



Government Girls' Polytechnic, Bilaspur

Name of the Lab: **Electrical & Electronic
Measurement Lab**

Practical: **Network Analysis Lab**

Class : **3rd Semester (ET&T)**

Teachers Assessment: 10
Examination: 50

End Semester

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 VB1a 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 = 20k 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. 10^{-6} A).

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

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 ($I = V/R$):

$$I_1 = \text{-----}$$

$$I_2 = \text{-----}$$

$$I_3 = \text{-----}$$

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

For Kirchoffs Voltage Law.

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

$I_1 R_1$ ----- is -ve (fall in potential)

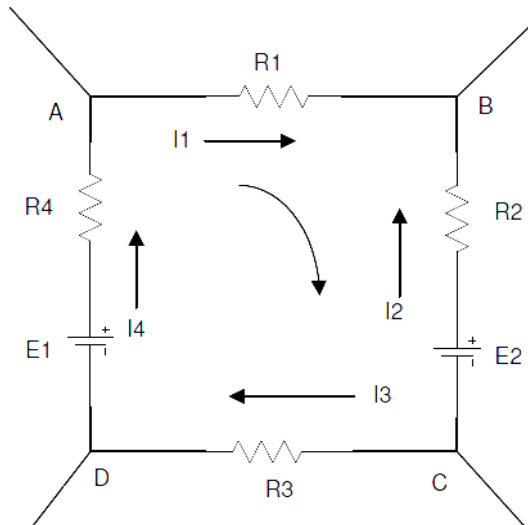
$I_2 R_2$ ----- is +ve (rise in potential)

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

$I_3 R_3$ ----- is -ve (fall in potential)

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

$I_4 R_4$ ----- is -ve (fall in potential)



According to KVL

$$I_1 R_1 - I_2 R_2 - E_2 - I_3 R_3 + E_1 - I_4 R_4 = 0$$

$$-I_1 R_1 + I_2 R_2 - I_3 R_3 - I_4 R_4 = E_2 - E_1$$

$$I_1 R_1 - I_2 R_2 + I_3 R_3 + I_4 R_4 = E_1 - E_2$$

$$VR_1 - VR_2 + VR_3 + VR_4 = E_1 - E_2$$

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 resistor 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 R 1	V R 2	V R 3	V R 4	V R 5	V1= VR1 +VR 2+V R5	V2= VR3 +VR 4+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

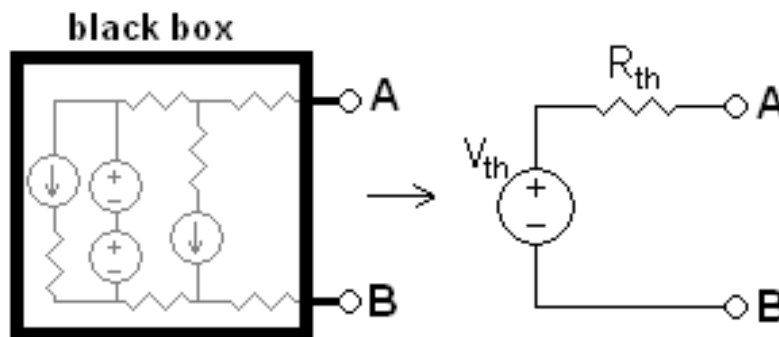
Objective :- Apply the theiving theorem for finding current in a complex electrical circuit

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 V_{Th} .
2. Calculate the output current, I_{AB} , when the output terminals are short circuited (load resistance is 0). R_{Th} equals V_{Th} divided by this I_{AB} .

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

Step 2 could also be thought of as:

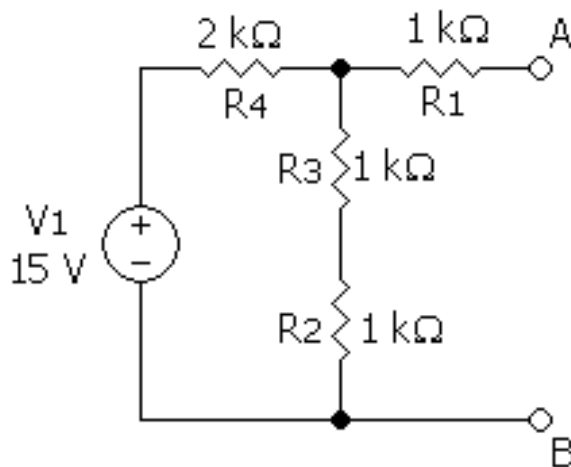
2a. Replace voltage sources with short circuits, and current sources with open circuits.

2b. Calculate the resistance between terminals A and B. This is R_{Th} .

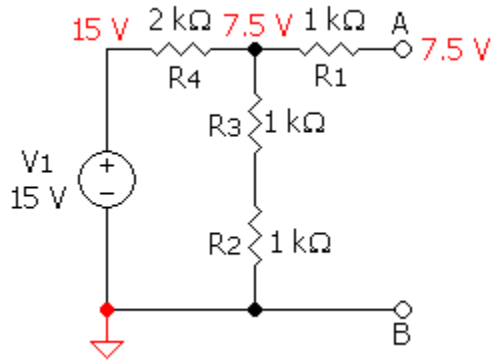
The Thévenin-equivalent voltage is the voltage at the output terminals of the original circuit. When calculating a Thévenin-equivalent voltage, the voltage divider principle is often useful, by declaring one terminal to be V_{out} and the other terminal to be at the ground point.

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.

