltage V_{Th} =

Thevenin's Equivalent Resistance R_{Th} =

Current through the load resistor I_L =

Measured

Open circuit voltage V_{Th} =

Thevenin's Equivalent Resistance R_{Th} =

Current through the load resistor I_L =

1. Thevenin's Norton's and Maximum Power Transfer Theorems.
Part - A
THEVENIN'S THEOREM

AIM :

To verify Thevenin's Theorem for a linear network.

APPARATUS:

	NAME	RANGE	QUANTITY
1.	Bread Board.		
2.	Resistors -	1.8KΩ	3 No.s
		1KΩ	1 No.
3.	Voltmeter	(0-20) V	1 No.
4.	Ammeter	(0-20mA)	1 No.
5.	Multi meter		1 No.
6.	Connecting wires.		

THEORY:

Statement: Any linear bilateral network containing one or more voltage sources can be replaced by an equivalent circuit consisting of a single voltage source whose value is equal to the open circuit voltage across the output terminals, in series with Thevenin's equivalent resistance. The Thevenin's equivalent resistance is equal to the effective resistance measured between the output terminals, with the load resistance removed and with all the energy sources are replaced by their internal resistances.

PROCEDURE:

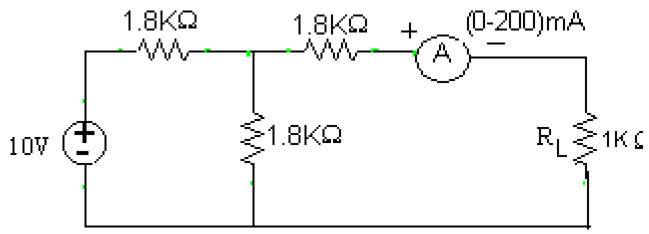
1. Connect the circuit as per CIRCUIT-1. Note down the current through the load resistance.
2. Calculate the value of open circuit voltage, Thevenin's equivalent resistance and the current through the load resistance using Thevenin's theorem.
3. Find out R_{Th} by shorting the voltage source and measuring the equivalent resistance across open circuited R_L (1KΩ) terminals using a multimeter. Compare this value with the calculated value.
4. Remove the load resistor R_L and connect the circuit as per CIRCUIT-2 and Note down the reading of voltmeter as V_{Th}.
5. Connect the Thevenin's equivalent circuit as shown in CIRCUIT-3 and Note down the reading of ammeter.

6. If current through the load resistance using Thevenin's theorem is equal to the measured value of the current from circuit1, Thevenin's Theorems is verified.

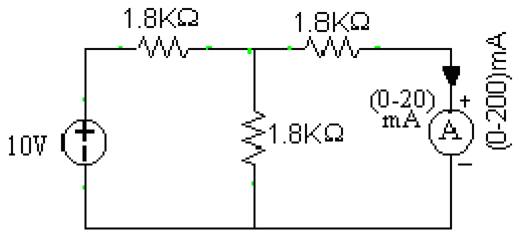
RESULT :

CONCLUSION :

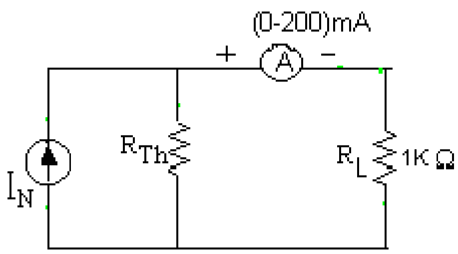
CIRCUIT DIAGRAM:



CIRCUIT – 1



CIRCUIT – 2



CIRCUIT - 3

OBSERVATIONS :

Theoretical:

Short circuit current I_N =

Norton's Equivalent Resistance R_N =

Current through the load resistor I_L =

Measured:

Short circuit current I_N =

Norton's Equivalent Resistance R_N =

Current through the load resistor I_L =

Part - B

NORTON'S THEOREM

AIM :

To verify Norton's Theorems for a linear network.

APPARATUS:

	NAME	RANGE	QUANTITY
1.	Bread Board.		
2.	Resistors	1.8KΩ	3 No.s
		1 KΩ	1 No
3.	Voltmeter	(0-20) V	1 No
4.	Ammeter	(0-20mA)	1 No
5.	Multi meter		1 No.
6.	Connecting wires.		

THEORY :

Statement:

Any linear bilateral network containing one or more current sources can be replaced by an equivalent circuit consisting of an equivalent current source, in parallel with an equivalent resistance. Value of current source is equal to the short circuit current through the output terminals with the load resistance shorted. The Norton's equivalent resistance is equal to the effective resistance measured between the output terminals with the load resistance removed, and with all the energy sources replaced by their internal resistances.

PROCEDURE:

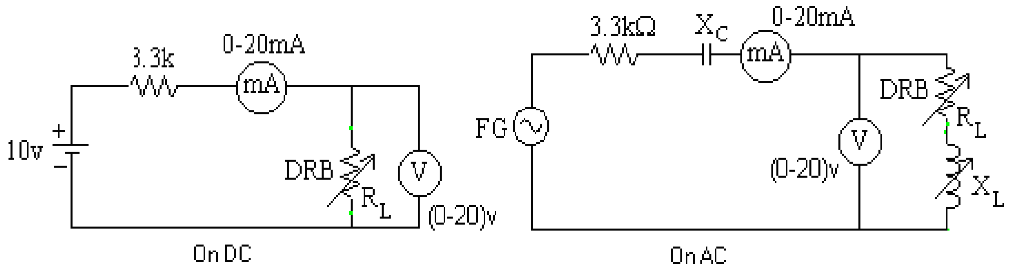
1. Connect the circuit as per CIRCUIT-1. Note down the current through the load resistance.
2. Calculate the value of short circuit current, Norton's equivalent resistance and the current through the load resistance using Norton's theorem.
3. Find out R_N by shorting the voltage source and measuring the equivalent resistance across open circuited R_L (1KΩ) terminals using a multimeter. Compare this value with the calculated value.
4. Remove the load resistor R_L and connect the circuit as per CIRCUIT-2 and Note down the reading of ammeter as I_N
5. Connect the Norton's equivalent circuit as shown in CIRCUIT-3 and Note down the reading of ammeter.

6. If current through the load resistance using Norton's theorem is equal to the measured value of the current from circuit1, Norton's Theorems is verified.

