TITLE: LIST OF EXPERIMENTS

1. Verification of KCL & KVL (i) Experiment (ii) Simulation

2. Verification of Maximum Power Transfer Theorem (i) Experiment (ii) Simulation

3. i. Verification of Reciprocity Theorem (i) Experiment (ii) Simulation
   ii. Verification of Superposition Theorem (i) Experiment (ii) Simulation

4. Verification of Thevenin's Theorem (i) Experiment (ii) Simulation

5. Verification of Norton's Theorem (i) Experiment (ii) Simulation

6. Determination of Self & Mutual Inductance

7. RLC Series & Parallel Resonance

8. RL & RC Locus Diagrams

9. i.Verification of compensation theorem (i) Experiment (ii) Simulation
   ii. Verification of Milliman's theorem   (i) Experiment (ii) Simulation

10. Determination of average value, rms value, peak and phase values of AC System.

11. Determination of Z & Y Parameters

12. Determination of ABCD Parameters

13. Digital simulation of an electric circuit (Including dependant sources) to find node voltages and branch currents using PSpice

14. Digital simulation of an electric circuit to find transient response

15. Digital simulation of series and parallel resonance using PSpice

----

TITLE: VERIFICATION OF KCL	GPREC/EEE/EXPT.No. ECP-01
& KVL	DATE:18-10-2021

OBJECTIVE: To verify Kirchhoff's voltage law and Kirchhoff's current law by (a) Experiment (b) Simulation in PSpice software.

APPARATUS:

Sl.No.	Unit	Range	Type	No.
1	Regulated power supply	(0-30)V/(0-2)A		1
2	Digital Multimeter			1
3	Ammeter	(0-2)A	MC	4
4	Rheostats			4
5	Connecting Wires			Reqd.

KCL STATEMENT: *Kirchoff's current law states that the algebraic sum of currents at a node or junction is equal to zero. It can also be stated as the sum of currents entering a node is equal to sum of currents coming out of that node.*

THEORY- KCL: This law is also called **Kirchhoff's first law**, **Kirchhoff's point rule**, or **Kirchhoff's junction rule** or **nodal rule**.

Consider the circuit shown in Fig.1 below. It can be seen that the currents flowing through the branches are I_1, I_2, I_3 whereas the net current is given by *I*. So, using Kirchoff's current law, we can write
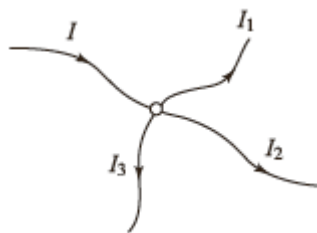
$$I = I_1 + I_2 + I_3$$

Fig.1

TITLE: VERIFICATION OF KCL	GPREC/EEE/EXPT.No. ECP-01
& KVL	DATE:18-10-2021

APPLICATIONS:

1. Matrix version of Kirchhoff's current law is the basis of most circuit simulation software, such as PSPICE.

2. Kirchhoff's current law combined with Ohm's Law is used in nodal analysis.

3. KCL is applicable to any lumped network irrespective of the nature of the network; whether unilateral or bilateral, active or passive, linear or non-linear.

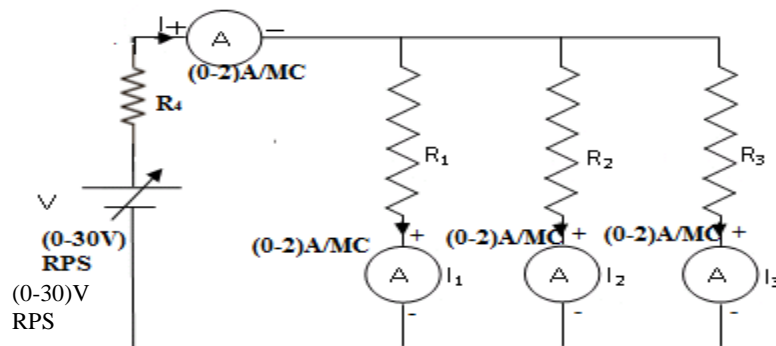**CIRCUIT DIAGRAM:**
   a. **Experiment**
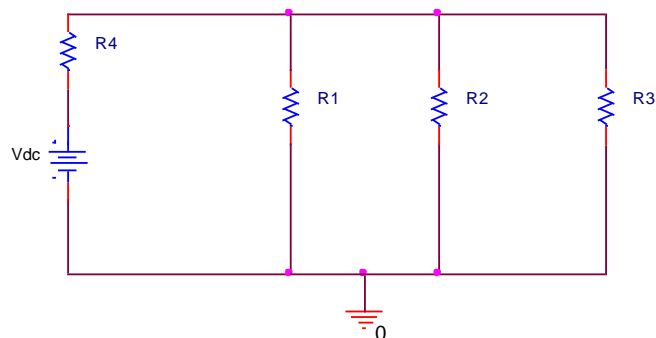

Fig. 1 KCL CIRCUIT DIAGRAM

   b. **Simulation in PSpice**

Fig. 2 Simulation Model of KCL in PSpice

TITLE: VERIFICATION OF KCL	GPREC/EEE/EXPT.No. ECP-01
& KVL	DATE:18-10-2021

PROCEDURE FOR KCL:

1. Connections are made as per the circuit diagram shown in Fig.2 & Fig.3

2. By varying the input voltage note down all meter readings.

3. Verify KCL.

KVL STATEMENT: *Kirchoff's voltage law (KVL) states that the algebraic sum of voltages in a closed loop is equal to zero.*

THEORY - KVL:

As current flows through resistor there will be a voltage drop. This voltage drop of all resistors along with the source voltage should be zero. Consider the circuit shown in Fig.4. Two resistors R_1 and R_2 are connected to each other and are supplied from a voltage source of *V* volts. Assume that a current of *I* amps flows through the circuit.
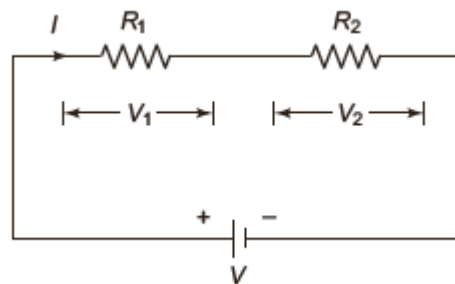

Fig.4 KVL circuit diagram

The value of current *I* flowing through the resistors is the same in the above circuit. Due to the flow of current there will be a voltage drop in both the resistors. Let the voltage drops in both the resistors be V_1 and V_2. So, in order to write the voltage equations along with the polarities the following notation can be considered.

(i) Starting at one end of the circuit (preferably the ground terminal) follow the current directionand note down the first sign that comes in that direction along with that value of voltage.

(ii) Follow the same steps for the remaining elements also.

Prepared by	Approved by	Page 3 of 7
Dr.G.Venkata Subba Reddy	Dr. T.Bramhananda Reddy	Revision No.0
	HEAD, EEE Dept.	

TITLE: VERIFICATION OF KCL	GPREC/EEE/EXPT.No. ECP-01
& KVL	DATE:18-10-2021

Using the above methodology, the voltage equations may be written as

$$-V + V_1 + V_2 = 0$$

But $V_1 = IR_1$ and $V_2 = IR_2$, which implies

$$-V + IR_1 + IR_2 = 0$$

Or

$$IR_1 + IR_2 = V$$

Or

$$I = \frac{V}{R_1 + R_2}$$

APPLICATIONS OF KVL:

1. KVL is used to determine the values of unknown currents and their direction.

2. Useful to find the unknown values in complex circuits and networks (not suitable for high frequency circuit).

3. Kirchhoff's Laws are useful in understanding the transfer of energy through an electric circuit.
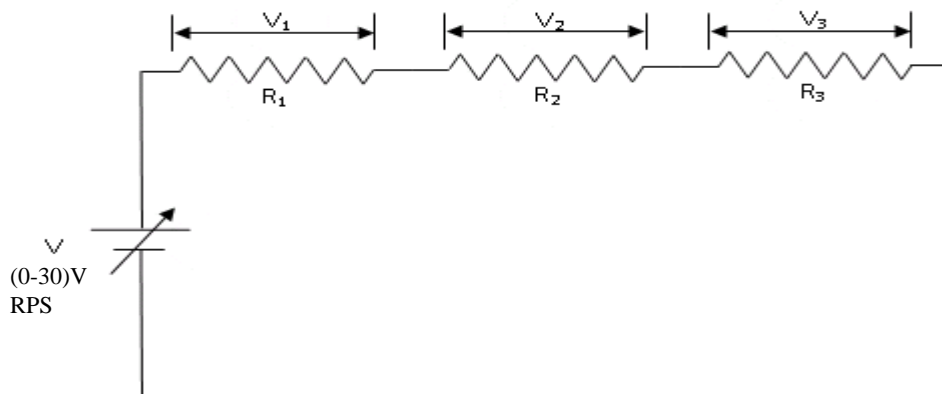
a. **Experiment**

Fig. 3 KVL CIRCUIT DIAGRAM

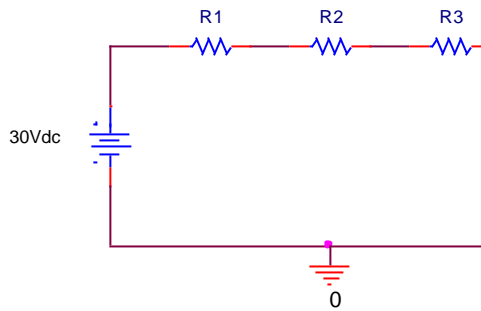| **TITLE:** VERIFICATION OF KCL	GPREC/EEE/EXPT.No. ECP-01
& KVL	DATE:18-10-2021

b.Simulation in PSpice

Fig. 4 Simulation Model of KVL in PSpice

PROCEDURE FOR KVL:

1. Connections are made as per the circuit diagrams shown in Fig.5 & Fig.6.
2. By varying the input voltage note down all meter readings.
3. Verify KVL.

TABULAR COLUMN:

    FOR KCL:

S.No.	V(V)	I(A)		I_1(A)		I_2(A)		I_3(A)		$I_1 + I_2 + I_3$ (A)	
		Th	Pr	Th	Pr	Th	Pr	Th	Pr	Th	Pr

Prepared by   Approved by   Page 5 of 7
Dr.G.Venkata Subba Reddy   Dr. T.Bramhananda Reddy   Revision No.0
                           HEAD, EEE Dept.

TITLE: VERIFICATION OF KCL	GPREC/EEE/EXPT.No. ECP-01
& KVL	DATE:18-10-2021

FOR KVL:

S.No.	V (V)		V₁(V)		V₂(V)		V₃(V)		V₁ +V₂ +V₃ (V)	
	Th	Pr	Th	Pr	Th	Pr	Th	Pr	Th	Pr

RESULT: Kirchhoff's laws are verified.

TITLE: VERIFICATION OF KCL
& KVL

GPREC/EEE/EXPT.No. ECP-01

DATE:18-10-2021

Questions on KCL & KVL:

1. Define KVL
2. Define KCL
3. Define Ohm's law
4. What are the applications of KVL and KCL?
5. In nodal analysis which law will be used?
6. In mesh analysis which law will be used?

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 7 of 7
Revision No.0

TITLE: VERIFICATION OF MAXIMUM POWER TRANSFER THEOREM	GPREC/EEE/EXPT.No. ECP-02 DATE: 18-10-2021

OBJECTIVE: To **v**erify Maximum Power Transfer theorem by (a) Simulation in PSpice software (b) Experiment.

APPARATUS:

Sl.No.	Unit	Range	Type	No. Required
1	Voltmeter	(0-100V)	MC	1
2	Ammeter	(0-5)A	MC	1
3	Rheostat		-	2
4	Connecting Wires			Reqd.

STATEMENT: *Maximum power transfer theorem states that the power delivered by an active network to a load connected across its terminals is maximum, when the impedance of the load is the complex conjugate of the active network impedance.*

THEORY:

In this theorem we shall consider different cases

Case (i): When the load is purely resistive

Consider a network delivering power to the load as shown in figure given below
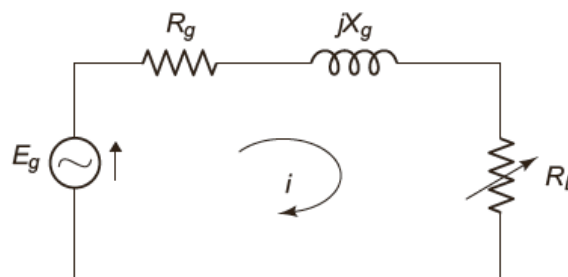

Fig. 1

Current delivered to the load is $i = \dfrac{E_g}{(R_g + R_L) + jX_g}$

TITLE: VERIFICATION OF MAXIMUM POWER TRANSFER THEOREM	GPREC/EEE/EXPT.No. ECP-02 DATE: 18-10-2021

Power delivered to the load is $i^2 R_L$

$$P = \frac{E_g^2 R_L}{(R_g + R_L)^2 + X_g^2} \text{ watts}$$

The condition when the power delivered to the load is maximum, can be formed if

$$\frac{dP}{dR_L} = 0$$

$$\frac{d}{dR_L}\left[\frac{E_g^2 R_L}{(R_g + R_L)^2 + X_g^2}\right] = 0$$

Simplifying we get $R_L^2 = R_g^2 + X_g^2$ or $R_L = \sqrt{R_g^2 + X_g^2}$

Therefore load resistance $R_L = Z_g$

If $X_g = 0$ then $R_L = R_g$

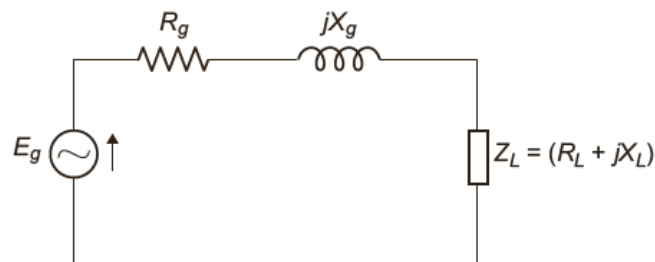**Case (ii)** Let the load be an impedance with fixed resistance and variable reactance

Fig. 2

The current i is given as $i = \dfrac{E_g}{(R_g + R_L) + j(X_g + X_L)}$

The power delivered to the load is given as $P = \dfrac{E_g^2 R_L}{(R_g + R_L)^2 + (X_g + X_L)^2}$

TITLE: VERIFICATION OF MAXIMUM POWER TRANSFER THEOREM	GPREC/EEE/EXPT.No. ECP-02
	DATE: 18-10-2021

The condition can be found by using $\dfrac{dP}{dX_L} = 0$

i.e. $\dfrac{d}{dX_L}\left[\dfrac{E_g^2 R_L}{(R_g + R_L)^2 + (X_g + X_L)^2} \right] = 0$

Solving this we get

$$X_L = -X_g$$

The reactance of the load is of the same magnitude as the reactance of the network but is of opposite sign.

Case (iii) Let the load be an impedance with fixed reactance and variable resistance

The power $P = \dfrac{E_g^2 R_L}{(R_g + R_L)^2 + (X_g + X_L)^2}$

The condition can be found by $\dfrac{dP}{dR_L} = 0$

$$\dfrac{d}{dR_L}\left[\dfrac{E_g^2 R_L}{(R_g + R_L)^2 + (X_g + X_L)^2} \right] = 0$$

Solving this we get

$$R_L = (R_g + jX_g) + jX_L \quad or \quad R_L = Z_g + jX_L$$

General case: If the load is an impedance with variable reactance and resistance, then the maximum power would be delivered when $Z_L = Z_g$ i.e $R_L = R_g \quad and \quad X_L = -X_g$

Procedure to verify Maximum Power Transfer Theorem

i. In order to verify maximum power transfer theorem the network is reduced in such a way that the network consists of a single source, active network impedance and the load impedance.

TITLE: VERIFICATION OF MAXIMUM POWER TRANSFER THEOREM	GPREC/EEE/EXPT.No. ECP-02 DATE: 18-10-2021

ii. The condition for maximum power transfer is then verified by checking at the condition when the impedance of the load is the complex conjugate of the active network impedance.

Application:

1. In communication system, maximum power transfer is always sought. For example in public address system, the circuit is adjusted for maximum power transfer by making load resistance (speaker) equal to the source resistance (amplifier). When source and load have the same resistance, they are said to be matched.

2. In car engines, the power delivered to the starter motor of the car will depend upon the effective resistance of the motor and the internal resistance of the battery. If the two resistances are equal, maximum power will be transferred to the motor to turn to the engine.

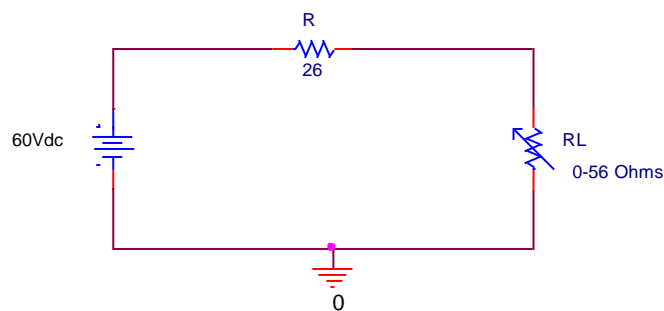**CIRCUIT DIAGRAM:**

a. **Simulation in PSpice**

Fig. 3

PROCEDURE:

1. Simulation model is prepared according to the circuit.

2. Load resistance is varied in steps and voltage, current and power values are tabulated.

3. Graph is drawn between load resistance and power, to find the load resistance value where maximum power transfer takes place from source to load.