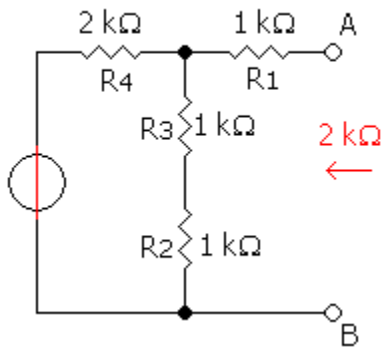
Example



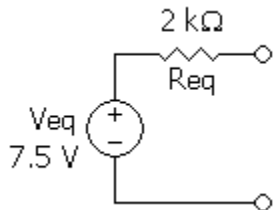
Step 0: The original circuit



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:

$$V_{th} = \left[\frac{R_2 + R_3}{R_2 + R_3 + R_4} \right] V_1$$

$$V_{th} = \left[\frac{1\text{ k}\Omega + 1\text{ k}\Omega}{1\text{ k}\Omega + 1\text{ k}\Omega + 2\text{ k}\Omega} \right] 15\text{V}$$

$$= \frac{1}{2}(15) = 7.5\text{V}$$

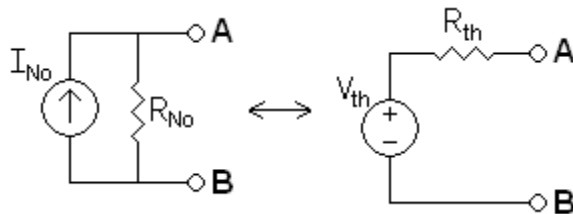
(notice that R_1 is not taken into consideration, as above calculations are done in an open circuit condition between A and B, therefore no current flows through this part, which means there is no current through R_1 and therefore no voltage drop along this part)

Calculating equivalent resistance:

$$\begin{aligned} R_{th} &= R_1 + [(R_2 + R_3) \parallel R_4] \\ &= 1 \text{ k } \Omega + [(1 \text{ k } \Omega + 1 \text{ k } \Omega) \parallel 2 \text{ k } \Omega] \\ &= 2 \text{ k } \Omega \end{aligned}$$

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 the Thévenin 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 point of 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

Theory

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

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).

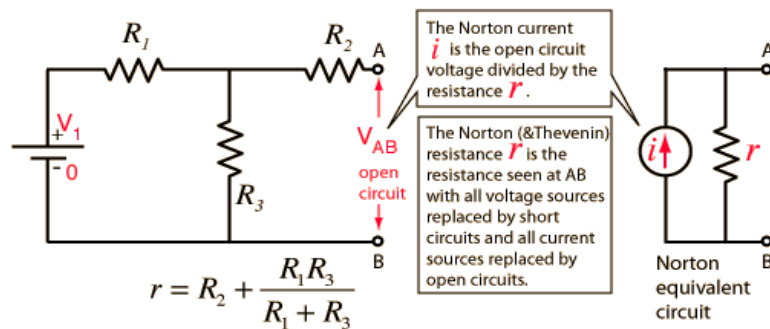
Calculation of a Norton equivalent circuit

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} .

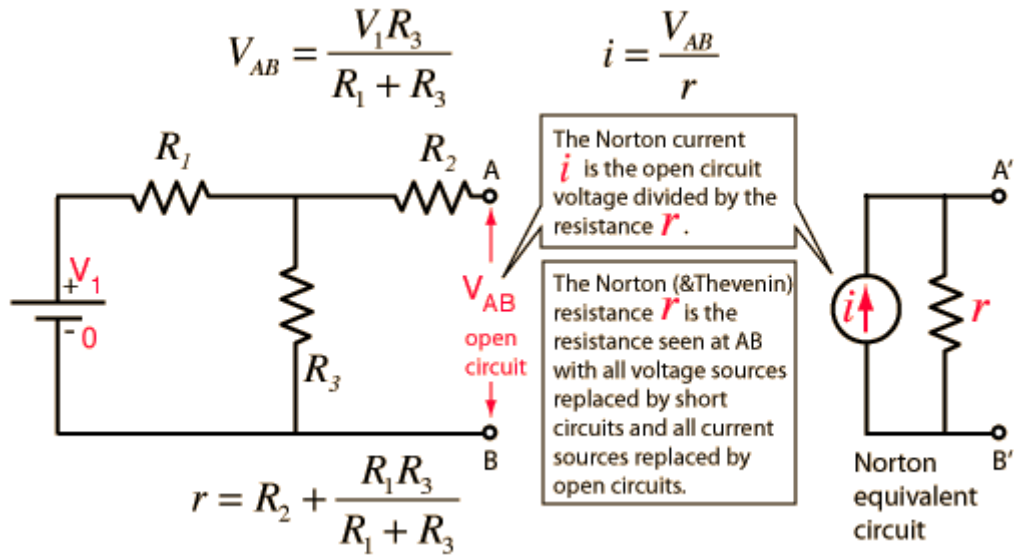
Calculate the output voltage, V_{AB} , when in open circuit condition (i.e., no load resistor — meaning infinite load resistance). R_{No} equals this V_{AB} divided by I_{No} . OR Replace independent voltage sources with short circuits and independent current sources with open circuits. The total resistance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

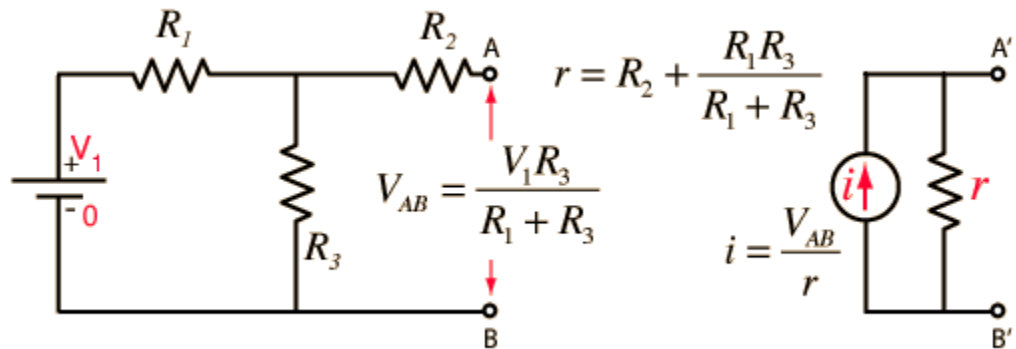
The value i for the current used in Norton's Theorem is found by determining the open circuit voltage at the terminals AB and dividing it by the Norton resistance r .