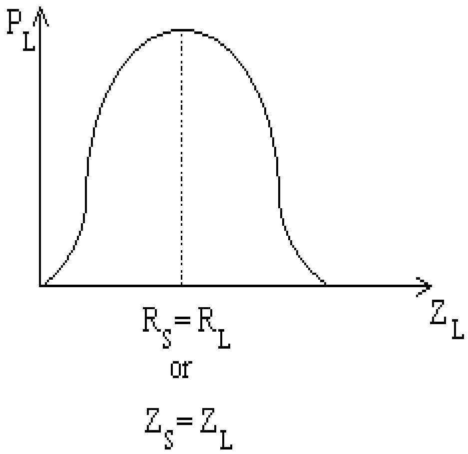
RESULT :

CONCLUSION :

CIRCUIT DIAGRAM:



EXPECTED GRAPH:



Part - C

MAXIMUM POWER TRANSFER THEOREM

AIM :

To verify the Maximum Power Transfer theorem on DC and AC.

APPARATUS :

1. Resistor - 3.3KΩ.
2. Resistance Box
3. Inductor - 45mH.
4. Inductance Box.
5. Capacitor - 0.01μF
6. AC milli Ammeter - 0-20mA
7. AC Voltmeter - 0-20 V
8. DC milli Ammeter - 0-20mA
9. DC Voltmeter - 0-20V
10. Function Generator
11. Regulated Power Supply.
12. Connecting wires.

THEORY :

Statement:

D.C.:- The maximum power is said to be delivered from the source to the load when the load resistance is equal to the source resistance.

For the given circuit maximum power delivered to the load is given by

$$P_{max} = \frac{V_s^2}{4R_L}$$

A.C.:- The maximum power is said to be delivered to the load when the source impedance is complex conjugate of load impedance.

$$P_{max} = \frac{V_s^2}{4R_L}$$

The maximum power transfer theorem finds its application in a radio speaker system supplying the input signals to voltage pre-amplifiers it is necessary to transfer maximum voltage, current or power to the load.

TABULAR FORM:

On DC

S.No	R_L (Ω)	I (A)	V (V)	P_L (W)

On AC

S.No	R_L (Ω)	Z_L (Ω)	I (A)	V (V)	P_L (W)

PROCEDURE:

On DC:-

1. Connect the circuit as shown in diagram.
2. Apply 10V DC from the RPS.
3. Take the readings of the milli Ammeter and voltmeter while varying R_L in suitable steps.
4. Tabulate the readings and plot the graph.
5. Verify the maximum power transfer theorem.

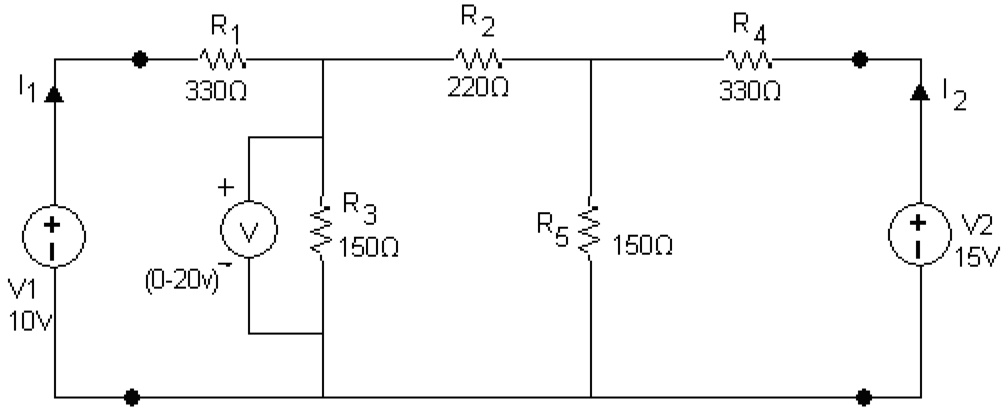
On AC:-

1. Connect the circuit as shown in the diagram.
2. Apply 20v (pp) from the FG.
3. Keep X_L=X_C. Take the readings of milli Ammeter and voltmeter while varying the R_L in suitable steps.
4. Plot the graph P_L Vs X_L.
5. Verify that the maximum power transfer occurs at the values predicted by the theorem.

RESULT :

CONCLUSION :

CIRCUIT DIAGRAM :



2. Super position Theorem and RMS value of complex wave.

Part - A
SUPERPOSITION THEOREM

AIM :

To verify the Super position Theorem on DC circuit.

APPARATUS :

	Name	Range	Quantity
1.	Bread Board.		
2.	Resistors	150Ω	2 No.s
		330 Ω	2 No.s
		150 Ω	1 No.
3.	M.C. Voltmeter	0-20 V	1 No.

Theory :

Statement : The Super position theorem states that in any linear network containing two or more sources, the response in any element is equal to the sum of the responses caused by individual sources acting alone while the other sources are made inoperative.

We define a linear element as a passive element that has a linear voltage-current relationship. i.e. multiplying the time-varying current through the element by a constant K results in the multiplication of the time-varying voltage across the element by the same constant K.

$V(t) = R\,I\,(t)$.

PROCEDURE :

1. Connect the circuit as shown in circuit diagram and Note down the reading of the voltmeter as V.
2. Short-circuit the voltage source V_I and Note down the reading of voltmeter as V_I.
3. Now short-circuit the voltage source V_2, keeping V_I in the circuit and Note the reading of voltmeter as V^{II}.
4. If $V = V^{I} + V^{II}$, Super position theorem is verified.

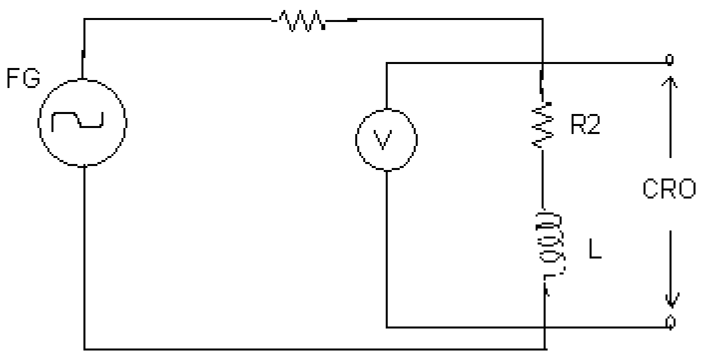
OBSERVATIONS :

$V =$ $\qquad$ $V^{I} =$ $\qquad$ $V^{II} =$

RESULT :

CONCLUSION :

CIRCUIT DIAGRAM :



OBSERVATIONS & CALCULATIONS :

Peak value (V)	Calculated RMS value (V)	Measured value (V)

Part - B

RMS VALUE OF COMPLEX WAVE

AIM :

To calculate the RMS value of a complex wave.