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 4 of 7
Revision No.0

| TITLE: VERIFICATION OF MAXIMUM POWER TRANSFER THEOREM | GPREC/EEE/EXPT.No. ECP-02 |
| | DATE: 18-10-2021 |

TABULAR COLUMN:

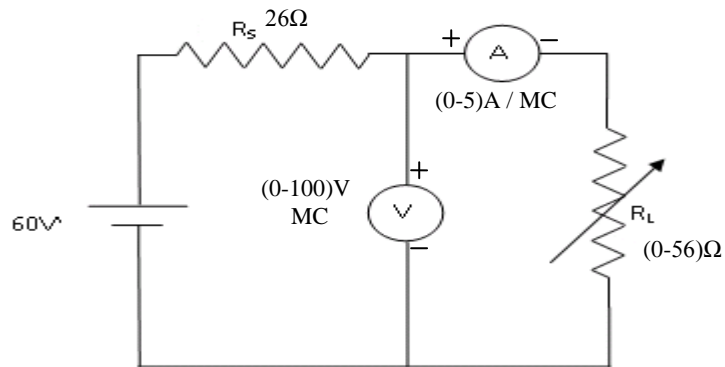| R	V	I	P	R_L

b. **Experiment**

Fig. 4

PROCEDURE:

1. Connections are made as per the circuit diagram.
2. Keep the load resistance (R_L) at maximum position.
3. Apply a DC voltage of 60V.
4. Vary R_L in steps and note all the meter readings.
5. Calculate the power (P) drawn by the load for different values of R_L and tabulate.
6. Plot a graph, Power Vs load resistor R_L. Where $P = I^2 R_L$; $I = \dfrac{V_S}{R_S + R_L}$
7. Observe that the maximum power is drawn when the load resistance is equal to the source resistance.

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 5 of 7
Revision No.0

TITLE: VERIFICATION OF MAXIMUM POWER TRANSFER THEOREM	GPREC/EEE/EXPT.No. ECP-02 DATE: 18-10-2021

TABULAR COLUMN:

S.No.	V	I	P	R_L

EXPECTED GRAPH:
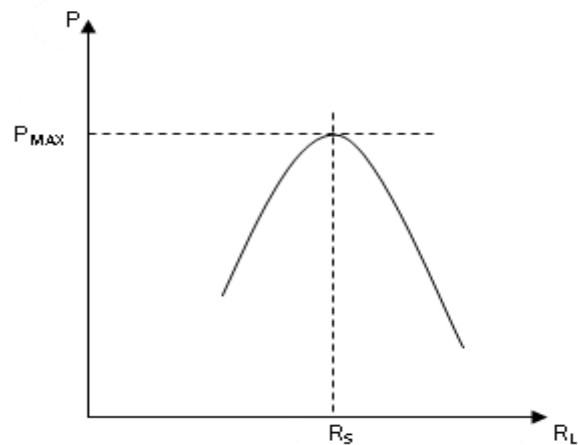
Fig. 5

RESULT: Maximum power transfer theorem is verified.

TITLE: VERIFICATION OF MAXIMUM POWER TRANSFER THEOREM	GPREC/EEE/EXPT.No. ECP-02 DATE: 18-10-2021

QUESTIONS

1. Define maximum power transfer theorem.

2. What is the condition, at which the power transferred from source to load is maximum, when the load is pure resistive.

3. What is the condition, at which the power transferred from source to load is maximum, when the load is impedance with fixed resistance and variable reactance.

4. What is the condition, at which the power transferred from source to load is maximum, when the load is impedance with fixed reactance and variable resistance.

5. What is the condition, at which the power transferred from source to load is maximum, when the load is impedance with resistance and reactance are variable.

6. What are the applications of Maximum power transfer theorem?

TITLE: VERIFICATION OF RECIPROCITY THEOREM	GPREC/EEE/EXPT.No. ECP-03 DATE: 18-10-2021

OBJECTIVE: To Verify Reciprocity theorem by (a) Experiment (b) Simulation in PSpice software.

APPARATUS:

Sl.No.	Unit	Range	Type	No.
1	Regulated Power Supply	(0-30)V/(0-2)A		1
2	Ammeter	(0-2)A	MC	1
3	Rheostat			3
4	Digital Multimeter			1
	Connecting Wires			Reqd.

STATEMENT: *In a linear bilateral network, the ratio of excitation to response is equal in the case even though the positions of excitation and response are interchanged. However, if the excitation is a voltage source, the response must be a current and vice-versa.*
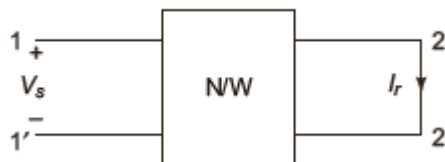
THEORY:

Considering two port networks as shown in Fig.1.

Fig. 1

The reciprocity theorem can be illustrated using this network. Considering the network has k loops and the network is excited by V_s (voltage source) at s_{th} loop and current should be determined at r_{th} loop then

$$I_r = \frac{A_{sr}}{D_z}V_s \text{ which implies } \frac{V_s}{I_r} = \frac{D_z}{A_{sr}} \tag{a}$$

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 1 of 6
Revision No.0

TITLE: VERIFICATION OF RECIPROCITY THEOREM	GPREC/EEE/EXPT.No. ECP-03 DATE: 18-10-2021

where A_{sr} is a cofactor

D_z is the impedance of the network

$\dfrac{A_{sr}}{D_z}$ is the admittance

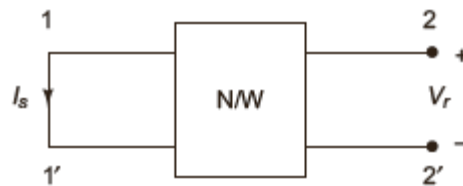As we interchange the positions of response and excitation, the circuit is

Fig. 2

$I_s = \dfrac{A_{rs}}{D_z} V_r$ which implies

$$\dfrac{V_r}{I_s} = \dfrac{D_z}{A_{rs}} \qquad\qquad\qquad (b)$$

From equations a and b

$\dfrac{V_s}{I_r} = \dfrac{D_z}{A_{sr}}$ and $\dfrac{V_r}{I_s} = \dfrac{D_z}{A_{rs}}$

If A_{sr} and A_{rs} are equal we can have a symmetric network with RLC elements and no controlled sources

$$\dfrac{V_s}{I_r} = \dfrac{V_r}{I_s}$$

Hence the theorem can be proved

Procedure to verify Reciprocity theorem

  i. In order to verify reciprocity theorem consider a network in which excitation is placed at terminals $11^{'}$ and response is to be found out at $22^{'}$.

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 2 of 6
Revision No.0

TITLE: VERIFICATION OF RECIPROCITY THEOREM	GPREC/EEE/EXPT.No. ECP-03 DATE: 18-10-2021

ii. The response is found at the terminals of $22'$ using various techniques discussed earlier and noted

iii. The excitation is now shifted to across the terminals $22'$ and the response is found out across the terminals $11'$.

iv. If the ratio of excitation to response in both the cases is considered and is found to be the same, then reciprocity theorem is said to be satisfied.
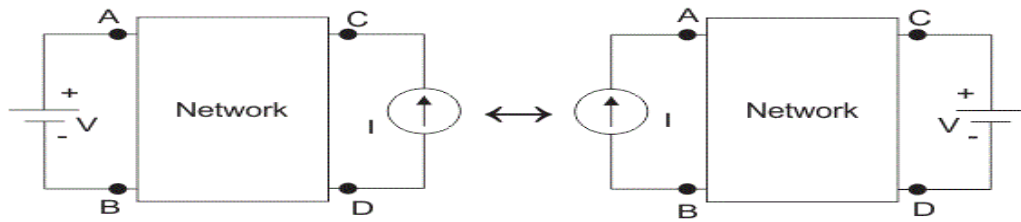

Fig. 3

CIRCUIT DIAGRAM:

a. **Simulation Model**
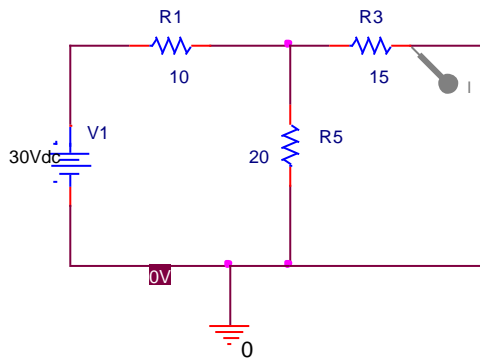
Fig.4. Pspice Model

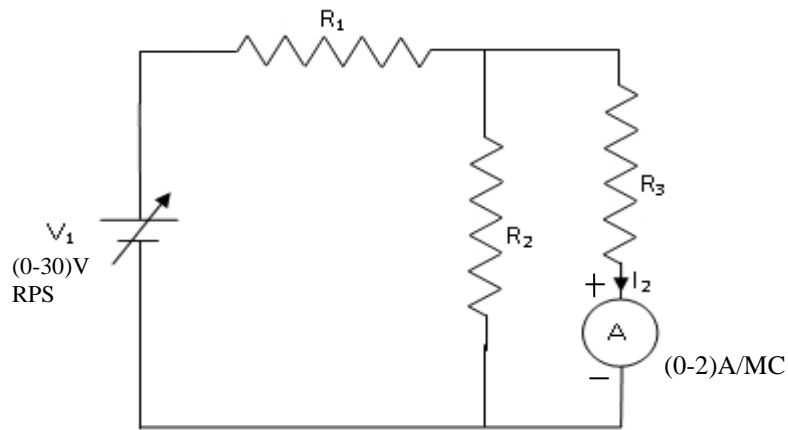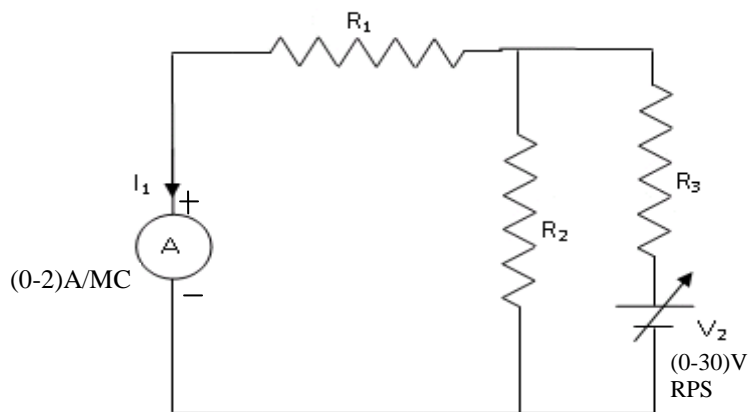|	
TITLE: VERIFICATION OF RECIPROCITY THEOREM	GPREC/EEE/EXPT.No. ECP-03 DATE: 18-10-2021
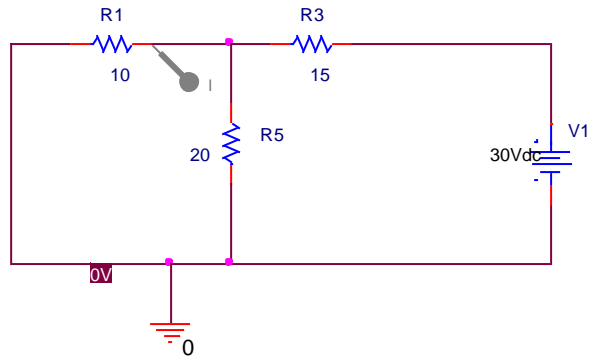
Fig.5. Pspice Model

b. Experiment

Fig.6

Fig.7

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 4 of 6
Revision No.0

TITLE: VERIFICATION OF RECIPROCITY THEOREM	GPREC/EEE/EXPT.No. ECP-03 DATE: 18-10-2021

PROCEDURE:

1. Connections are made as per the circuit diagram shown in Fig.6.
2. Apply voltage in steps.
3. Note down meter readings and calculate the ratio of voltage to current.
4. Repeat the procedure for the circuit shown in Fig.7 and calculate the ratio.
5. Verify that the ratio remains same in both the procedures.

TABULAR COLUMNS:

Table.1 for Fig.6

S.No.	V	I		V / I	
		Th.	Pr.	Th.	Pr.

Table.1 for Fig.7

S.No.	V	I		V / I	
		Th.	Pr.	Th.	Pr.

RESULT: Reciprocity theorem is verified.

TITLE: VERIFICATION OF	GPREC/EEE/EXPT.No. ECP-03	
RECIPROCITY THEOREM	DATE: 16-08-2021	

Questions:

1. Define reciprocity theorem.

2. What are the applications of reciprocity theorem?

3. What are the limitations of reciprocity theorem?

TITLE: VERIFICATION OF SUPERPOSITION THEOREM	GPREC/EEE/EXPT.No. ECP-03 DATE: 18-10-2021

OBJECTIVE: To Verify Superposition theorem by (a) Experiment (b) Simulation in PSpice software.

APPARATUS:

Sl.No.	Unit	Range	Type	No.
1	Regulated power supply	(0-30)V/(0-2)A		2
2	Ammeter	(0-2)A	MC	1
3	Rheostat			3
4	Connecting Wires		-	Reqd.

STATEMENT: *In any linear network with several independent sources and dependent sources, the overall response in any part of the network is equal to the sum of the individual responses due to each independent source with all other independent sources reduced to zero.*

(or)

"According to superposition theorem, in any linear network containing two or more sources, the response in any element is equal to the algebraic sum of the responses caused by the individual sources acting alone while the other sources are non-operative."

This means all independent voltages are shorted and all independent current sources are open circuited. If the sources contain internal impedances, the sources are replaced by the impedances.

THEORY: Superposition theorem is valid only for linear systems. Superposition theorem is not applicable to power. This theorem can be applicable to voltages and currents. The above statement can be proved by the following explanation.

In linear systems, currents are propositional to the voltages $I\alpha V$

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 1 of 6
Revision No.0

TITLE: VERIFICATION OF SUPERPOSITION THEOREM	GPREC/EEE/EXPT.No. ECP-03 DATE: 18-10-2021

In this theorem, we add individual currents of a branch due to each source taking at a time to get the resultant current. This is similar to voltages.

$\therefore$ We can have $I = I_1 + I_2 + ...$

$$V = V_1 + V_2 + ...$$

But $P \neq P_1 + P_2$.

Procedure to verify Superposition Theorem

   i.   While evaluating the superposition theorem only one source (voltage source or current source) is kept in the network while making the other sources equal to zero. This implies to each source present in the network

   ii. If there are 2 voltage sources, V_1, V_2 and one current source I in a network, first V_1 is considered to be present in the network while making the other voltage source and current source equal to zero

   iii. The corresponding parameter (current or voltage) is found in the particular element when V_1 is present.

   iv. In the next step V_2 is considered while making the other sources constant and the corresponding parameter in the particular element is found out

   v.   The same procedure is followed when the current source is considered

   vi. The values of parameter across the particular element obtained during various sources is added and verified with the value of parameter obtained when all the three sources are present in the network

   vii. The theorem is said to be satisfied if both the values are equal

CIRCUIT DIAGRAM:

   a.   **PSpice Simulation**

TITLE: VERIFICATION OF SUPERPOSITION THEOREM

GPREC/EEE/EXPT.No. ECP-03

DATE: 18-10-2021

Fig.1

Fig.2

Fig.3

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 3 of 6
Revision No.0

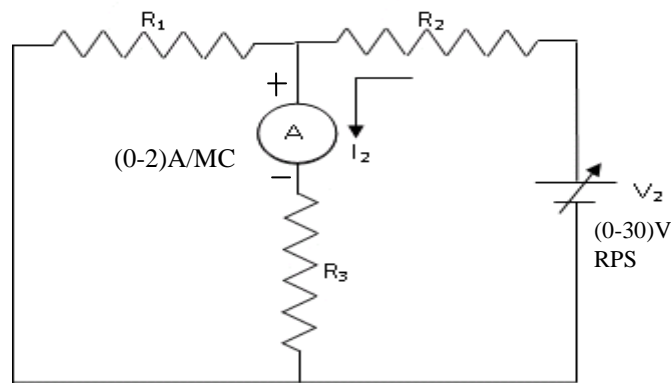| TITLE: VERIFICATION OF SUPERPOSITION THEOREM	GPREC/EEE/EXPT.No. ECP-03 DATE: 18-10-2021

b. **Experiment**

Fig.4

Fig.5

Fig.6

TITLE: VERIFICATION OF SUPERPOSITION THEOREM	GPREC/EEE/EXPT.No. ECP-03 DATE: 18-10-2021

PROCEDURE:

1. a. Connections are made as per the circuit diagram shown in Fig.4.

   b. By varying V_1 & V_2 in steps note down the ammeter reading (I).

2. a. Connections are made as per the circuit diagram shown in Fig.5.

   b. By varying V_1 in steps note down the ammeter reading (I_1).

3. a. Connections are made as per the circuit diagram shown in Fig.6.

   b. By varying V_2 in steps note down the ammeter reading (I_2).

4. Verify the superposition theorem ($I = I_1 + I_2$).

TABULAR COLUMNS:

Table.1 for Fig.4

Sl.no.	V_1(V)	V_2(V)	I(A)	
			Th.	Pr.

Table.2 for Fig.5

Sl.no.	V_1(V)	I_1(A)	
		Th.	Pr.

Table.3 for Fi.6

Sl.no.	V_2(V)	I_2(A)	
		Th.	Pr.

Table.4 To verify Superposition theorem

Sl.no.	V_1(V)	V_2(V)	I(A)		I=(I_1+ I_2)(A)	
			Th.	Pr.	Th.	Pr.

RESULT: **Superposition theorem is verified.**

TITLE: VERIFICATION OF GPREC/EEE/EXPT.No. ECP-03

SUPERPOSITION THEOREM DATE: 18-10-2021

Questions:

1. Define superposition theorem

2. Super position theorem is applicable for the circuits which consists of _____ of sources.

3. Why superposition theorem is not applicable for powers.

4. What are the applications of superposition theorem?

TITLE: VERIFICATION OF THEVENIN'S THEOREM	GPREC/EEE/EXPT.No. ECP-04	
	DATE: 18-10-2021	

OBJECTIVE: To verify Thevenin's theorem by (a) Simulation in PSpice software (b) Experiment.

APPARATUS:

Sl.No.	Unit	Range	Type	No.
1	Regulated power supply	(0-30)V/(0-2)A		1
2	Ammeter	(0-2)A	MC	1
3	Rheostat			4
4	Digital Multimeter			1
5	Connecting Wires			Reqd.

STATEMENT:

Thevenin's theorem states that any two terminal linear network having a number of sources and resistances can be replaced by a simple equivalent circuit consisting of a single voltage source in series with a resistance, where the value of the voltage source is equal to the open circuit voltage across the two terminals of the network, and resistance is equal to the equivalent resistance measured between the terminals with all the energy sources are replaced by their internal resistances.
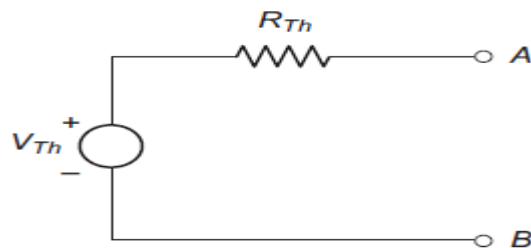

Fig.1 Thevenin's Equivalent Circuit

THEORY:

In electrical circuit theory, Thevenin's theorem for linear electrical networks states that any combination of voltage sources, current sources and resistors with two terminals is electrically equivalent to a single voltage source V and a single series resistor R. For

----

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 1 of 6
Revision No.0

TITLE: VERIFICATION OF THEVENIN'S THEOREM	GPREC/EEE/EXPT.No. ECP-04 DATE: 18-10-2021

single frequency AC systems, the theorem can also be applied to general impedances, not just resistors. Any complex network can be reduced to a Thevenin's equivalent circuit consist of a single voltage source and series resistance connected to a load.

