Network Analysis Lab
3rd Semester (ET&T)

Government Polytechnic Jashpur

Department of Electronics & Telecommunication Engg.

A.K.Singh

Lecturer(Et&T)

EXPERIMENT No- 1

OBJECTIVE: - Apply the Kirchhoff's law for finding current in a complex electrical circuit

EQUIPMENT REQUIRED: - Batteries, Resistors, Multimeter, Battery Holders, Connecting Leads, Alligator clips.

THEORY: - KCL: - This law is also called Kirchhoff's point rule, Kirchhoff's junction rule (or nodal rule), and Kirchhoff's first rule. Kirchhoff's first law states that: ―the sum of the currents flowing through a node must be zero‖. This law is particularly useful when applied at a position where the current is split into pieces by several wires. The point in the circuit where the current splits is known as a node. Or

―The algebraic sum of current at any node of a circuit is zero‖. The direction of incoming currents to a node being positive the outgoing current should be taken negative.

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

n is the total number of branches with currents flowing towards or away from the node.

PROCEDURE: - For Current Law:-

1. Using the multimeter, measure the value of the resistance of each of the three resistors provided by setting the scale of the multimeter on the 200K scale.
2. Use the multimeter to measure the voltage from the battery(s) in the single D battery holders and the two D battery holder.
3. Set up the circuit shown in Figure. In this circuit, use one of the single D battery holder for VB1 and the two D battery holder for VB2. For the resistors in the circuit, use the resistors closest to the following values: R1 = 50 k ohm, R2 = 20 k ohm, and R3 = 10 k ohm
4. Set the multimeter on 200 u on the current scale (i.e. `the `A" scale). Attach a black lead to the COM terminal and a red lead to the mA terminal. With these settings, the multimeter is set to read the current in the circuit in micro Amperes (i.e. ¹A).
5. Measure the current flowing into the top node of the circuit from each of the three branch wires. To measure the current you will have to break the circuit to insert the multimeter. You must also measure the polarity of the current in a consistent manner. If the current flows into the node, then the current should be measured from the positive (red/mA) terminal to the negative (black/COM) terminal. Record these measured currents on your data table.
6. In the space provided on the data table, add the three currents to check that the sum of the current is zero (or close to zero).
7. Measure the current flowing into the top node of the circuit from each of the three branch wires. To measure the current you will have to break the circuit to insert the multimeter. You must also measure the polarity of the current in a consistent manner. If the current flows into the node, then the current should be measured from the positive (red/mA) terminal to the negative (black/COM) terminal. Record these measured currents on your data table.
8. In the space provided on the data table, add the three currents to check that the sum of the currents is zero (or close to zero).

For KVL:-

1) Connect the circuit as per the circuit diagram on bread board supply using connecting wire sand chords.
2) Switch on the power supply and adjust V1 and V2
3) Measure VR1 & VR2 , VR3 ,VR4 & VR5.
4) Select any desired loop say loop (1), apply KVL as per given in observation table and verify the result.
5) Repeat step (2) onwards for different V1 and V2.

OBSERVATION: - For Current/Node Law:-

(a) Current across individual resistor R_1, R_2 & R_3 (I= V/R):

I_1 =………., I_2 =…………, I_3 =……………..

(b) Verifying KCL using measured values: $I_1 + I_2 + I_3$ = --------

For Kirchoffs Voltage Law-

Starting from point A , if we go around the mesh in clockwise direction the different EMF's & I.R drop will have following values and signs.

I_1R_1------------ is –ve (fall in potential)

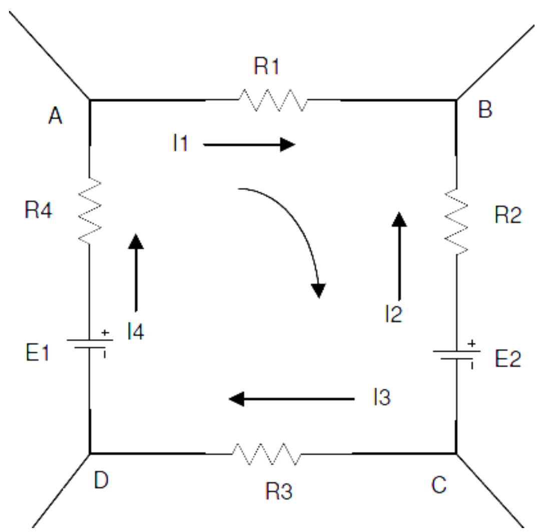
I_2R_2--------------is +ve (rise in potential)

E_2----------- is –ve (fall in potential

I_3R_3-------------is –ve (fall in potential)

E_1----------------is +ve (rise in potential)

I_4R_4-----------is –ve (fall in potential)



According to KVL

$I_1R_1 - I_2R_2 - E_2 - I_3R_3 + E_1 - I_4R_4 = 0$

$- I_1R_1 + I_2R_2 - I_3R_3 - I_4R_4 = E_2 - E_1$

$I_1R_1 - I_2R_2 + I_3R_3 + I_4R_4 = E2 - E_1$

$V_{R1} - V_{R2} + V_{R3} + V_{R4} = E_1 - E_2$

Determination of algebraic sign

1. Battery EMF

While going round a loop (in a direction of our own choice) if we go from the -ve terminal of battery to its +ve terminal , there is rise in potential , hence this EMF should be given as + ve signal .On the other hand if we go from its + ve terminal its -ve terminal , there's a fall in potential , hence this battery EMF should be given as -ve sign.

2. IR drops in series

If we go through a circuit in the same direction as its current, then there is a fall or decrease in potential for the simple reason that current always flow from higher to lower potential.
Hence this IR drop should be taken as -ve. However, if we go around the loop in direction opposite to that of the current there is a rise in voltage. Hence these IR should be taken as +ve. It clears that the algebraic sign of IR drop across a resister depends on the direction of current through that resistor.

Observation table

For KCL:

S. NO.	V_1	V_2	I_1	I_2	I_3	$I_3 = I_1+I_2$
1						
2						

For KVL

S No.	V_1	V_2	V_{R1}	V_{R2}	V_{R3}	V_{R4}	V_{R5}	$V_1= V_{R1}+V_{R2}+V_{R5}$	$V_2= V_{R3}+V_{R4}+V_{R5}$
1									
2									

RESULT: - The current approaching to the junction is equal to currents leaving from the junction. So KCL is verified similarly voltage supplied to desired loop equals to voltage drop by same loop so KVL is verified.

PRECAUTIONS: -

1) All the connection should be tight.
2) Ammeter is always connected in series in the circuit while voltmeter is parallel to the conductor.
3) The electrical current should not flow the circuit for long time, Otherwise its temperature will increase and the result will be affected.
4) It should be care that the values of the components of the circuit is does not exceed to their ratings (maximum value).
5) Before the circuit connection it should be check out working condition of all the Component.

EXPERIMENT No- 2

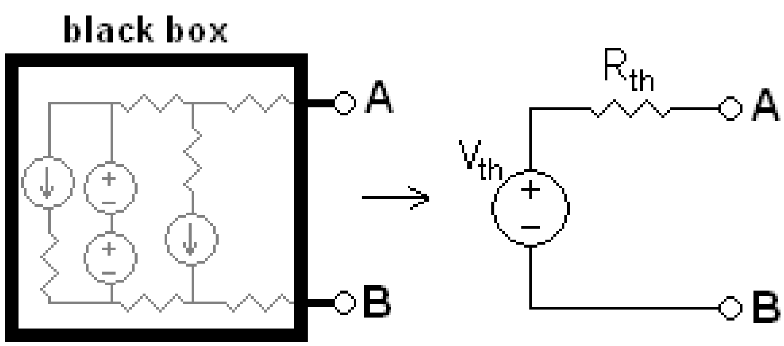
Objective: - Apply the thevenin's theorem for finding current in a complex electrical circuit

Theory

Thévenin's theorem:-

In circuit theory, Thévenin'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 single frequency AC systems the theorem can also be applied to general impedances, not just resistors. The theorem was first discovered by German scientist Hermann von Helmholtz in 1853,[1] but was then rediscovered in 1883 by French telegraph engineer Léon Charles Thévenin (1857–1926).

This theorem states that a circuit of voltage sources and resistors can be converted into a Thévenin equivalent, which is a simplification technique used in circuit analysis. The Thévenin equivalent can be used as a good model for a power supply or battery (with the resistor representing the internal impedance and the source representing the electromotive force). The circuit consists of an ideal voltage source in series with an ideal resistor.



Any black box containing only voltage sources, current sources, and other resistors can be converted to a Thévenin equivalent circuit, comprising exactly one voltage source and one resistor.

Calculating the Thévenin equivalent

To calculate the equivalent circuit, the resistance and voltage are needed, so two equations are required. These two equations are usually obtained by using the following steps, but any conditions placed on the terminals of the circuit should also work:

1. Calculate the output voltage, V_{AB}, when in open circuit condition (no load resistor—meaning infinite resistance). This is Vth.
2. Calculate the output current, I_{AB}, when the output terminals are short circuited (load resistance is 0). R_{Th} equals Vth divided by this I_{AB}.