APPARATUS :

	Name	Range	Quantity
1.	Resistors	100Ω	2 Nos
2.	Inductor	1 mH	1 No
3.	Function Generator		1 No
4.	Multimeter		1 No
5.	CRO		1 No

THEORY :

RMS (Root Mean Square) value of an ac wave is the mean of the root of the square of the voltages at different instants. For an ac wave it will be $1/\sqrt{2}$ times the peak value.

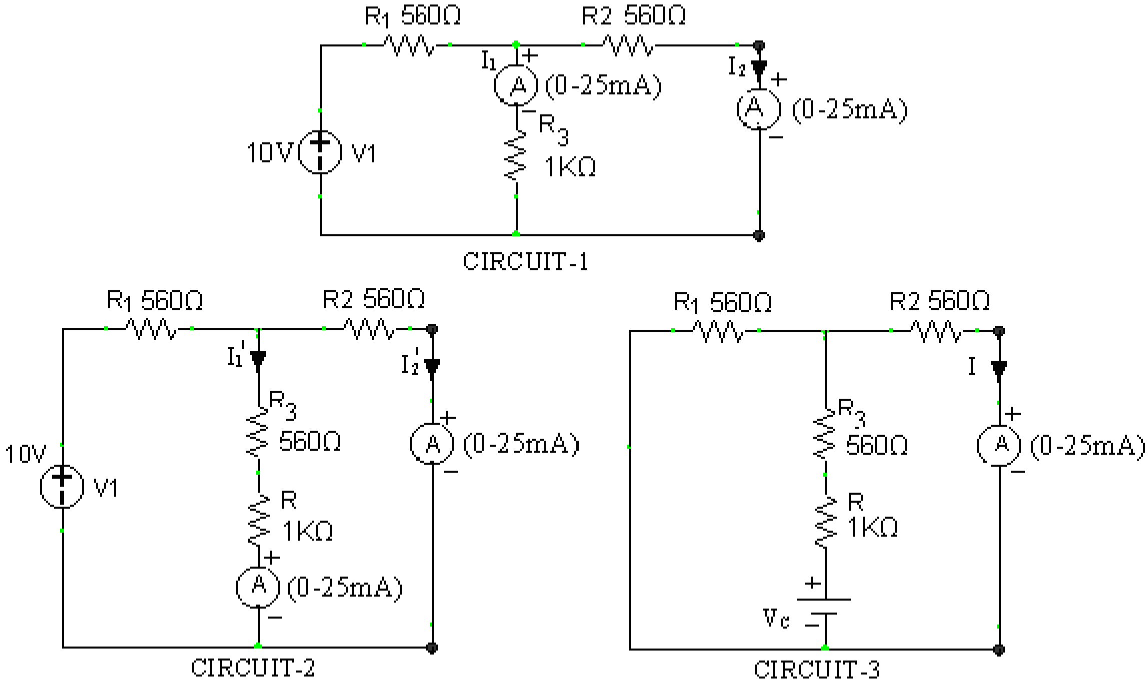
PROCEDURE :

1. Connect the circuit as per the circuit diagram.
2. Apply the sinusoidal wave as input from the Function Generator.
3. Observe the output waveform in the CRO. Note down the peak value of the output wave, from the CRO.
4. Calculate the RMS value and compare with the measured value.
5. Switch OFF the supply.

RESULT :

CONCLUSION :

CIRCUIT DIAGRAM:



OBSERVATIONS:

I_1 (mA)	I_2 (mA)	I'_1 (mA)	I'_2 (mA)	Vc (V)	Calculated I (mA)	Measured I (mA)

3. VERIFICATION OF COMPENSATION THEOREM

AIM :

To verify the Compensation Theorem.

APPARATUS :

NAME	RANGE	QUANTITY
Bread Board.		
Resistors	1K	3 No.s
	560Ω	1 No
Ammeter	(0-25mA) MC	2 Nos

THEORY :

Compensation theorem states that any element in the linear ,bilateral network can be replaced by a voltage source of magnitude equal to the current passing through the element multiplied by the value of current , provided the currents and voltages of the other parts of the circuit remain unaltered.

This theorem is useful in finding the changes in current or voltage when the value of resistance is changed in the circuit. If the resistance of any branch of a network is changed from R to (R+▲R) where the current flowing in that branch originally is I, the change of current in the other branches can be calculated by placing a voltage source of the value I(▲R) in the modified branch with all the other sources made ineffective. This theorem is particularly useful in analyzing the networks where the values of the branch elements are varied and for studying the effect of tolerance on such values.

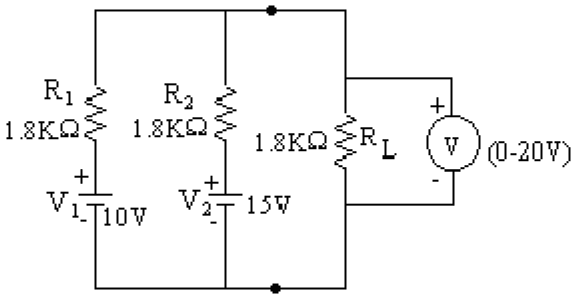
PROCEDURE :

1. Connect the circuit as shown in CIRCUIT-1, Note down the values of I_1 and I_2 using milli Ammeters.
2. Connect the circuit as shown in CIRCUIT-2, Note down the value of $I_2^|$.
3. Connect the circuit as shown in CIRCUIT-3, where V_C(Compensating voltage)=($I_2^|$ - I_2) 560Ω.
4. Note down the reading of ammeter as I.
5. If I = $I_2^|$ - I_2 , Compensating Theorem is verified.

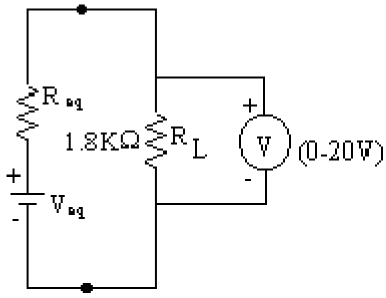
RESULT :

CONCLUSION:

CIRCUIT DIAGRAM:



CIRCUIT-1



CIRCUIT-2

OBSERVATIONS:

V_{L1} (V)	V_{L2} (V)

4. RECIPROCITY, MILLMAN'S THEOREMS
PART - A
MILLMAN'S THEOREM

AIM :

To verify the Millman's Theorem.