Procedure to obtain Thevenin's equivalent circuit

i. Temporally remove the load resistance whose current is required.

ii. Find the open circuit voltage V_{OC} which appear across the two terminals from where the load resistance has been removed. It is also called Thevenin's voltage V_{th}.

iii. Calculate the resistance of the whole network as looked into from these two terminals, after all voltage sources are replaced by short circuit and current sources are replaced by open circuit leaving internal resistance (if any). It is also called Thevenin's resistance R_{th}.

iv. Replace the entire network by a single Thevenin's voltage source whose voltage is V_{th} and whose resistance is R_{th}.

v. Connect the load resistance (R_L) back to its terminals from where it was previously removed.

vi. Finally calculate the current flowing though R_L by using equation.

$$I_L = \frac{V_{th}}{R_{th} + R_L}$$

Applications of Thevenin's Theorem:

- Thevenin's Theorem is especially useful in analyzing power systems and other circuits where one particular resistor in the circuit (called the "load" resistor) is subject to change, and re-calculation of the circuit is necessary with each trial value of load resistance, to determine voltage across it and current through it.

- Source modeling and resistance measurement using the Wheatstone bridge provide applications for Thevenin's theorem.

TITLE: VERIFICATION OF	GPREC/EEE/EXPT.No. ECP-04
THEVENIN'S THEOREM	DATE: 18-10-2021

CIRCUIT DIAGRAM:

a. Simulation Model in PSpice
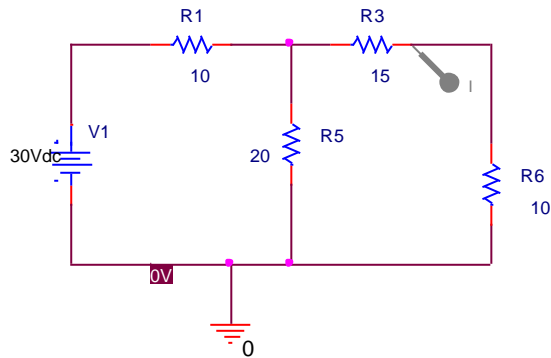
Fig.2
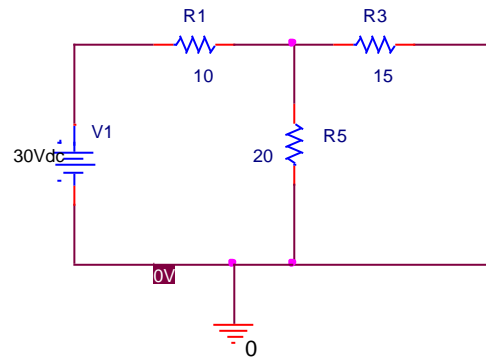
Fig.3
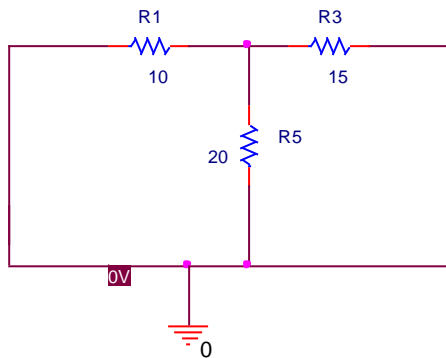
Fig.4
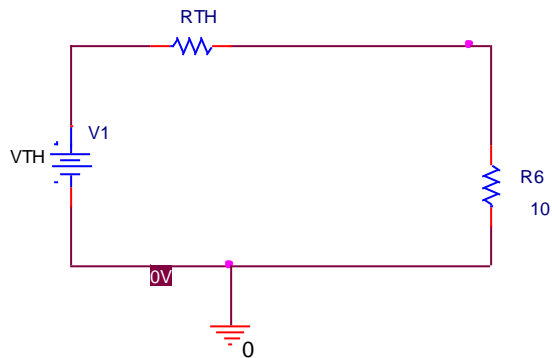
Fig.5

b. Experiment
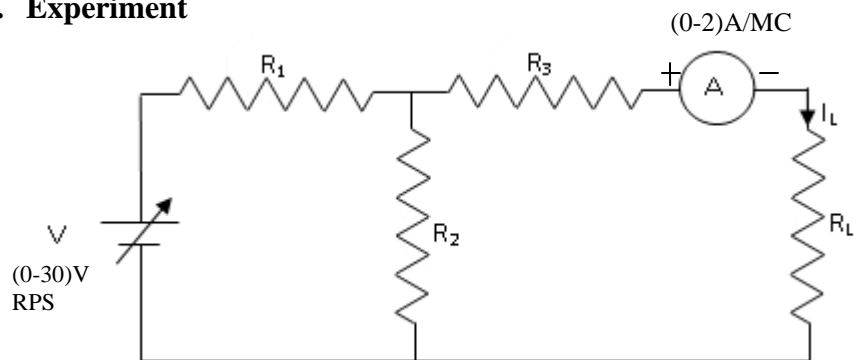
Fig.6

TITLE: VERIFICATION OF THEVENIN'S THEOREM	GPREC/EEE/EXPT.No. ECP-04 DATE: 18-10-2021
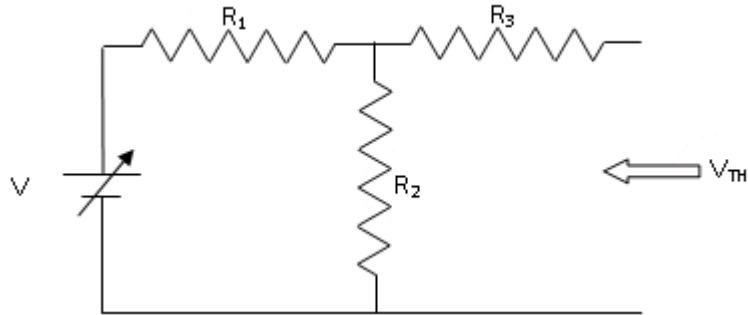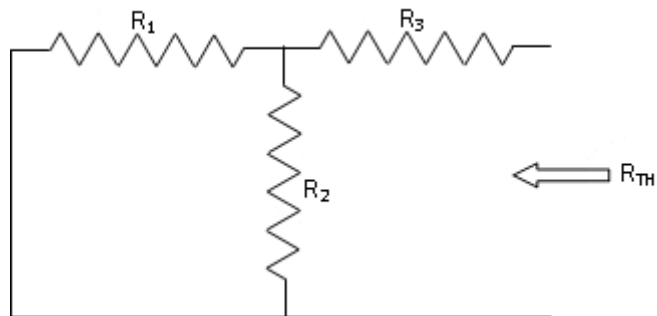
Fig.7

Fig.8
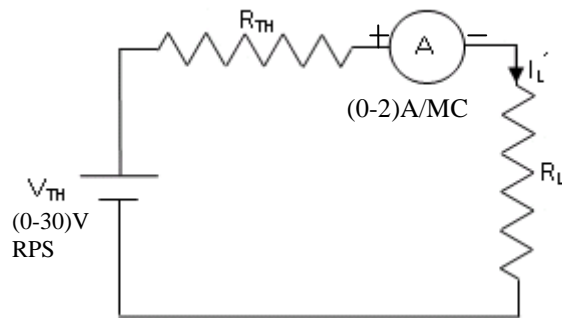
Fig.9

TABULAR COLUMN:

Table.1 for Fig.6

S.No.	V (V)	$I_L(A)$	
		Th.	Pr.

Table.2 for Fig.7

S.No.	V(V)	$V_{TH}(V)$	
		Th.	Pr.

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 4 of 6
Revision No.0

TITLE: VERIFICATION OF GPREC/EEE/EXPT.No. ECP-04

THEVENIN'S THEOREM DATE: 18-10-2021

Table.3 for Fig.8

$R_{TH}(\Omega)$

Table.4 for Fig.9

Sl.no.	$V_{TH}(V)$	$I_L'(A)$	
		Th.	Pr.

PROCEDURE:

1. a. Connections are made as per the circuit shown in Fig.6.

   b. By varying the voltage in steps note down the readings.

2. a. Connections are made as per the circuit shown in Fig.7.

   b. By varying the voltage in steps note down the readings.

3. Connections are made as per the circuit shown in Fig. 8 to find R_{TH}.

4. a. Connections are made as per the circuit shown in Fig. 9.

   b. Vary the voltage in steps of V_{TH} and note down the readings of I_L'.

The above procedure is followed for experiment also.

RESULT: Thevenin's theorem is verified.

TITLE: VERIFICATION OF THEVENIN'S THEOREM	GPRECD/EEE/EXPT.No.ECP-04
	DATE: 16-08-2021

QUESTIONS:

1. Define Thevenins theorem

2. What are the applications of Thevenins theorem.

3. Explain the procedure of solving the circuits using Thevenins theorem.

4. Draw the equivalent circuit of Thevenin's theorem.

5. State the difference between Thevenin's and Norton's theorem?

TITLE: VERIFICATION OF NORTON'S THEOREM	GPRECD/EEE/EXPT.No.ECP-05 DATE: 18-10-2021

OBJECTIVE: To verify Norton's theorem by (a) Experiment (b) Simulation in PSpice software.

APPARATUS:

Sl.No.	Unit	Range	Type	No.
1	Regulated power supply	(0-30)V/(0-2)A		2
2	Rheostat			4
3	Digital multimeter			1
4	Ammeter	(0-2)A	MC	1
5	Connecting Wires			Reqd.

NORTON'S THEOREM:

STATEMENT: *Norton's Theorem states that any two terminal linear network with current sources, voltage sources and resistances can be replaced by an equivalent circuit consisting of a current source in parallel with a resistance. The value of the current source is that short circuit current between the two terminals of the network and the resistance is the equivalent resistance measured between the terminals of the network with all the energy sources replaced by their internal resistance.*
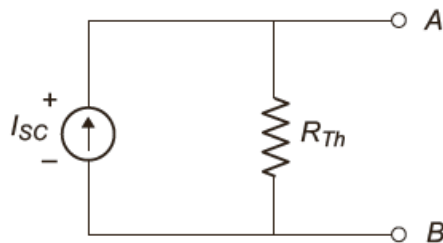

Fig. Norton's Equivalent Circuit

THEORY: This theorem is just alternative of Thevenin theorem.

Prepared by	Approved by	Page 1 of 6
Dr. G.Venkata Subba Reddy	Dr. T.Bramhananda Reddy HEAD, EEE Dept.	Revision No.0

TITLE: VERIFICATION OF NORTON'S THEOREM	GPRECD/EEE/EXPT.No.ECP-05 DATE: 18-10-2021

Procedure to obtain Norton's equivalent circuit

i. Temporarily remove the load resistance (if any) across the two given terminals and put a short circuit across them.

ii. Calculate the short circuit current I_{sc} or I_N

iii. Calculate the resistance of the whole network as looked into these two terminals, after all voltage sources are replaced by short circuit and current sources are replaced by open circuit leaving internal resistance (if any). It is called Thevenin's resistance R_{th} or Norton resistance R_N.

iv. Replace the entire network by a single Thevenin's current source whose short circuit current is I_{sc} or I_N and parallel with Thevenin's resistance R_{th} or R_N

v. Connect the load resistance (R_L) back to its terminals from where it was previously removed.

vi. Finally calculate the current flowing through R_L by using current division formula.

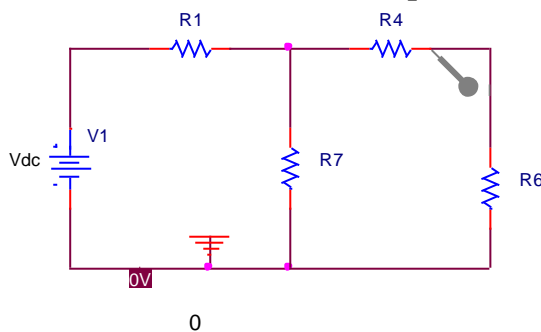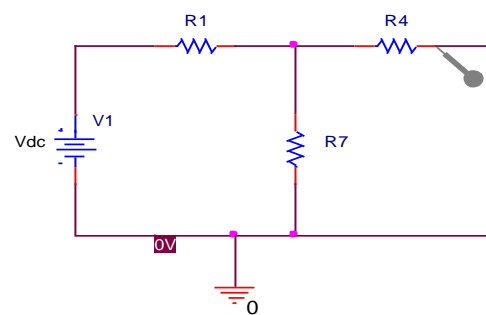**CIRCUIT DIAGRAM:**
  a. Simulation Model in PSpice

Fig.1   Fig.2

TITLE: VERIFICATION OF NORTON'S THEOREM

GPRECD/EEE/EXPT.No.ECP-05

DATE: 18-10-2021

Fig.3

Fig.4

b. Experiment

Fig.5

Fig.6

TITLE: VERIFICATION OF NORTON'S THEOREM

GPRECD/EEE/EXPT.No.ECP-05

DATE: 18-10-2021

Fig.7

Fig.8

TABULAR COLUMNS:

Table.1 for Fig.5

S.No.	V (V)	$I_L(A)$	
		Th.	Pr.

Table.2 for Fig.6

S.No.	V(V)	$I_{SC}(A)$	
		Th.	Pr.

Prepared by
Dr. G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 4 of 6
Revision No.0

TITLE: VERIFICATION OF NORTON'S THEOREM	GPRECD/EEE/EXPT.No.ECP-05 DATE: 18-10-2021

Table 1 for Fig.7

R$_N$

Table.4 for Fig.8

S.No.	I$_{SC}$(A)	I$_L$'(A)	
		Th.	Pr.

PROCEDURE:

1. a. Connections are made as per the circuit shown in Fig.5.

   b. By varying the voltage in steps note down the readings.

2. a. Connections are made as per the circuit shown in Fig.6.

   b. By varying the voltage in steps note down the readings of I$_{sc}$.

3. a. Connections are made as per Fig.7 to find R$_N$.

4. a. Connections are made as per the circuit shown in Fig.8.

   b. By varying the current I$_{sc}$ in steps note down the readings of I$_L'$.

RESULT: Norton's theorem is verified.

TITLE: VERIFICATION OF GPRECD/EEE/EXPT.No.ECP-05

NORTON'S THEOREM DATE: 18-10-2021

Questions:

1. State Norton's theorem

2. State the applications of Norton's theorem.

3. To Linear and bilateral network, Norton's theorem can be applied.

4. Differentiate unilateral and bilateral, linear and non-linear.

5. Differentiate single port and two port networks.

TITLE: DETERMINATION OF SELF& MUTUAL INDUCTANCE	GPREC/EEE/EXPT.No. EC(P)-06 DATE: 18-10-2021		

OBJECTIVE: To Determine of Self Inductance, Mutual Inductance and coefficient of coupling of an air core transformer.

APPARATUS:

Sl.No.	Unit	Range	Type	No.
1	Air core transformer	1KVA		1
2	Ammeter	0-5A	MI	1
3	Voltmeter	0-300V	MI	1
4	Wattmeter	5A, 300V	LPF	1
			UPF	1
5	Connecting Wires			Reqd.

THEORY:

Self Inductance:

The property of self-inductance is a particular form of electromagnetic induction. Self inductance is defined as the induction of a voltage in a current-carrying wire when the current in the wire itself is changing. In the case of self-inductance, the magnetic field created by a changing current in the circuit itself induces a voltage in the same circuit. Therefore, the voltage is self-induced.

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

From the concepts of basic electrical engineering, we know that inductance may also be expressed as

$$L = \frac{N\phi}{I}$$

Therefore self inductance is represented as

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 1 of 7
Revision No.0

TITLE: DETERMINATION OF SELF& MUTUAL INDUCTANCE	GPREC/EEE/EXPT.No. EC(P)-06 DATE: 18-10-2021

$$L = N \frac{d\phi}{dt}$$

The unit for self inductance is Henry.

Mutual Inductance:

Mutual inductance is defined as the constant of proportionality between the rate of change in current in one circuit and the resulting e.m.f induced in another circuit. A varying current in coil 1 produces a magnetic field. This magnetic field links with coil 2 and inducing an e.m.f between its ends. Such an action in which a varying quantity in one circuit causes the development of a quantity in a different circuit is called mutual action or transfer action. Figure shows two coils which carry current I_1 and I_2. The coils will have leakage flux Φ_{11} and Φ_{22} for coils 1 and coil 2, respectively, and a mutual flux Φ_{21} where the flux of coil 2 links coil 1 or flux of coil 1 links coil 2. The induced voltage of coil 2 may be written as

$$V_{L2} = N_2 \frac{d\phi_{12}}{dt}$$

Fig. 1

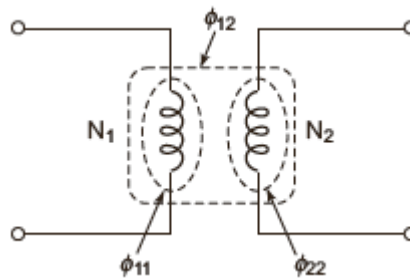As Φ_{12} is related to current of coil 1, the induced voltage is proportional to the rate of change of I_1. This implies

$$V_{L2} = M \frac{di_1}{dt}$$

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 2 of 7
Revision No.0

TITLE: DETERMINATION OF SELF& MUTUAL INDUCTANCE	GPREC/EEE/EXPT.No. EC(P)-06
	DATE: 18-10-2021

where *M* is constant of proportionality between the two coils, also known as mutual inductance.

When the coils are linked with air as medium, the flux and current are linearly proportional to each other and the expressions of mutual inductance are written as

$$M = N_2 \frac{\phi_{12}}{I_1}$$

$$M = N_1 \frac{\phi_{21}}{I_2}$$

Coefficient of Coupling

Coefficient of coupling is defined as the ratio of mutual inductance actually present between the two coils to the maximum possible value.

$$K = \frac{M}{\sqrt{L_1 L_2}}$$

Where

      K is the coupling coefficient and $0 \le k \le 1$

      L_1 is the inductance of the first coil, and

      L_2 is the inductance of the second coil.

