NETWORK ANALYSIS AND SIMULATION LABORATORY–23EC51

Prepared By
Mr.G.Venkata Rao
Associate Professor,

Mr.K.V.Ashok
Sr. Assistant Professor

Mr.M.Sambasiva Reddy
Sr. Assistant Professor

Department of Electronics and Communication Engineering
LAKIREDDY BALI REDDY COLLEGE OF ENGINEERING
(AUTONOMOUS)
L.B.REDDY NAGAR, MYLAVARAM-521230

Affiliated to JNTUK, Kakinada & Approved by AICTE, New Delhi
NAAC Accredited with "A" grade, Accredited by NBA, Certified by ISO 21001-2018

Course Objectives:

• To gain hands on experience in verifying Kirchhoff’s laws and network theorems
• To analyze transient behavior of circuits
• To study resonance characteristics
• To determine 2-port network parameters

Course Outcomes:

CO1: Demonstrate fundamental circuit laws, network theorems, node and mesh analysis of electrical circuits (**Apply**).
CO2: Design resonance circuit for given specifications (**Analyze**).
CO3: Measure time constants of RL & RC circuits (**Apply**).
CO4: Analyze the 1^{st} and 2^{nd} order circuits with respect to parameter variation (**Analyze**).
CO5: Characterize and model the network in terms of all network parameters (**Apply**).

Hardware Requirements: Regulated Power supplies, Analog/Digital Function Generators, Digital Multimeters, Decade Resistance Boxes/Rheostats, Decade Capacitance Boxes, Ammeters (Analog or Digital), Voltmeters (Analog or Digital), Active & Passive Electronic Components.

Software requirements: Multisim/ Pspice/Equivalent simulation software tool, Computer Systems with required specifications.

References:
1. Network Analysis – ME Van Valkenburg, Prentice Hall of India, revised 3rd Edition, 2019. 2. Engineering Circuit Analysis by William H. Hayt, Jack Kemmerly, Jamie Phillips, Steven M. Durbin, 9th Edition 2020.

Network Analysis and Simulation Laboratory

List of experiments

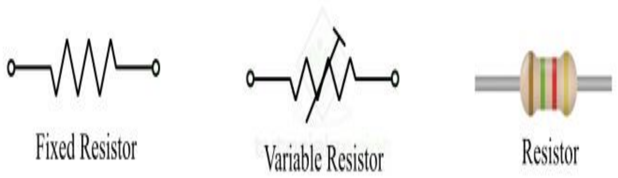
1. Study of components of a circuit and Verification of KCL and KVL.
2. Verification of mesh and nodal analysis for AC circuits
3. Verification of Superposition, Thevenin's & Norton theorems for AC circuits.
4. Verification of maximum power transfer theorem for AC circuits.
5. Verification of Tellegen's theorem for two networks of the same topology.
6. Study of DC transients in RL, RC and RLC circuits.
7. To study frequency response of various 1st order RL & RC networks.
8. To study the transient and steady state response of a 2^{nd} order circuit by varying its various parameters and studying their effects on responses.
9. Find the Q Factor and Bandwidth of a Series and Parallel Resonance circuit.
10. Determination of open circuit (Z) and short circuit (Y) parameters.
11. Determination of hybrid (H) and transmission (ABCD) parameters.
12. To measure two port parameters of a twin-T network and study its frequency response.

1. Study of components of a circuit and Verification of KCL and KVL

Aim: To study various components of a circuit and verify KCL and KVL.
Apparatus Required:

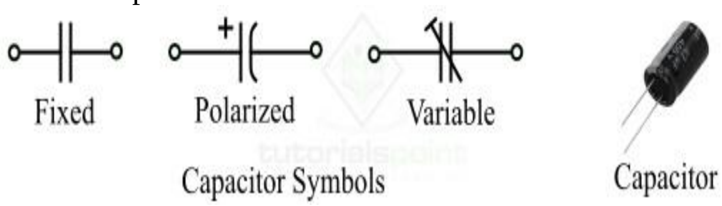
Sl.No.	Apparatus	Range	Quantity
1	RPS (regulated power supply)	(0-30V)	1
2	Resistance	5KΩ,10KΩ,15KΩ	3
3	Ammeter	(0-30mA)MC	3
4	Voltmeter	(0-30V)MC	3

. **Theory:**
Resistor: An electric circuit element that introduces an electrical friction or resistance in the path of electric current is called a resistor. The characteristic by which it oppose the flow of current is known as resistance. The resistance of a resistor is denoted by symbol *R* and measured in Ohms (ΩΩ). The typical circuit symbol of a resistor is shown in the following figure.

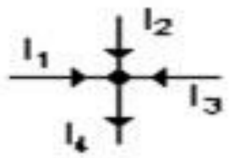


Inductor: Inductor is basically a wire of finite length twisted into a coil. An inductor is also a basic circuit element that used to introduce inductance in an electrical or electronic circuit. The inductor has a property, known as inductance, which opposes any change in the electric current. The circuit symbol of a typical inductor is shown in the following figure.

Capacitor: An electric circuit element that has an ability of storing electrical energy in the form of electric field is called a capacitor. The property of the capacitor by virtue of which it store electrical energy is known as capacitance.



Kirchhoff's Current Law: This law is also called Kirchhoff's point rule, Kirchhoff's junction rule (or nodal rule), and Kirchhoff's first rule. It states that, "In any network of conductors, the algebraic sum of currents meeting at a point (or junction) is zero".



$$I_1 + I_2 + I_3 = I_4$$

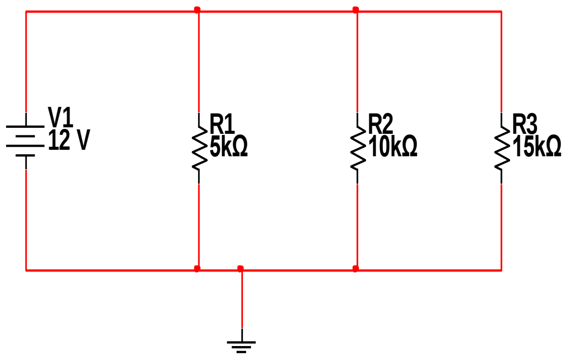
Kirchhoff's Voltage Law: This law is also called Kirchhoff's second law, Kirchhoff's loop (or mesh) rule, and Kirchhoff's second rule and states that, "The algebraic sum of all IR drops and EMFs in any closed loop (or mesh) of a network is zero".

$$\sum_{k=1}^{n} V_k = 0$$

Here, n is the total number of voltages measured.

Together, Kirchhoff's Voltage and Current Law is a formidable pair of tools useful in analysing electric circuits.

CIRCUIT DIAGRAM FOR KCL:



Procedure for KCL:

1. Give the connections as per the circuit diagram.
2. Set a particular value in RPS.
3. Note down the corresponding ammeter readings.
4. Repeat the same for different voltages.

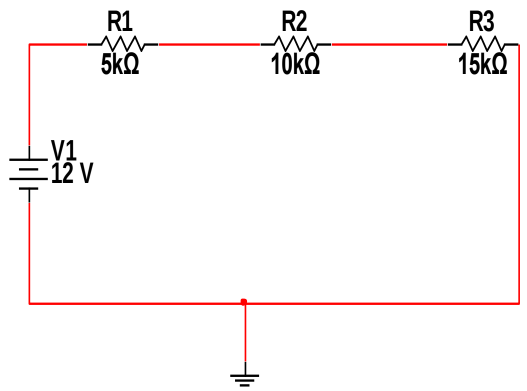
KCL - Theoretical Values:

S.No	Voltage E	Current			I1 = I2 + I3
		I1	I2	I3	
	Volts	mA	mA	mA	mA
1					
2					
3					
4					
5					

KCL - Practical Values:

S.No	Voltage E	Current			I1 = I2 + I3
		I1	I2	I3	
	Volts	mA	mA	mA	mA
1					
2					
3					
4					
5					

Circuit diagram for KVL:



Procedure for KVL:

1. Give the connections as per the circuit diagram.
2. Set a particular value in RPS for both E1 and E2.
3. Note all the voltage reading for V1, V2 and V3.
4. Repeat the same for different voltages.

KVL – Theoretical Values

S.No	RPS		Current			KVL E1 = V1 + V2
	E1	E2	V1	V2	V3	
	Volts	Volts	Volts	Volts	Volts	Volts
1						
2						
3						
4						
5						

KVL – Practical Values

S.No	RPS		Current			KVL E1 = V1 + V2
	E1	E2	V1	V2	V3	
	Volts	Volts	Volts	Volts	Volts	Volts
1						
2						
3						
4						
5						

Result: Thus KVL, KCL verified theoretically and practically.

2. Verification of mesh and nodal analysis for AC circuits

Aim: To verify mesh and nodal analysis for AC circuits.

Apparatus Required:

1.AC Power supply
2.Resistor-3Ω
3.Decade Inductance Box-3No.
4.Decade Capacitance Box-2No.