The equivalent circuit is a voltage source with voltage Vth in series with a resistance R_{Th}.

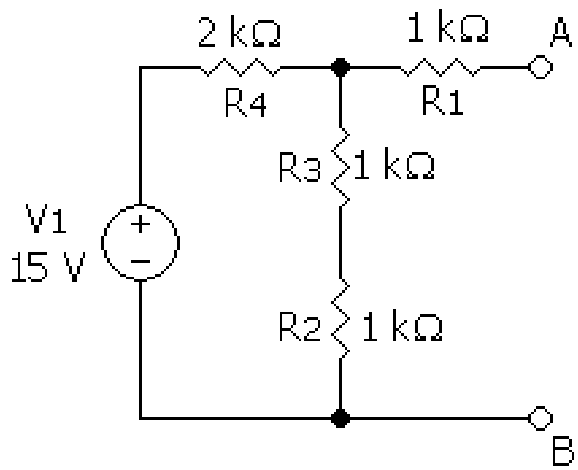
Step 2 could also be thought of as:

2a. Replace voltage sources with short circuits, and current sources with open circuits.
2b. Calculate the resistance between terminals A and B. This is R_{Th}.

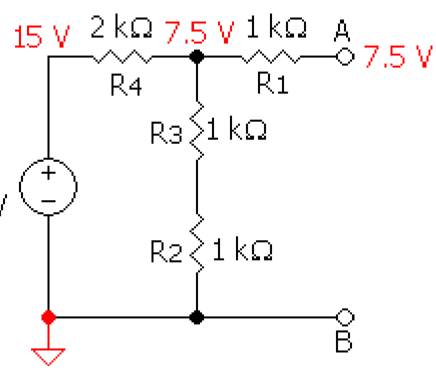
The Thévenin-equivalent voltage is the voltage at the output terminals of the original circuit. When calculating a Thévenin-equivalent voltage, the voltage divider principle is often useful, by declaring one terminal to be V_{out} and the other terminal to be at the ground point.

The Thévenin-equivalent resistance is the resistance measured across points A and B "looking back" into the circuit. It is important to first replace all voltage- and current- sources with their internal resistances. For an ideal voltage source, this means replace the voltage source with a short circuit. For an ideal current source, this means replace the current source with an open circuit. Resistance can then be calculated across the terminals using the formulae for series and parallel circuits. This method is valid only for circuits with independent sources. If there are dependent sources in the circuit, another method must be used such as connecting a test source across A and B and calculating the voltage across or current through the test source.

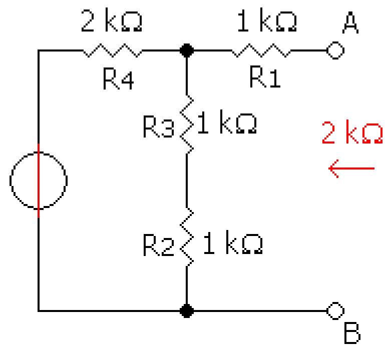
Example



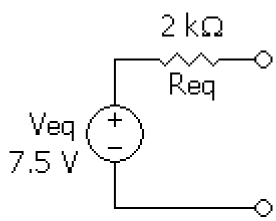
Step 0: The original circuit



Step 1: Calculating the equivalent output voltage



Step 2: Calculating the equivalent resistance



Step 3: The equivalent circuit

In the example, calculating the equivalent voltage:

V_{th}=[R2+R3/(R2+R3=R4)]V1

V_{th}=[1k Ω +1 k Ω /(1 k Ω +1 k Ω +2 k Ω)] 15V

= ½(15) = 7.5V

(notice that R_1 is not taken into consideration, as above calculations are done in an opencircuit condition between A and B, therefore no current flows through this part, which means there is no current through R_1 and therefore no voltage drop along this part)

Calculating equivalent resistance:

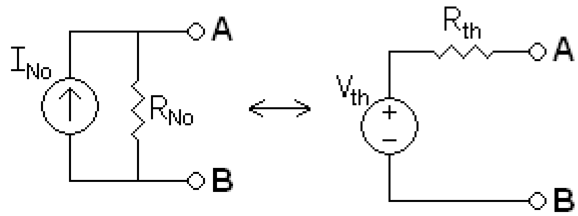
$R_{th} = R_1 + [(R_2+R_3) \| R_4]$

= 1 k Ω + [(1 k Ω +1 k Ω) ||2 k Ω]

=2 k Ω

Conversion to a Norton equivalent

A Norton equivalent circuit is related to the Thévenin equivalent by the following:



$R_{th}=R_{no}$

$V_{th}=I_{no} R_{no}$

$I_{no}=V_{th}/R_{th}$

Practical limitations

- Many, if not most circuits are only linear over a certain range of values, thus theThévenin equivalent is valid only within this linear range and may not be valid outside the range.
- The Thévenin equivalent has an equivalent I-V characteristic only from the pointof view of the load.
- The power dissipation of the Thévenin equivalent is not necessarily identical to the power dissipation of the real system. However, the power dissipated by an external resistor between the two output terminals is the same however the internal circuit is represented.

PRECAUTIONS: -

1) All the connection should be tight.
2) It should be care that the values of the components of the circuit is does not exceed to their ratings (maximum value).
3) Before the circuit connection it should be check out working condition of all the Component.

Experiment No.- 3

Aim - Verify the Norton's theorem

EQUIPMENT REQUIRED 1. Power supply 2. Resistors 3. Connecting wires 4. Bread board 5. Multimeters/ammeter

Theory:

Statement - Norton's theorem for linear electrical networks, states that any collection of voltage sources, current sources, and resistors with two terminals is electrically equivalent to an ideal current source, I, in parallel with a single resistor, R.

For single-frequency AC systems the theorem can also be applied to general impedances, not just resistors. The Norton equivalent is used to represent any network of linear sources and impedances, at a given frequency. The circuit consists of an ideal current source in parallel with an ideal impedance (or resistor for non-reactive circuits).

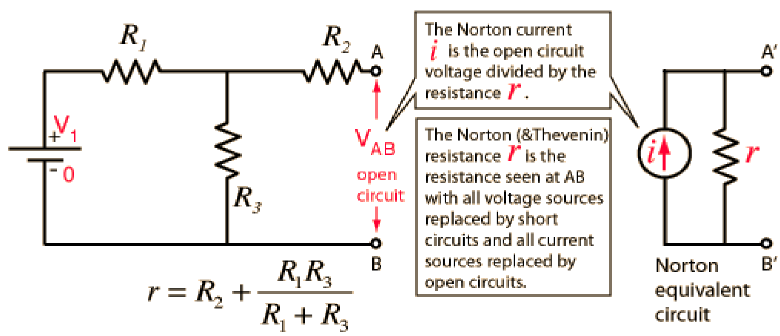
Calculation of a Norton equivalent circuit

The Norton equivalent circuit is a current source with current I_{No} in parallel with a resistance R_{No}. To find the equivalent,

1. Find the Norton current I_{No}. Calculate the output current, I_{AB}, with a short circuit as the load (meaning 0 resistances between A and B). This is I_{No}.
2. Find the Norton resistance R_{No}. When there are no dependent sources (i.e., all current and voltage sources are independent), there are two methods of determining the Norton impedance R_{No}.

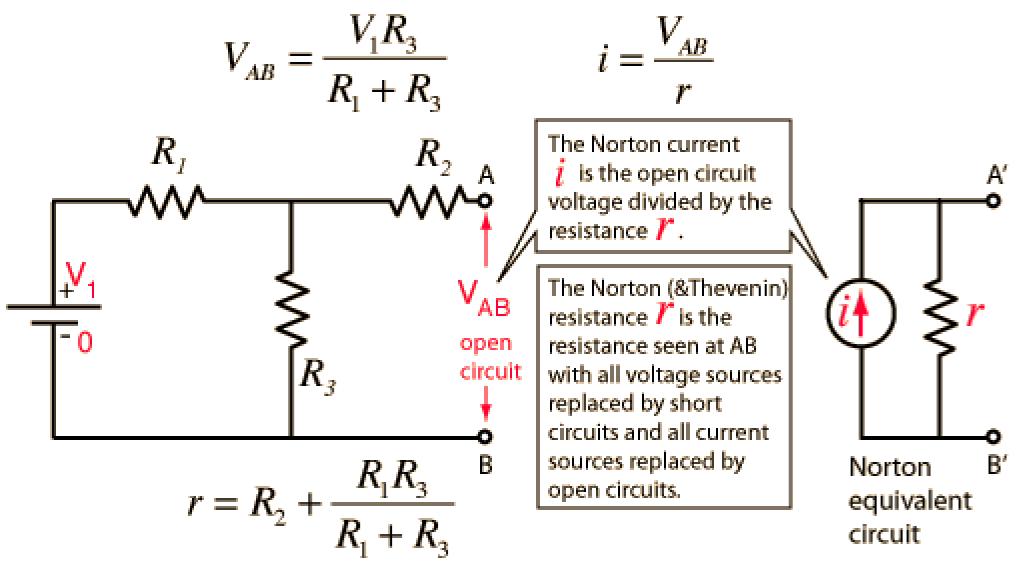
Calculate the output voltage, V_{AB}, when in open circuit condition (i.e., no load resistor — meaning infinite load resistance). R_{No} equals this V_{AB} divided by I_{No}. OR Replace independent voltage sources with short circuits and independent current sources with open circuits. The total resistance across the output port is the Norton impedance R_{No}.