Prepared by   Approved by   Page 3 of 7
Dr.G.Venkata Subba Reddy   Dr. T.Bramhananda Reddy   Revision No.0
                          HEAD, EEE Dept.

TITLE: DETERMINATION OF SELF& MUTUAL INDUCTANCE

GPREC/EEE/EXPT.No. EC(P)-06

DATE: 18-10-2021

CIRCUIT DIAGRAM:
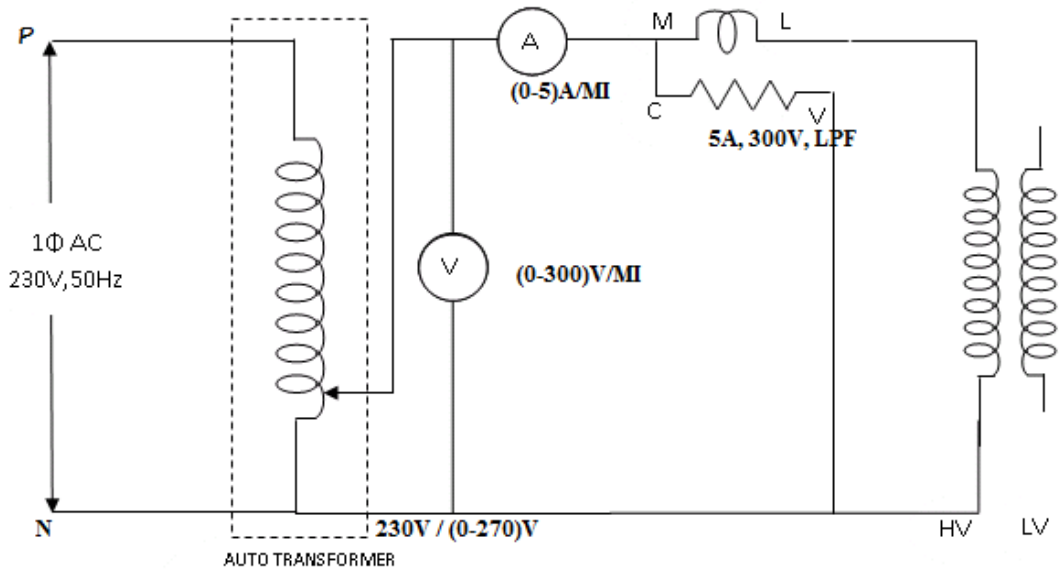
Fig. 2 To find L_{HV}

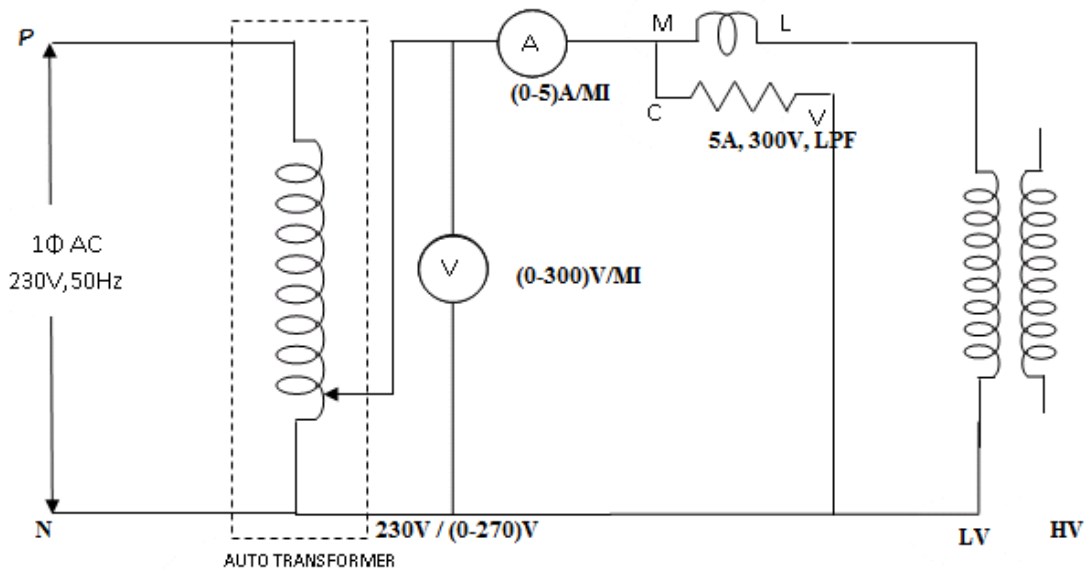Fig. 3 To find L_{LV}

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 4 of 7
Revision No.0

TITLE: DETERMINATION OF SELF& MUTUAL INDUCTANCE

GPREC/EEE/EXPT.No. EC(P)-06
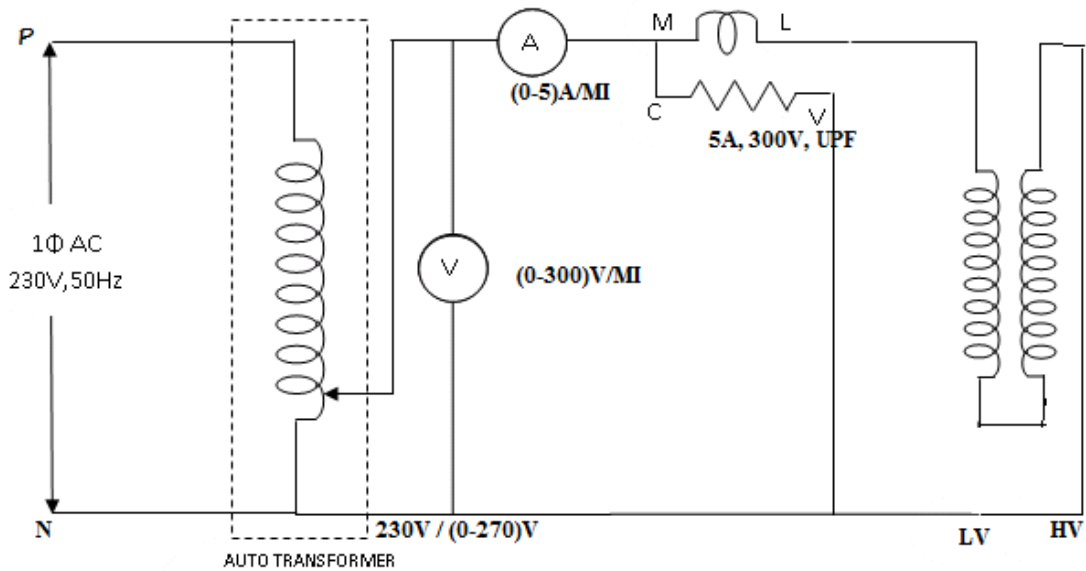
DATE: 18-10-2021

Fig.4 To find M & K

PROCEDURE:

1. Connections are made as per the circuit shown in Fig.2.

2. Using auto transformer, apply voltage in steps.

3. Note down all meter readings.

4. Calculate self inductance L_1 of high voltage (HV) winding.

5. Above procedure is repeated for circuit diagram shown in Fig.3 & Fig.4 to find self inductance L_2 of low voltage (LV) winding and mutual inductance of both windings.

TABULAR COLUMN:

S.No	V(V)	I_1(A)	W_1(W)	Φ_1	Z_1(Ω)	L_1(H)

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 5 of 7
Revision No.0

TITLE: DETERMINATION OF SELF& MUTUAL INDUCTANCE

GPREC/EEE/EXPT.No. EC(P)-06

DATE: 18-10-2021

Sl.No	V(V)	I_2(A)	W_2(W)	Φ_2	Z_2(Ω)	L_2(H)

Sl.No	V(V)	I(A)	W(W)	Φ	Z(Ω)	L(H)	M=(L-(L_1+L_2))/2

RESULT: Self inductance, mutual inductance and coefficient of coupling are determined.

TITLE: DETERMINATION OF SELF& MUTUAL INDUCTANCE	GPREC/EEE/EXPT.No. EC(P)-06 DATE: 18-10-2021

Questions:

1. Define self inductance.

2. Define mutual inductance.

3. Define coefficient of coupling.

4. Differentiate air core and iron core transformer.

5. If the transformer core is made of wood, what will be the coupling coefficient.

TITLE:RLC SERIES & PARALLEL RESONANCE	GPREC/EEE/EXPT.No. ECP-07 DATE: 18-10-2021

OBJECTIVE: To verify RLC Series resonance & Parallel resonance.

APPARATUS:

Sl.No.	Unit	Range	Type	No. Required
1	Function Generator			1
2	Digital Multimeter			1
3	Resistor	1KΩ		1
4	Capacitor	0.1μF		1
5	Inductance box	25mH		1
6	Connecting Wires			Reqd.

THEORY:

In an electrical circuit consisting of various active sources and passive sources, the state at which the current is maximum is called as resonance. In a series RLC circuit the condition of current lagging or leading the applied voltage depends upon the values of X_L and X_C. If one of the parameters of the series RLC circuit is varied in such a way that the current in the circuit is in phase with the applied voltage, then the circuit is said to be in resonance. In other words resonance can also be defined as when the net or total current in an electrical circuit is in phase with the applied voltage, then the circuit is said to undergo resonance

Fig.1 shows a series RLC circuit where the parameters R, L, C are laid in series with each other. The voltage of the circuit is given by

$$V = I(R + j(X_L - X_C))$$

Total impedance of the series RLC circuit is given by

$$Z = R + j(X_L - X_C)$$

TITLE:RLC SERIES & PARALLEL RESONANCE

GPREC/EEE/EXPT.No. ECP-07

DATE: 18-10-2021

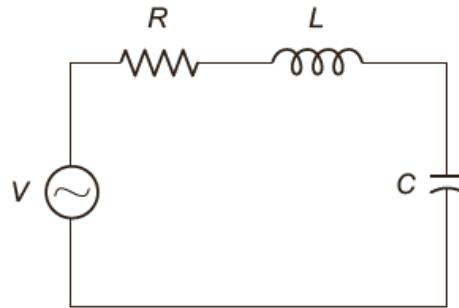$$Z = R + j(\omega L - \frac{1}{\omega C})$$

Fig.1. RLC series circuit

The circuit is said to be in resonance if the current is in phase with the applied voltage. In a series RLC circuit resonance occurs when $X_L = X_C$. The frequency at which the resonance occurs is called resonant frequency.

Resonant frequency

When the condition of series resonance is considered i.e. $X_L = X_C$, the impedance in a series RLC circuit becomes purely resistive. At resonant frequency (f_o), the voltages across capacitance and inductance are equal in magnitude i.e. $V_L = V_C$.

At resonance, $X_L = X_C$

$$\omega L = \frac{1}{\omega C}$$

$$2\pi f_o L = \frac{1}{2\pi f_o c} \quad \text{where } f_o \text{ is frequency at resonance}$$

$$f_o = \frac{1}{2\pi\sqrt{LC}}$$

Since $2\pi f_o = \omega_o$ it can be written as $\omega_o = \frac{1}{\sqrt{LC}}$

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 2 of 9
Revision No.1

TITLE:RLC SERIES & PARALLEL RESONANCE	GPREC/EEE/EXPT.No. ECP-07 DATE: 18-10-2021

The relation of resonant frequency changes when the R-L-C parameters are placed in parallel with each other. Two cases are majorly considered in the concept of parallel resonance.

   i.   Considering the internal resistance of both L and C

  ii.   Considering the internal resistance of only L

Parallel Resonance Considering Internal Resistance of L and C

Normally parallel resonance occurs at $X_C = X_L$. Consider the parallel circuit as shown in Fig 2. The internal resistances of both L and C have been incorporated along with the parameters.
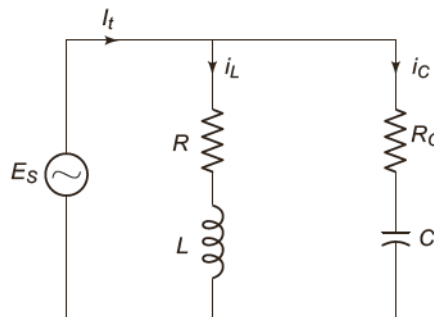

Fig.2. Parallel Circuit

According to KCL

$$I_t = i_L + i_C$$

$$I_t = V_S(Y_L + Y_C)$$

Where $i_L = V_S \cdot Y_L$

$$i_C = V_S \cdot Y_C$$

Y_L & Y_C are branch admittances

$$\therefore Y_L = \frac{1}{Z_L} = \frac{1}{R_L + j\omega L}$$

Rationalizing this we get

TITLE:RLC   SERIES   &   GPREC/EEE/EXPT.No. ECP-07

PARALLEL RESONANCE   DATE: 18-10-2021

$$Y_L = \frac{R_L}{R_L^2 + \omega^2 L^2} - j\frac{\omega L}{R_L^2 + \omega^2 L^2}$$

Similarly

$$Y_C = \frac{1}{Z_C} = \frac{1}{R_C - j(\frac{1}{\omega C})}$$

Rationalizing this we get, $Y_C = \dfrac{R_C + j(\frac{1}{\omega C})}{R_C^2 + (\frac{1}{\omega^2 C^2})}$

$$= \frac{R_C}{R_C^2 + (\frac{1}{\omega^2 C^2})} + \frac{j(\frac{1}{\omega C})}{R_C^2 + \frac{1}{\omega^2 C^2}}$$

$\therefore$ Substituting the values of Y_L and Y_C we get

$$I_t = V_S \left(\frac{R_L}{R_L^2 + \omega^2 L^2} + \frac{R_C}{R_C^2 + (\frac{1}{\omega^2 C^2})} \right) - j \left(\frac{\omega L}{R_L^2 + \omega^2 L^2} - \frac{1/\omega C}{R_C^2 + (\frac{1}{\omega^2 C^2})} \right)$$

The circuit is to resonance if the net susceptance is zero.

$\therefore$ From the above equation equating the imaginary part equal to zero we get

$$\frac{\omega_o L}{R_L^2 + \omega_o^2 L^2} - \frac{Y\omega_o C}{R_C^2 + \frac{1}{\omega_o^2 C^2}} = 0$$

Where ω_o is the resonant frequency

Solving the above equation we get $\omega_o = \sqrt{\dfrac{1}{LC}\left(\dfrac{L - CR_L{}^2}{L - CR_C{}^2} \right)}$

$\therefore$ Substituting $\omega_o = 2\pi f_o$ is above equation

----

Prepared by   Approved by   Page 4 of 9
Dr.G.Venkata Subba Reddy   Dr. T.Bramhananda Reddy   Revision No.1
                      HEAD, EEE Dept.

TITLE:RLC SERIES & PARALLEL RESONANCE	GPREC/EEE/EXPT.No. ECP-07 DATE: 18-10-2021

$$f_o = \frac{1}{2\pi} \sqrt{\frac{1}{LC}\left(\frac{L-CR_L{}^2}{L-CR_C{}^2}\right)}$$

If $R_L = R_C$ then, $f_o = \dfrac{1}{2\pi\sqrt{LC}}$

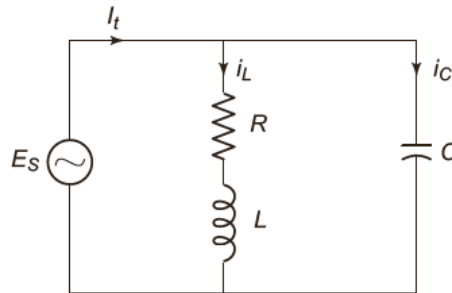**Parallel Resonance Considering Internal Resistance of only L**

Fig.3. Parallel Circuit with $R_C = 0$

Figure shows the parallel L-C circuit where the internal resistance of C is considered to be zero and only the internal resistance of L is considered. The above figure can also be referred to as a tank circuit because the circuit stores energy in the magnetic field of the coil and in the electric field of the capacitor. Normally the circuit is asid to be in resonant condition when the susceptance part of the admittance is zero

The total admittance of the above circuit is

$$Y = \frac{1}{R_L + jX_L} + \frac{j}{X_C}$$

Simplying further

$$Y = \frac{R_L - jX_L}{R_L^2 + X_L^2} + \frac{j}{X_C}$$

$$= \frac{R_L}{R_L^2 + X_L^2} + j\left[\frac{1}{X_C} - \frac{X_L}{R_L^2 + X_L^2}\right] = G + jB$$

In order to satisfy the condition of resonance, the suseptance part, B should be zero

----

TITLE:RLC SERIES & PARALLEL RESONANCE	GPREC/EEE/EXPT.No. ECP-07
	DATE: 18-10-2021

$$\therefore \qquad \frac{1}{X_C} - \frac{X_L}{R_L^2 + X_L^2} = 0$$

which implies $\qquad \dfrac{1}{X_C} = \dfrac{X_L}{R_L^2 + X_L^2}$

$$\omega C = \frac{\omega L}{R_L^2 + \omega^2 L^2}$$

From the above expression

$$R_L^2 + \omega^2 L^2 = \frac{L}{C}$$

$$\omega^2 L^2 = \frac{L}{C} - R_L^2$$

$$\omega = \sqrt{\frac{1}{LC} - \frac{R_L^2}{L^2}}$$

The resonant frequency for the tank circuit is given by $\quad f_o = \dfrac{1}{2\pi}\sqrt{\dfrac{1}{LC} - \dfrac{R_L^2}{L^2}}$

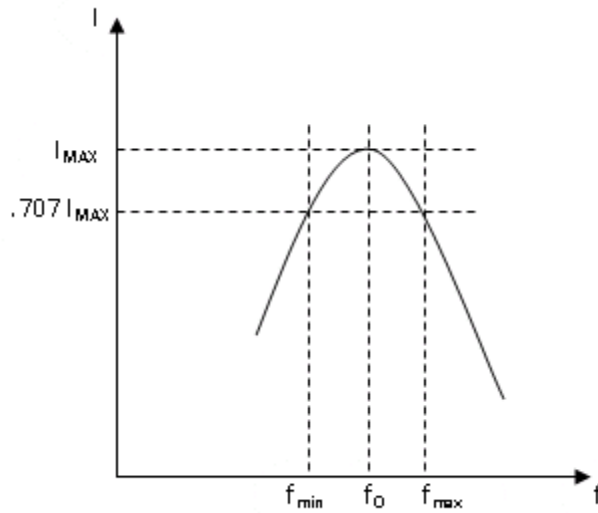## CIRCUIT DIAGRAM & EXPECTED GRAPH:

RLC Series Resonance:

Fig.4 RLC series circuit

Prepared by   Approved by   Page 6 of 9
Dr.G.Venkata Subba Reddy   Dr. T.Bramhananda Reddy   Revision No.1
HEAD, EEE Dept.