Theory:

In Mesh analysis, we will consider the currents flowing through each mesh. Hence, Mesh analysis is also called as **Mesh-current method**.
A **branch** is a path that joins two nodes and it contains a circuit element. If a branch belongs to only one mesh, then the branch current will be equal to mesh current.
If a branch is common to two meshes, then the branch current will be equal to the sum (or difference) of two mesh currents, when they are in same (or opposite) direction.
In Nodal analysis, we will consider the node voltages with respect to Ground. Hence, Nodal analysis is also called as **Node-voltage method**.

Circuit Diagrams:

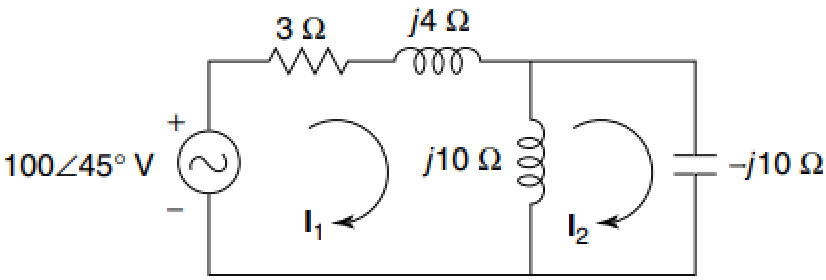


Fig-1: Circuit diagram for mesh analysis

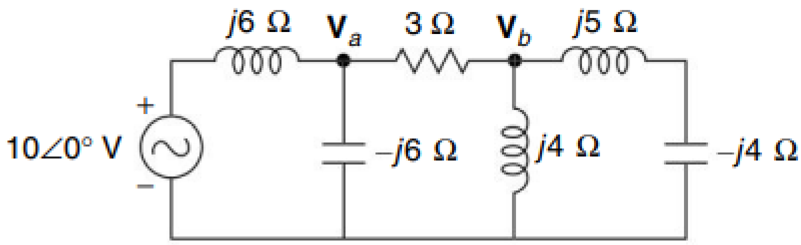


Fig-2: Circuit diagram for nodal analysis

Procedure :

1. Connect the circuits as shown in Fig-1 and 2.
2. Measure the mesh currents I_1,I_2 and node voltages V_a,V_b
3. Note down the values.

TABULAR COLUMNS :

Mesh Analysis

S.No	I_1(A)	I_2(A)
1		

Nodal Analysis

S.No	V_a(V)	V_b(V)
1		

Result: Thus Mesh and Nodal Analysis techniques are used and verified for AC circuits.

3. Verification of Superposition, Thevenin's & Norton theorems for AC circuits

Aim: To verify Superposition, Thevenin's & Norton theorems for AC circuits.

Apparatus Required:

S.NO.	NAME OF THE EQUIPMENT	RANGE	QUANTITY
1.	Regulated Power Supply (RPS)	(0 – 30) V	1
2.	Rheostat	50 **Ω**/5 A	1
		25 **Ω**/5 A	1
		28 **Ω**/5 A	1
3.	Ammeter	(0-2) A , MC	1
	Voltmeter	(0-30) V , MC	1
4.	Connecting Wires	--	Required

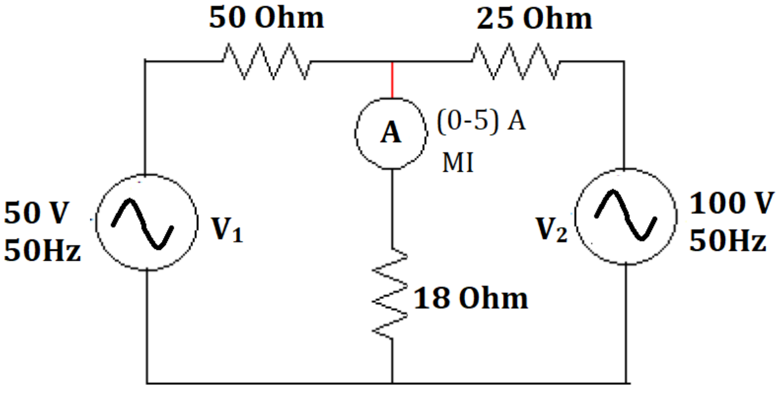
Theory:

Thevenin's theorem states that any two terminal linear networks having anumber of voltage-current sources and resistances(impedances) can be replaced by a simpleequivalent circuit consisting of a single voltage source in series with aresistance(impedance), 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 theterminals with all the energy sourcesare replaced by their internalresistances.

The superposition theorem states that 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 individual sources acting alone, while the other sources are non-operative; that is, while considering the effect of individual sources, other ideal voltage sources and ideal current sources in the network are replaced by short circuit and open circuit across their terminals respectively. This theorem is valid only for linear systems.

Norton's theorem states that any two terminal linear networks with voltage-current sources and resistances (impedances) can be replaced by an equivalent circuit consisting of a current source in parallel with a resistance (impedance). The value of current source is the short circuit current (Norton's current) between the two terminals of the network and resistance (Norton's resistance or impedance) is the equivalent resistance (impedance) measured between the terminals of the network with all energy sources are replaced by their internal resistances.

Circuit diagrams:



(b)

Fig-1: Circuit diagram for verifying superposition theorem

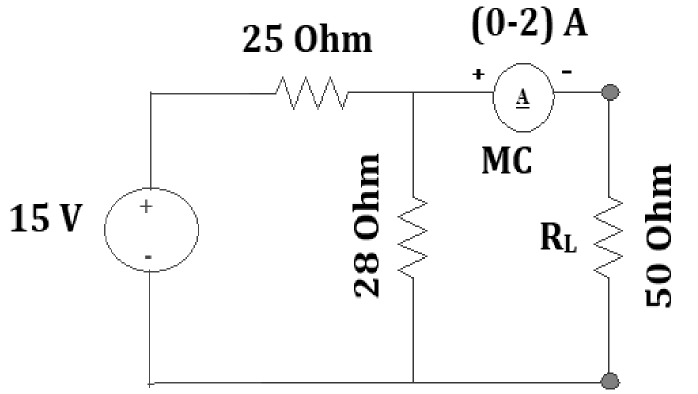


Fig-2: Circuit diagram for verifying Thevenin's theorem

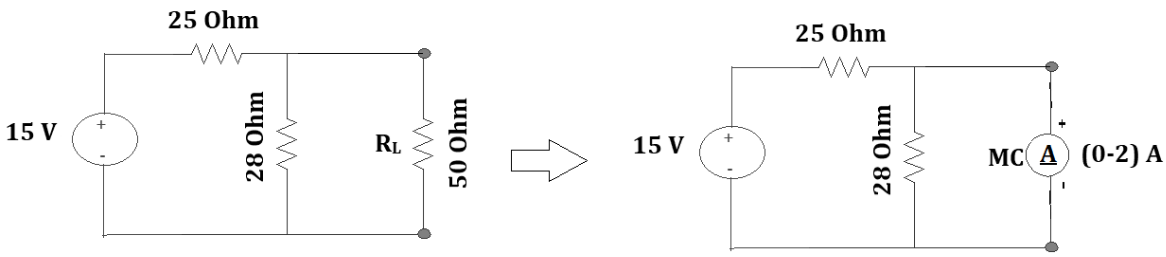


Fig-3: Circuit diagram for verifying Norton's theorem

Procedure:

Superposition theorem

1. Connect the circuit as shown in the Circuit diagram (a) or (b)
2. Switch ON the supply voltage (DC or AC) and apply V_1 and V_2.
3. Note down the Ammeter reading as I.
4. Make V_1= 0and apply V_2, note down the ammeter reading as I_1.
5. Make V_2= 0and apply V_1, note down the ammeter reading as I_2.
6. Verify the condition $I = I_1 + I_2$

Thevenin's theorem

1. Connect the circuit as shown in the Circuit diagram
2. Switch ON the supply voltage and apply the voltage as 15V.
3. Note down the Ammeter reading as I.
4. Open the load resistance terminals and measure the voltage (V_{Th}) across the open circuited terminals.
5. Short the voltage source and measure the Thevenin's resistance (R_{Th}) across the open circuited terminals.
6. Connect the Thevenin's equivalent circuit by connecting the load resistance across it.
7. Measure the load current (I_L) through the load resistance (R_L).
8. Verify $I = I_L$.

Norton's theorem

1. Connect the circuit as shown in the Circuit diagram
2. Short the load resistance terminals and measure the current (I_N) through the short- circuited terminals by connecting an ammeter.
3. Open the load terminals and short the voltage source.
4. Measure the Norton's resistance (R_N) across the open circuited terminals.
5. Draw the Norton's equivalent circuit by connecting the load resistance across it.

Observation Tables:

Superposition theorem

S.NO.	Parameter	V_1 = 50V, V_2 = 100V	V_1 = 0V, V_2 = 100V	V_1 = 50V, V_2 = 0V
1.	Current through 18 Ohm (Theoretical)			
2.	Current through 18 Ohm (Practical)			

Thevenin's theorem

Parameter	Theoretical	Practical
V_{Th} (V)		
R_{Th}(Ω)		
I_L (A)		

Norton's theorem

Parameter	Theoretical	Practical
I_N (A)		
R_N (Ω)		

Result: Thus, Superposition, Thevenin's & Norton theorems for AC circuits verified practically.

4. Verification of maximum power transfer theorem for AC circuits.

Aim: To verify the maximum power transfer theorem for AC circuit.