Any collection of batteries and resistances with two terminals is electrically equivalent to an ideal current source i in parallel with a single resistor r. The value of r is the same as that in the Thevenin equivalent and the current i can be found by dividing the open circuit voltage by r.



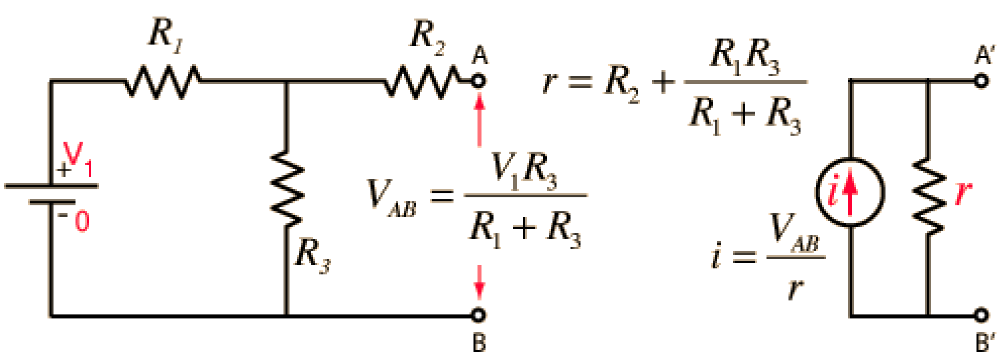
Norton Current

The value i for the current used in Norton's Theorem is found by determining the open circuit voltage at the terminals AB and dividing it by the Norton resistance r.



Norton Example

Replacing a network by its Norton equivalent can simplify the analysis of a complex circuit. In this example, the Norton current is obtained from the open circuit voltage (the Thevenin voltage) divided by the resistance r. This resistance is the same as the Thevenin resistance, the resistance looking back from AB with V_1 replaced by a short circuit.



For R_1 = …. Ω

R_2 =….. Ω

R_3 =…… Ω

And voltage V_1 =…… V

The open circuit voltage is = $V_{AB} = \frac{V_1 R_3}{R_1 + R_3}$ = ……V

Since R_1 and R_3 form a simple voltage divider

The Norton resistance is = $r = R_2 + \frac{R_1 R_3}{R_1 + R_3}$ =...........Ω.

And the resulting Norton current is = $i = \frac{V_{AB}}{r}$ =..........A

Circuit diagram:

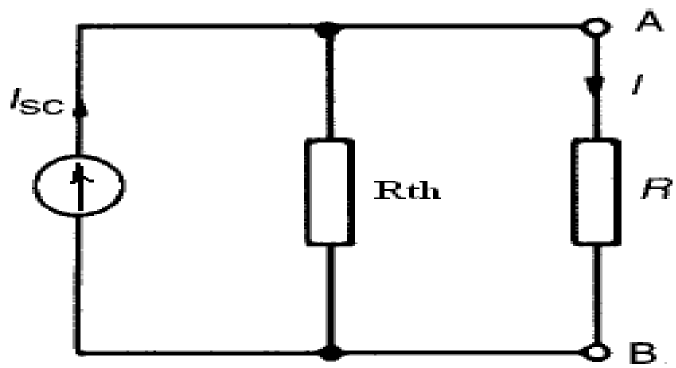


Figure : Norton's equivalent circuit

Observation table-

S. N.	Rth	Vth	I	
1.				Theoretical
2.				Practical

Result- Norton theorem is verified.
PRECAUTIONS: -

1) All the connection should be tight
2) It should be care that the values of the components of the circuit is does not exceed to their ratings (maximum value).
3) Before the circuit connection it should be check out working condition of all the Component

Experiment no – 4

Objective – Verify the theorems

a. Super position theorems

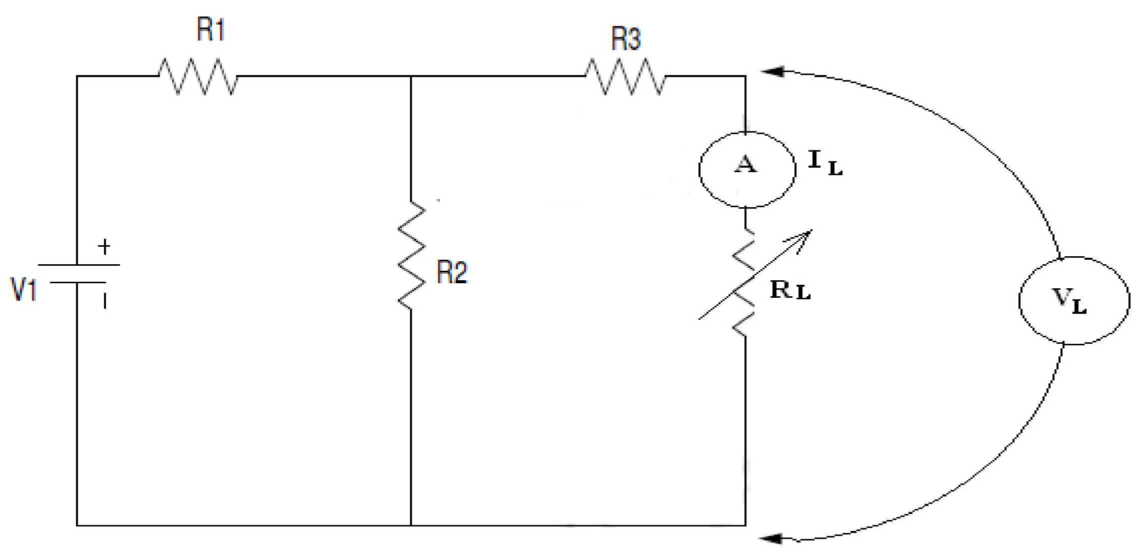
b. Maximum power transfer theorem force circuits

Apparatus required –1.Power supply 2.Variable resistors 3.Connecting wires 4. Bread board 5. Multimeters/ammeter

Theory - The maximum power transfer theorem states that a load resistance will abstract maximum power from the network when the load resistance is equal to the internal resistance. For maximum power transfer

Load resistance $R_L=R_{th}$
Where R_{th} equivalent resistance of the remaining circuit
Maximum power = $P_{max} = V^2/4R_L$
Where V is the dc supply voltage.

Circuit diagram -



PROCEDURE:

1. Connect the circuit diagram as shown in fig.
2. Take the readings of voltmeter and ammeter for different values of RL
3. Verify that power is maximum when $R_L = R_I$

OBSERVATION TABLE:

S.N.	R	V_L	I_L	P= V_LX I_L
1.	R1Ω			
2.	R2Ω			
3.	R3Ω			
4.	R4Ω=Rth			

RESULT:
Maximum power transfer theorem has been verified.

Superposition theorem

Apparatus required –
1) 2mm Patch cords, 2) Digital Multimeter,3) Superposition Theorem Kit

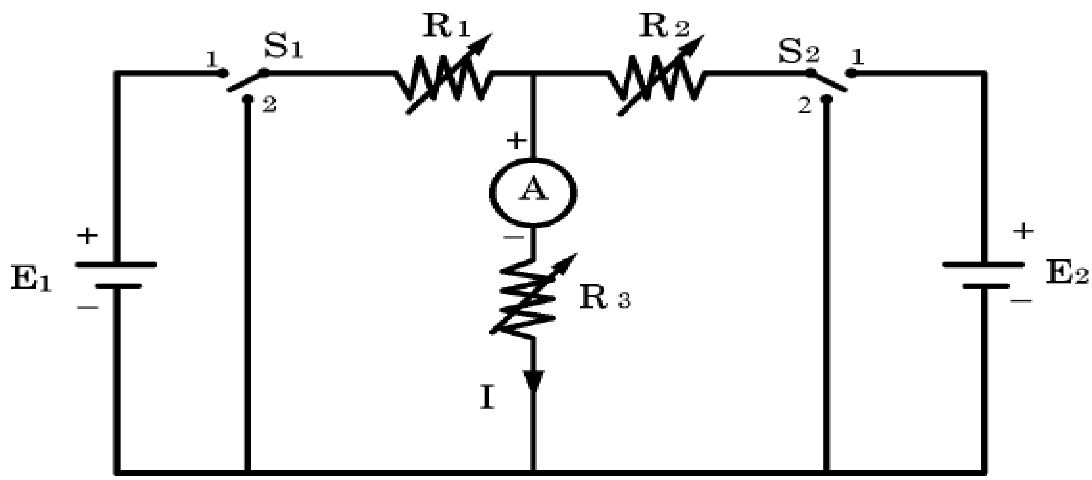
Statement- In a linear, bilateral, active network containing more than one source of emf/current, the current flowing through any branch/ point, is the algebraic sum of all the currents, flowing through that branch / point, considering that a single source of emf / current is in acting ata time, while all the other sources of emf / current are replaced by their internal resistances (for the time being)

EXPLANATION:

1. Select only one source and replace all other sources by their internal resistances. (If the source is the ideal current source replace it by open ckt. if the source is the ideal voltage source replaces it by short ckt.)