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 $R_1 = \dots \Omega$

$R_2 = \dots \Omega$

$R_3 = \dots \Omega$

And voltage $V_1 = \dots \text{V}$

The open circuit voltage is $= \frac{V_1 R_3}{R_1 + R_3} = \dots \text{V}$

Since R_1 and R_3 form a simple voltage divider

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

and the resulting Norton current is $= i = \frac{V_{AB}}{r} = \dots \text{A}$

circuit diagram

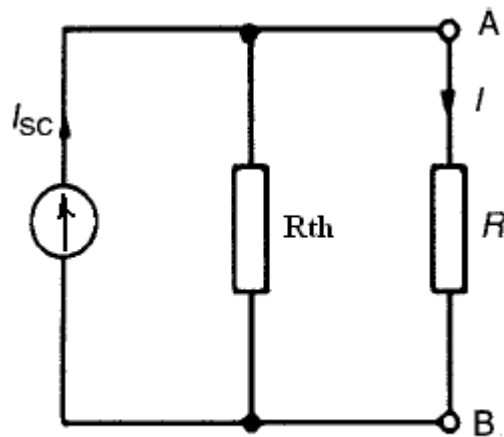


Figure : Norton's equivalent circuit

Observation table-

S. N.	R_{th}	V_{th}	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).
- 4) 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 supply2. Variable resistors3. Connecting wires4. Bread board5. 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

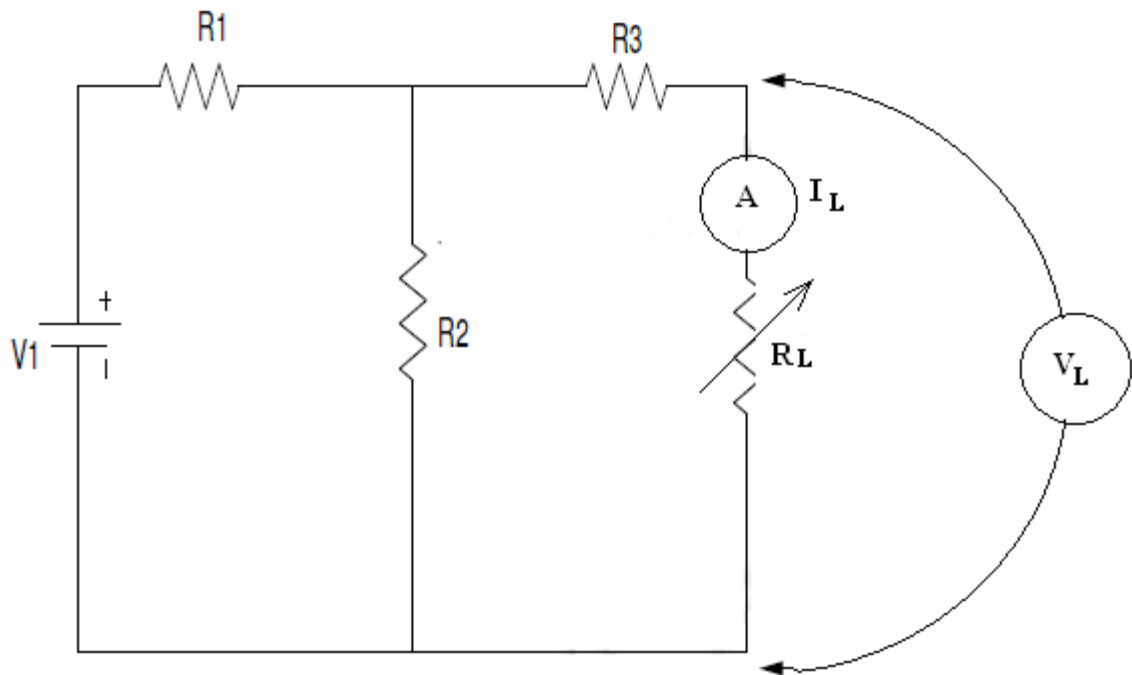
Load resistance $R_L = R_{th}$

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 R_L
3. Verify that power is maximum when $R_L = R_1$

OBSERVATION TABLE:

S.N.	R	V_L	I_L	$P = V_L \times I_L$
1.	$R1\Omega$			
2.	$R2\Omega$			
3.	$R3\Omega$			
4.	$R4\Omega = R_{th}$			

RESULT:

Maximum power transfer theorem has been verified.

Superposition theorem

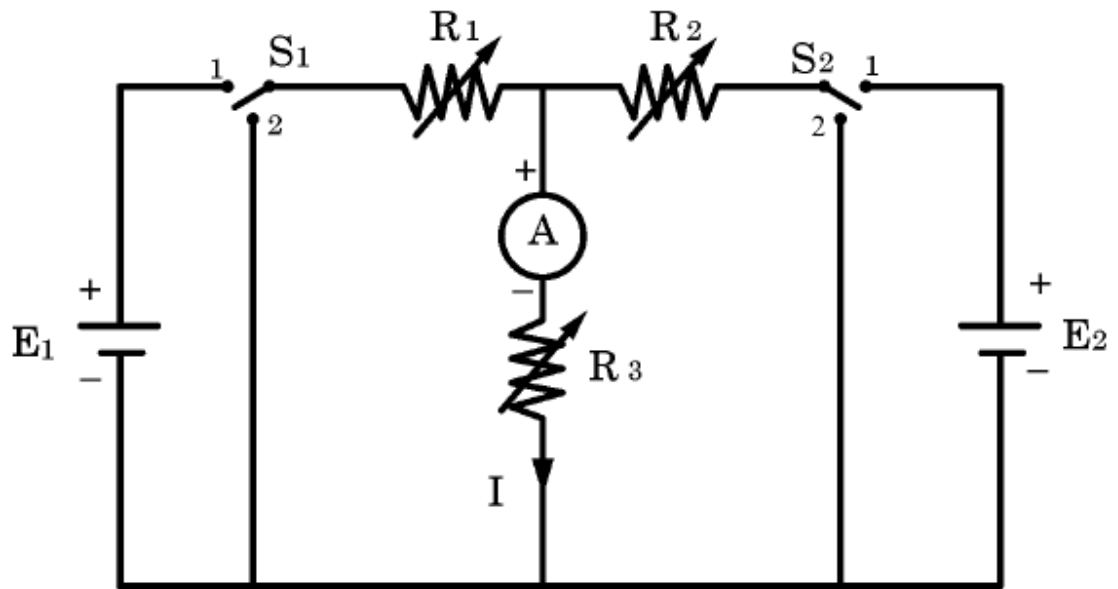
Apparatus required –

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 at a 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 :

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

OBSERVATIONS 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).
- 5) Before the circuit connection it should be check out working condition of all the Component.