APPARATUS :

	NAME	RANGE	QUANTITY
1.	Bread Board.		
2.	Resistors	1.8KΩ	3No.s
3.	Voltmeter	(0-20)V	1 No.

THEORY :

Millman's theorem states that in any network, if the voltage sources V_1, V_2, ….. … V_n in series with internal resistances R_1, R_2,…. R_n respectively are in parallel, then these sources may be replaced by a single voltage source V_{eq} in series with R' where value of the voltage source V_{eq} can be given by

$$V_{eq} = \frac{V_1 G_1 + V_2 G_2 + \ldots\ldots.. V_n G_n}{G_1 + G_2 + \ldots.. G_n}$$

Where Gn is the conductance of the nth branch,

And $R' = 1 / (G_1 + G_2 + \ldots\ldots + G_n)$

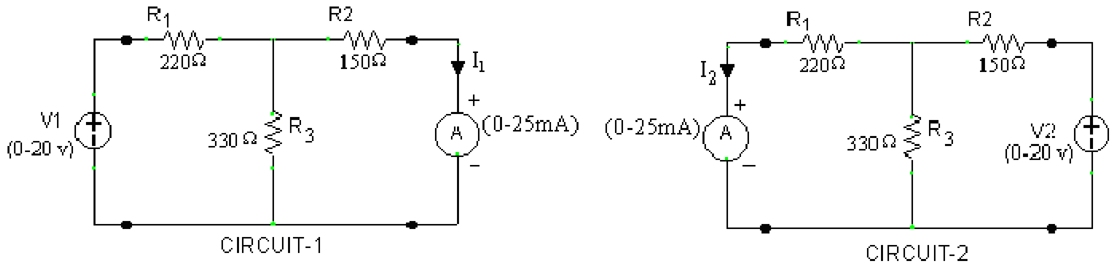
PROCEDURE :

1. Connect the circuit as shown in CIRCUIT-1 and Note down the reading of voltmeter as V_{L1}.
2. Connect the equivalent circuit as shown in CIRCUIT-2 , by calculating
3. $V_{eq}=(V_1G_1 +V_2G_2)/ (G_1 + G_2)$ and $R_{eq}= 1/ (G_1+G_2)$
4. Note down the reading of the voltmeter as V_{L2}.
5. If $V_{L1} = V_{L2}$, the Milliman's Theorem is verified.

RESULT :

CONCLUSION :

CIRCUIT DIAGRAM :



OBSERVATIONS :

I_1 (mA)	I_2 (mA)

PART - B

RECIPROCITY THEOREM

AIM :

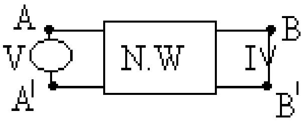
To verify the Reciprocity Theorem.

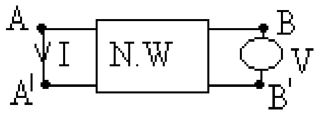
APPARATUS :

	NAME		RANGE	QUANTITY
1.	Bread Board			
2.	Resistors	-	150Ω, 220Ω, 330Ω.	I No each
3.	M.C.Ammeter	-	(0-20) mA	I No

THEORY :

Statement: In any bilateral, linear network, if we apply some input to a circuit the ratio of response (output) in any element to the input is constant even when the position of the input and output are interchanged.





Another way of stating the above is that the receiving and sending points are interchangeable.

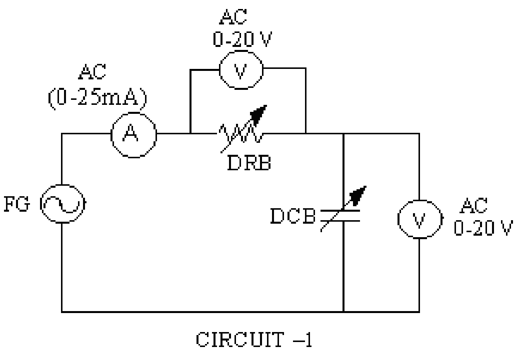
PROCEDURE :

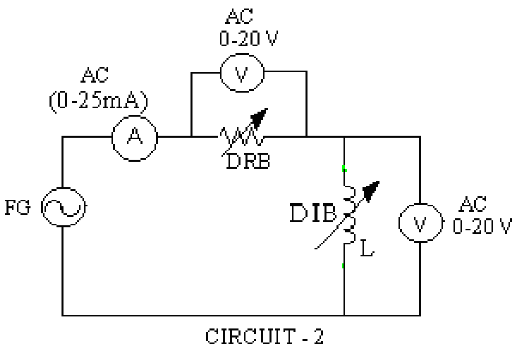
1. Connect the circuit as shown in CIRCUIT-1 and take the reading of Ammeter as I_1.
2. Now change the voltage source to the right hand side as shown in CIRCUIT-2 and measure the current with the help of Ammeter as I_2.
3. If $I_1 = I_2$, then the Theorem is verified.

RESULT:

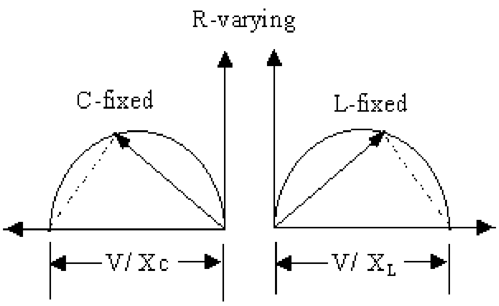
CONCLUSION :

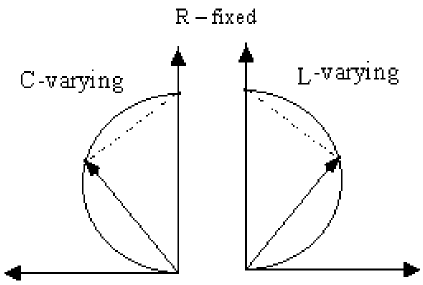
CIRCUIT DIAGRAM :





EXPECTED WAVE FORMS:





5. LOCUS DIAGRAMS OF RL AND RC SERIES CIRCUITS.

AIM :

To Plot the current locus diagrams for RL and RC circuits.