2. Find the current and its direction through the desired branch.3. Add all the branch currents to obtain the actual branch current.

Circuit diagram:-



STEPWISE PROCEDURE:

Make the connections as per the circuit diagram.

1. Keep both Switches S1 and S2 at position 1
2. Switch ON both the D.C. sources.
3. Adjust the Rheostats such that the ammeter will read properly.
4. Note down the value of current from ammeter A as I ampere.
5. Switch OFF the sources.
6. Keep the Switch S1 at position 1 and S2 at position 2 then Switch ON the supply source E1.
7. Note down the value of current from Ammeter as I' amp
8. Switch OFF the supply and Keep the Switch S1 at position 2 Switch S2 at position 1
9. Switch ON the supply source E2 only.
10. Note down the value of current from Ammeter as I''.
11. Switch OFF the supply.
12. Adjust the values of Rheostats and repeat the above procedure.
13. Note down two more sets of readings.
14. Switch OFF the supply and disconnect the circuit.

Observation table:

Sr. No.	Both sources E_1 and E_2 in the circuit I Amp	Only source E_1 in the circuit I' Amp	Only source E_2 in the circuit I" Amp	Calculated value of I = I' + I" Amp
1.				
2.				
3.				

CALCULATIONS:
Calculate branch current I = I' + I" for each reading and tabulate the result

RESULT:
Superposition theorem has been verified

PRECAUTIONS: -
1) All the connection should be tight
2) It should be care that the values of the components of the circuit is does not exceed to their ratings (maximum value).
3) Before the circuit connection it should be check out working condition of all the Component.

Experiment no 5

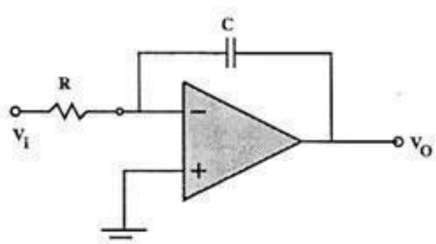
Objective- Observe the wave shape of an integrating circuit on the CRO.

Apparatus required: Four 6 volt batteries, Operational amplifiers, One 10 kΩ potentiometer, linear tape, two capacitors, 0.1 µF each, non-polarized, Two 100 kΩ resistors, Three 1 MΩ resistors

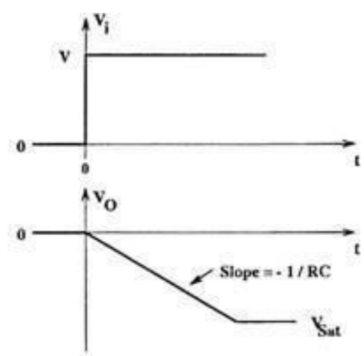
Theory- Connect a voltmeter between the op-amp's output terminal and the circuit ground point. Slowly move the potentiometer control while monitoring the output voltage. The output voltage should be changing at a rate established by the potentiometer's deviation from zero (center) position. To use calculus terms, we would say that the output voltage represents the integral (with respect to time) of the input voltage function. That is, the input voltage level establishes the output voltage rate of change over time. This is precisely the opposite of differentiation, where the derivative of a signal or function is its instantaneous rate of change. If you have two voltmeters, you may readily see this relationship between input voltage and output voltage rate of change by measuring the wiper voltage (between the potentiometer wiper and ground) with one meter and the output voltage (between the op-amp output terminal and ground) with the other. Adjusting the potentiometer to give zero volts should result in the slowest output voltage rate-of- change.

Conversely, the more voltage input to this circuit, the faster its output voltage will change, or "ramp."

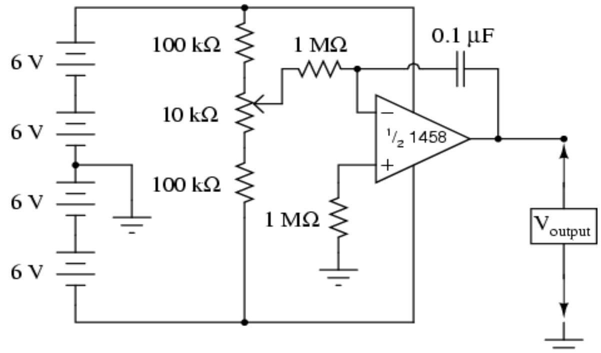
Circuit diagram –

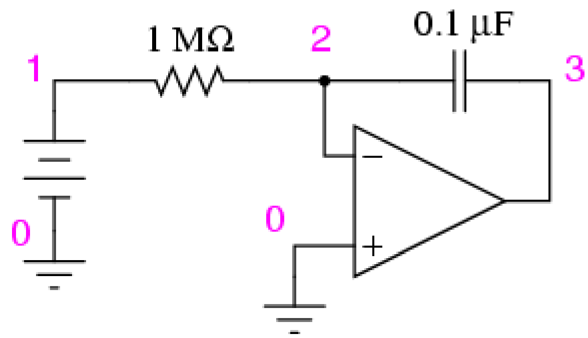


(a) Integrator Diagram



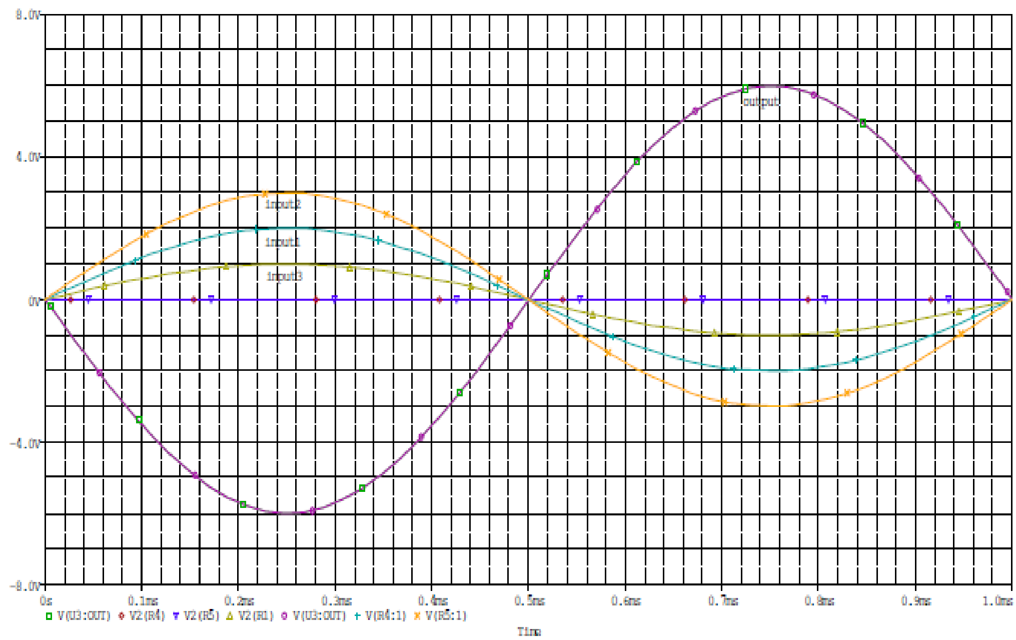
(b) Input and Output waveforms





As soon as the "grounding" resistor is shorted with a jumper wire, the op-amp's output voltage will start to change, or drift. Ideally, this should not happen, because the integrator circuit still has an input signal of zero volts. However, real operational amplifiers have a very small amount of current entering each input terminal called the bias current. These bias currents will drop voltage across any resistance in their path. Since the 1 MΩ input resistor conducts some amount of bias current regardless of input signal magnitude, it will drop voltage across its terminals due to bias current, thus "offsetting" the amount of signal voltage seen at the inverting terminal of the op-amp. If the other (non-inverting) input is connected directly to ground as we have done here, this "offset" voltage incurred by voltage drop generated by bias current will cause the integrator circuit to slowly "integrate" as though it were receiving a very small input signal.