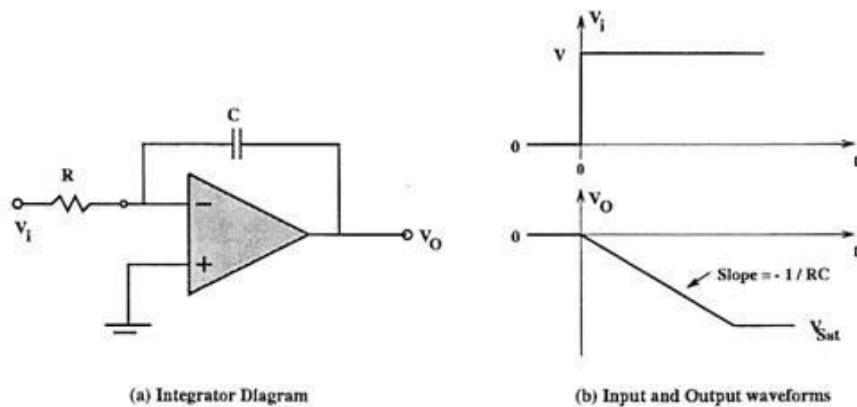
Experiment no 5

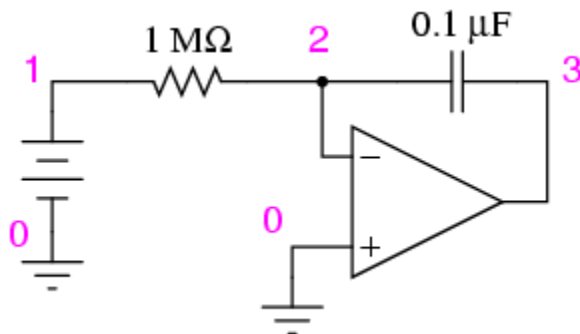
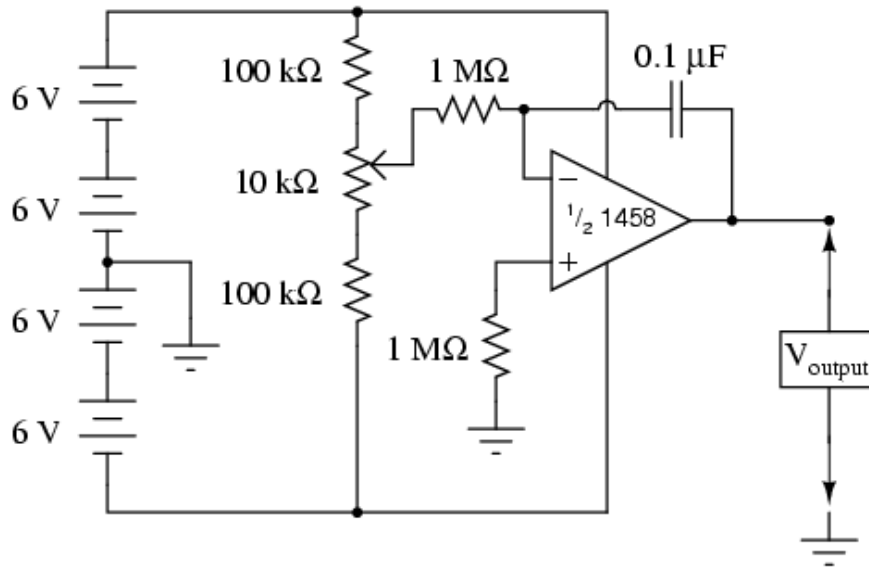
Objective- Observe the wave shape of an integrating ckt on the CRO.

Apparatus required Four 6 volt batteries
Operational amplifier, model 1458 recommended (Radio Shack catalog # 276-038)
One 10 k Ω potentiometer, linear taper (Radio Shack catalog # 271-1715)
Two capacitors, 0.1 μ F each, non-polarized (Radio Shack catalog # 272-135)
Two 100 k Ω resistors
Three 1 M Ω 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."