TITLE:RLC SERIES & PARALLEL RESONANCE	GPREC/EEE/EXPT.No. ECP-07 DATE: 18-10-2021

RLC Parallel Resonance:

Fig.5 RLC Parallel Circuit

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 7 of 9
Revision No.1
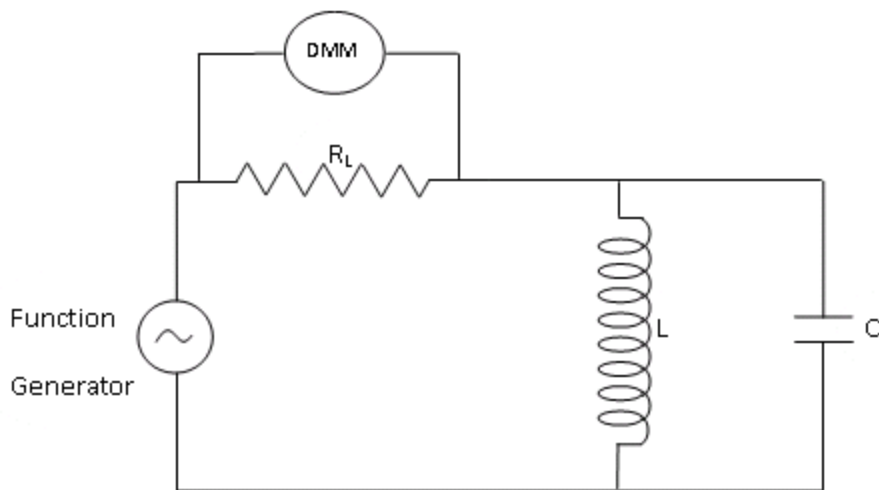
TITLE:RLC SERIES & PARALLEL RESONANCE	GPREC/EEE/EXPT.No. ECP-07
	DATE: 18-10-2021
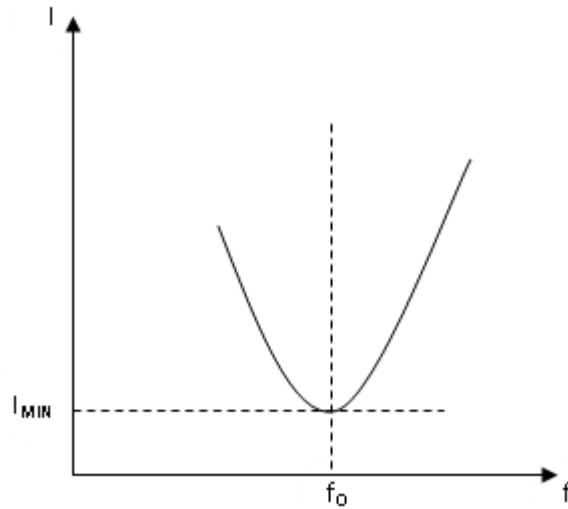
PROCEDURE:

RLC Series & Parallel Resonance:

1. Connections are made as per the circuit diagram shown in Fig.4 & Fig.5, for series and parallel resonance respectively.

2. By varying the frequency of the function generator, note down the voltmeter reading.

3. Find the current I by using R value

4. Plot f Vs I for series resonance and parallel resonance.

5. Observe that the resonance occur at resonant frequency.

TABULAR COLUMN:

RLC Series & Parallel Resonance:

Sl.no.	f(Hz)	V(V)	I(A)

RESULT: RLC Series Resonance & Parallel Resonance graphs are plotted and resonant frequency is found.

TITLE:RLC SERIES & PARALLEL RESONANCE	GPREC/EEE/EXPT.No. ECP-07 DATE: 18-10-2021

Questions:

1. Define resonance.

2. What is the condition for resonance in RLC series and parallel circuits.

3. Define bandwidth

4. Define Selectivity

5. Define Q factor.

TITLE:RL & RC LOCUS DIAGRAMS	GPREC/EEE/EXPT.No. ECP-8 DATE: 18-10-2021

OBJECTIVE: To plot RL & RC locus diagrams.

APPARATUS:

Sl.No.	Unit	Range	Type	No.
1	Auto Transformer	230/0-280V		1
2	Voltmeter	0-300V	MI	1
3	Ammeter	0-5/10A	MI	1
4	Rheostat	26Ω/4A		1
	Inductor	150mH		1
5	Capacitor	50µF		1
6	Connecting Wires			Reqd.

THEORY:

(1) Series RL Locus

Consider a RL series circuit as shown in figure below.
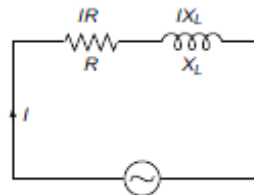
Fig.1 RL series circuit

Let I be the current flowing through the circuit, IR be the voltage drop across R, and IX_L be the voltage drop across X_L. The current flowing through the circuit can be resolved into two componenets I cos θ and I sin θ. We know that

$$I = \frac{V}{\sqrt{R^2 + X_L^2}} \text{ and}$$

$$\cos \theta = \frac{R}{\sqrt{R^2 + X_L^2}}$$

Let us suppose that $X = I \cos \theta$ which can be written as

----

Prepared by	Approved by	Page 1 of 8
Dr.G.Venkata Subba Reddy	Dr. T.Bramhananda Reddy	Revision No.0
	HEAD, EEE Dept.	

TITLE:RL & RC LOCUS DIAGRAMS	GPREC/EEE/EXPT.No. ECP-8 DATE: 18-10-2021

$$X = \frac{V}{\sqrt{R^2 + X_L^2}} \times \frac{R}{\sqrt{R^2 + X_L^2}}$$

$$= \frac{VR}{R^2 + X_L^2}$$

$$R^2 + X_L^2 = \frac{VR}{X}$$

Similarly, let $Y = I \sin \theta$

$$= \frac{V}{\sqrt{R^2 + X_L^2}} \times \frac{X_L}{\sqrt{R^2 + X_L^2}}$$

$$= \frac{VX_L}{R^2 + X_L^2}$$

Therefore, $X^2 + Y^2$ can be written as

$$X^2 + Y^2 = \frac{V^2 R^2}{(R^2 + X_L^2)^2} + \frac{V^2 X_L^2}{(R^2 + X_L^2)^2}$$

$$= \frac{V^2(R^2 + X_L^2)}{(R^2 + X_L^2)^2}$$

$$X^2 + Y^2 = \frac{V^2}{R^2 + X_L^2}$$

And there fore

$$X^2 + Y^2 = \frac{V^2 X}{VR}$$

$$X^2 + Y^2 = \frac{V^2 X}{VR}$$

This can be modified as

$$\left(X - \frac{V}{2R}\right)^2 + Y^2 = \left(\frac{V}{2R}\right)^2$$

The above expression represents a equation of a circle.

Locus diagram of R–L circuit when R is varying

Consider an R–L circuit in which R is varying from 0 to ∞.

The maximum current can be written as

TITLE:RL & RC LOCUS DIAGRAMS	GPREC/EEE/EXPT.No. ECP-8
	DATE: 18-10-2021

$$I_{max} = \frac{V}{X_L}$$

when R=0. Thus, here *I*max lags the voltage by an angle of 90°. The procedure to draw the locus diagram in this case can be described as given below.

(i) The reference vector *V* is drawn as shown

(ii) Since *I*max lags the voltage by an angle of 90°, it is drawn at an angle of 90°.

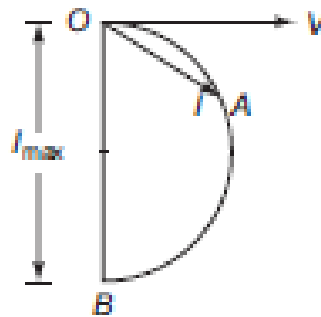(iii) With (0, Imax /2) as the center and the radius, a semi-circle is drawn approximately.

Fig.2 Locus Diagram for RL circuit

Locus diagram of *R–C* circuit when *R* is varying

Consider an *R–C* circuit in which *R* is varying from 0 to ∞

The maximum current can be

$$I_{max} = \frac{V}{X_C}$$

Thus, here *I*max leads the voltage by an angle of 90°. The procedure to draw the locus diagram can therefore be

(i) The reference vector *V* is drawn as shown

(ii) Since *I*max leads the voltage by an angle of 90°, it is drawn at an angle of 90°.

TITLE:RL & RC LOCUS DIAGRAMS	GPREC/EEE/EXPT.No. ECP-8
	DATE: 18-10-2021

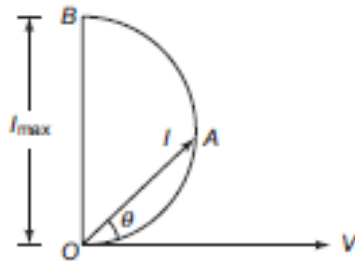(iii) With (0, Imax /2) as the center and the radius a semi-circle is drawn approximately.

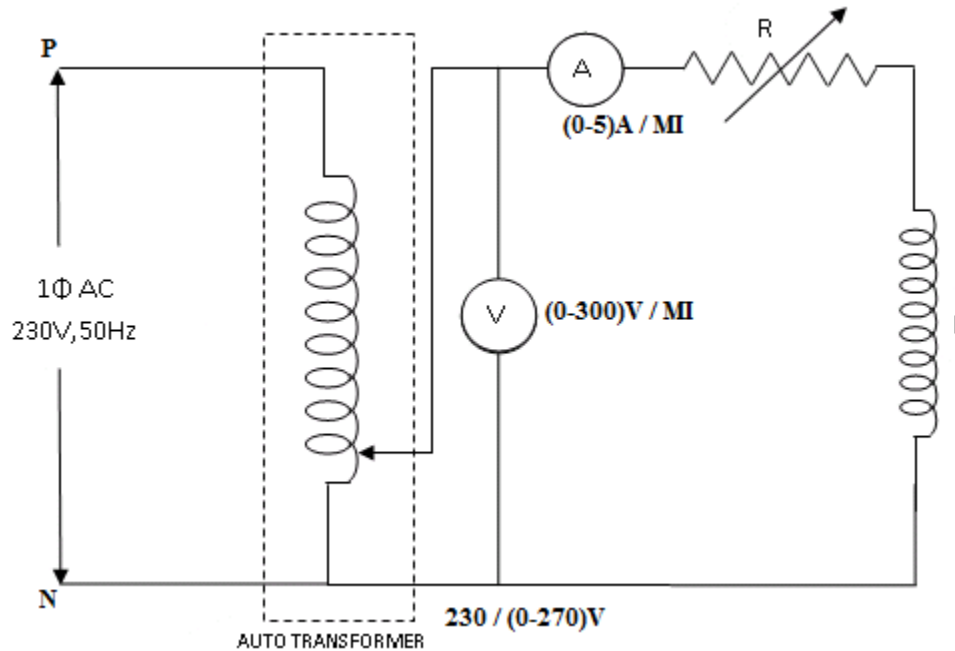**Fig.3 Locus diagram for RC circuit**

CIRCUIT DIAGRAM & MODEL GRAPH:

 RL:

Fig.4 Circuit Diagram for RL

TITLE:RL & RC LOCUS DIAGRAMS	GPREC/EEE/EXPT.No. ECP-8 DATE: 18-10-2021
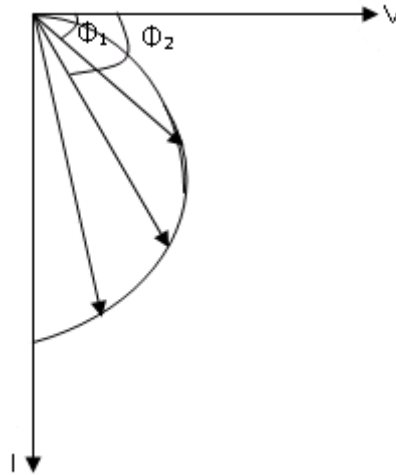
Fig.5 Locus Diagram for RL circuit

RC:
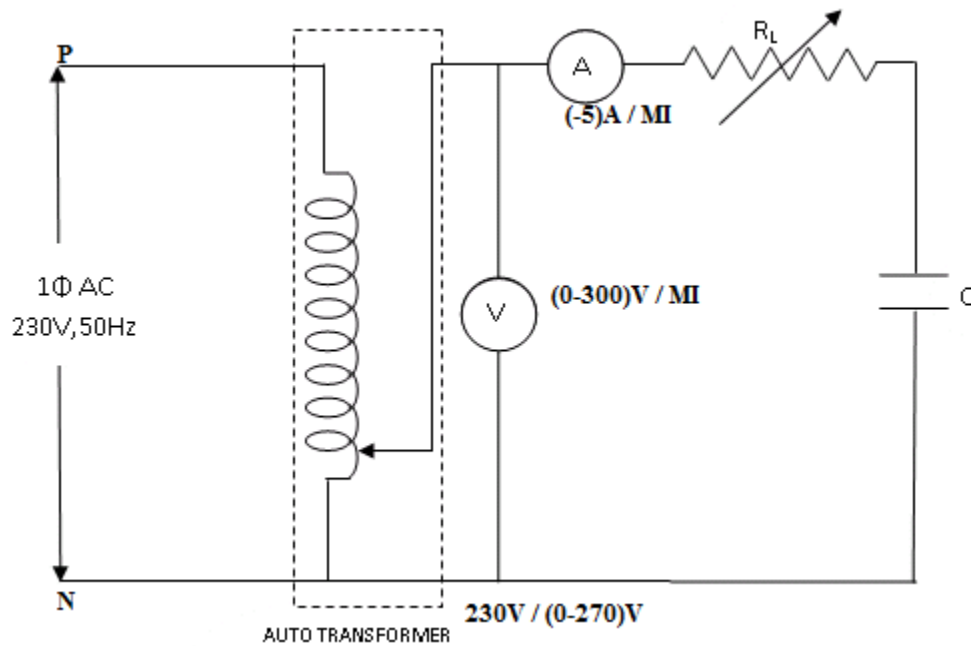
Fig.6 Circuit Diagram for RC

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 5 of 8
Revision No.0

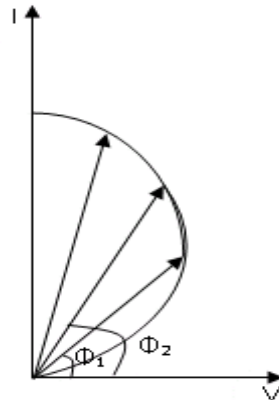| **TITLE:**RL & RC LOCUS DIAGRAMS	GPREC/EEE/EXPT.No. ECP-8
	DATE: 18-10-2021

Fig.7 Locus Diagram for RC circuit

PROCEDURE:

RL LOCUS DIAGRAM:

1. Connections are made as per the circuit shown in Fig. 4
2. Apply a voltage of 140V using autotransformer.
3. By adjusting the value of R note down the corresponding readings.
4. Draw the current locus diagram.

RC LOCUS DIAGRAM:

1. Connections are made as per the circuit shown in Fig. 6
2. Apply a voltage of 140V using autotransformer.
3. By adjusting the value of R note down the corresponding readings.
4. Draw the current locus diagram.

TABULAR COLUMN for RL:

S.No.	V(v)	I(A)	Z(Ω)	Φ

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 6 of 8
Revision No.0

TITLE: RL & RC LOCUS DIAGRAMS	GPREC/EEE/EXPT.No. ECP-8 DATE: 18-10-2021

TABULAR COLUMN for RC:

S.No.	V(v)	I(A)	Z(Ω)	Φ

RESULT: RL and RC locus diagrams are drawn.

TITLE:RL & RC LOCUS DIAGRAMS	GPREC/EEE/EXPT.No. ECP-8 DATE: 18-10-2021

Questions:

1. Differentiate RL and RC locus diagrams.

2. What is the power factor in RL and RC series circuits?

3. Write the equation for impedance in RL and RC series circuits.

4. How the loci could differ from RL and RC parallel circuits from RL and RC series circuits.

TITLE:	VERIFICATION	OF	GPRECD/EEE/EXPT.No.ECP-09
COMPENSATION THEOREM			DATE: 18-10-2021

OBJECTIVE: To verify Compensation theorem by (a) Simulation in PSpice software

(b) Experiment.

APPARATUS:

Sl.No.	Unit	Range	Type	No.
1	Regulated power supply	(0-30)V/(0-2)A		1
2	Ammeter	(0-2)A	MC	1
3	Voltmeter	(0-30)V	MC	1
3	Rheostat			3
4	Digital Multimeter			1
5	Connecting Wires			Reqd.

STATEMENT:

Compensation theorem states that any element in electrical network can be replaced by its equivalent voltage source, whose value is equal to product of current flowing through it and its value. (Compensation theorem got the importance of determining the change in current flowing through element or circuit because of change in the resistance value).

THEORY:

Any network consisting of linear or bilateral impedances and independent sources. If the branch having a current I and impedance Z that increased by ΔZ, then a change of voltage and current in other branches of the network is same as the voltage or current produced by an opposing voltage source of value IΔZ placed in that branch after replacing original source by their internal impedance.

TITLE: VERIFICATION OF COMPENSATION THEOREM	GPRECD/EEE/EXPT.No.ECP-09 DATE: 18-10-2021

In network theory, it is important to know or study the effect of change in impedance or resistance in one of it's branches. It will affect the corresponding voltage and currents of the network or circuit. The Compensation theorem theorem gives information about the change in network

The compensation theorem works on the basic concept of Ohm's law. According to Ohm's law, When a current passes through the resistor. Some amount of voltage drop occurs across the resistor. This voltage drop will oppose the source voltage.Therefore,we connect an extra voltage source in opposite polarity compared to the source voltage and magnitude is equal to the voltage drop..Compensation theorem works on this concept.