Waveform:



Result - Wave shape of an integrating ckt on the CRO is observed

PRECAUTIONS: -

1) All the connection should be tight
2) It should be care that the values of the components of the circuit is does notexceed to their ratings (maximum value).
3) Before the circuit connection it should be check out working condition of allthe Component.

Experiment no 6

Objective- Observe the wave shape of a differentiating ckt.

Apparatus required Resistors, Capacitors, Op-Amp IC 741, Connecting wires and CRO probes

Theory:-

Differentiator: One of the simplest of the op-amp circuits that contain capacitor is the Differentiating Amplifier, or Differentiator. As the name suggests, the circuit performs the mathematical operation of differentiation, that is, the output waveform is the derivative of input waveform. A differentiator circuit is shown in fig. The node N is at virtual ground potential i.e., $V_n = 0$. The current i_C the capacitor is, $i_C = C_1$ d/dt $(V_i - V_n) = C_1\, dV_i/dt$
(1)The current if through the feedback resistor is Vo/Rf and there is no current into the opamp. Therefore, the nodal equation at node N is,
$C_1 (dV_i/dt) + Vo/V_f = 0$

From which we have

$Vo = - R_f C_1 (dV_i/dt)$ --------------------(2)

Thus the output voltage V_o is a constant $(-R_fC_1)$ times the derivative of the input voltageVi and the circuit is a differentiator. The sign indicates an 180^o phase shift of the output waveform V_0 with respect to the input signal.
The phasor equivalent of Eq. (2) is, $V_o(s) = - R_fC_1\, s\, V_i(s)$ where V_o and V_i is the phasor representation of V_o and V_i. In steady state, put s = j_. We may now write the magnitude of gain A of the differentiator as,

$A = V_o/V_i = -$ j_ $R_f C_1$ = _ $R_f C_1$ ------------------------------ (3)

From Eq. (3), one can draw the frequency response of the op-amp differentiator.
Equation (3) may be rewritten as$A = f / f_a$
Where, $f_a = 1/2\pi R_f C_1$

f= operating frequency

CIRCUIT DIAGRAM:

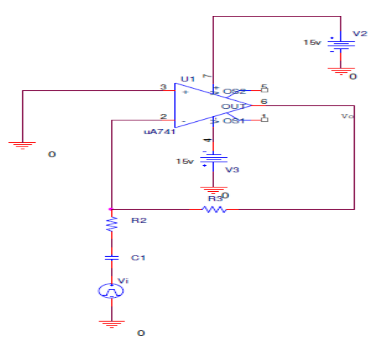


Fig. Differentiator

Waveform:

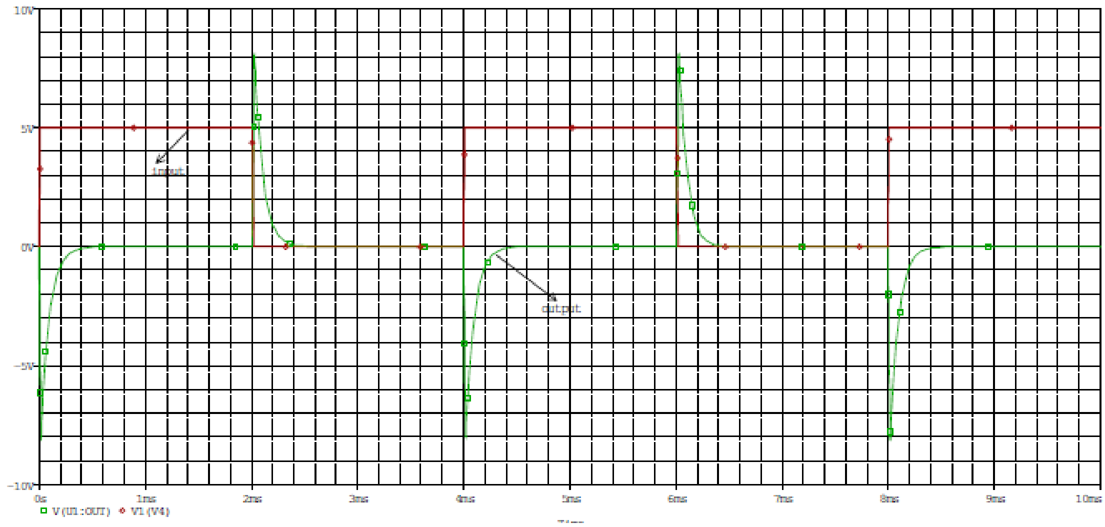


Fig. Input & Output waveforms

Result- Wave shape of a differentiating circuit is observed.

PRECAUTIONS: -

1) All the connection should be tight.
2) It should be care that the values of the components of the circuit is does not exceed to their ratings (maximum value).
3) Before the circuit connection it should be check out working condition of all the Component

Experiment No. – 7

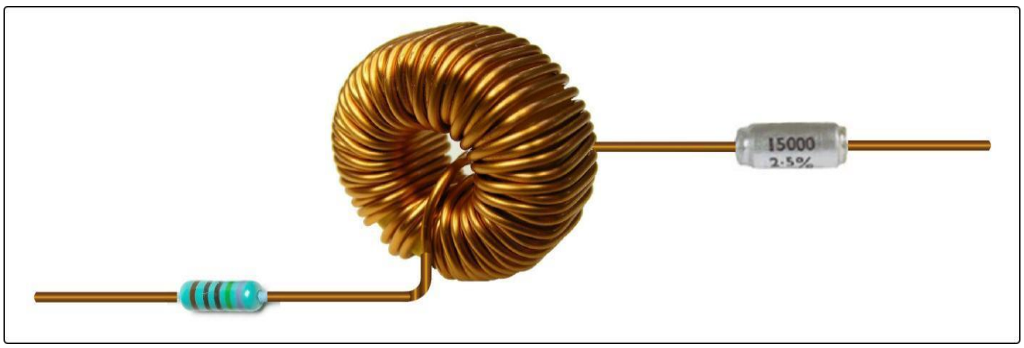
Objective - Find different electrical parameter in R.L.:R.C.:R.L.C. series circuits and draw the Phasor diagram and

a. Determine current and P.F. in each case
b. Determine and observe the resonance condition.

Apparatus required: Voltmeters, Watt meter, Rheostat, Inductor, Capacitor, 1 KVA, An autotransformer

Theory

An RLC circuit (or LCR circuit) is an electrical circuit consisting of a resistor, an inductor, and a capacitor, connected in series or in parallel. The RLC part of the name is due to those letters being the usual electrical symbols for resistance, inductance and capacitance respectively. The circuit forms a harmonic oscillator for current and will resonate in just the same way as an LC circuit will. The difference that the presence of the resistor makes is that any oscillation induced in the circuit will die away over time if it not kept going by a source. This effect of the resistor is called damping. Some resistanceis unavoidable in real circuits, even if a resistor is not specifically included as a component. A pure LC circuit is an ideal which really only exists in theory.



A series RLC circuit: a resistor, inductor, and a capacitor

Resonance

An important property of this circuit is its ability to resonate at a specific frequency, the resonance frequency. Frequencies are measured in units of hertz. In this article, however, angular frequency, is used which is more mathematically convenient. This is measured in radians per second. They are related to each other by a simple proportion,

Natural frequency

The resonance frequency is defined in terms of the impedance presented to a driving source. It is still possible for the circuit to carry on oscillating (for a time) after the driving source has been removed or it is subjected to a step in voltage (including a step down to

Zero). This is similar to the way that a tuning fork will carry on ringing after it has been struck, and the effect is often called ringing. This effect is the undriven natural resonance frequency of the circuit and in general is not exactly the same as the driven resonance frequency, although the two will usually be quite close to each other. Various terms are used by different authors to distinguish the two, but resonance frequency unqualified usually means the driven resonance frequency. The driven frequency may be called the undamped resonance frequency or undamped natural frequency and the undriven frequency may be called the damped resonance frequency or the damped natural frequency. The reason for this terminology is that the driven resonance frequency in a series or parallel resonant circuit has the value.