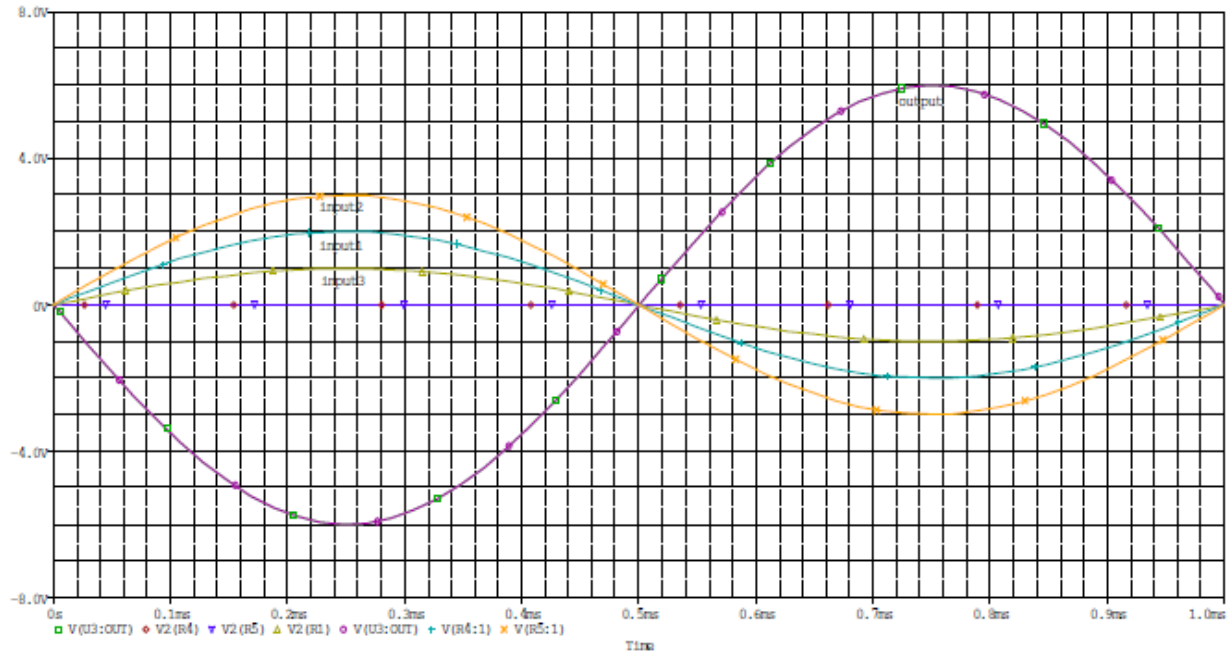
Circuit diagram –





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 $1\text{ M}\Omega$ 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 (noninverting) 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 not exceed to their ratings (maximum value).
- 6) Before the circuit connection it should be check out working condition of all the 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 through the capacitor is, $i_C = C_1 \frac{d}{dt} (V_i - V_n) = C_1 \frac{dV_i}{dt}$ -----(1) The current through the feedback resistor is V_o/R_f and there is no current into the opamp. Therefore, the nodal equation at node N is,

$$C_1 \left(\frac{dV_i}{dt} \right) + \frac{V_o}{R_f} = 0$$

From which we have

$$V_o = - R_f C_1 \left(\frac{dV_i}{dt} \right) \text{-----}(2)$$

Thus the output voltage V_o is a constant $(-R_f C_1)$ times the derivative of the input voltage V_i and the circuit is a differentiator. The sign indicates a 180° phase shift of the output waveform V_o with respect to the input signal.

The phasor equivalent of Eq. (2) is, $V_o(s) = - R_f C_1 s V_i(s)$ where V_o and V_i is the phasor representation of V_o and V_i . In steady state, put $s = j\omega$. We may now write the magnitude of gain A of the differentiator as,

$$A = \frac{V_o}{V_i} = - j\omega R_f C_1 = \omega R_f C_1 \text{-----}(3)$$

From Eq. (3), one can draw the frequency response of the op-amp differentiator.

Equation

(3) may be rewritten as

$$A = \frac{f}{f_a}$$

$$\text{Where, } f_a = \frac{1}{2\pi R_f C_1}$$

f = operating frequency

CIRCUIT DIAGRAM:

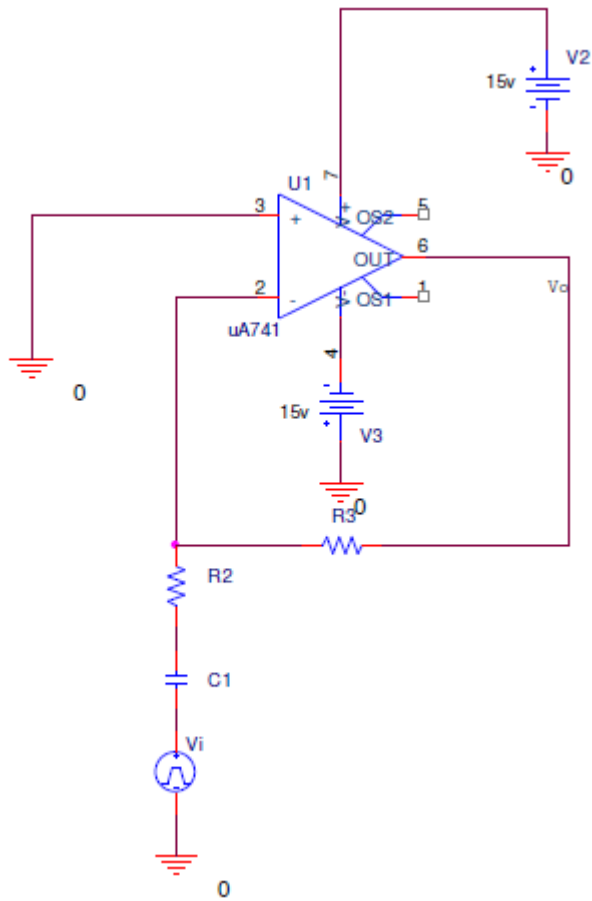


Fig. Differentiator

Waveform

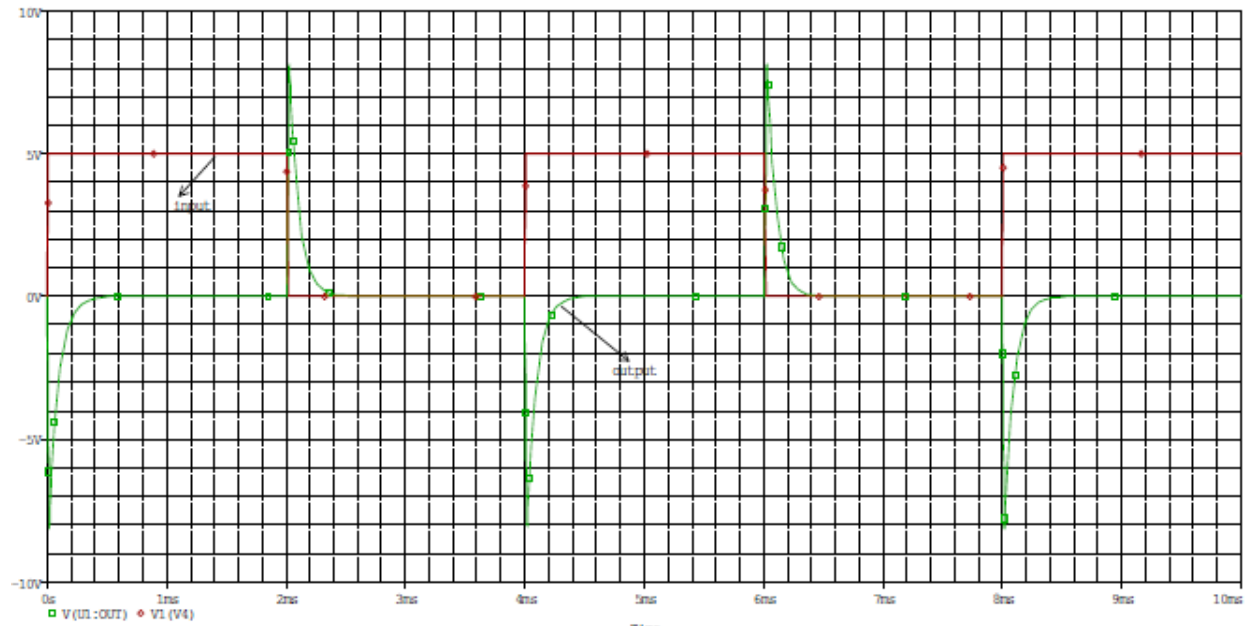


Fig. Input & Output waveforms

Result- wave shape of a differentiating ckt 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).