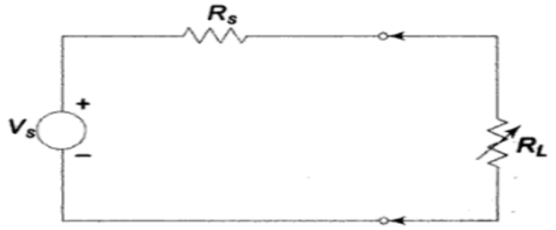
Apparatus Required:

S.NO.	NAME OF THE EQUIPMENT	RANGE	QUANTITY
1.	Regulated Power Supply (RPS)	(0 – 30) V	1
2.	Rheostat	50 **Ω**/5 A	1
		25 **Ω**/5 A	1
3.	Ammeter	(0-2) A , MC	1
		(0-2) A , MI	1
4.	Voltmeter	(0-30) V , MC	1
		(0-30) V , MI	1
5.	Dimmerstat	(0-230) V	1
6.	Connecting Wires	--	Required

Theory:

The maximum Power Transfer Theorem states that maximum power is delivered from a source to a load when the load resistance is equal to the source resistance.

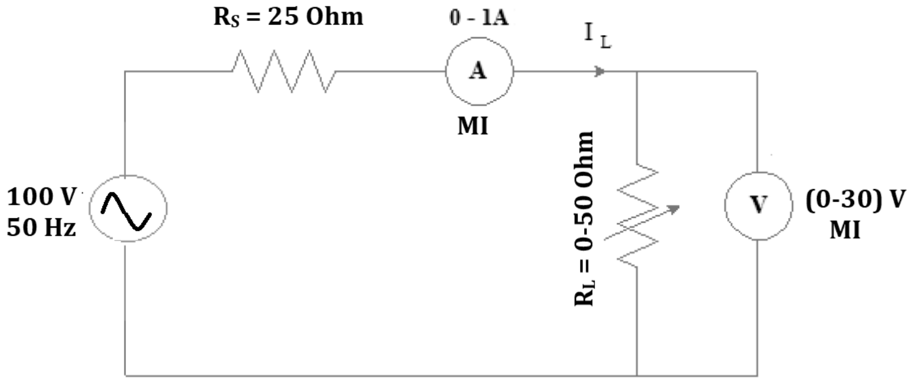
In Fig., assume that the load resistance is variable.



Maximum Power delivered to the load resistance,

$$P_{max} = \frac{V_S^2}{4R_S}$$

CIRCUIT DIAGRAM :



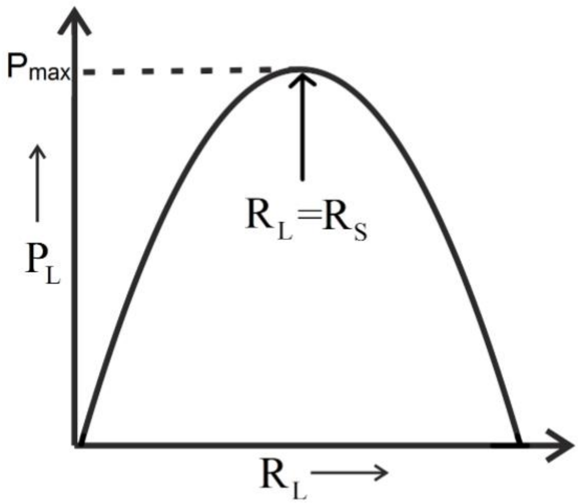
PROCEDURE :

1. Connect the circuit as shown in the Circuit diagram.
2. Switch ON the supply voltage (DC or AC) and apply the given voltage.
3. Vary the load resistance (R_L) from 0 to maximum value in steps and note down the readings of volt meter (V_L) and ammeter (I_L).
4. Calculate the power (P_L) consumed by R_L at each step.
5. Tabulate the readings.
6. Draw the graph between Load resistance (R_L) and Power (P_L).

TABULAR COLUMNS :

AC EXCITATION			
R_L (Ω)	V_L (V)	I_L (A)	$P_L = V_L I_L$ (W)

MODEL GRAPH:



PRECAUTIONS :

1. Check the connections before switching ON the supply.
2. The terminals of the rheostat should be properly connected.
3. Disconnect the connections properly after the completion of experiment.

RESULT :

The maximum power transfer theorem is verified for AC circuits theoretically and practically.

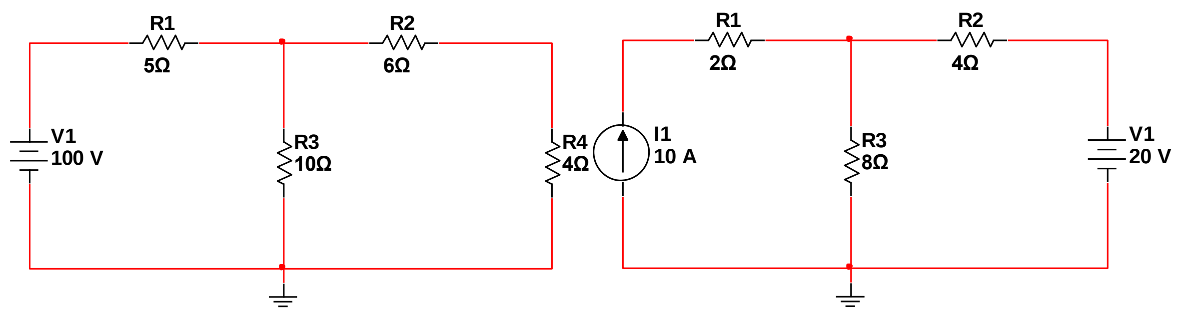
5. Verification of Tellegen's theorem for two networks of the same topology

AIM: Verification of Tellegen's theorem for two networks of the same topology.

APPARATUS REQUIRED:

1. Bread Board
2. RPS (0-30)V
3. Decade Resistance Box- 4No.
4. Connecting Wires

CIRCUIT DIAGRAMS:



THEORY

For any given time, the sum of power delivered to each branch of any electric network is zero. Thus for K^{th} branch, this theorem states that,

$$\sum_{k=0}^{n} v_k' i_k = 0 \qquad \text{(or)} \qquad \sum_{k=0}^{n} v_k i_k' = 0$$

PROCEDURE:

1. First measure branch currents and voltages.
2. Apply Tellegen's theorem and find the sum of product of voltages in one circuit and currents in other circuit.
3. The sum of all power i.e. equal or less than zero.
4. Hence, the Tellegen's theorem is verified.

TABULATION:

S.No	V_1i_1'	V_2i_2'	V_3i_3'	V_4i_4'	V_5i_5'	$S_1= V_1i_1'+ V_2i_2'+ V_3i_3'+ V_4i_4'+ V_5i_5'$
1.						
2.						
3.						
4.						
5.						
6.						
7.						

RESULT :

Tellegen's theorem is verified using both hardware.

6. Study of DC transients in RL, RC and RLC circuits

AIM: To study the transient response of a series RL,RC and RLC circuits and understand the time constant concept with DC Power Supply.

APPARATUS:

1.Multisim Simulation Software
2. DC Voltage Source
3.Resistor – 1KΩ
4. Inductor- 50µH,50mH
5.Capacitor - 50µF

CIRCUIT DIAGRAM:

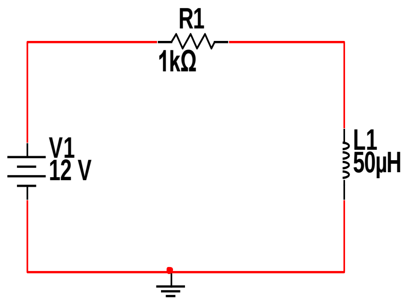


Fig1: RL Circuit

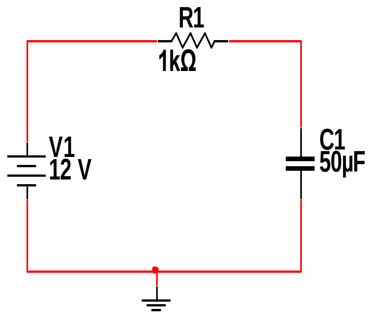


Fig2: RC Circuit

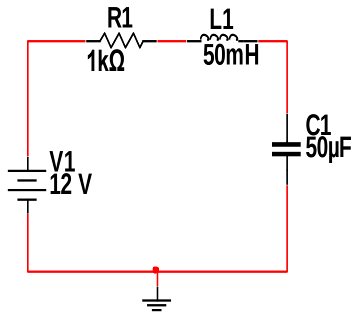
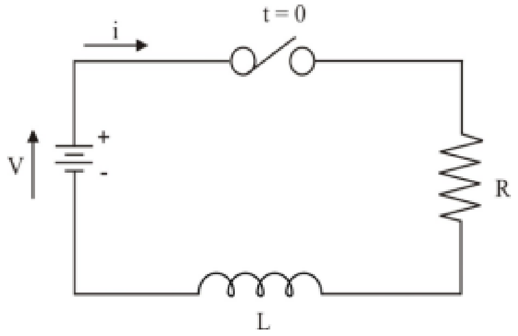


Fig3: RLC Circuit

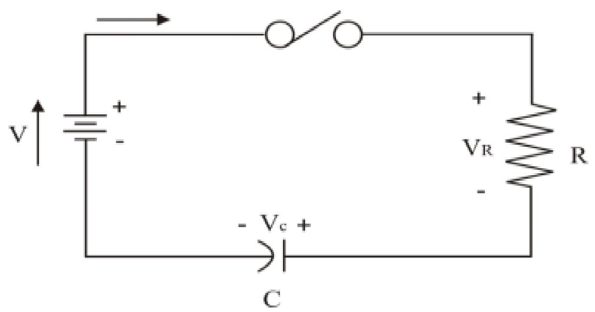
THEORY

When a circuit is switched from one condition to another either by a change in the applied voltage or a change in one of the circuit elements, there is a transitional period during which the branch currents and voltage drops change from their former values to new ones. After this transition interval called the transient, the circuit is said to be in the steady state.