Damping

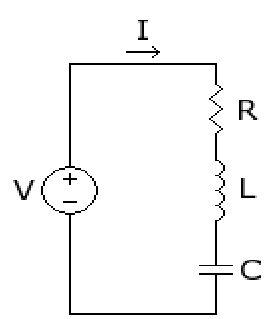
Damping is caused by the resistance in the circuit. It determines whether or not the circuit will resonate naturally (that is, without a driving source). Circuits which will resonate in this way are described as underdamped and those that will not are overdamped.
Damping attenuation (symbol α) is measured in nepers per second. However, the unitless damping factor (symbol ζ) is often a more useful measure, which is related to $\alpha$by

Bandwidth

The resonance effect can be used for filtering, the rapid change in impedance near resonance can be used to pass or block signals close to the resonance frequency. Bothband-pass and band-stop filters can be constructed and some filter circuits are shown later in the article. A key parameter in filter design is bandwidth. The bandwidth is measured between the 3dB-points, that is, the frequencies at which the power passed through the circuit has fallen to half the value passed at resonance. There are two of these half-power frequencies, one above, and one below the resonance frequency

Q factor

The Q factor is a widespread measure used to characterise resonators. It is defined as the peak energy stored in the circuit divided by the average energy dissipated in it per cycle at resonance. Low Q circuits are therefore damped and lossy and high Q circuits are underdamped. Q is related to bandwidth; low Q circuits are wide band and high Q circuits are narrow band. In fact, it happens that Q is the inverse of fractional bandwidth



Series RLC circuit

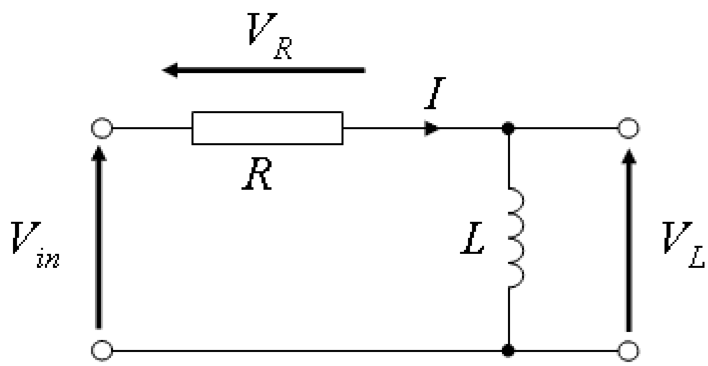
. RLC series circuit
- V - The voltage of the power source
- I - the current in the circuit
- R - The resistance of the resistor L - the inductance of the inductor
- C - The capacitance of the capacitor

In this circuit, the three components are all in series with the voltage source. The governing differential equation can be found by substituting into Kirchhoff's voltage law(KVL) the constitutive equation for each of the three elements. From KVL, Where are the voltages across R, L and C respectively and is the time varying voltage from the source. Substituting in the constitutive equations,

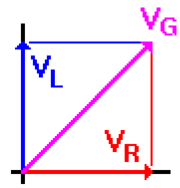
A resistor-inductor circuit (RL circuit), or RL filter or RL network, is one of the simplest analogue infinite impulse response electronic filters. It consists of a resistor and an inductor, either in series or in parallel, driven by a voltage source.

The fundamental passive linear circuit elements are the resistor (R), capacitor (C) and inductor (L). These circuit elements can be combined to form an electrical circuit in four distinct ways: the RC circuit, the RL circuit, the LC circuit and the RLC circuit with the abbreviations indicating which components are used. These circuits exhibit important types of behaviour that are fundamental to analogue electronics. In particular, they areable to act as passive filters. This article considers the RL circuit in both series and parallel as shown in the diagrams.

In practice, however, capacitors (and RC circuits) are usually preferred to inductors since they can be more easily manufactured and are generally physically smaller, particularly for higher values of components



The Vectors in an RL Series Circuit



The vectors for this example circuit are shown to the right. This time the composite phaseangle is positive instead of negative, because V_L leads I_L. But to determine just what that phase angle is, we must start by determining X_L and then calculating the rest of the circuitparameters.

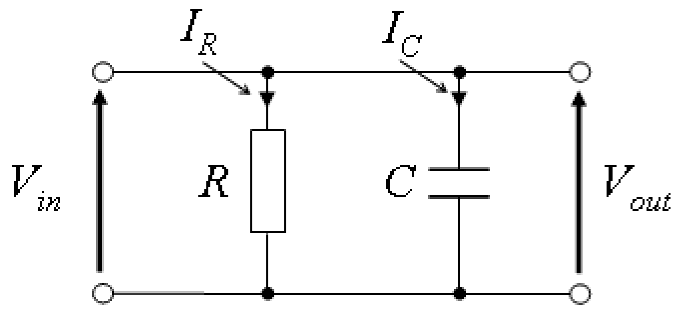
$X_{L=}2\pi fL$

I=E/Z

$V_{L=}$ I* X_L

$V_{R=}$ I × R

RC Circuits

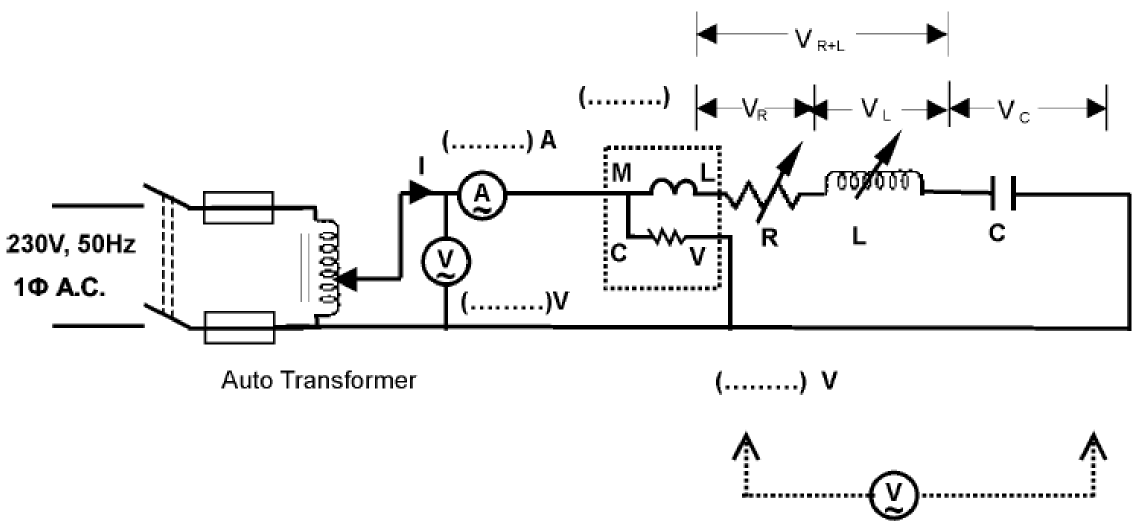


A parallel RC Circuit

No, RC does not stand for "Remote Control". An RC circuit is a circuit that has both a resistor (R) and a capacitor (C). Like the RL Circuit, we will combine the resistor and thesource on one side of the circuit, and combine them into a thevening source. Then if we apply KVL around the resulting loop, we get the following equation:

$$v_{source} = RC\frac{dv_{capacitor}(t)}{dt} + v_{capacitor}(t)$$

Circuit diagram



STEPWISE PROCEDURE:

– Discharge the capacitor before and after use.
– Connect the circuit as shown in figure.
– Initially set the autotransformer to zero position and rheostat to maximum position.Switch
- ON the supply.
– Apply suitable voltage so that desired current flows in the circuit.
– Record the values of V, I, VR, VL, VC by varying resistance, keeping inductance unchanged.
– Record the values of V, I, VR, VL, VC by varying inductance, Keeping resistance unchanged.
– Reduce the voltage to zero and switch ―OFF the supply.
— Draw the phasor diagram for each reading

Observation table

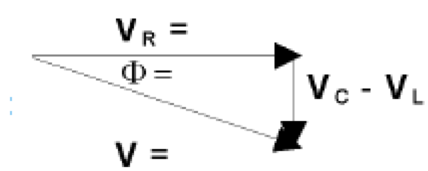
Resistance of inductor = r =...............

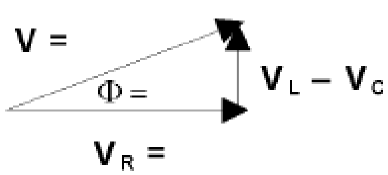
Table for measured and calculated values of V, I etc

Sr. No.	V (V)	I (A)	V_R (V)	V_L (V)	V_{R+L} (V	V_C (V)	Z= $\frac{V}{I}$ Ω	R= $\frac{VR}{I}$ Ω	Z_L= $\frac{VR+L}{I}$ Ω	X_C= $\frac{VC}{I}$ Ω	X_L Ω	Φ from phasor diagram
1												
2												
3												
4												
5												
6												