CIRCUIT DIAGRAM:
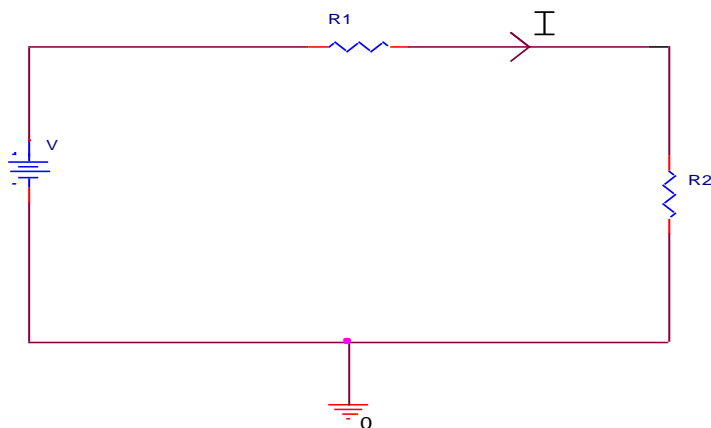
    a. **Simulation Model in PSpice**

     Fig.1 Basic Circuit

Prepared by   Approved by   Page 2 of 7
Dr.G.Venkata Subba Reddy   Dr. T.Bramhananda Reddy   Revision No.0
                           HEAD, EEE Dept.

TITLE: VERIFICATION OF GPRECD/EEE/EXPT.No.ECP-09

COMPENSATION THEOREM DATE: 18-10-2021
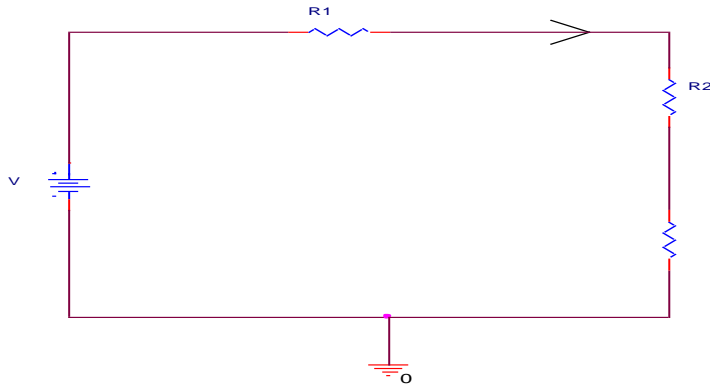
Fig.2 After Change in Resistance Circuit
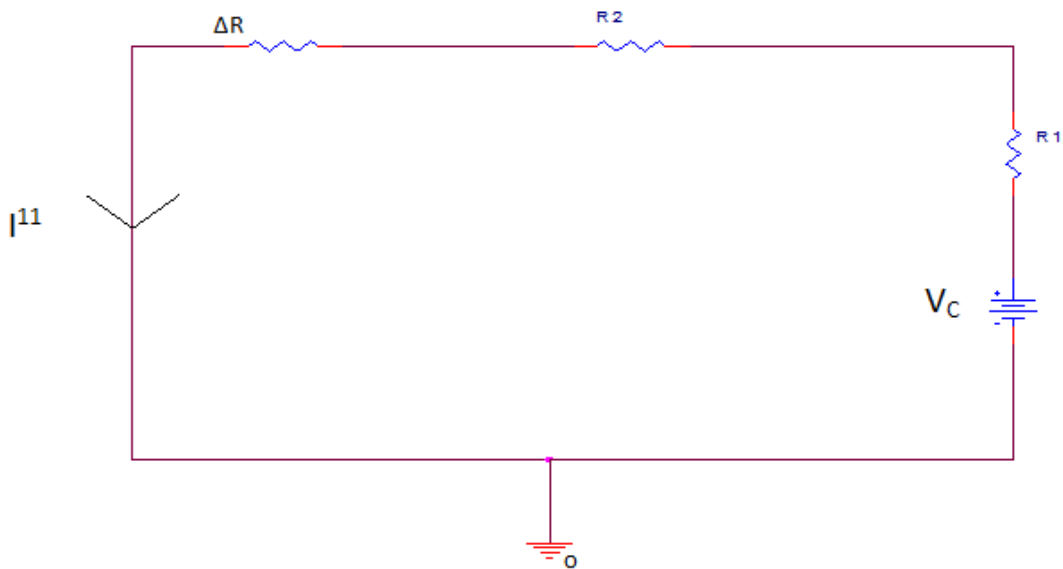
Fig .3 Compensation theorem circuit

TITLE: VERIFICATION OF COMPENSATION THEOREM

GPRECD/EEE/EXPT.No.ECP-09

DATE: 18-10-2021

b. **Experiment**
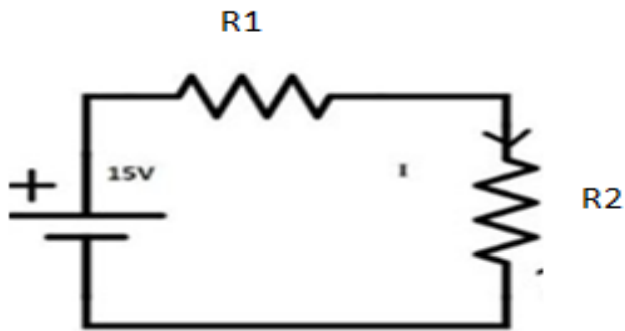
Fig.1 Basic Circuit

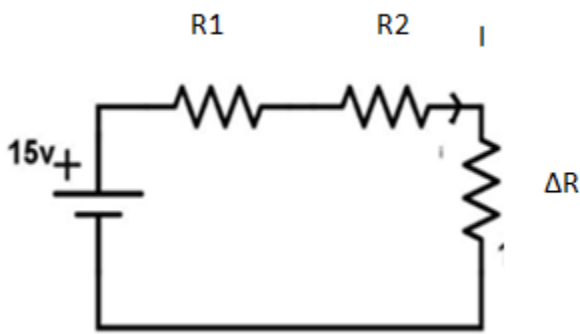**Fig.2 After Change in Resistance Circuit**

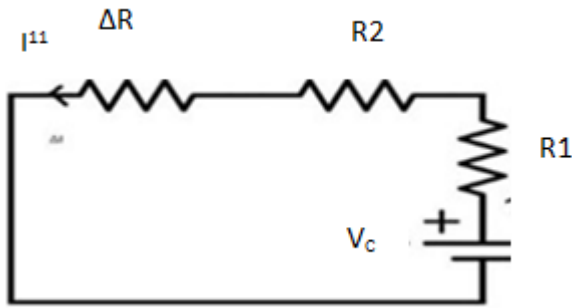|	
TITLE: VERIFICATION OF COMPENSATION THEOREM	GPRECD/EEE/EXPT.No.ECP-09 DATE: 18-10-2021

Fig .3 Compensation theorem circuit

PROCEDURE:

1. Connect the circuit as shown in Fig.1

2. Measure the current I.

3. Connect the circuit as shown in Fig.2 by increasing the circuit resistance(ΔR), Measure the current I_1.

4. The change in current in the circuit can be found by connecting a voltage source equal to $V_C = I_1\Delta R$ as shown in Fig.3.

5. Measure the current I" i.e., the change in current.

6. Observe that I"= I- I_1.

TITLE: VERIFICATION OF COMPENSATION THEOREM	GPRECD/EEE/EXPT.No.ECP-09 DATE: 18-10-2021

TABULAR COLUMN:

Table.1 for Fig.1

V	I	
	Theoretical value	Practical value

Table.2 for Fig.2

V	I_1	
	Theoretical value	Practical value

Table.3 for Fig.3

V_C	I^{11}	
	Theoretical value	Practical value

TITLE: VERIFICATION OF	GPRECD/EEE/EXPT.No.ECP-09
COMPENSATION THEOREM	DATE: 18-10-2021

RESULT: Compensation Theorem is verified.

QUESTIONS:

1. How to verify compensation theorem.
2. What are the applications of Compensation theorem.
3. Explain the procedure of solving the circuits using Compensation Theorem.
4. What is meant by the compensation emf.
5. State Compensation Theorem?.

TITLE: VERIFICATION OF MILLIMAN'S THEOREM	GPREC/EEE/EXPT.No. ECP-09 DATE: 18-10-2021

OBJECTIVE: To verify the Milliman's Theorem by (a) Experiment (b) Simulation in PSpice software

APPARATUS:

Sl.No.	Unit	Range	Type	No.
1	Regulated Power supply	(0-30) V/(0-2)A		1
2	Ammeter	(0-2)A	MC	1
4	Rheostat			3
5	Connecting Wires			Reqd.

STATEMENT:

This theorem states that in any network, if the voltage sources $V_1, V_2, \ldots V_n$ in series with their internal resistances $R_1, R_2 \ldots R_n$ respectively are in parallel, then these sources may be replaced by a single voltage source V' in series-with R_1.

$$V' = V_1G_1 + V_2G_2 + \text{-----} V_nG_n / (G_1 + G_2 + \text{-------} G_n)$$

$$R_1 = 1 / (G_1 + G_2 + \text{-----} G_n)$$

THEORY:

In electrical circuit theory, Milliman's Theorem states that for any linear electrical network states that any parallel combination of voltage sources in series with resistances can be replaced by a single voltage source V^1 and a single series resistor R1 .For single frequency AC systems, the theorem can also be applied to general impedances, any complex network can be reduced to a Milliman's equivalent circuit consist of a single voltage source and series impedance connected to a load.

Millman's Theorem is very convenient for determining the voltage across a set of parallel branches, where there are enough voltage sources present to preclude solution via

----

Prepared by   Approved by   Page 1 of 4
Dr.G.Venkata Subba Reddy   Dr. T.Bramhananda Reddy   Revision No.0
HEAD, EEE Dept.

TITLE: VERIFICATION OF MILLIMAN'S THEOREM	GPREC/EEE/EXPT.No. ECP-09 DATE: 18-10-2021

regular series-parallel reduction method. It also is easy in the sense that it doesn't require the use of simultaneous equations. However, it is limited in that it only applied to circuits which can be re-drawn to fit this form. It cannot be used, for example, to solve an unbalanced bridge circuit. And, even in cases where Millman's Theorem can be applied, the solution of individual resistor voltage drops can be a bit daunting to some, the Millman's Theorem equation only providing a single figure for branch voltage.
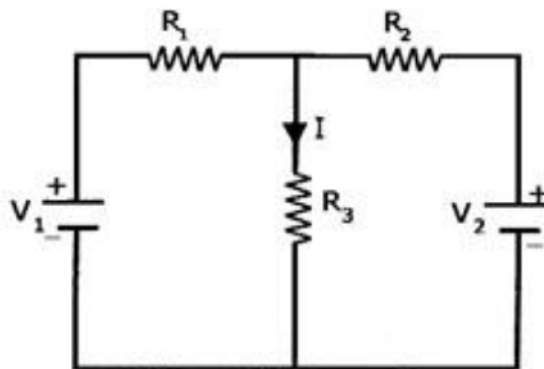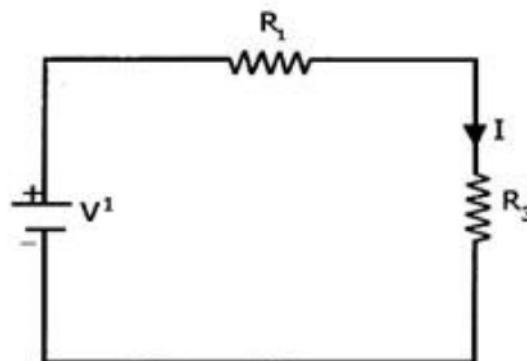
CIRCUIT DIAGRAM:

Fig.1 Basic Circuit

Fig.2 Milliman's Circuit

TITLE: VERIFICATION OF MILLIMAN'S THEOREM	GPREC/EEE/EXPT.No. ECP-09 DATE: 18-10-2021

b.Simulation Model in PSpice

Fig.1

Fig.2

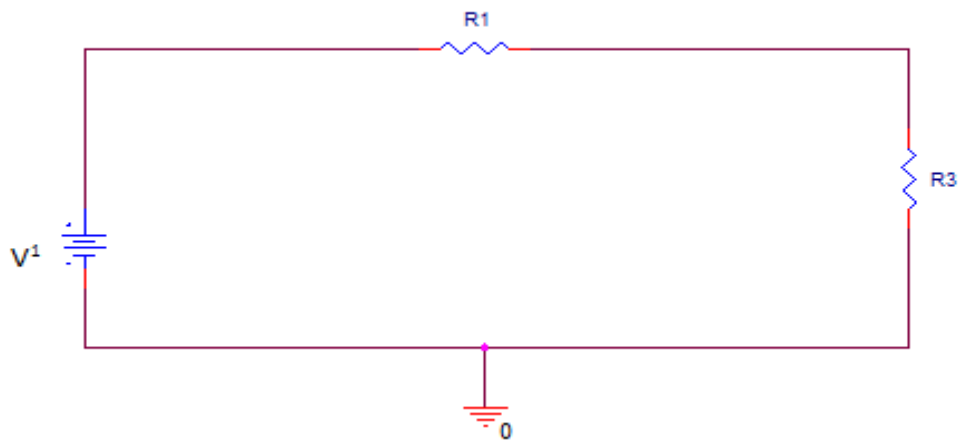---

TITLE:	VERIFICATION	OF	GPREC/EEE/EXPT.No. ECP-09
MILLIMAN'S THEOREM			DATE: 18-10-2021

TABULAR COLUMN:

S.No	V$_1$	V$_2$	I	
			Th.	Pr.

S.No	V^1	R1	I	
			Th.	Pr.

PROCEDURE:
1. Connect the circuit as shown in fig.1
2. Measure the current through the resistor R3.
3. Connect the circuit as shown in fig.2 and measure the current through R3.
4. Observe that the two currents are same.

RESULT:

Questions:
1. Define Milliman's Theorem.
2. What are the applications of Milliman's Theorem.
3. Explain the procedure of solving the circuits using Milliman's Theorem
4. Draw the equivalent circuit of Milliman's Theorem.
5. State the difference between Milliman's Theorem and Tellegen's Theorem.

TITLE: Determination of Average ,RMS, Peak and Phase values of AC Systems	GPREC/EEE/EXPT.No. ECP-10 DATE:18-10-2021

**OBJECTIVE:** To determine Average, RMS, Peak and Phase values of AC Systens.

.

APPARATUS:

Sl.No.	Unit	Range	Type	No.
1	Digital Multimeter		DVVM	1
2	Resistors	100Ω		1
3	Capacitor	100μF		1
4	Inductor	1 mH		1
5	CRO			1
6	Function Generator			1
5	Connecting Wires and patch cards			Reqd.

THEORY- The Oscilloscope is a device for observing and taking measurements of electrical signals and waveforms. The analog oscilloscope consists of a cathode ray tube (CRT) which displays a graph, primarily voltage versus time. It also has one or more amplifiers to supply voltage signals to the CRT and a time base system for generating the time scale. Some of the modern digital oscilloscopes use a liquid crystal display screen for the same purpose.

Sine WaveThe sine wave is a common type of alternating current (ac) and alternating
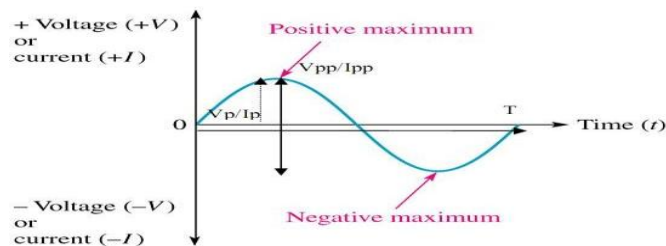
voltage.

Fig. 1 Sinusoidal signal

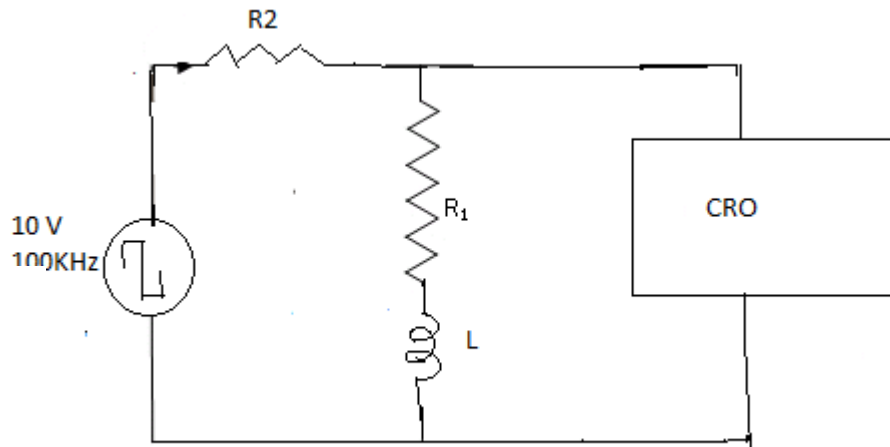|	
TITLE: Determination of Average ,RMS, Peak and Phase values of AC Systems	GPREC/EEE/EXPT.No. ECP-10 DATE:18-10-2021

CIRCUIT DIAGRAM:

Fig. 1 CIRCUIT DIAGRAM

PROCEDURE FOR AC Peak, RMS, and Phase Measurement:

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

2.   Set the value of frequency say 100 Hz in the function generator.

3.   Adjust the ground of channel 1 and 2 of Cathode Ray

    Oscilloscope and then set it into DC mode.

4.   Connect CRO across the load in DC mode and observe the waveform.

    Adjust the DC offset of function generator.

5.   Note down the amplitude and frequency.

6.   Set the multimeter into AC mode and measure input voltage and

    voltage across point AB. This value gives RMS value of sinusoidal AC.

7.   Calculate the average value.

8.   Repeat experiment for different frequency and different peak to peak
    voltage.

9.   Measure the RMS and Average value of DC signal also where

    instead of function generator you can use DC supply.