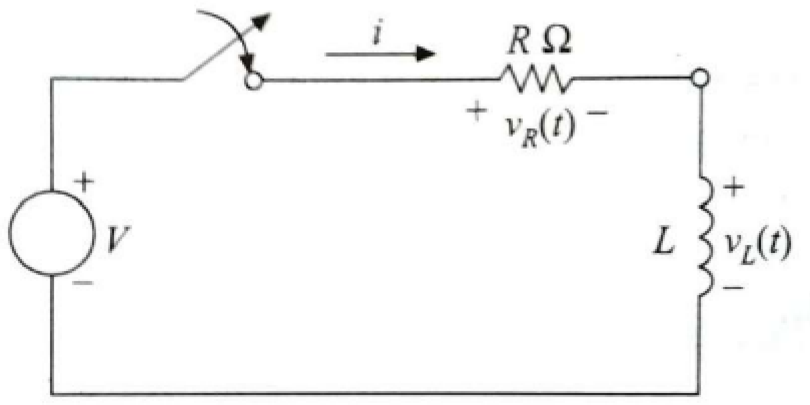
a)For RL Circuit:



b) For RC Circuit



c) For RLC Circuit



PROCEDURE:

Make the connections on the NV6514 Transient Analysis of RC/RL Circuits as shown in figure.

1. Make sure that the toggle switch connected across the DC Supply is in downward position.
2. Connect +5 V DC Power Supply to the input of RL Circuit i.e. connect +5 V terminal to terminal 3 and Gnd terminal to terminal 4.
3. Connect the mains cord to the Trainer and switch ‘On’ the mains supply.
4. Now switch ‘On’ the power switch of the trainer.
5. Switch the toggle switch in upward direction so that DC Supply will connect to the RL circuit.
6. Connect DSO across inductor i.e. across TP1 and TP2. Keep DSO at 200 µs or 500 µs Time Base.
7. Observe the transient response (firstly sudden increase in voltage and then exponentially decaying) on DSO. Now immediately press RUN/STOP Switch of DSO to hold the response shown on the DSO screen.
8. Now switch the toggle switch in downward direction so that resistor, R will short with Inductor, L.
9. Now observe the response till it (first sudden increase of voltage in negative direction and then exponentially rising towards reference level) reaches reference level of DSO.

Calculations :

a)For RL Circuit:
Theoretically,

Time Constant, TC =L/R = ……………..
Where,
L = 141.37mH, R =1k

Practically (on DSO screen),
In the charging circuit, One Time Constant is the time by which the inductor attains the 36.8% of maximum voltage (in our case, +5 V).
Time Constant or Time required to decay to 36.8% of 5 V (i.e. 1.84 V)= …………
b)For RC circuit:
Theoretically,

Time Constant, TC = R C = ……………..
where R = 10 k , C = 1000 µF.

Practically (on DSO screen),
In the charging circuit, Time Constant is the time by which the capacitor attains the 63.2% of steady state voltage or final value (in our case, +5 V).

Time Constant or Time required to rise to 63.2% of 5 V (i.e. 3.16 V) =…………
In the discharging circuit, Time Constant is time by which the capacitor discharges to 36.8% of its initial steady state voltage (in our case, +5 V).
Time Constant or Time required to decay to 36.8% of 5V (i.e. 1.84 V) = ………….
2. Similarly, 2TC is the time required to achieve 86.5% of final or initial value of voltage.

Practically, 2TC = ……….
Theoretically, 2TC =……...

3. After 5TC, the voltage reach their final values which is also called steady state response..

Practically, 5TC = ……….
Theoretically, 5TC =……..

Result: The Transient Response of Series RL,RC & RLC cCircuits are verified.

7. To study frequency response of various 1st order RL & RC networks.

Aim: To study the characteristics and frequency response of various 1st order RL & RC networks.

Apparatus Required:

- Function generator
- Jumper wires
- Oscilloscope
- Resistor: 1.2kΩ
- Probes
- Capacitor: 1μF
- Inductor: 1mF
- Bread board

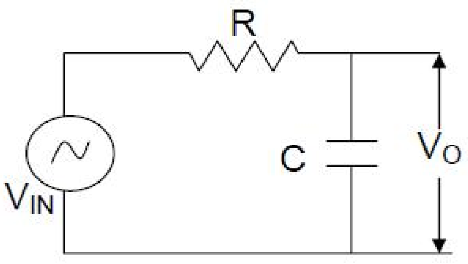
Theory:

The impedance of an inductor is proportional to frequency and the impedance of a capacitor is inversely proportional to frequency. These characteristics can be used to select or reject certain frequencies of an input signal. This selection and rejection of frequencies is called filtering, and a circuit which does this is called a filter.

If a filter passes low frequencies and rejects high ones, it is called a low-pass filter. An RC low pass filter is shown in Figure 1.A frequency is considered passed if its magnitude (voltage amplitude) is within 70% (or 1/√2) of the maximum amplitude passed and rejected otherwise. The 70% frequency is called corner frequency, roll-off frequency, break frequency, cutoff frequency or half-power frequency. The corner frequency for the RC filter is given as:

$\mathbf{f_C = 1 / 2\pi RC}$

At cut off frequency, $R=X_C$ i.e. voltage $V_R=V_C$ and phase angle between input and output voltage will be 45^0.



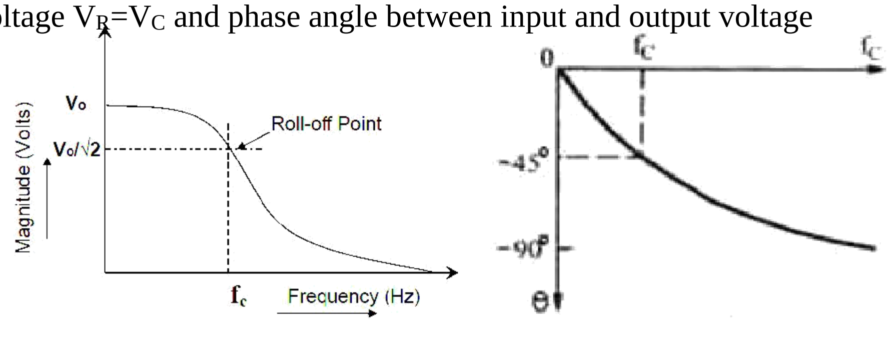
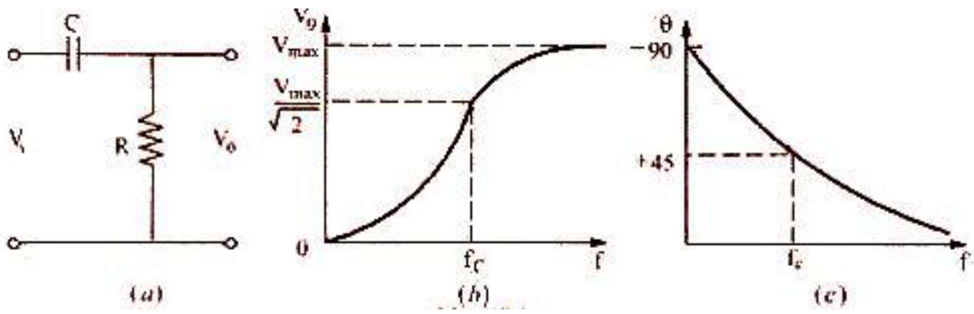
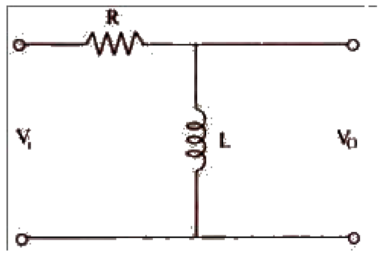


Figure 1: Low Pass FilterFigure 2: Amplitude and Phase Response of Low Pass RC Filter





If a filter allows signals of higher frequencies to pass from input to the output while blocking the lower frequencies, this filter is called a high pass filter. The minimum frequency it allows to pass is called cutoff frequency f_C. A high pass filter may be RL or RC as shown in Figures below.