7) Before the circuit connection it should be check out working condition of all

the Component

Experiment no 7

Objective- Use the filter circuit in musical light system.

Theory -

Play a song from your audio source and probe the signal with your oscilloscope. Adjust the horizontal and vertical scales so you can see several seconds of the signal at once. The energy of a bass drum lies mainly in frequencies under 200Hz. To isolate the bass drum, we need to eliminate the song's high frequencies with a low pass filter. The ideal low pass filter has the magnitude response shown in Figure 2(a), where all frequencies below the cutoff are passed through untouched, and all frequencies above the cutoff are blocked completely. This magnitude response is impossible in reality, but you will implement one of the filters shown in Figure 2(b) as an approximation.

When you design your low pass filter, you will be faced with many important choices. As an engineer, you must guide your decisions by considering the cost and effectiveness of each implementation:

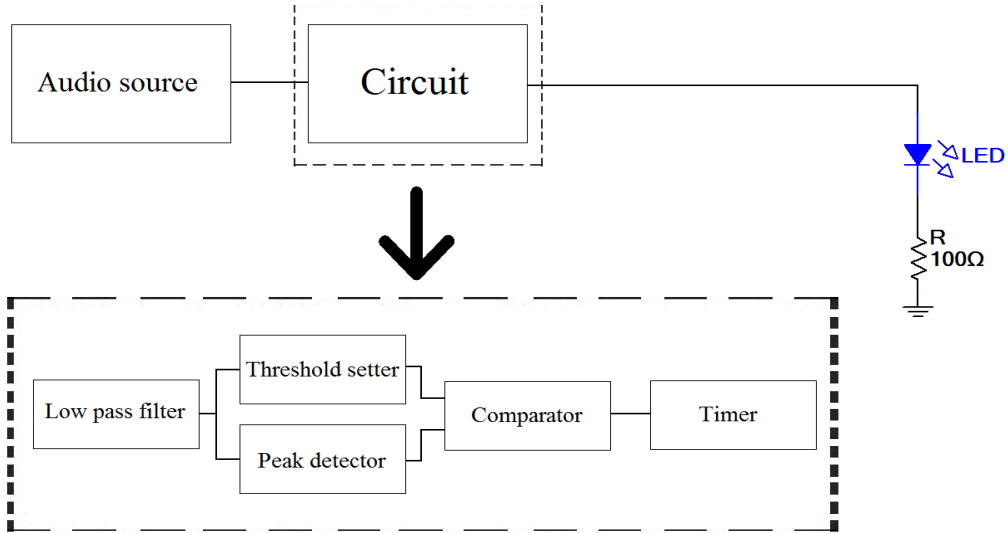
1. Effectiveness. The filter must attenuate the high frequencies so the LED blinks only in response to the bass drum, and not any other instruments.

2. Cost. The filter must be as simple as possible (but still be effective). We could use a monstrous 10th order filter, but that is far more complicated than necessary. In the real world, higher complexity translates to higher cost.

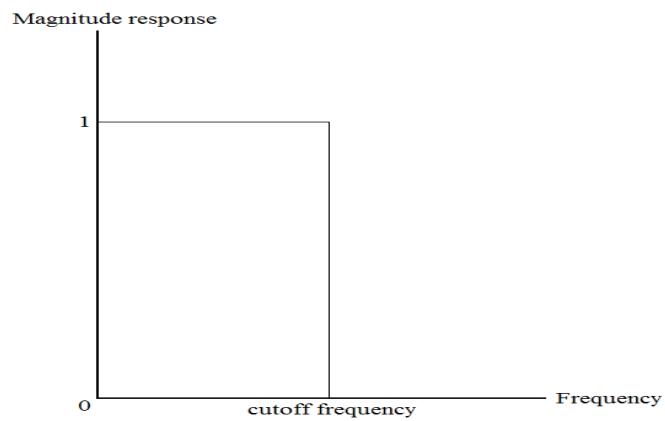
1.