TITLE: Determination of Average ,RMS, Peak and Phase values of AC Systems	GPREC/EEE/EXPT.No. ECP-10 DATE:18-10-2021

5   Tabular Column:

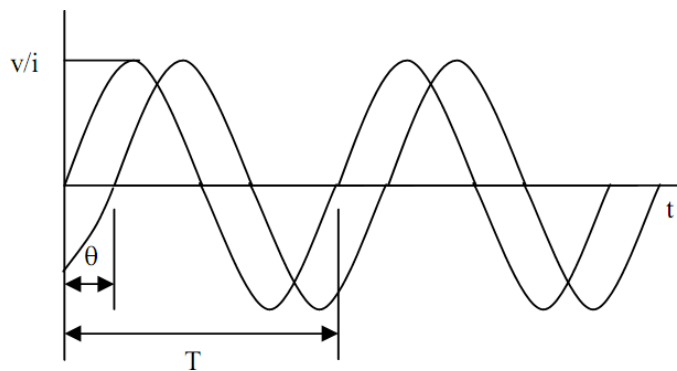| Peak value (V)	RMS value (V)	Average value (V)

Figure :Two sinusoidal waves with phase difference

CALCULATIONS:

Average Value= $2V_m/\pi$ for sinusoidal signal

RMS Value= $V_m/\sqrt{2}$ for sinusoidal signal

Average Value= $1/2 * V_m$ for Triangular signal signal

RMS Value= $V_m/\sqrt{3}$ for Triangular signal

TITLE: Determination of Average ,RMS, Peak and Phase values of AC Systems	GPREC/EEE/EXPT.No. ECP-10 DATE:18-10-2021

Average Value= V_m for square signal

RMS Value= V_m for square signal

RESULT:

VIVA QUESTIONS:

1. Define Average,RMS values of AC System.

2. Define Form Factor and Crest Factor.

3. What are the Network Elements.

4. Define Phase and Phase Difference?

5. What are the Phase relations between V and I in R,L and C elements.

TITLE: Determination of Average ,RMS, Peak and Phase values of AC Systems

GPREC/EEE/EXPT.No. ECP-10

DATE:18-10-2021

Prepared by   Approved by   Page 5 of 5
Dr.G.Venkata Subba Reddy   Dr. T.Bramhananda Reddy   Revision No.0
                                        HEAD, EEE Dept.

TITLE: DETERMINATION OF Z   GPREC/EEE/EXPT.No. ECP-11

& Y PARAMETERS   DATE: 18-10-2021

OBJECTIVE: To determine Impedance (Z) & Admittance (Y) parameters.

APPARATUS:

Sl.No.	Unit	Range	Type	No.
1	Regulated power supply	(0-30) V/(0-2)A		1
2	Rheostats		-	
3	Voltmeter	(0-30) V	MC	2
4	Ammeter	(0-2)A	MC	2
5	Connecting Wires			Reqd.

THEORY: Consider the two-port network shown in Fig.1. In the figure, $1-1'$ represents input port where the voltage is V_1 and the conventional current is 'I_1' as shown and $2-2'$ represents out port where the voltage is V_2 and the conventional current is 'I_2'. The defining equations can be expressed by considering any two variables as independent variables and expressing the other two variables in terms of these two variables and network parameters.
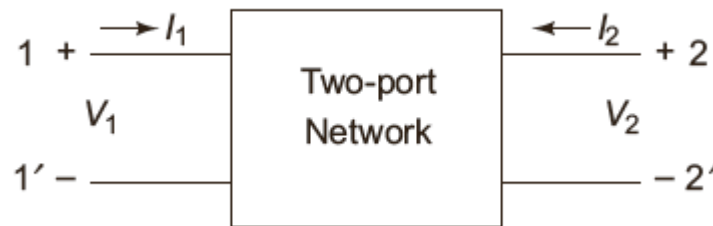

Fig.1 Two port network

∴   The defining equations of Z parameters are

$$V_1 = Z_{11} I_1 + Z_{12} I_2$$

And $$V_2 = Z_{21} I_1 + Z_{22} I_2$$

These two equations can be expressed in the matrix form as

TITLE: DETERMINATION OF Z

& Y PARAMETERS

GPREC/EEE/EXPT.No. ECP-11

DATE: 18-10-2021

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} Z_{11} & Z_{12} \\ Z_{21} & Z_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix}$$

Where Z_{11}, Z_{12}, Z_{21}, Z_{22} are network parameters called Z parameters, I_1 and I_2 are independent variables, V_1 and V_2 are dependent variables. If the output port $2-2'$ is open circuited then $I_2 = 0$

$$\therefore \quad V_1 = Z_{11} I_1 + 0$$

$$\Rightarrow \quad Z_{11} = \frac{V_1}{I_1} \bigg|_{I_2 = 0}$$

Z_{11} is called as ***driving point impedance*** because current I_1 develops voltage V_1 at the input port. Also we have

$$V_2 = Z_{21} I_1 + 0$$

$$\Rightarrow \quad Z_{21} = \frac{V_2}{I_1} \bigg|_{I_2 = 0}$$

Z_{21} is called as **"*Forward transfer impedance*"** because current I_1 develops voltage V_2 at the open circuited port. Similarly Z_{12} and Z_{22} can be found by considering $I_1 = 0$ i.e., by open circuiting the input port terminals.

By substituting $I_1 = 0$, we get

$$V_1 = 0 + Z_{12} I_2$$

$$\Rightarrow \quad Z_{12} = \frac{V_1}{I_2} \bigg|_{I_1 = 0}$$

Z_{12} is called as **"*Reverse Transfer impedance*"** because current I_2 develops voltage V_1 at the open circuited port.   Also we have

$$V_2 = 0 + Z_{22} I_2$$

$$\Rightarrow \quad Z_{22} = \frac{V_2}{I_2} \bigg|_{I_1 = 0}$$

Z_{22} is called as **driving point impedance**.

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 2 of 9
Revision No.0

TITLE: DETERMINATION OF Z & Y PARAMETERS	GPREC/EEE/EXPT.No. ECP-11 DATE: 18-10-2021

Note: The unit for Z parameters is Ohms(Ω)

Methods of finding the Z – PARAMETERS:

The defining equations for Z Parameters are

$$V_1 = Z_{11} I_1 + Z_{12} I_2$$

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

There are two methods for finding Z – parameters (i) Direct method and (ii) Indirect method

(i)Direct Method

Step (i) : By open circuiting of the output port, I_2 will tends to zero and so express V_1 and V_2 in terms of I_1 and calculated as

$$Z_{11} = \frac{V_1}{I_1} \qquad \text{and} \qquad Z_{21} = \frac{V_2}{I_1}$$

Step (ii) : By open circuiting of the input port, I_1 will tends to zero and so express V_1 and V_2 in terms of I_2 and calculate as

$$Z_{12} = \frac{V_1}{I_2} \quad \text{and} \ \ Z_{22} = \frac{V_2}{I_2}$$

(ii)Indirect Method

Using node analysis or loop analysis, expresses V_1 and V_2 in terms of I_1 and I_2 then calculate Z parameters.

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 3 of 9
Revision No.0

TITLE: DETERMINATION OF Z	GPREC/EEE/EXPT.No. ECP-11
& Y PARAMETERS	DATE: 18-10-2021

Z PARAMETERS (Open Circuit Impedance Parameters):
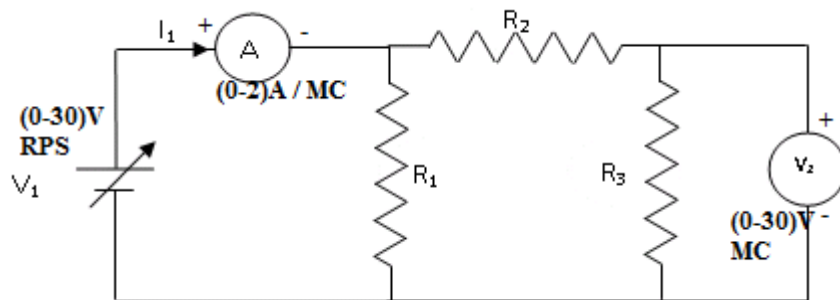
CIRCUIT DIAGRAM:

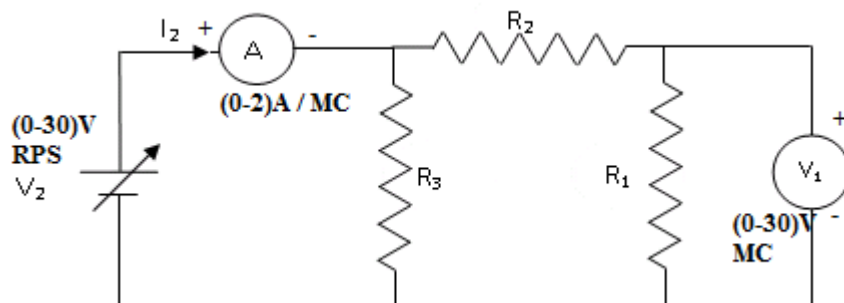**Fig. 2 To find Z_{11} & Z_{21}**

Fig. 3 To find Z_{12} & Z_{22}

PROCEDURE:

1. Connections are made as per the circuit shown in Fig. 2 and Fig.3.

2. By varying the RPS , corresponding readings are noted down.

3. The impedance parameters Z_{11} , Z_{21}, Z_{22} & Z_{12} are calculated.

TITLE: DETERMINATION OF Z & Y PARAMETERS

GPREC/EEE/EXPT.No. ECP-11

DATE: 18-10-2021

TABULAR COLUMN:

S.No.	V_1	I_1	I_2	$Z_{11}=V_1/I_1$		$Z_{21}=V_2/I_1$	
				Th.	Pr.	Th.	Pr.

S.No.	V_2	I_1	I_2	$Z_{12}=V_1/I_2$		$Z_{22}=V_2/I_2$	
				Th.	Pr.	Th.	Pr.

Y PARAMETERS (Short Circuit Admittance Parameters):

THEORY:

The Y-Parameters of a two port network are generally defined by expressing the port Currents I_1 and I_2 in terms of their voltages V_1 and V_2. The defining equations are expressed by considering any two variables as independent variables and expressing the other two variables in terms of these two variables and network parameters. The defining equations for hybrid parameters are

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

and $\quad I_2 = Y_{21} V_1 + Y_{22} V_2$

By short circuiting the output port $V_2 = 0$

$$\therefore \quad I_1 = Y_{11} V_1 + 0$$

Prepared by	Approved by	Page 5 of 9
Dr.G.Venkata Subba Reddy	Dr. T.Bramhananda Reddy	Revision No.0
	HEAD, EEE Dept.	

TITLE: DETERMINATION OF Z   GPREC/EEE/EXPT.No. ECP-11

& Y PARAMETERS   DATE: 18-10-2021

$$\Rightarrow \quad Y_{11} = \frac{I_1}{V_1}\Big|_{V_2=0}$$

Y_{11} is called as **Input admittance**.

And also when $V_2 = 0$

$$I_2 = Y_{21} \, V_1 + 0$$

$$\Rightarrow \quad Y_{21} = \frac{I_2}{V_1}\Big|_{V_2=0}$$

Y_{21} is called as **reverse transfer admittance**.

Similarly by short circuiting the input port $V_1 = 0$

$$\therefore \quad I_1 = Y_{12} \, V_2$$

$$Y_{12} = \frac{I_1}{V_2}\Big|_{V_1=0}$$

Y_{12} is called as **Forward Transfer admittance**

And also when $V_1 = 0$

$$I_2 = Y_{22} \, V_2$$

$$\Rightarrow \quad Y_{22} = \frac{I_2}{V_2}\Big|_{V_1=0}$$

Y_{22} is called as **Output admittance**

Methods of finding the Y PARAMETERS:

The defining equations for Y parameters are

$$I_1 = Y_{11} \, V_1 + Y_{12} \, V_2$$

$$I_2 = Y_{21} \, V_1 + Y_{22} + V_2$$

There are two methods for finding Y parameters (i) Direct method (ii) Indirect method

----

Prepared by   Approved by   Page 6 of 9
Dr.G.Venkata Subba Reddy   Dr. T.Bramhananda Reddy   Revision No.0
HEAD, EEE Dept.

TITLE: DETERMINATION OF Z & Y PARAMETERS	GPREC/EEE/EXPT.No. ECP-11 DATE: 18-10-2021

(i) Direct Method:-

Step (i) : By short circuiting the output port, V_2 will tends to zero and so express I_1 and I_2 in terms of V_1 and calculate

$$Y_{11} = \frac{I_1}{V_2} \ and \ Y_{22} = \frac{I_2}{V_1}$$

Step (ii) : By short circuiting the input port, V_1 will tends to zero and so express I_1 and I_2 in terms of V_2 and calculate

$$Y_{12} = \frac{I_1}{V_2} \ and Y_{22} = \frac{I_2}{V_2}$$

(ii) Indirect Method:-

Using node analysis or loop analysis express I_1 and I_2 in terms of V_1 and V_2 and then calculate Y parameters and these are expressed in mho.

CIRCUIT DIAGRAM:
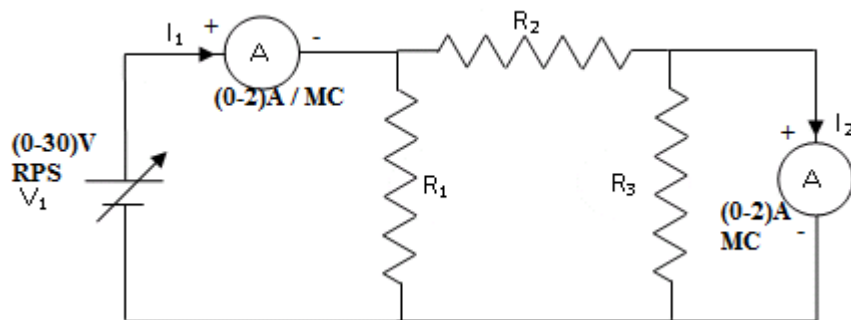
Fig. 4 To find Y_{11} & Y_{21}

TITLE: DETERMINATION OF Z   GPREC/EEE/EXPT.No. ECP-11
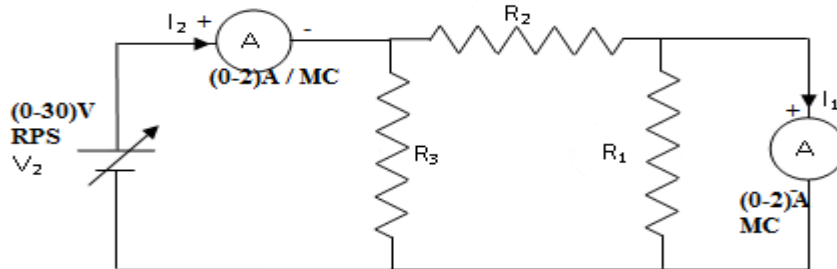
& Y PARAMETERS   DATE: 18-10-2021

Fig. 5 To find Y_{22} & Y_{12}

PROCEDURE:

1. Connections are to be made as per the circuit diagram shown in Fig.4.

2. Output port is short circuited and by varying V_1, readings are noted down.

3. Y_{11} & Y_{21} are calculated.

4. Input port is short circuited and by varying V_2 as per the circuit diagram shown in Fig.5, readings are noted down to calculate Y_{22} & Y_{12} .

TABULAR COLUMN:

S.No.	V_1	I_1	I_2	$Y_{11}=I_1/V_1$		$Y_{21}=I_2/V_1$	
				Th.	Pr.	Th.	Pr.

Sl.no.	V_2	I_1	I_2	$Y_{12}=I_1/V_2$		$Y_{22}=I_2/V_2$	
				Th.	Pr.	Th.	Pr.

TITLE: DETERMINATION OF Z & Y PARAMETERS	GPREC/EEE/EXPT.No. ECP-11
	DATE: 18-10-2021

RESULT: Impedance and admittance parameters are determined.

Questions

1. What are the defining equations of Z and Y parameters
2. What are the conditions for symmetry
3. What are the conditions for reciprocity
4. What is the condition to be used to find Z parameters
5. What is the condition to be used to find Y parameters

TITLE: DETERMINATION OF ABCD PARAMETERS			GPREC/EEE/EXPT.No. ECP-12 DATE: 18-10-2021	

OBJECTIVE: To Determine Transmission line parameters (ABCD)

APPARATUS:

Sl.No.	Unit	Range	Type	No.
1	Regulated Power supply	(0-30) V/(0-2)A		1
2	Voltmeter	(0-30) V	MC	1
3	Ammeter	(0-2)A	MC	2
4	Rheostat			3
5	Connecting Wires			Reqd.

THEORY:

ABCD-PARAMETERS

In transmission parameters the voltage and current at the sending end will be expressed with the voltage and current at the receiving end. These parameters will be used to study the performance of the transmission lines.

The defining equations for ABCD parameters are

$$V_1 = AV_2 - BI_2$$

And $I_1 = CV_2 - DI_2$

Here $-I_2$ is considered because the direction of I_2 is opposite to that of I_1.

By open circuiting the output port, $I_2 = 0$

$$\therefore V_1 = AV_2$$

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

A is called as **open circuit voltage gain.**

Also when $I_2 = 0$

$$I_1 = CV_2$$

TITLE: DETERMINATION OF ABCD PARAMETERS	GPREC/EEE/EXPT.No. ECP-12 DATE: 18-10-2021

$$\therefore \quad C = \frac{I_1}{V_2}\Big|_{I_2=0}$$

C is called as **open circuit transfer impedance**.

Similarly by short circuiting the out put port $V_2 = 0$

$$V_1 = -BI_2$$

$$\Rightarrow \quad -B = \frac{V_1}{I_2}\Big|_{V_2=0}$$

$$\Rightarrow \quad -\frac{1}{B} = \frac{I_2}{V_1}\Big|_{V_2=0}$$

It is also called as **short circuit transfer admittance**.

Also when $V_2 = 0$.

$$I_1 = -DI_2$$

$$\Rightarrow \quad -D = \frac{I_1}{I_2}\Big|_{V_2=0}$$

$$\Rightarrow \quad -\frac{1}{D} = \frac{I_2}{I_1}\Big|_{V_2=0}$$

It is also called as **short circuit current gain**.

Method of Finding ABCD Parameters

Transmission parameters are also known as ABCD parameters. The defining equations for ABCD parameters are

$$V_1 = AV_2 - BI_2$$

$$I_1 = CV_2 - Di_2$$

$$\begin{bmatrix} V_1 \\ I_1 \end{bmatrix} = \begin{bmatrix} A & B \\ C & D \end{bmatrix} \begin{bmatrix} V_2 \\ -I_2 \end{bmatrix}$$

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 2 of 5
Revision No.0

TITLE: DETERMINATION OF ABCD PARAMETERS	GPREC/EEE/EXPT.No. ECP-12 DATE: 18-10-2021

(i) Direct Method

Step (i): By open circuiting the out put port, I_2 tends to zero and so A and C can be calculated

$$A = \frac{V_1}{V_2} \ , \qquad \text{Called reverse voltage ratio}$$

$$C = \frac{I_1}{V_2} \ , \qquad \text{Called reverse transfer admittance.}$$

Step (ii): By short circuiting the out put port, V_2 tends to zero and so B and D can be calculated.

$$- B = \frac{V_1}{I_2} \qquad \text{Called reverse transfer impedance}$$

$$- D = \frac{I_1}{I_2} \qquad \text{Called reverse current ratio}$$

A and D parameter has no units, parameter C in mho and parameter B is ohm.