APPARATUS :

1. Resistance Box
2. Inductance Box
3. Capacitance Box
4. Ammeter AC -- (0-20mA) --- 1 No
5. Volt meters AC -- (0-20V) --- 2 No.s
6. Function Generator
7. CRO

THEORY :

Locus diagrams are useful in determining the behaviour or response of an RLC circuit, when one of its parameters is varied while the frequency and voltage are kept constant. The magnitude and phase of the current vector in the circuit depend upon the values of R, L and C and frequency at the fixed source voltage. The path travelled by the tip of the current vector when the parameters R, L or C are varied while frequency and voltage are kept constant is called the locus diagram.

R-varying :

If R = 0, then I = V/ X_L or V/ X_C and has maximum value. It will lag or lead the voltage by 90^0 depending on whether the reactance is inductive or capacitive. The angle θ represents the phase angle ($\theta = \tan^{-1}(V_X/ V_R)$). As R is increased from zero value, I and θ decrease. In the limiting case when R= ∞, then I = 0 and θ = 0^0. The locus of end-point qitl is a semi-circle of radius V / X.

R-Fixed :

If X = 0, then I = V/ R and has maximum value. The current will be in phase with voltage as it is a purely resistive circuit i.e., the phase θ is zero. As X is increased depending on whether the reactance is inductance or apacitance the current starts lagging or leading V i.e., the current I decreases and phase angle θ increases. In the limiting case when X = ∞, then I = 0 and θ = 90^0. The locus of end point of I is a semi-circle of radius V/ R.

TABULAR COLOUMN :

f = **$V_{i/p}$ =**

Circuit – 1

R – fixed					
S.No	R (Ω)	C (μF)	V_R (V)	V_C (V)	I (mA)
C – fixed					
S.No	R (Ω)	C (μF)	V_R (V)	V_C (V)	I (mA)

Circuit – 2

R – fixed					
S.No	R (Ω)	C (μF)	V_R (V)	V_L (V)	I (mA)
L – fixed					
S.No	R (Ω)	C (μF)	V_R (V)	V_L (V)	I (mA)

PROCEDURE :

Circuit – 1:

1. Connect the circuit as shown in CIRCUIT – 1. Note down the values of applied voltage and frequency.
2. Fix the resistance at a suitable value (say 1KΩ).
3. Note down the values of V_R, V_C and I for at least three different values of the Capacitor.
4. Now fix the capacitance at a suitable value (say 0.01μF).
5. Note down the values of V_R, V_C and I for at least three different values of the resistor.
6. Draw the corresponding current locus diagrams.

Circuit – 2 :

1. Connect the circuit as shown in CIRCUIT – 2. Note down the values of applied voltage and frequency.
2. Fix the resistance at a suitable value (say 1KΩ).
3. Note down the values of V_R, V_L and I at least three different values of the Inductor.
4. Now fix the Inductance value at a suitable value (say 45mH).
5. Note down the values of V_R, V_L and I at least three different values of the resistor.
6. Draw the corresponding current locus diagrams.

RESULT :

CONCLUSION :

6. SIMULATION OF DC CIRCUITS

AIM :

To simulate a simple DC circuits using PSpice

APPARATUS : PC (in working conduct) with PSpice software.

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.

Result :

Conclusion :

7. DC TRANSIENT RESPONSE

AIM :

To simulate a simple DC circuits using PSpice

APPARATUS : PC (in working conduct) with PSpice software.

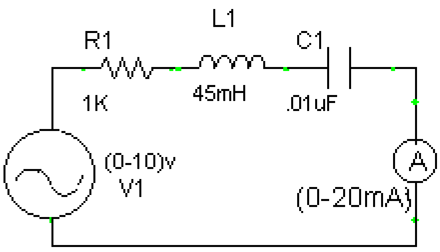
Procedure :

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.

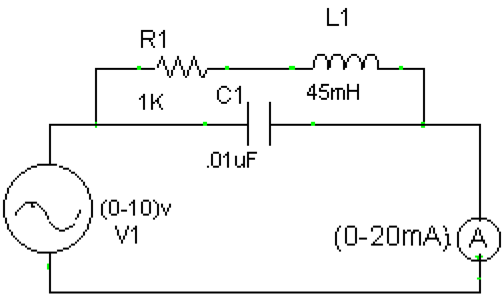
Result :

Conclusion :

CIRCUIT DIAGRAM :

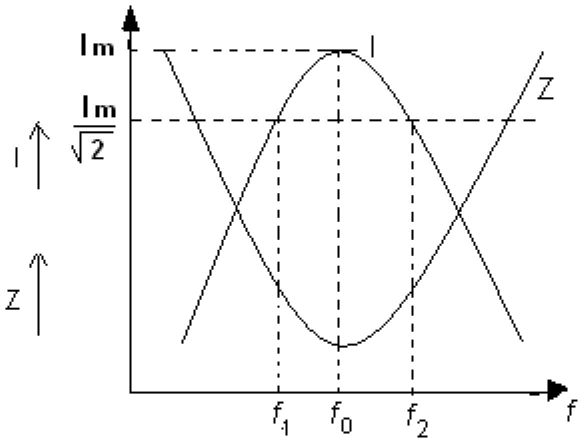


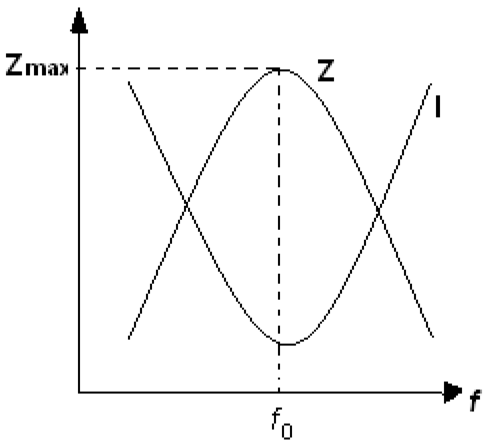
CIRCUIT –1



CIRCUIT-2

EXPECTED GRAPH :





8. SERIES AND PARALLEL RESONANCE

AIM :

To determine resonant frequency, band width and Q-factor for series and parallel RLC circuits

APPARATUS :

	NAME	RANGE	QUANTITY
1.	Resistor	1KΩ	1 No.
2.	Inductor	45mH	1 No
3.	Capacitor	0.01μF	1 No
4.	Milli Ammeter	0-20mA (AC)	1 No
5.	Function generator		

THEORY :

An AC circuit is said to be in Resonance when the applied voltage and current are in phase. Resonance circuits are formed by the combination of reactive elements connected in either series or parallel.