....Power consumed by circuit = Watt

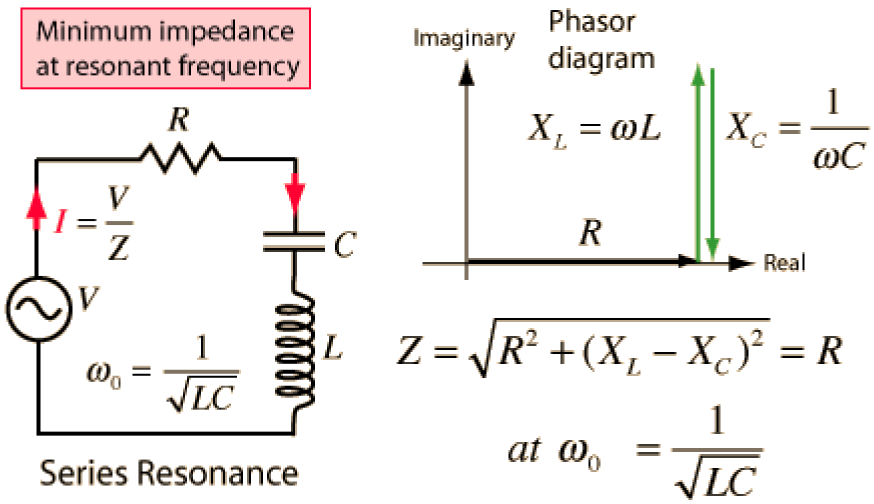
Phasor diagram





Series Resonance

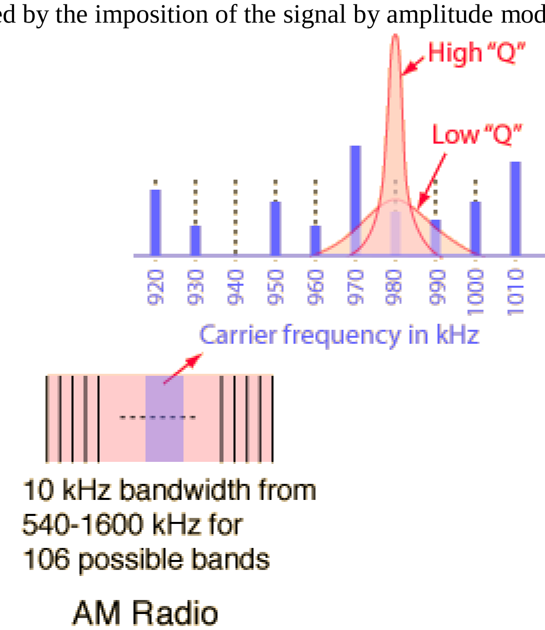
The resonance of a series RLC circuit occurs when the inductive and capacitive reactances are equal in magnitude but cancel each other because they are 180 degrees apart in phase. The sharp minimum in impedance which occurs is useful in tuning applications. The sharpness of the minimum depends on the value of R and is characterized by the "Q" of the circuit.



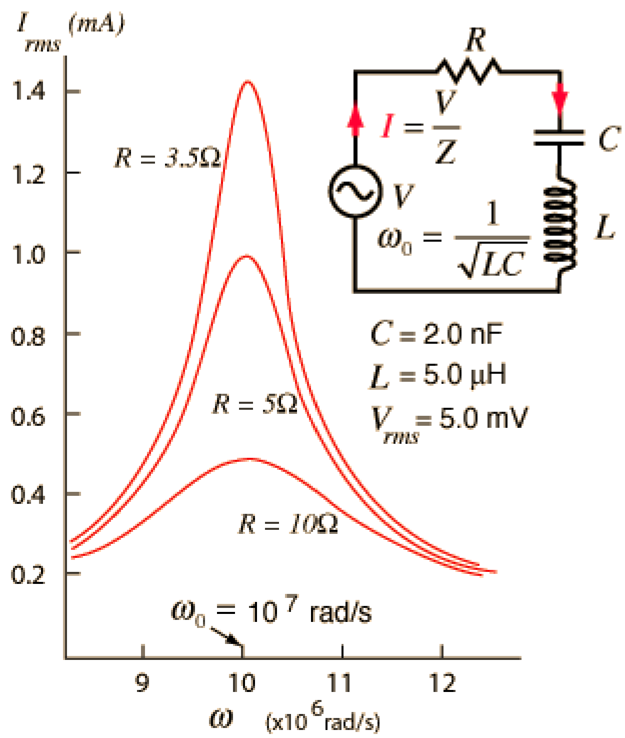
Selectivity and Q of a Circuit

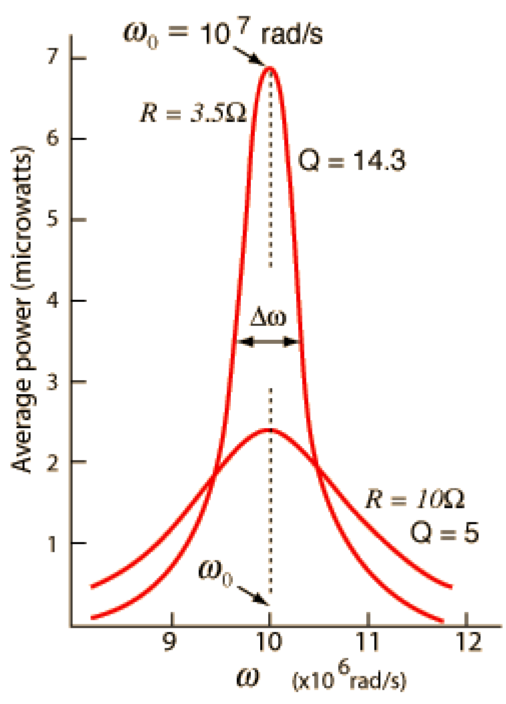
Resonant circuits are used to respond selectively to signals of a given frequency while discriminating against signals of different frequencies. If the response of the circuit is more narrowly peaked around the chosen frequency, we say that the circuit has higher "selectivity". A "quality factor" Q, as described below, is a measure of that selectivity, and we speak of a circuit having a "high Q" if it is more narrowly selective.

An example of the application of resonant circuits is the selection of AM radio stations by the radio receiver. The selectivity of the tuning must be high enough to discriminate strongly against stations above and below in carrier frequency, but not so high as to discriminate against the "sidebands" created by the imposition of the signal by amplitude modulation.



The selectivity of a circuit is dependent upon the amount of resistance in the circuit. The variations on a series resonant circuit at right follow an example in Serway & Beichner. The smaller the resistance, the higher the "Q" for given values of L and C. The parallel resonant circuit is more commonly used in electronics, but the algebra necessary to characterize the resonance is much more involved.





.

The quality factor Q is defined by

$$Q = \frac{\omega_0}{\Delta\omega}$$

where Δω is the width of the resonant power curve at half maximum.
Since that width turns out to be Δω =R/L, the value of Q can also be expressed as

$$Q = \frac{\omega_0 L}{R}$$

The Q is a commonly used parameter in electronics, with values usually in therange of Q=10 to 100 for circuit applications.

Power in a Series Resonant Circuit

The average power dissipated in a series resonant circuit can be expressed in terms ofthe rms voltage and current as follows:

$$P_{avg} = I_{rms}^2 R = \frac{V_{rms}^2}{Z^2} R = \frac{V_{rms}^2 R}{R^2 + (X_L - X_C)^2}$$

Using the forms of the inductive reactance and capacitive reactance, the term involvingthem can be expressed in terms of the frequency.

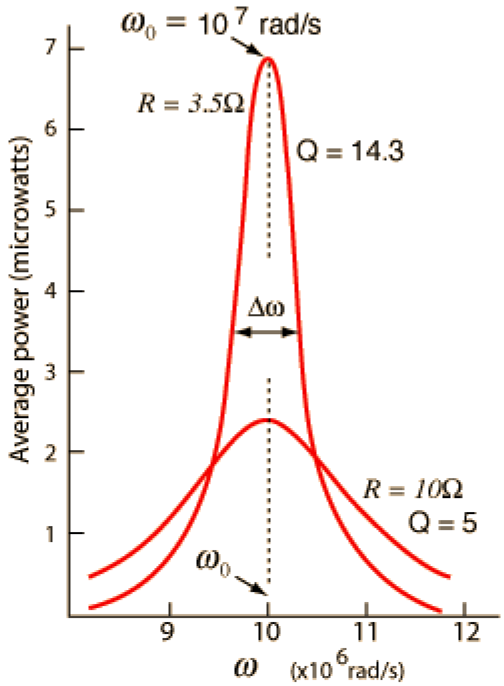
$$(X_L - X_C)^2 = \left(\omega L - \frac{1}{\omega C}\right)^2 = \frac{L^2}{\omega^2}(\omega^2 - \omega_0^2)^2$$

where use has been made of the resonant frequency expression

$$\omega_0 = \frac{1}{\sqrt{LC}}$$

Substitution now gives the expression for average power as a function of frequency.

$$P_{avg} = \frac{V_{rms}^2 R\omega^2}{R^2\omega^2 + L^2(\omega^2 - \omega_0^2)^2}$$



This power distribution is plotted at left using the same circuit parameters as wereused in the example on the Q factor of theseries resonant circuit

The average power at resonance is just

$$P_{avg} = \frac{V_{rms}^2}{R}$$

since at the resonant frequency ω_0 the reactive parts cancel so that the circuit appears as just the resistance R.

Result -different electrical parameter in R.L.:R.C.:R.L.C. series circuits are found

PRECAUTIONS: -

1) All the connection should be tight
2) It should be care that the values of the components of the circuit is does not exceed to their ratings (maximum value).
3) Before the circuit connection it should be check out working condition of all the Component

Experiment No.- 8

Objective – Find different electrical parameter in R-C & R-L-C parallel circuit and draw the 1) Phasor diagram. 2) Find power and P.F. of the circuit 3) Observe parallel resonance condition

Apparatus required – RLC kit, wires.