CIRCUIT DIAGRAM:
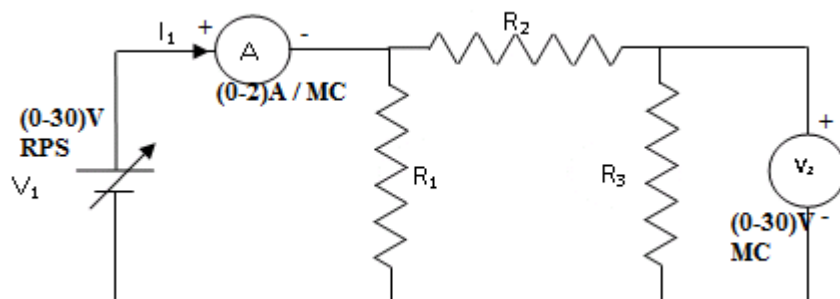
Fig.1 To find Parameter 'A' & 'C'

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 3 of 5
Revision No.0

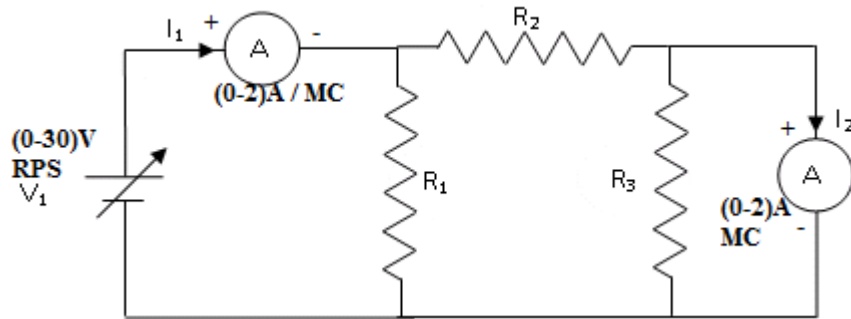| TITLE: DETERMINATION OF ABCD PARAMETERS	GPREC/EEE/EXPT.No. ECP-12 DATE: 18-10-2021

Fig.2 To find Parameter 'B' & 'D'

PROCEDURE:

TO FIND A, C PARAMETERS:

1. Connections are made as per the circuit diagram shown in Fig.1.
2. By varying V_1, the corresponding readings of V_2 and I_1 are noted down.
3. A and C parameters are calculated.

TO FIND B, D PARAMETERS:

1. Connections are made as per the circuit diagram shown in Fig.2.
2. By varying V_1, the corresponding readings of I_1 and I_2 are noted down.
3. B and D parameters are calculated.

TABULAR COLUMN:

S.No	V_1	V_2	I_1	A		C	
				Th.	Pr.	Th.	Pr.

S.No	V_1	I_1	I_2	B		D	
				Th.	Pr.	Th.	Pr.

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 4 of 5
Revision No.0

TITLE: DETERMINATION OF ABCD PARAMETERS

GPREC/EEE/EXPT.No. ECP-12

DATE: 18-10-2021

RESULT: ABCD parameters are determined.

Questions:
1. What are the defining equations of ABCD parameters
2. What is the significance of the two port parameters?
3. What is the applications of ABCD parameters?
4. What is the condition for symmetrical network?

TITLE: DIGITAL SIMULATION OF AN ELECTRIC CIRCUIT (INCLUDING DEPENDANT SOURCES) TO FIND NODE VOLTAGES AND BRANCH CURRENTS USING PSPICE.	GPREC/EEE/EXPT.No. ECP-13 DATE: 18-10-2021

OBJECTIVE: To simulate the given electrical circuit and to find the node voltages and branch currents using Pspice software.

CIRCUIT DIAGRAM:
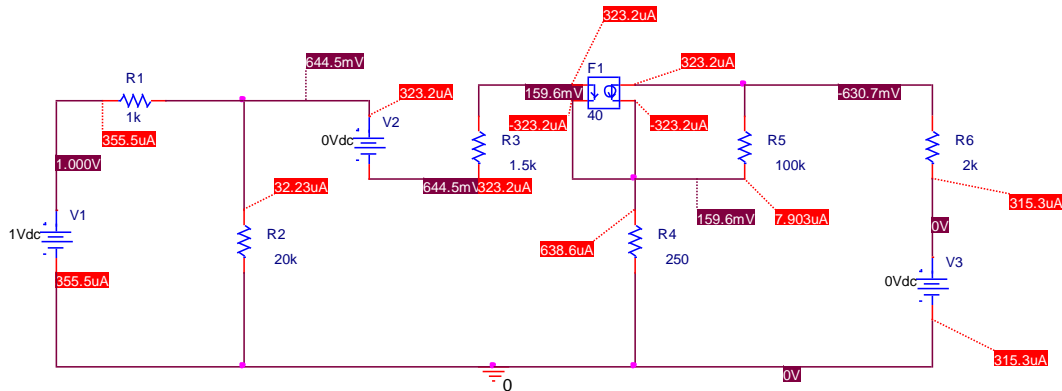
Fig.1 Circuit Diagram

Output:

NODE	VOLTAGE	NODE	VOLTAGE	NODE	VOLTAGE
(N00295)	1.0000	(N00325)	.6445	(N00355)	.6445
(N00382)	.1596	(N00417)	.1596	(N00506)	-.6307	(N00560)	0.0000

PROCEDURE:

1. Connections are made as per the circuit diagram shown in Fig.1
2. With the applied voltage simulate the circuit.
3. Observe the node voltages and branch currents.

RESULT: Given circuit is simulated and the node voltages and branch currents are observed.

Prepared by   Approved by   Page 1 of 2
Dr.G.Venkata Subba Reddy   Dr. T.Bramhananda Reddy   Revision No.0
HEAD, EEE Dept.

TITLE: DIGITAL SIMULATION OF AN ELECTRIC CIRCUIT (INCLUDING DEPENDANT SOURCES) TO FIND NODE VOLTAGES AND BRANCH CURRENTS USING PSPICE.	GPREC/EEE/EXPT.No. ECP-13 DATE: 18-10-2021

Questions:

1. Define node.

2. Explain the procedure to find branch currents using current division.

3. Give the difference between ideal and dependent sources?

4. Give the syntax for dependent sources using Pspice?

5. Why the dummy voltage is mentioned with 0V and what is the purpose of it?

TITLE: DIGITAL SIMULATION OF	GPREC/EEE/EXPT.No. ECP-14
AN ELECTRIC CIRCUIT TO FIND	DATE: 18-10-2021
TRANSIENT RESPONSE	

OBJECTIVE: To simulate the given electrical circuit and to find the transient response using Pspice software.
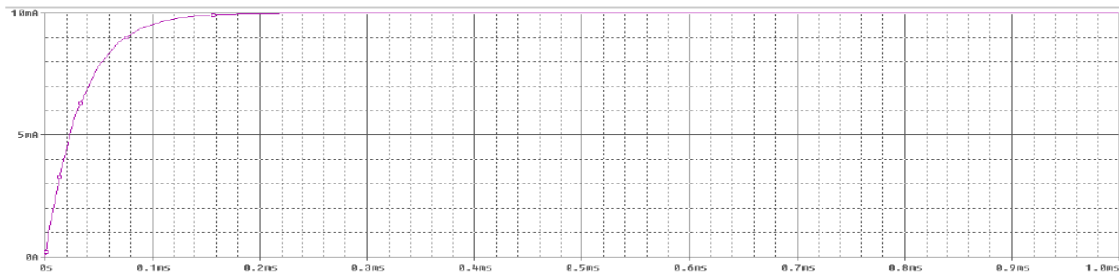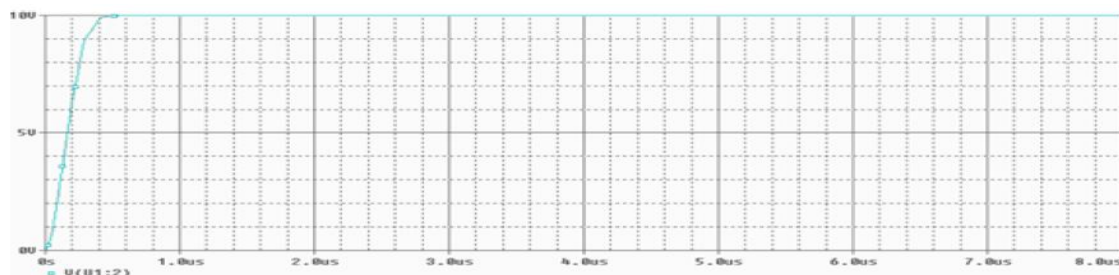
RL SERIES CIRCUIT:
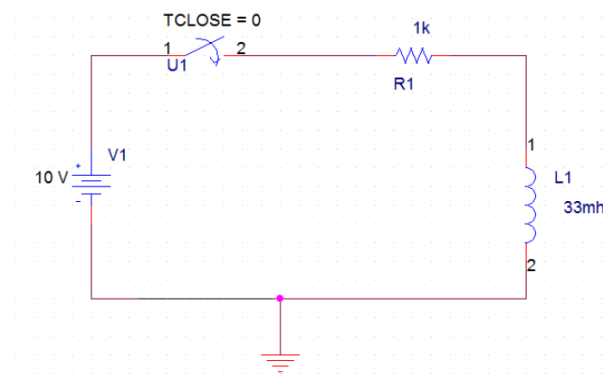
Fig.1 RL series circuit

Output:

Fig.2 Current i(t)

TITLE: DIGITAL SIMULATION OF AN ELECTRIC CIRCUIT TO FIND TRANSIENT RESPONSE

GPREC/EEE/EXPT.No. ECP-14

DATE: 18-10-2021

Fig.3 Voltage across resistor

RC SERIES CIRCUIT:
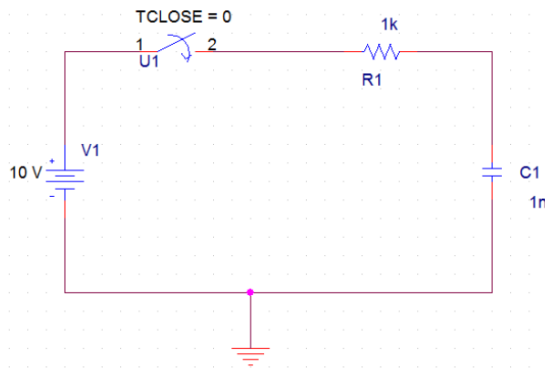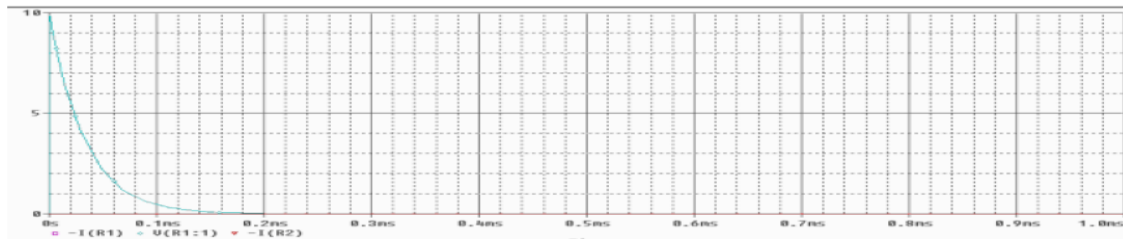

Fig.4 RC series circuit
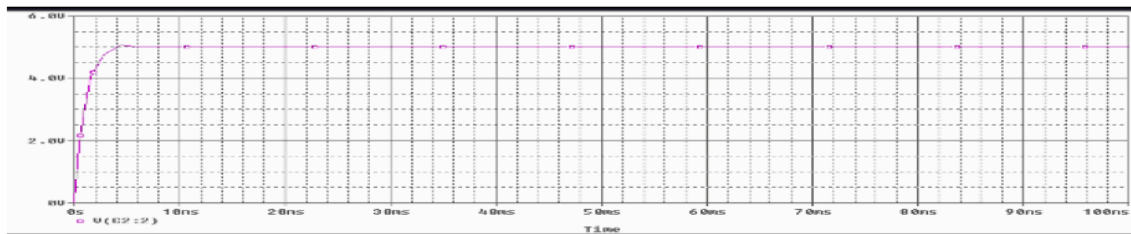
Output:

Fig.5 Current i(t)

Fig.6 Voltage across resistor

PROCEDURE:

1. Connections are made as per the circuit diagram shown in Fig.1 and Fig.4

2. With the applied voltage simulate the circuit.

TITLE: DIGITAL SIMULATION OF AN ELECTRIC CIRCUIT TO FIND TRANSIENT RESPONSE

GPREC/EEE/EXPT.No. ECP-14

DATE: 18-10-2021

3. Observe the current waveform and voltage across the resistor.

RESULT: RL and RC circuits are simulated and the transient response is observed.

Questions:

1. Define time constant of RL Circuit.

2. Define time constant of RC Circuit.

3. What is the time taken for the current to reach its steady state value in RL charging circuit?

4. What is the time taken for the voltage to reach its steady state value in RC charging circuit?

5. What is the voltage across the capacitor after 10τ sec in an RC discharging circuit?

TITLE: DIGITAL SIMULATION OF SERIES AND PARALLEL RESONANCE USING PSPICE	GPREC/EEE/EXPT.No. ECP-15 DATE: 18-10-2021

OBJECTIVE: To simulate the given RLC series and parallel circuit and to draw the frequency response curves.

RLC SERIES CIRCUIT:

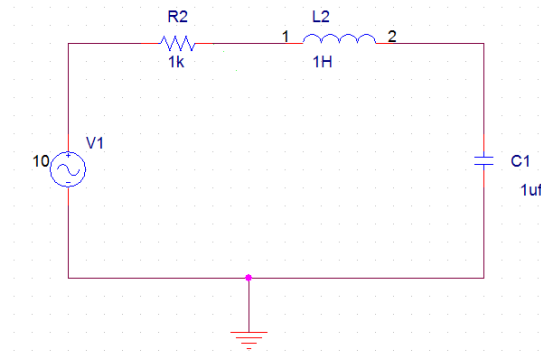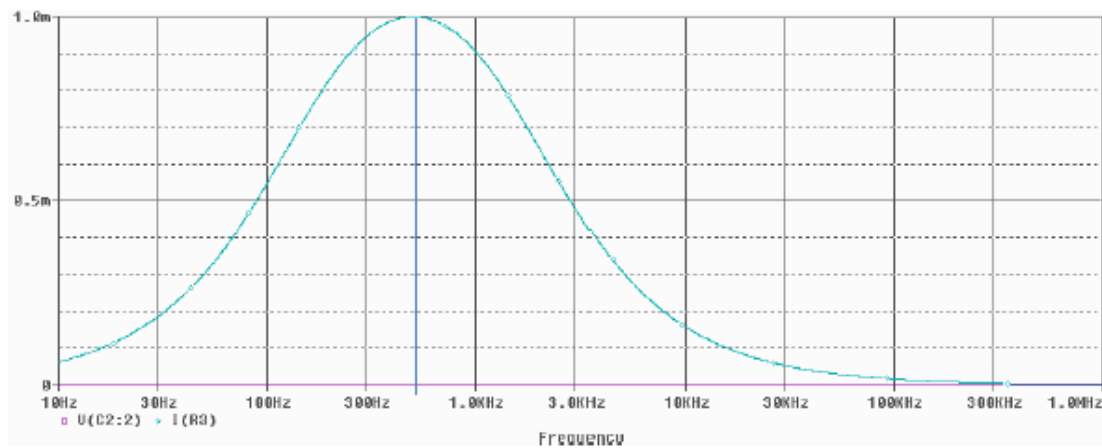**Fig.1 RLC series circuit**

Output:

Fig.2 Frequency response curve

TITLE: DIGITAL SIMULATION OF SERIES AND PARALLEL RESONANCE USING PSPICE

GPREC/EEE/EXPT.No. ECP-15

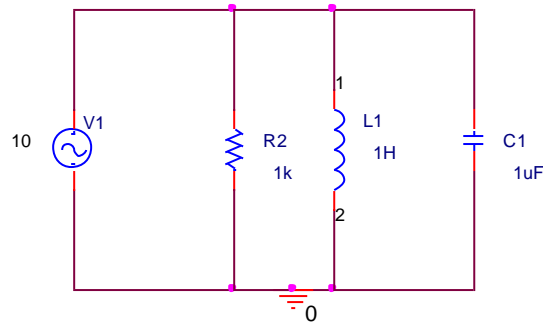DATE: 18-10-2021

RLC PARALLEL CIRCUIT:
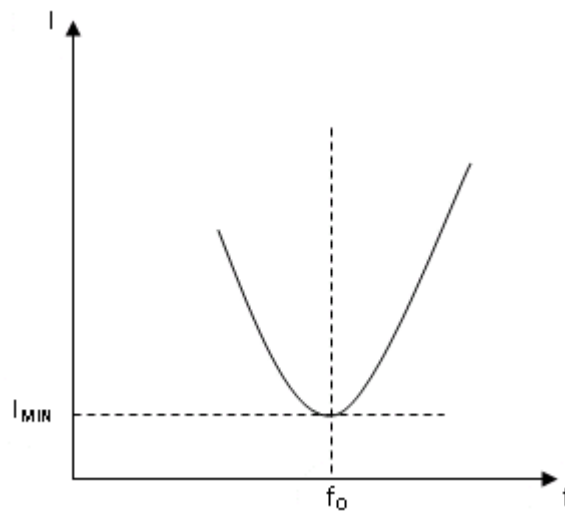

Fig.3 RLC Parallel circuit

Output:

Fig.4 Current i(t)

PROCEDURE:

1. Connections are made as per the circuit diagram shown in Fig.1 and Fig.3

2. With the applied voltage simulate the circuit.

3. Observe the frequency Vs. current waveform and find the resonant frequency.

Prepared by
Dr.G.Venkata Subba Reddy

Approved by
Dr. T.Bramhananda Reddy
HEAD, EEE Dept.

Page 2 of 3
Revision No.0

TITLE: DIGITAL SIMULATION OF SERIES AND PARALLEL RESONANCE USING PSPICE	GPREC/EEE/EXPT.No. ECP-15 DATE: 18-10-2021

RESULT: RLC series and parallel circuits are simulated and the resonant frequency is evaluated.

Questions:

1. Derive the formula for resonance frequency
2. Write the formula for quality factor.
3. Write the relation between fr, Q and band width.
4. At the time of resonance, what is the power factor of the circuit?
5. At the time of resonance, current in RLC series circuit will be maximum. Why?