Resonance frequency in series circuit is given by $f_r = 1/(2\pi\sqrt{LC})$ Hz

The impedance of the RLC circuit is

$$Z = R + j(\omega_L - 1/\omega_C) = R + jX$$

The circuit is in resonance when X = 0 ie., when $\omega_L = 1/\omega_C$

In series RLC circuit the current lags behind or leads the applied voltage depending upon the value of X_L and Xc. When X_L is greater than Xc the circuit is inductive and when X_c is greater than X_L, the circuit is capacitive.

Quality factor (Q-factor) or (Selectivity) :

Quality factor can be defined as ,

= 2 π (maximum energy stored)/ (energy dissipated per cycle).

$$= (f_2 - f_1) / f_r$$

Band width: Band width of a resonance circuit is defined as the band of frequencies on either sides of resonance frequency. This frequency range can be obtained by dropping a vertical in the graph at its half power value, i.e., $1/\sqrt{2}$ times of maximum value.

$$\text{Band width} = f_2 - f_1$$

TABULAR COLOUMN :

Series

S. No	I/P V	Frequency, Hz	I_L, Amp	Z=V/I_L Ω

Parallel

S. No	V i/p	Frequency, Hz	I_L, Amp	Z=V/I_L Ω

CALCULATIONS:

For Series Resonance :

$$f_0 = \frac{1}{(2\pi\sqrt{LC})}$$

$$f_1 = f_0 - (R/4\pi L)$$

$$f_2 = f_0 + (R/4\pi L)$$

$$\text{Band width} = f_2 - f_1$$

Q-factor =

__For Parallel Resonance :

$$f_0 = \frac{1}{2\pi}\sqrt{\frac{1}{LC} - \frac{R^2}{L^2}}$$

$$f_1 = f_0 - (R/4\pi L)$$

$$f_2 = f_0 + (R/4\pi L)$$

$$\text{Band width} = f_2 - f_1$$

Q-factor =

THEORETICAL CALCULATIONS :

Series

$$f_0 = \frac{1}{(2\pi\sqrt{LC})}$$

$$f_1 = f_0 - (R/4\pi L)$$

$$f_2 = f_0 + (R/4\pi L)$$

$$\text{Band width} = f_2 - f_1$$

Q-factor =

Parallel

$$f_0 = \frac{1}{2\pi}\sqrt{\frac{1}{LC} - \frac{R^2}{L^2}}$$

$$f_1 = f_0 - (R/4\pi L)$$

$$f_2 = f_0 + (R/4\pi L)$$

$$\text{Band width} = f_2 - f_1$$

Q-factor =

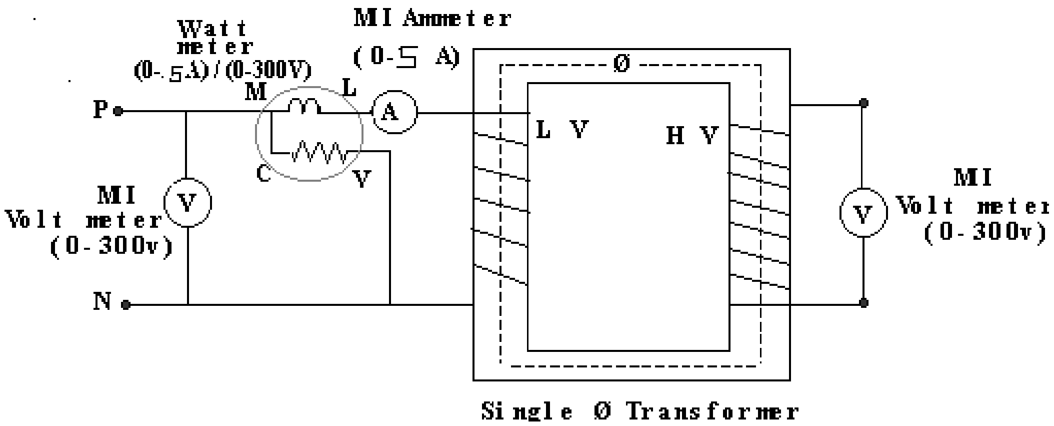
PROCEDURE :

1. Connect the circuit as shown in diagram(1).
2. Apply 20V (peak to peak) from the Function Generator.
3. Vary the input frequency in suitable steps (starting from 1K Hz to 10K Hz in step of 500 Hz).
4. Note down the readings of the milli Ammeter for different values of frequency.
5. Calculate the Impedance Z.
6. Plot the graphs for current Vs frequency and Z Vs frequency.
7. Identify the values of f_0 , f_1 and f_2 from the graph, Calculate the Q-factor and Band width.
8. Compare with theoretical values.
9. Connect the circuit as per diagram(2).
10. Repeat steps (2) & (3).
11. Note down the readings of the voltmeter and milliammeter for different frequencies.
12. Calculate the Impedance Z.
13. Plot the graphs for current Vs frequency and Z Vs frequency
14. Also plot the graph of Voltage Vs Frequency.
15. Identify the values of f_0 , f_1 and f_2 from the graph, Calculate the Q-factor and Band width.

RESULT :

CONCLUSION :

CIRCUIT DIAGRAM :



OBSERVATIONS:

When connected to LV side :

S No.	Voltage V_1 (volt)	Current I (Amp)	Power W (watt)	Voltage V_2(volt)

When connected to HV side :

S No.	Voltage V_1 (volt)	Current I (Amp)	Power W (watt)	Voltage V_2(volt)

CALCULATIONS :

When connected to LV side :

Power factor $\cos \Phi = W / V_L * I$

Power factor angle $\Phi = \cos^{-1} (W / V_L * I)$

Magnetizing current $I\mu = I * \sin \Phi$

Self Inductive reactance $X_{L1} = V_L / I\mu$

Self Inductance $L_1 = X_{L1} / 2 \pi f$

9. DETERMINATION OF SELF & MUTUAL INDUCTANCES AND COEFFICIENT OF COUPLING

AIM:

To determine the self and mutual inductances and coefficient of coupling for two inductive coils.