Theory -

Parallel Resonance

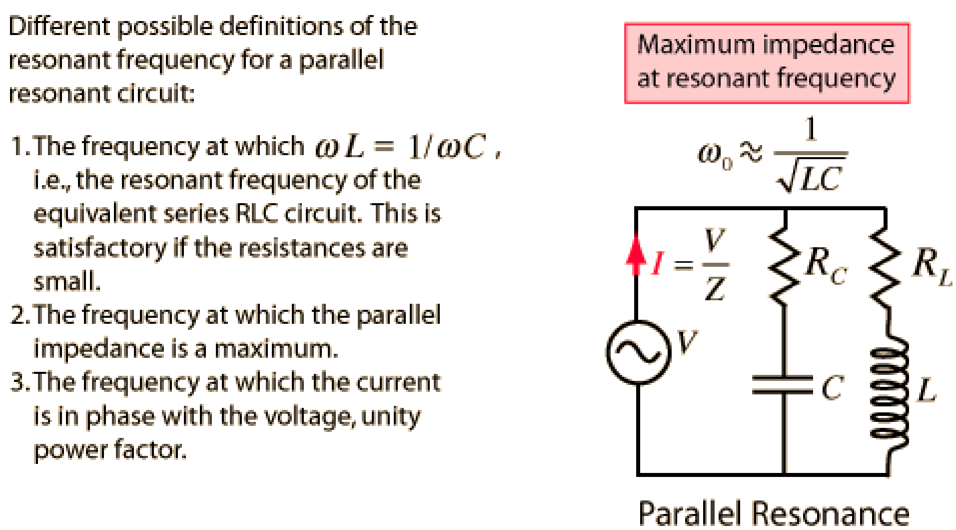
The resonance of a parallel RLC circuit is a bit more involved than the series resonance. The resonant frequency can be defined in three different ways, which converge on the same expression as the series resonant frequency if the resistance of the circuit is small.

In simple reactive circuits with little or no resistance, the effects of radically altered impedance

$$f_{resonant} = \frac{1}{2\pi\sqrt{LC}}$$

will manifest at the resonance frequency predicted by the equation given earlier. In a parallel (tank) LC circuit, this means infinite impedance at resonance. In series LC circuit, it means zero impedance at resonance:

However, as soon as significant levels of resistance are introduced into most LC circuits, this simple calculation for resonance becomes invalid. We'll take a look at several LC circuits with added resistance, using the same values for capacitance and inductance as before: 10 µF and 100 mH, respectively. According to our simple equation, the resonant frequency should be 159.155 Hz. Watch, though, where current reaches maximum or minimum in the following SPICE



Resonance: Impedance Maximum

One of the ways to define resonance for a parallel RLC circuit is the frequency at which the impedance is maximum. The general case is rather complex, but the special case where the resistances of the inductor and capacitor are negligible can be handled readily by using the concept of admittance.

Resonance: Phase Definition

Defining the parallel resonant frequency as the frequency at which the voltage and current are in phase, unity power factor, gives the following expression for the resonantfrequency:

$$\omega_0 = \frac{1}{\sqrt{LC}}\left[\frac{R_L^2C - L}{R_C^2C - L}\right]^{\frac{1}{2}}$$

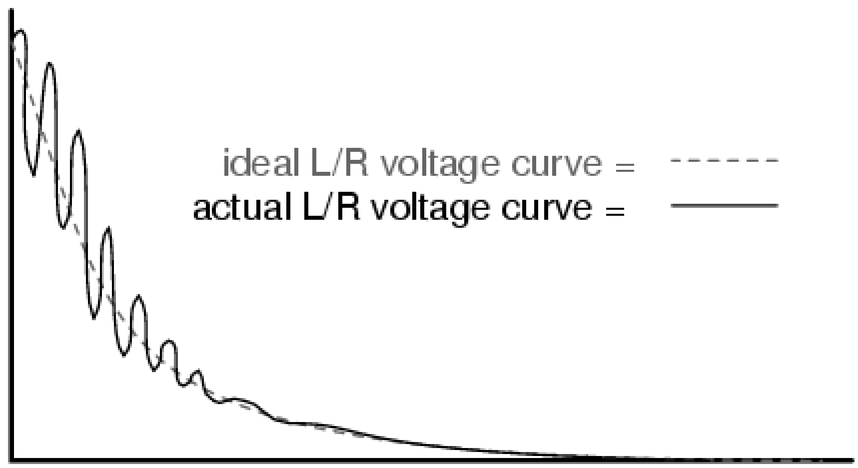
The above resonant frequency expression is obtained by taking the impedance expressions for the parallel RLC circuit and setting the expression for X_{eq} equal to zero to force the phase to zero.

Admittance

Although the impedance Z is a far more common way to characterize the voltage-current relationships in an AC circuit, there are times when the admittance is a valuable construct. For a given circuit element, the admittance is just the reciprocal of the impedance.

The admittance has its most obvious utility in dealing with parallel AC circuits where there are no series elements. The equivalent admittance of parallel elements is the sum of the admittances of the components.

Response curve



Inductor ringing due to resonance with stray capacitance.

All inductors contain a certain amount of stray capacitance due to turn-to-turn and turn-to-core insulation gaps. Also, the placement of circuit conductors may create stray capacitance. While clean circuit layout is important in eliminating much of this stray capacitance, there will always be some that you cannot eliminate. If this causes resonant problems (unwanted AC oscillations), added resistance may be a way to combat it. If resistor R is large enough, it will cause a condition of anti-resonance, dissipating enough energy to prohibit the inductance and stray capacitance from sustaining oscillations for very long.

As was mentioned before, the angle of this ―power triangle‖ graphically indicates the ratio between the amount of dissipated (or consumed) power and the amount of absorbed/returned power. It also happens to be the same angle as that of the circuit's impedance in polar form. When expressed as a fraction, this ratio between true power and apparent power is called the power factor for this circuit. Because true power and apparent power form the adjacent and hypotenuse sides of a right triangle, respectively, the power factor ratio is also equal to the cosine of that:

$$\text{Power factor} = \frac{\text{True power}}{\text{Apparent power}}$$

$$\text{Power factor} = \frac{119.365\ \text{W}}{169.256\ \text{VA}}$$

$$\text{Power factor} = 0.705$$

$$\cos 45.152^\circ = 0.705$$

Phase angle.

The "power loss" within the capacitor is that portion of the total power applied to the capacitor that is not stored by the capacitor on an instantaneous basis. As the current passes through the series resistance element, it generates heat, which is the "power loss." It should be noted that all energy losses (due to leads, dielectric polarization, connections, and eddy currents in the electrode material) are taken into account by the "equivalent series resistance" element. Analysis of the power equation shows:

$$(EI)^2 = (E_R I)^2 + (E_C I - E_L I)^2$$

substituting: $(I^2 Z)^2 = (I^2 R)^2 + (I^2 X_C - I^2 X_L)^2$

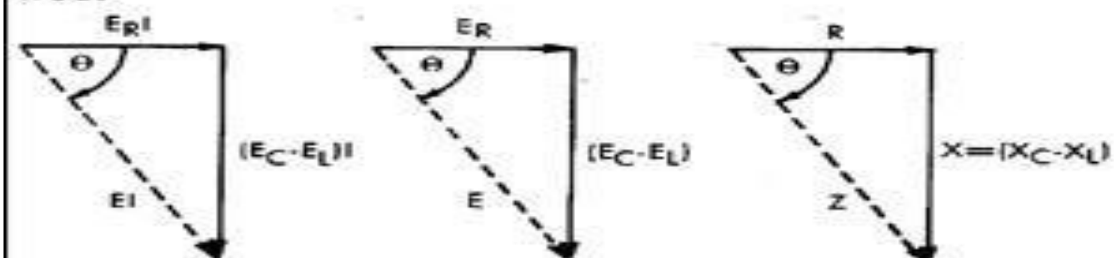
$$(\text{Power in})^2 = (\text{Power losses})^2 + (\text{power out})^2$$

And therefore the power factor by definition is:

$$PF = \frac{\text{Power Losses}}{\text{Power In}} = \frac{I^2 R}{I^2 Z} = \frac{E_R I}{EI} = \cos\theta$$

(from the vector diagram)

Note also that the following vector relationships hold true:



Power Diagram　　Voltage Diagram　　Impedance Diagram

$$PF = \cos\theta = \frac{E_R I}{EI} = \frac{E_R}{E} = \frac{R}{Z}$$

Result –.

PRECAUTIONS: -

1) All the connection should be tight
2) It should be care that the values of the components of the circuit is does not exceed to their ratings (maximum value).
3) Before the circuit connection it should be check out working condition of all the components.