The cutoff frequency for the RC filter is given as:

$$\mathbf{V_0=Vi\times[R/(R-jXc)]}$$

$$\mathbf{f_c = 1 / 2\pi RC}$$

At fc, R=Xc and the phase angle between Vo and Vi is +45 as shown in Figure 1(c).It can be seen that high pass filter can be obtained merely by interchanging the positions of R and C in low pass RC filter. In high pass filter, all the frequencies above f_c are passed and below are attenuated.
The cutoff frequency for the RL filter is given as:

$$\mathbf{V_0=Vi\times[jX_L/(R+jX_L)]}$$

$$\mathbf{f_c = R / 2\pi L}$$

Procedure:

1. Connect the components on bread board according the circuit diagram.
2. Apply sinusoidal input voltage of 5 V peak to peak with 20 Hz frequency from function generator and note the peak to peak voltage across resistor from oscilloscope.
3. Also measure the phase difference ("θ") between input and resistor output voltage by oscilloscope.
4. Increase the frequency at regular steps and fill the table shown below.
5. Draw the graph between output voltage and frequency.
6. Mark the cut off frequency on the graph.
7. Draw the graph between θ and frequency.
8. Mark the value of θ at cut off frequency.

Observations:

S.No	Peak to peak value of input voltage	Frequency of input voltage (Hz)	Peak to peak value of resistor output voltage from oscilloscope ($V_0=V_R$)	Phase angle between input and output voltage ("θ" degrees)
1	5 Vp-p			
2	5 Vp-p			
3	5 Vp-p			
4	5 Vp-p			
5	5 Vp-p			
6	5 Vp-p			

7	5 Vp-p				
8	5 Vp-p				
9	5 Vp-p				
10	5 Vp-p				
11	5 Vp-p				
12	5 Vp-p				
13	5 Vp-p				
14	5 Vp-p				
15	5 Vp-p				

Result: The characteristics and frequency response of various 1st order RL & RC networks arte studied.

8. To study the transient and steady state response of a 2nd order circuit by varying its various parameters andstudying their effects on responses

Aim: To study the step response of second order circuits and to understand the difference between overdamped, critically damped and underdamped responses.

Apparatus Required:

- Breadboard
- Function generator
- Oscilloscope
- Digital multimeter (DMM)

Theory:

Second-order circuits are RLC circuits that contain two energy storage elements. They can be represented by a second-order differential equation. A characteristic equation, which is derived from the governing differential equation, is often used to determine the natural response of the circuit. The characteristic equation usually takes the form of a quadratic equation, and it has two roots s1 and s2.

$$a_2 s^2 + a_1 s + a_1 = 0$$

$$s_1 = \frac{-a_1 + \sqrt{a_1^2 - 4a_2 a_0}}{2a_2}$$

$$s_2 = \frac{-a_1 - \sqrt{a_1^2 - 4a_2 a_0}}{2a_2}$$

When these roots are rewritten as,

$$s_1 = -\alpha + \sqrt{\alpha^2 - \omega_0^2}$$

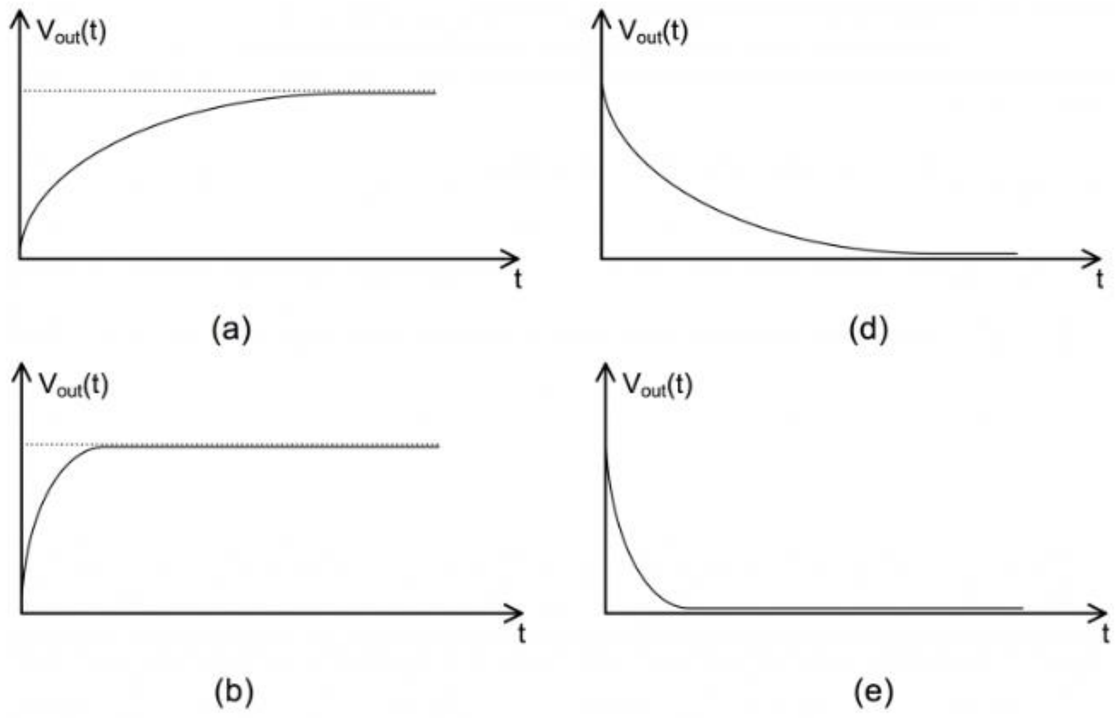
$$s_2 = -\alpha - \sqrt{\alpha^2 - \omega_0^2}$$

then the natural response of the circuit is determined by:

$\alpha^2 > \omega_0^2$, there are two real and distinct roots →Overdamped, as shown in Figure 1(a) and (d).

$\alpha^2 = \omega_0^2$, there are two real equal roots → Critically damped, as shown in Figure 1 (b) and (e).

$\alpha^2 < \omega_0^2$, there are two complex roots →Underdamped, as shown in Figure 1(c) and (f).



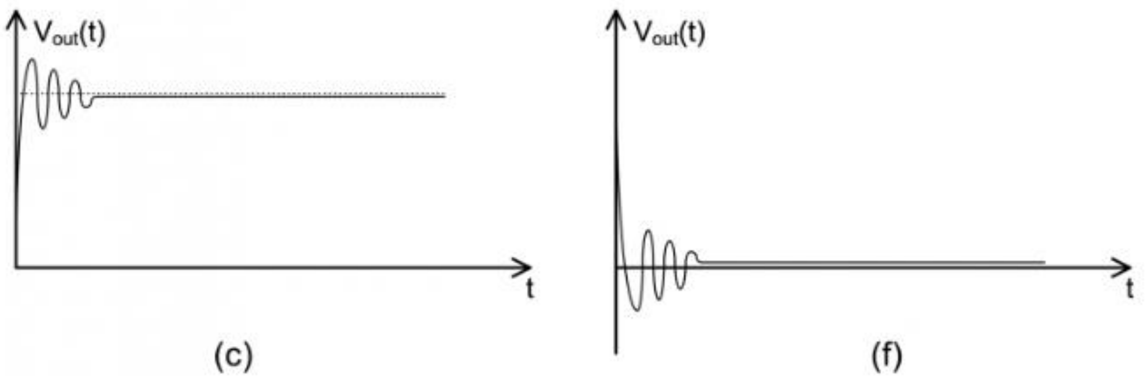


Figure 1: Second order circuits natural responses

Circuit Diagrams:

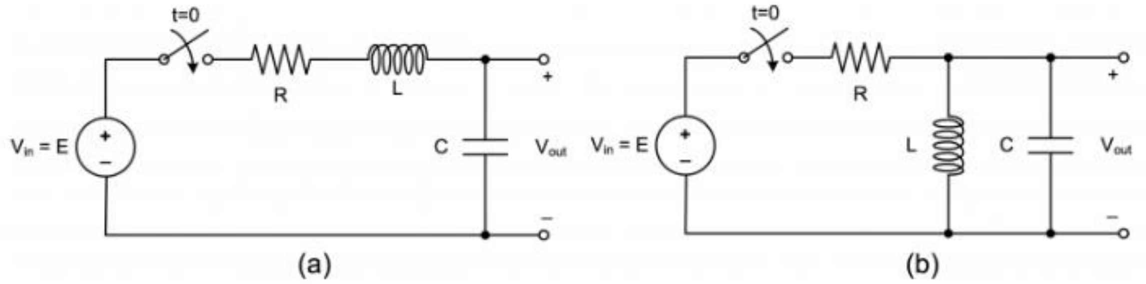


Figure 2: Second order circuits with step input

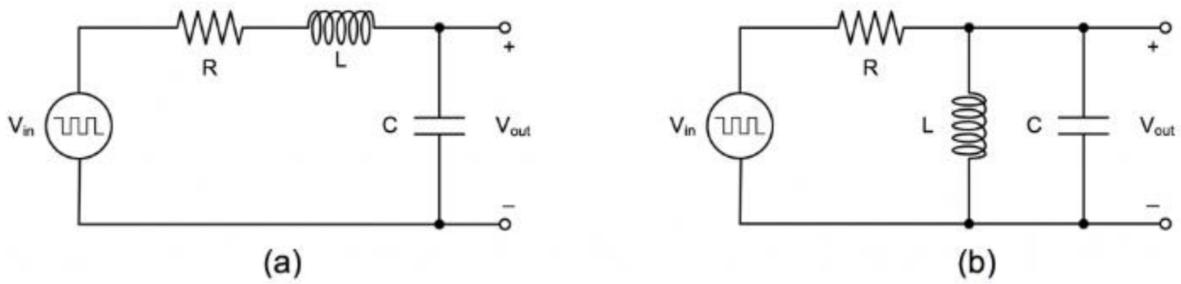


Figure 3: Second order Circuits with square wave input

Procedure:

Perform the following circuit simulations using a circuit simulator.