APPARATUS :

Name	Type / Range	Quantity
Single phase variac	5 KVA, 230/ (0 – 270) V	1 No
Volt meter	MI , (0 – 300) V	2 No
Ammeter	MI , (0 – 5) A	1 No
Wattmeter	LPF , 300 V , 5 A	1 No.

THEORY :

The property by which a coil opposes any change in the current passing through it is known as self inductance. Whenever current passes through an inductor, it produces a magnetic field around the coil and if the current is alternating it produces an emf in the coil. Thus the self inductive reactance and the self inductance can be found out by measuring the emf induced and the current required to produce it.

Mutual inductance is the property by which a coil opposes any change in the current passing through a neighbouring coil. Thus the mutual inductive reactance and the mutual inductance can be found out by measuring the emf induced in the neighbouring coil and the current required to produce it.

The amount of coupling between the inductively coupled coils is expressed in terms of the coefficient of coupling.

PROCEDURE :

(1) Connect the apparatus as per the circuit diagram.

(2) Adjust the variac and apply a voltage of 115 V at primary.

(3) Note down the voltmeter , ammeter and wattmeter readings in the primary and secondary sides.

(4) Interchange the HV and LV sides.

Mutual Inductive reactance = $X_{M12} = V_H / I\mu$

Mutual Inductance $M_{12} = X_{M12} / 2\pi f$

When connected to HV side :

Power factor $\cos\Phi = W / V_H * I$

Power factor angle $\Phi = \cos^{-1}(W / V_H * I)$

Magnetizing current $I\mu = I * \sin\Phi$

Self Inductive reactance $X_{L2} = V_H / I\mu$

Self Inductance $L_1 = X_{L1} / 2\pi f$

Mutual Inductive reactance = $X_{M12} = V_L / I\mu$

Mutual Inductance $M_{12} = X_{M12} / 2\pi f$

Coefficient of Coupling , $K = M / \sqrt{(L_1 * L_2)}$

(5) Note down the voltmeter , ammeter and wattmeter readings in the primary and secondary sides.

(6) Switch off the supply and disconnect the circuit.

RESULT :

CONCLUSION:

CIRCUIT DIAGRAM:

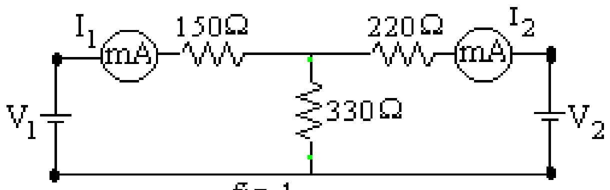


fig-1

OBSERVATIONS & CALCULATIONS :

For Z-parameters

When V_2 is open circuited,

V_1 (volt)	V_2 (volt)	I_1 (A)	I_2 (A)

$Z_{11} = V_1 / I_1$ ($I_2 = 0$)

$Z_{21} = V_2 / I_1$ ($I_2 = 0$)

When V_1 is open circuited,

V_1 (volt)	V_2 (volt)	I_1 (A)	I_2 (A)

$Z_{12} = V_1 / I_2$ ($I_1 = 0$)

$Z_{21} = V_1 / I_1$ ($I_1 = 0$)

For Y -parameters

When V_2 is short circuited,

V_1 (volt)	V_2 (volt)	I_1 (A)	I_2 (A)

$Y_{11} = I_1 / V_1$ ($V_2 = 0$) =

$Y_{21} = I_2 / V_1$ ($V_2 = 0$)

When V_1 is short circuited,

V_1 (volt)	V_2 (volt)	I_1 (A)	I_2 (A)

$Y_{12} = I_1 / V_2$ ($V_1 = 0$)

$Y_{22} = I_2 / V_2$ ($V_2 = 0$)

10. Z & Y PARAMETERS

AIM :

To determine the Z, and Y parameters of a Two-port network.

APPARATUS :

	Name	Type / Range	Quantity
1.	Resistors	- 150Ω, 220Ω and 330Ω	each 1 No.s
2.	milli Ammeter	- (0-20mA)	2 No.s
3.	Voltmeter	- (0-20v)	1 No
4.	Regulated power Supply	- 30 V, 2A	1 No.
5.	Connecting wires.		

THEORY :

A network is having two pairs of accessible terminals, it is called a two port network. If voltage and current at the input and output terminals are V_1, I_1 and V_2, I_2 respectively, there are six sets of possible combinations generated by the four variables, describing a two - port network. Z - parameters and Y- parameters are two among them.

Using Z- parameters the circuit can be represented by the following equations

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

Using Y- parameters the circuit can be represented by the following equations

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

PROCEDURE :

Z- parameters:-

1. Connect the circuit as shown in diagram – 1.

For Z_{11} and Z_{21} :

1. Make $I_2 = 0$ by open circuiting the V_2 and Apply $V_1 = 10V$.
2. Note down the readings of V_1, V_2 and I_1.
3. Calculate Z_{11} and Z_{21} .
4. Verify with theoretical values.

For Z_{12} and Z_{22} :

1. Make $I_1 = 0$ by open circuiting the V_1 and Apply $V_2 = 15V$.
2. Note down the readings of V_1, V_2 and I_2.
3. Calculate Z_{12} and Z_{22}.
4. Verify with theoretical values.

<u>Y- parameters</u> :-

1. Connect the circuit as shown in diagram – 1.

For Y_{11} and Y_{21} :

1. Make $V_2 = 0$ by short circuiting it and Apply V_1=10v.
2. Note down the readings of I_1, I_2 and V_1.
3. Calculate Y_{11} and Y_{21} .
4. Verify with theoretical values.

For Y_{12} and Y_{22} :

1. Make $V_1 = 0$ by short circuiting it and Apply V_2 = 15v.
2. Note down the readings of I_1, I_2 and V_2.
3. Calculate Y_{12} and Y_{22}.
4. Verify with theoretical values.