Connection diagram

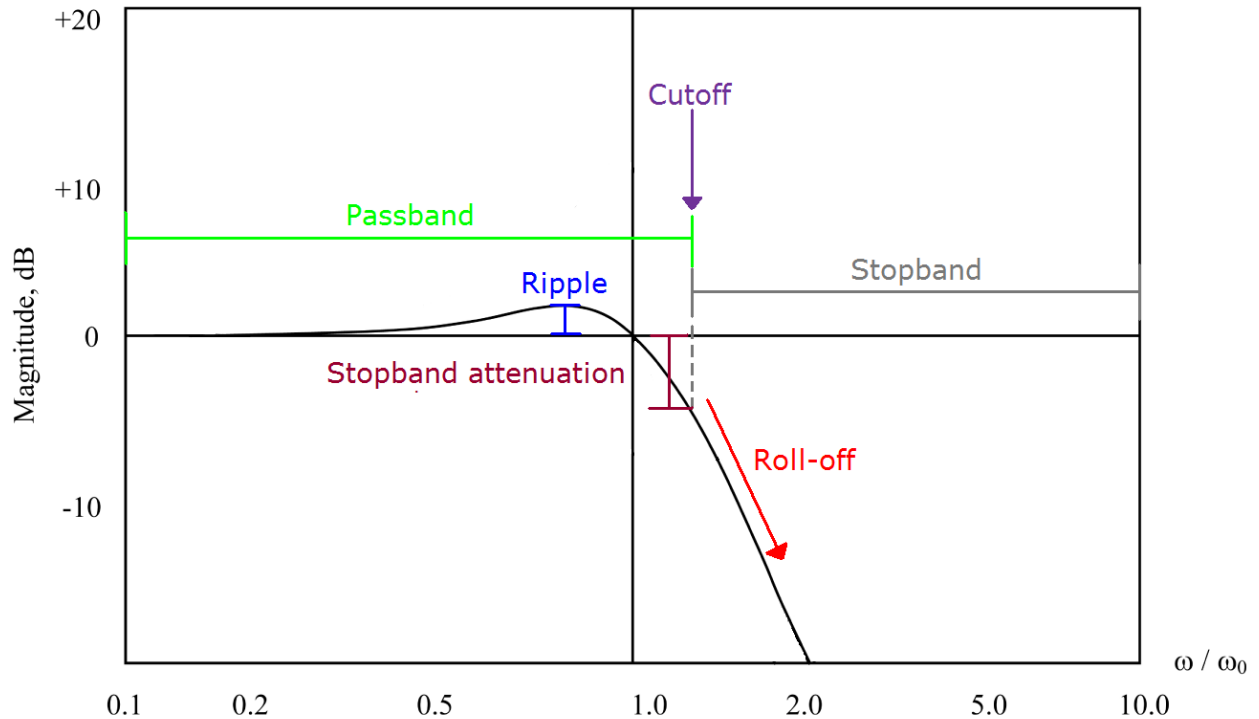
Figure



Block diagram of the circuit



Waveform



Example filter magnitude response and general terminology. The passband and stopband are the ranges of frequency that are nominally passed through and blocked by the filter. The cutoff frequency separates them. Stopband attenuation is the minimum attenuation of all stopband frequencies. Roll-off is how quickly stopband frequencies are attenuated as frequency increases. Ripple is how much the magnitude response ripples in passband or stopband.

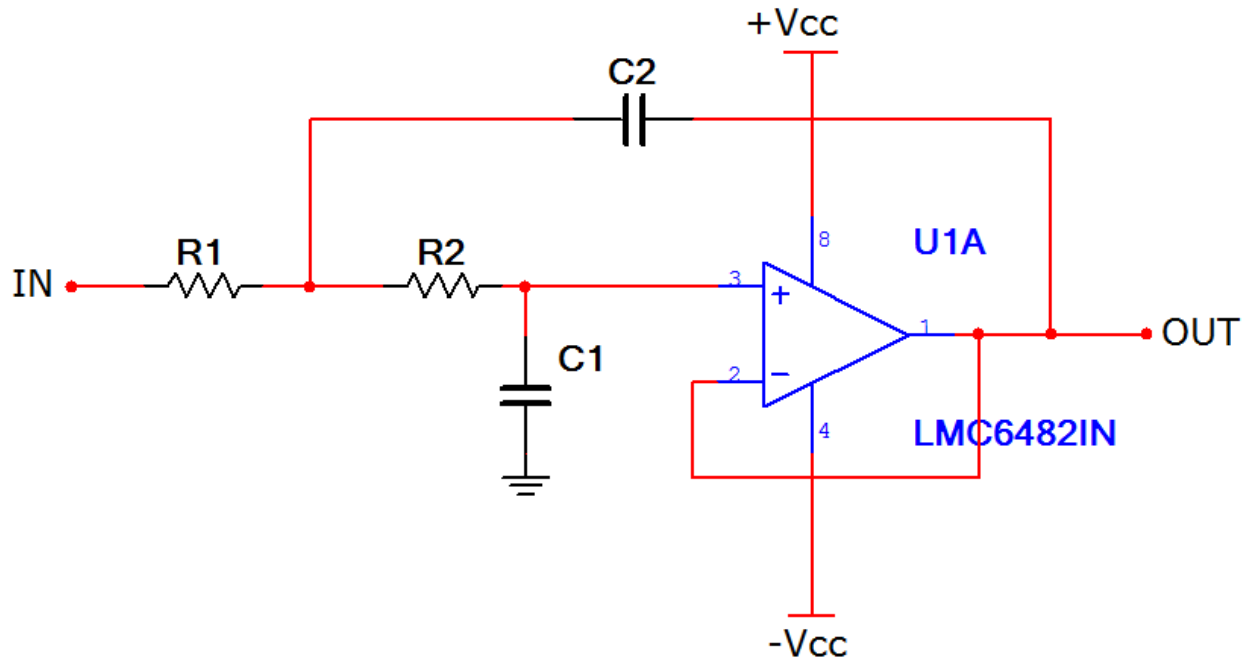
1. Bessel. The Bessel filter has the slowest roll-off of all 5 filters. Its chief advantage is its maximally flat group delay in the passband. Group delay is simply the derivative of the phase response (both are plotted vs. frequency). It is a measure of the time it takes for a signal to pass through a filter. Group delay increases with increasing filter order and decreasing cutoff frequency, which is bad. We do not want the LED to blink AFTER we hear the beat; we want the blink and the beat to coincide or be so close together we cannot perceive the delay. The Bessel's flat group delay means all frequencies in the passband are delayed the same amount, which means there is no signal distortion at the output due to different frequencies being delayed different amounts. The filter parameters stopband attenuation and ripple have no effect on this filter.

2. Butterworth. The Butterworth filter is the easiest of the 5 topologies to implement in analog circuits. Its magnitude response is characterized by a maximally flat passband, -3dB attenuation at the cutoff frequency, and a roll-off of $-20N$ dB/decade beyond cutoff, where N is the order. This roll-off is the slowest, with the exception of the Bessel filter. Stopband attenuation and ripple have no effect on the Butterworth.

3. Chebyshev/Inverse Chebyshev. These filters roll off more steeply than the Butterworth, but exhibit ripple in passband (Chebyshev) or stopband (Inverse).

The value of stopband attenuation has no effect on the Chebyshev filter; ripple has no effect on the Inverse Chebyshev.

3. Elliptic. The elliptic filter rolls off more steeply than all other filters, but exhibits ripple in both passband and stopband.



low pass filter schematic.