1. Build and simulate the circuits in Figure1. Set the input voltage to ±5V with a frequency of 1 kHz. Display both $V_{in}(t)$ and $V_{out}(t)$ on the oscilloscope. Compare the results with the plots from PREPARATION.
2. For the circuit in Figure 1(d), with the input voltage remains unchanged, display the voltage across the inductor. Compare this voltage waveform to the one obtained from the circuit in Figure 1(c). What do you observe? Why?
3. Build and simulate the circuits in Figure 3. Set the input voltage to ±4V with a frequency of 400 Hz. Display both $V_{in}(t)$ and $V_{out}(t)$ on the oscilloscope. Compare the results with the plots from PREPARATION.

On the function generator use the same square wave input settings as in SIMULATION. Build both circuits shown in Figure 3.

A. Natural responses

1. Measure the output voltage $V_{out}(t)$ for the following six cases.
   1. R = 22 kΩ in the circuit of Figure 4 – 6 (a)
   2. R = 6.3 kΩ in the circuit of Figure 4 – 6 (a)
   3. R = 2.2 kΩ in the circuit of Figure 4 – 6 (a)
   4. R = 680 Ω in the circuit of Figure 4 – 6 (b)
   5. R = 1.6 kΩ in the circuit of Figure 4 – 6 (b)
   6. R = 4.7 kΩ in the circuit of Figure 4 – 6 (b)

2. On the oscilloscope, connect Ch1 to the input and Ch2 to the output so that both the input and the output are displayed on the screen.
3. For each case, save the screen image with the associated measurements for both the input and the output on to a USB drive.
4. For each case, indicate if the output response is overdamped, critically damped or underdamped.

B. Damped natural frequency measurement

1. Set R = 470 Ω for the circuit in Figure 4 – 6 (a) and R = 22 kΩ for the circuit in Figure 4 – 6 (b). Measure and save the screen image for both the input and the output, and compare them with the results from PREPARATION and SIMULATION.
2. Zoom in on the output curve so that at least two whole oscillations (ripples) of the output from the beginning of an output cycle are displayed. Use the cursors to measure the time period T_d between the first two peaks (or between two zero phases). ω_d is calculated using:

$$\omega_d = \frac{2\pi}{T_d}$$

Result:

The step response of second order circuits and to understand the difference between overdamped, critically damped and underdamped responses are measured.

9.Find the Q Factor and Bandwidth of a Series and Parallel Resonance circuit

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

APPARATUS:

S.No	NameOftheEquipment	Range	Type	Quantity
1	Breadboard	-	-	1NO
2	Resistor	1k Ω	-	1NO
3	Inductor	50mH	-	1NO
4	Capacitors	0.1uF	-	1NO
5	CRO	20MHz.DualCH	-	1NO
6	Functiongenerator	100-10MHz	-	1NO
7	Ammeter	0-20mA	Digital	1NO
8	Connectingwires			

CIRCUIT DIAGRAM:

SERIESRESONANCE:

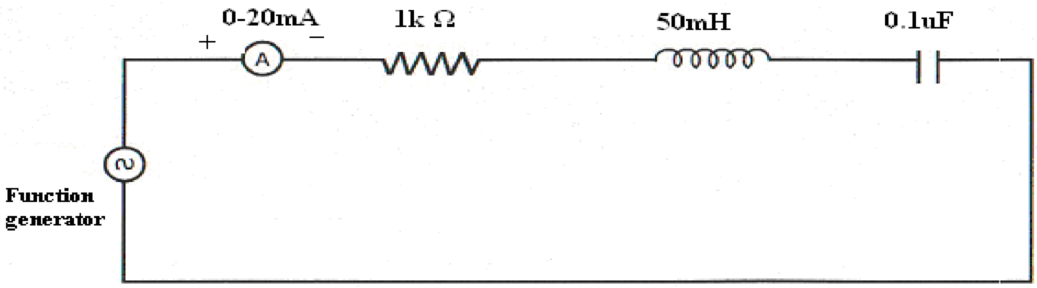


Fig.1

PARALLELRESONANACE:

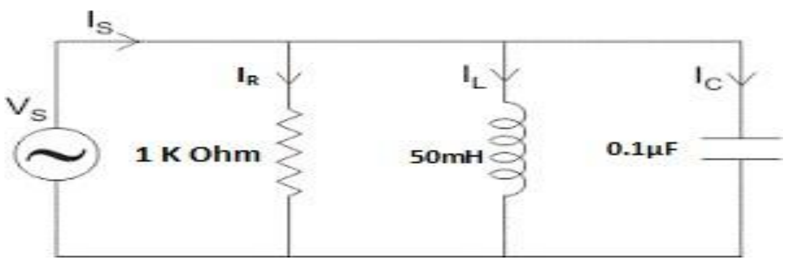


Fig.2

THEORY:

Resonance is a particular type of phenomenon inherently found normally in every kind ofsystem, electrical, mechanical, optical, Acoustical and even atomic. There are severaldefinitions of resonance. But, the most frequently used definition of resonance in electricalsystem is studied state operation of a circuit or system at that frequency for which the resultantresponseis in time phasewiththeforcingfunction.

SERIESRESONANCE:

Acircuitissaidtobeunderresonance,whentheappliedvoltage„V"andcurrentareinphase.Thus a series RLC circuit, under resonance behaves like a pure resistance network and thereactance of the circuit should be zero. Since V & I are in phase, the power factor is unity atresonance.

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

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

PARALLELRESONANCE:

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

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

Thus, at resonance the admittance(Y) is Minimum and voltage is Maximum. However,theperformanceofsuchacircuitisofinterestinthegeneralsubjectofresonance.Lower cut-offfrequencyisabovetheresonantfrequencyatwhichthecurrentisreducedto$\frac{1}{\sqrt{2}}$timesofitsminimum value. Upper cut-off frequency is above. Quality factor is the ratio of resistance toreactanceof inductor (or) capacitor. Selectivityis thereciprocal of thequalityfactors.

THEORITICALCALCULATIONS:

ForSeriesResonancecircuit:

1. Resonant frequency $f_r = \frac{1}{2\pi\sqrt{LC}}$
2. LowerCutoffFrequency$f_1=f_r-(R/4\pi L)$
3. UpperCutoff Frequency$f_2=f_r+(R/4\pi L)$
4. Band width = f_2-f_1 :
5. Quality factor $Q = \frac{w_0 L}{R} = \frac{2\pi fr.L}{R}$
6. Current at Resonance $I_o = V_{Ro}/R$

ForParallelResonancecircuit:

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

PROCEDURE:

1. Connectthecircuitasshowninfig.1forseriesresonantcircuit&fig.2forparallelresonantcircuit.
2. Setthevoltageofthesignalfromfunctiongeneratorto5V.
3. Vary the frequency of the signal over a wide range in steps and note down thecorrespondingammeterreadings.
4. Observe that the current first increases & then decreases in case of series resonant circuit &thevalueoffrequencycorrespondingtomaximumcurrent is equalto resonant frequency.
5. Observe that the current first decreases & then increases in case of parallel resonant circuit&thevalueoffrequencycorrespondingtominimumcurrentisequaltoresonant frequency.
6. Draw a graph between frequency and current & calculate the values of bandwidth &qualityfactor.

OBSERVATIONS:

SeriesResonance:

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

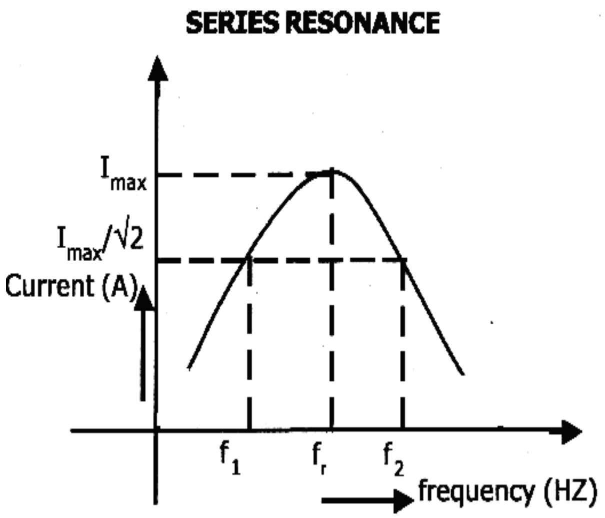
ParallelResonance:

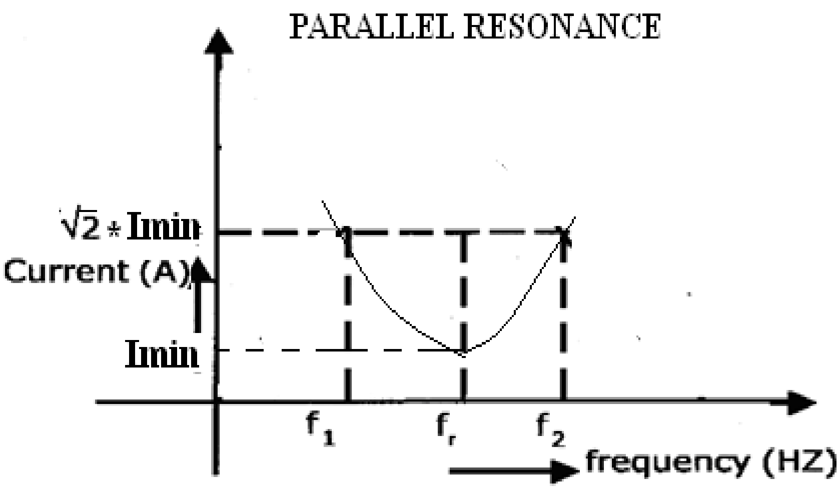
S.No.	**Frequency** (Hz)	**Current (Is)** (mA)

TabularColumn:

S.NO	**PARAMETER**	**Seriesresonantcircuit**		**Parallelresonantcircuit**	
		Theoretical	**Practical**	**Theoretical**	**Practical**
1	ResonantFrequency(f_r)				
2	Bandwidth				
3	Qualityfactor				

MODELGRAPHS:





f_1= lower cutoff frequency

f_2 = upper cutoff frequency

f_r=ResonantFrequency

PRECAUTIONS:

1. InitiallykeeptheRPS output voltageknobinzerovolt position.
2. Avoidlooseconnections.
3. AvoidshortcircuitofRPSoutputterminals.

RESULT:

The Resonant frequency, Quality factor and Bandwidth of a given series and parallel resonant circuits are measured.

VIVAQUESTIONS:

1) Whatisresonanceofcircuit?
2) Whatisseriesandparallelresonance?
3) Whatiscut-offfrequency?
4) Definebandwidth and Qualityfactor?

10. Determination of open circuit (Z) and short circuit (Y) parameters

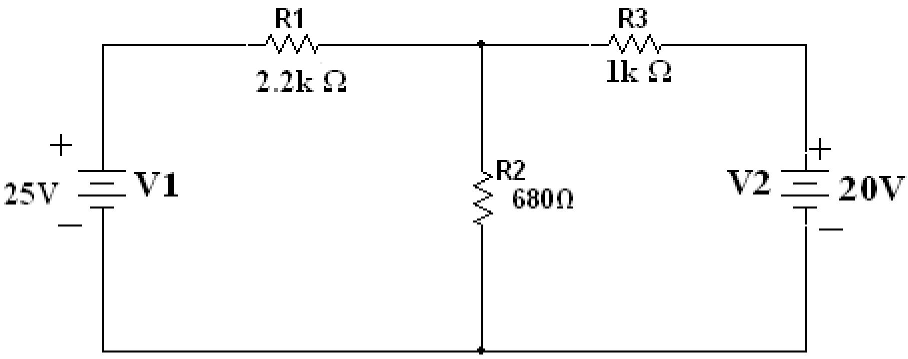
AIM: To determine the Impedance (Z) and admittance(Y) parameters of a two-portnetwork.

APPARATUS:

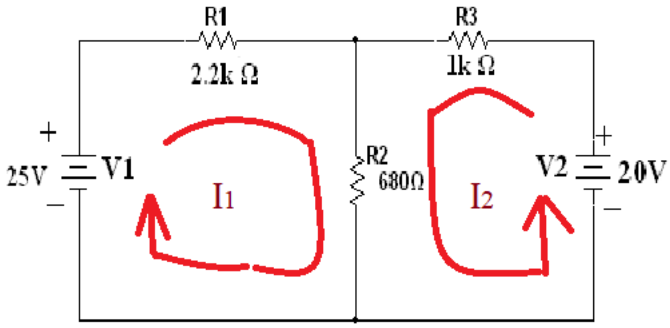
S.No	NameOfTheEquipment	Range	Quantity
1	Voltmeter	(0-20)V	1NO
2	Ammeter	(0-20)mA	1NO
3	RPS	0-30V	1NO
4	Resistors	2.2k Ω	1NO
		1k Ω	1NO
		680 Ω	1NO

CIRCUIT DIAGRAM:

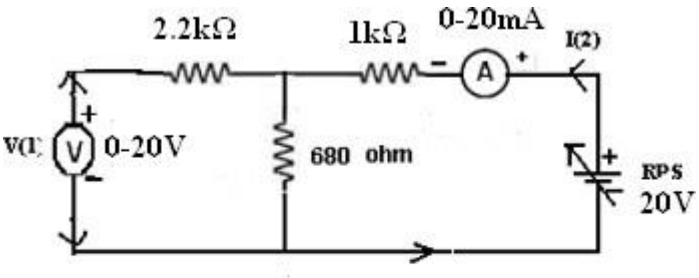
1. GIVENCIRCUIT:



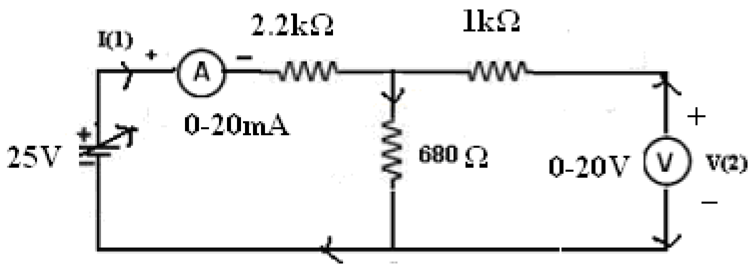
PRACTICALCIRCUITS:



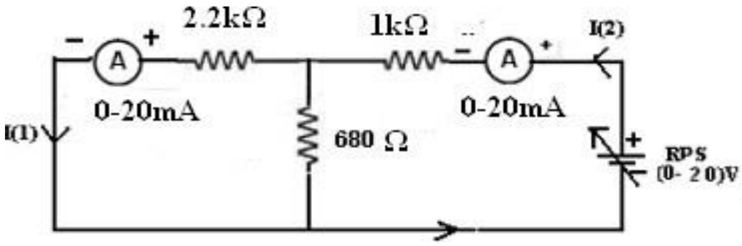
2. When $I_1 = 0$:



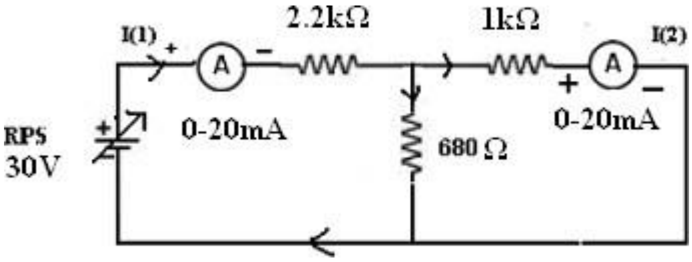
3. When $I_2 = 0$:



4.WhenV_1= 0:



5.When V_2 =0



THEORY:

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

If we relate the voltage of one port to the current of the same port, we get driving pointadmittance. On the other hand, if we relate the voltage of one port to the current at anotherport, we get transfer admittance. Admittance is a general term used to represent either theimpedance or the admittance of a network. We will consider a general two-port networkcomposed of linear, bilateral elements and independent sources. The voltage and current atport -1 are V_1 and I_1 and at port -2 are V_2 and I_2. The position of V_1 and V_2 and the directionsof I_1 and I_2 are customarily selected. Out of four variables only two are independent. The othertwo are expressed in terms of the independent variable of network parameters. The relationbetweenthe voltages and currents in terms ofZand Y parametersareasfollows.

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

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

Z-PARAMETERS:

$$Z11 = \frac{V1}{I1} / I2 = 0$$

$$Z12 = \frac{V1}{I2} / I1 = 0$$

$$Z21 = \frac{V2}{I1} / I2 = 0$$

$$Y22 = \frac{V2}{I2} / I1 = 0$$

Y-PARAMETERS:

$$Y11 = \frac{I1}{V1} / V2 = 0$$

$$Y12 = \frac{I2}{V1} / V1 = 0$$

$$Y21 = \frac{I2}{V1} / V2 = 0$$

$$Y22 = \frac{I2}{V2} / V1 = 0$$

PROCEDURE:

1. Connectionsaremadeasperthecircuitdiagram.
2. Opencircuittheport–1i.e., I_1=0,findthevaluesofV_1,I2andV_2.
3. Shortcircuittheport-1i.e. V_1=0, find the values of V_2,I_1andI_2.
4. Opencircuittheport–2i.e., I_2=0,findthevalues ofV_1,I1andV_2.
5. Shortcircuittheport-2i.e.V_2=0,find thevaluesofV_1,I_1andI_2.

5.Findthe ZandYparametersofthe given twoportnetwork.