CIRCUIT DIAGRAM :

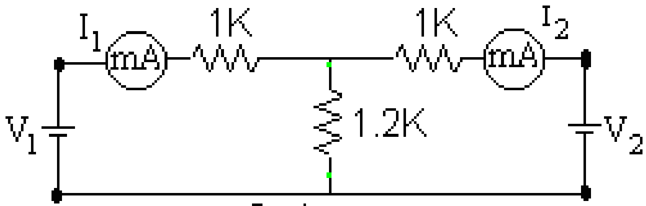


fig-1

OBSERVATIONS & CALCULATIONS :

For T-parameters

When V_2 is short circuited,

V_1 (volt)	V_2 (volt)	I_1 (A)	I_2 (A)

$B = V_1 / I_2 \; (V_2 = 0)$

$D = I_1 / I_2 \; (V_2 = 0)$

When V_2 is open circuited,

V_1 (volt)	V_2 (volt)	I_1 (A)	I_2 (A)

$A = V_1 / V_2 \; (I_2 = 0)$

$C = I_1 / V_2 \; (I_2 = 0)$

For h -parameters

When V_2 is short circuited,

V_1 (volt)	V_2 (volt)	I_1 (A)	I_2 (A)

$h_{11} = V_1 / I_1 \; (V_2 = 0)$

$h_{21} = I_2 / I_{21} \; (V_2 = 0)$

When V_1 is open circuited,

V_1 (volt)	V_2 (volt)	I_1 (A)	I_2 (A)

$h_{12} = V_1 / V_2 \; (I_1 = 0)$

$h_{22} = I_2 / V_2 \; (I_1 = 0)$

11. TRANSMISSION AND HYBRID PARAMETERS

AIM :

To determine the Transmission and Hybrid parameters of a Two-port network.

APPARATUS :

Name		Type / Range	Quantity
1. Resistors	-	150Ω, 220Ω and 330Ω	each 1 No.s
2. milli Ammeter	-	(0-20mA)	2 No.s
3. Voltmeter	-	(0-20v)	1 No
4. Regulated power Supply	-	30 V, 2A	1 No.
5. Connecting wires.			

THEORY :

A network is having two pairs of accessible terminals, it is called a two port network. If voltage and current at the input and output terminals are V_1, I_1 and V_2, I_2 respectively, there are six sets of possible combinations generated by the four variables, describing a two - port network. Transmission- parameters and Hybrid- parameters are two among them.

Using T- parameters the circuit can be represented by the following equations

$$V_1 = A V_2 - B I_2$$

$$I_1 = C V_2 - D I_2$$

Where A, B, C, D are the transmission parameters.

Using h- parameters the circuit can be represented by the following equations

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

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

h_{11}, h_{12}, h_{21}, h_{22} are the hybrid parameters.

PROCEDURE :

T- parameters :-

a. Connect the circuit as shown in diagram – 1.

b. Make $V_2 = 0$ by short-circuiting it and Apply V_1=10V.

c. Note down the readings of V_1, I_2 and I_1.

d. Calculate the values of B and D.

e. Make $I_2 = 0$ by open-circuiting V_2 and Apply V_1=10V

f. Note down the readings of V_1, V_2 and I_1.

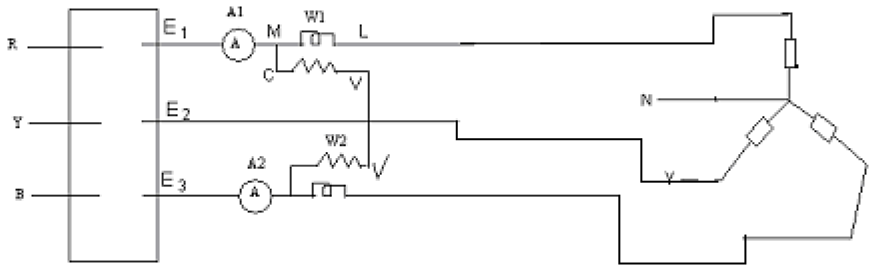
g. Calculate the values of A and C

h. Verify with theoretical values.

h – parameters :-

1. Connect the circuit as shown in diagram – 1.
2. Make $V_2 = 0$ by short-circuiting it and Apply V_1=10V.
3. Note down the readings of V_1, I_2 and I_1.
4. Calculate the values of h_{11} and h_{21}.
5. Make I_1=0 by open-circuiting V_1 and Apply V_2=10V
6. Note down the readings of V_1, V_2 and I_1.
7. Calculate the values of h_{12} and h_{22}.
8. Verify with theoretical values.

CIRCUIT DIAGRAM :



OBSERVATIONS & TABULAR COLUMN:

Phase Voltage ,V =

S No.	A_1 (A)	A_2 (A)	W_1 (W)	W_2 (W)	W (W)

12. MEASUREMENT OF 3 PHASE POWER BY 2 WATTMETER METHOD

AIM:

To measure the active power consumed by a 3 phase load, using 2 wattmeter method.

APPARATUS:

NAME	RANGE	QUANTITY
Wattmeter	10A/600V	2 Nos
Ammeter	(0-10)A	2Nos
Voltmeter	(0 -600V)	1 No
3-phase Auto transformer		
3-phase load		

THEORY:

A circuit is said to be unbalanced when the impedance in one or more phases differ from the impedances of the other phases.In such cases phase or line currents are different and are displaced from one another by unequal angles.

In two wattmeter method, we connect the current coil of the wattmeters in two different phases of the 3 phase circuit and the pressure coil will be connected between that particular phase and the third phase. The total power consumed by the load

$$W = W1 + W2$$

PROCEDURE:

1) Connect the circuit as shown in the circuit diagram. Ensure that the autotransformer is in the minimum position and the load applied is zero.
2) Switch ON the supply. Note down the meter readings.
3) Increase the load gradually and each time note down the meter readings.
4) Calculate the active power from the two wattmeter readings.
5) Gradually decrease the load to zero and switch OFF the supply.

RESULT :

CONCLUSION:

13. MESH ANALYSIS

AIM :

To simulate a simple DC circuits using PSpice

APPARATUS : PC (in working conduct) with PSpice software.

Procedure :

1. Open PSpice A/D windows
2. Create a new circuit file
3. Enter the program representing the nodal intercommoning variations comport
4. Run the program
5. Observe the Nodal voltages, current theory the elements in the output file
6. Observe the voltage, current graph in probe window.

Result :

Conclusion :

14. NODAL ANALYSIS

AIM :

To simulate a simple DC circuits using PSpice

APPARATUS : PC (in working conduct) with PSpice software.

Procedure :

1. Open PSpice A/D windows
2. Create a new circuit file
3. Enter the program representing the nodal intercommoning variations comport
4. Run the program
5. Observe the Nodal voltages, current theory the elements in the output file
6. Observe the voltage, current graph in probe window.

Result :

Conclusion :