THEORITICALVALUES:

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

PRACTICALVALUES:

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

Z-PARAMETERS:

Z-parameters	Theoretical	Practical
Z11		
Z12		
Z21		
Z22		

Y-PARAMETERS:

Y-Parameters	Theoretical	Practical
Y11		
Y12		
Y21		
Y22		

PRECAUTIONS:

1. InitiallykeeptheRPS output voltageknob in zerovolt position.
2. Avoidlooseconnections.
3. AvoidshortcircuitofRPSoutputterminals.

RESULT:

The Impedance (Z) and admittance(Y) parameters of a two-portnetwork are measured both theoretically and practicaslly.

VIVAQUESTIONS:

1. DefinePort?
2. DefineZ&Yparameters?
3. Whatistheconditionforsymmetryincase Z&Y parameters?
4. Definecharacteristicimpedance?
5. Whatistheconditionforreciprocityincase Z&Y parameters?

11. Determination of hybrid (H) and transmission (ABCD) parameters

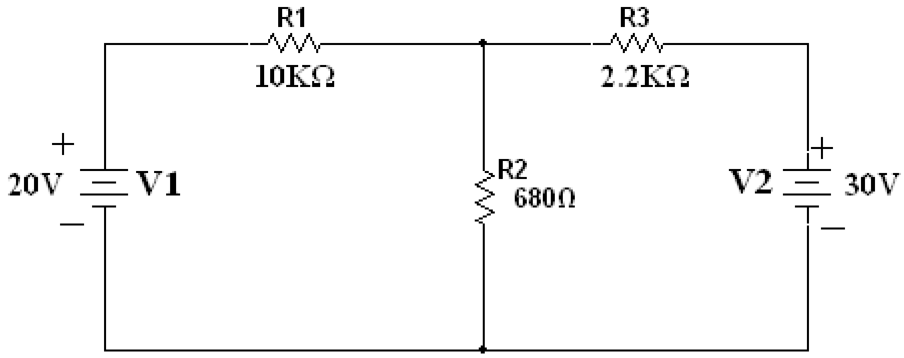
AIM: Todeterminethe Hybrid andTransmissionparametersofatwoportnetwork.

APPARATUS REQUIRED :

S.No	NameOfTheEquipment	Range	Quantity
1	Voltmeter	(0-20)V	1NO
2	Ammeter	(0-20)mA	1NO
3	RPS	0-30V	1NO
4	Resistors	10K Ω	1NO
		2.2Ω	1NO
		680 Ω	1NO
5	Breadboard	-	1NO
6	Connectingwires		Required number

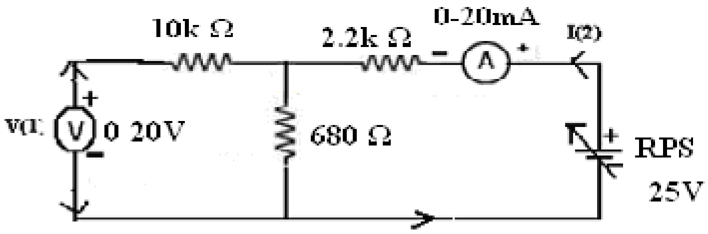
CIRCUIT DIAGRAM:

GIVENCIRCUIT:

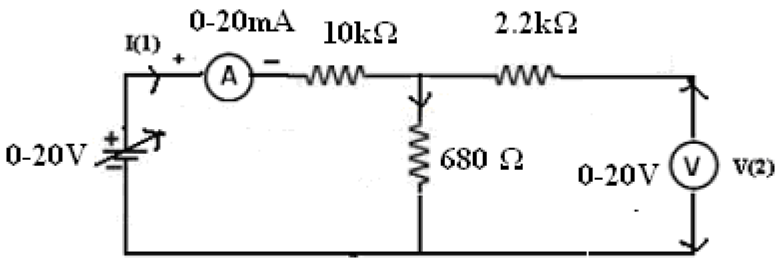


PRACTICALCIRCUITS:

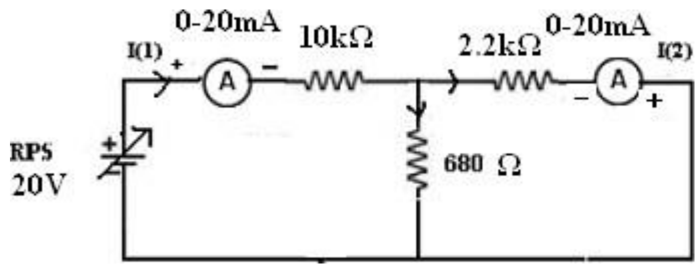
1. WhenI$_1$= 0:



2. When I$_2$=0:



3. When $V_2 = 0$:



THEORY :

Therelationbetweenthe voltagesandcurrentsofatwoportnetworkintermsofABCDandh–parametersis given asfollows.

H-PARAMETERS

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

ABCDPARAMETERS:

$$V_1 = AV_2 - BI_2$$
$$I_1 = CV_2 - DI_2$$

H-PARAMETERS:

$$h_{11} = \left.\frac{V_1}{I_1}\right|_{V_2=0} \quad h_{12} = \left.\frac{V_1}{V_2}\right|_{I_1=0} \quad h_{21} = \left.\frac{I_2}{I_1}\right|_{V_2=0} \quad h_{22} = \left.\frac{I_2}{V_2}\right|_{I_1=0}$$

ABCDPARAMETERS:

$$A = \left.\frac{V_1}{V_2}\right|_{I_2=0} \quad B = \left.\frac{V_1}{-I_2}\right|_{V_2=0} \quad C = \left.\frac{I_1}{V_2}\right|_{I_2=0} \quad D = \left.\frac{I_1}{-I_2}\right|_{V_2=0}$$

PROCEDURE:

1. Connectionsaremadeasperthecircuitdiagram.
2. Opencircuittheport–1i.e., I_1=0findthevaluesofV_1,I2andV_2.
3. Shortcircuittheport-1V_1=0find the values ofV_2,I_1andI_2.
4. Opencircuittheport–2i.e., I_2=0findthevalues ofV_1,I1andV_2.
5. Shortcircuittheport-2i.e.V_2=0 find the values of V_1,I_1andI_2

5.FindtheABCDandh-parameters ofthegiventwoportnetworkfromtheabovedata.

THEORITICALVALUES:

V_2 =0	V_1=	I_1=	I_2=
I_1 =0	V_1=	V_2=	I_2=
I_2 =0	V_1=	V_2=	I_1=

PRACTICALVALUES

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

H-PARAMETERS:

h-Parameters	Theoretical	Practical

ABCD-PARAMETERS:

T-parameters	Theoretical	Practical

PRECAUTIONS:

1. InitiallykeeptheRPSoutputvoltageknobinzero-volt position.
2. Avoidlooseconnections.
3. AvoidshortcircuitofRPSoutputterminals.

RESULT:

VIVAQUESTIONS

1. DefinePort?
2. Whatistheconditionforsymmetryincaseh-parameters&ABCD(T)parameters?
3. Definecharacteristicimpedance?
4. Whatistheconditionforreciprocityincase Hybrid (h) &ABCD(T)parameters?

—

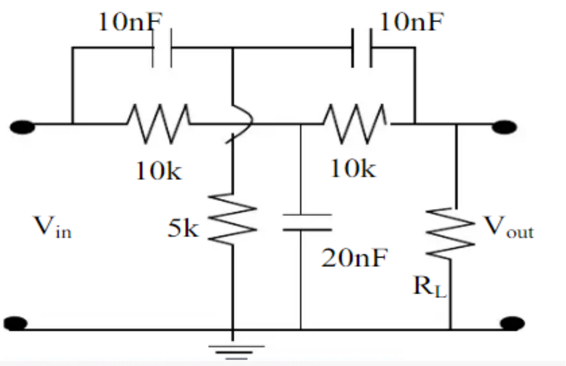
12. To measure two port parameters of a twin-T network and study its frequency response

AIM: To measure two port parameters of a twin-T network and study its frequency response.

APPARATUS REQUIRED :

1. Typhoon HIL Simulation Software
2. Resistors- 10KΩ,5KΩ
3. Capacitors- 10nF
4. DC Voltage Source

CIRCUIT DIAGRAM:



THEORY :

Twin-T is basically a frequency selective network. The amplitude and response of a twin –T network shows that the phase shift introduced by this network is zero at particular frequency. The Twin-T network acts as the phase lead-lag network. It introduces a phase shift that varies between +90 to -90 degrees.

PROCEDURE:

1. Open the typhoon HIL control centre on the system and click on additional tools.
2. Now from additional tool click on script editor.
3. Here in script editor one can write python code for bode plots for a given transfer function and see the response of filter on the bode plot.
4. Before writing the script, it is essential to find transfer function. Let R=20000 ohm and C= 1e-6 F. Hence Wn = 50 and transfer function becomes g(s) = (s +2500/ s2+10000s+2500)
5. Python script for bode plot can be written in the following manner.fromscipy import signal import matplotlib.

```
s1 = signal.lti([1, 0, 170], [1, 170, 289]) w,
mag, phase = signal.bode(s1)
plt.figure()
plt.semilogx(w, mag) # Bode magnitude plot
plt.figure()
plt.semilogx(w, phase) # Bode phase plot
plt.show()
```

RESULT: Two port network parameters of a twin-T network and study its frequency response was studied.