The timer

The final step is to set the length of each blink. Without the timer, the length of a blink is how long the peak from the peak detector signal stays above the threshold voltage. This time differs from beat to beat. We want to make this time the same for all beats. We can do this with the monostable timer circuit from Figure

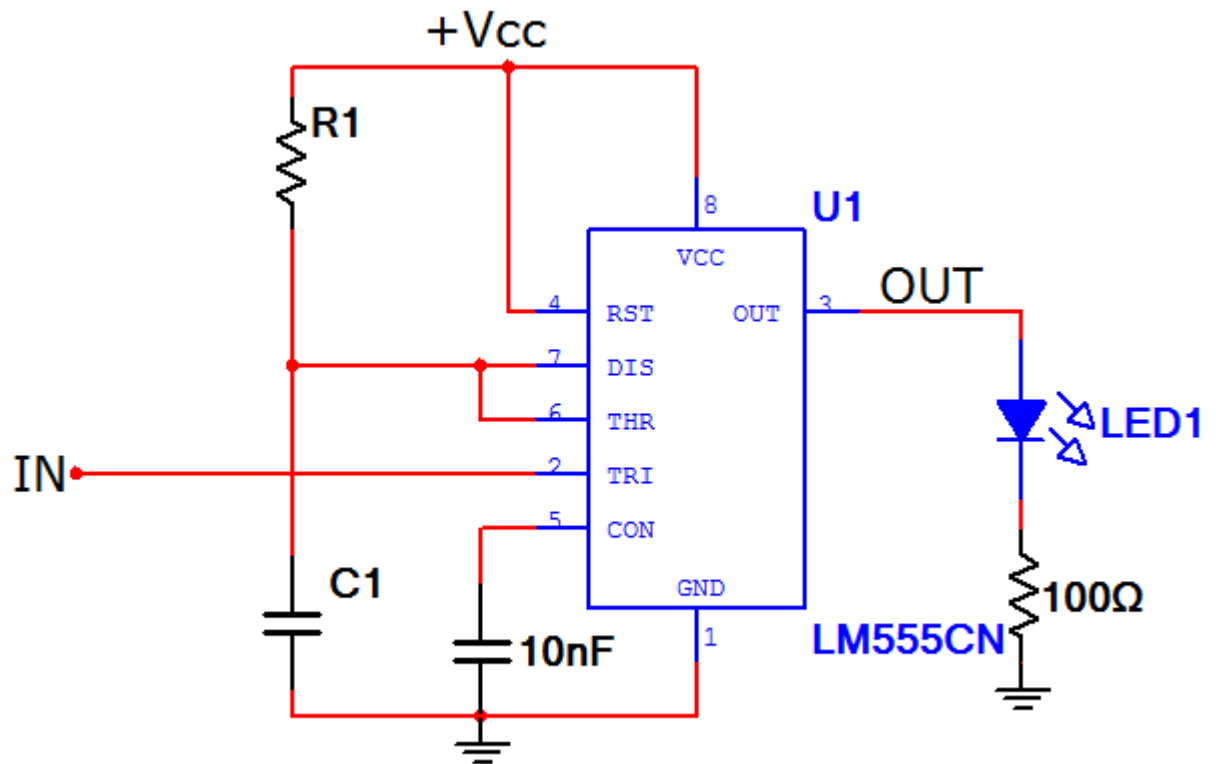


Figure 9 The monostable timer circuit.

Recall that OUT is +Vcc when $IN < V_{cc}/3$, i.e. OUT goes high when IN goes low. OUT remains high until C1, initially with zero voltage across its terminals, charges to $(2/3)V_{cc}$. The time it takes for C1 to charge to $(2/3)V_{cc}$ is $t_{pulse} = \ln(3) \cdot R1 \cdot C1$. Pick values for t_{pulse} , R1, and C1, and write them down.

The comparator signal goes high on the beat, but we need a trigger signal that goes low on the beat. How can we produce this trigger signal from the comparator output? Simply switch the two comparator inputs.

Wire the LED and the 100Ω resistor to the output of the timer and play your song.

Result - filter circuit is used in musical light system.

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- Develop a ckt for simple project based on N/W analysis.

Theory - This section deals with the resource aspects of network analysis, and shows how resources of men, machines etc could be scheduled over the project duration to be able to deal with resource and/or time constraints.

Procedure for project

Resources and Networks

The usefulness of networks is not confined only to the time and cost factors which have been discussed so far. Considerable assistance in planning and controlling the use of resources can be given to management by appropriate development of the basic network techniques.

Project resources

The resources (men of varying skills, machines of all types, the required materials, finance, and space) used in a project are subject to varying demands and loadings as the project proceeds. Management need to know what activities and what resources are critical to the project duration, and if resource limitations (eg shortage of materials, limited number of skilled personnel) might delay the project. They also wish to ensure, as far as possible, constant work rates to eliminate paying overtime at one stage of a project and having short time working at another stage.

Resource Scheduling Requirements

To be able to schedule the resource requirements for a project the following details are required.

- a. The customary activity times, descriptions and sequences as previously described.
- b. The resource requirements for each activity showing the classification of the resource and the quantity required.
- c. The resources in each classification that are available to the project. If variations in availability are likely during the project life, these must also be specified.
- d. Any management restrictions that need to be considered eg which activities may or may not be split or any limitations on labour mobility

Resources Scheduling Example Using a Gantt Chart

A simple project has the following time and resource data (for simplicity, only the one resource of labour is considered, but similar principles would apply to other types of interchangeable resources).

Result –objective is completed.

Experiment 9

Objective- Measurement of capacitance of a condenser without using R.L.C. bridge

Theory - A capacitor is a common electronic circuit component that consists of two parallel plates separated by an insulating material called a dielectric. When a battery with voltage V_o is connected across the capacitor, equal and opposite charges rapidly collect onto the plates due to the electric field created by the wires connecting the two plates¹. Once the plates are fully charged, one plate will have a net positive charge $+Q$ while the other plate will have a net negative charge $-Q$. It has been shown empirically that the charge on one of the plates, Q , is directly proportional to the applied potential difference, V_o , and the constant of proportionality is known as the capacitance², C . The amount of charge on each plate in this steady-state is then written as

$$Q = CV_o \quad (1)$$

where Q is in coulombs, V_o is in volts, and C is in farads (after 19th century British physicists Michael Faraday). That is, $1F = 1C/V$. A one-farad capacitor is physically quite large, so it is more common to see capacitors in the picofarad ($1pF = 10^{-12} F$) to microfarad ($1\mu F = 10^{-6} F$) range. Since the capacitor charges in some finite amount of time, the charge that exists on one of the plates at any time, that is before a steady-state exists, is given by

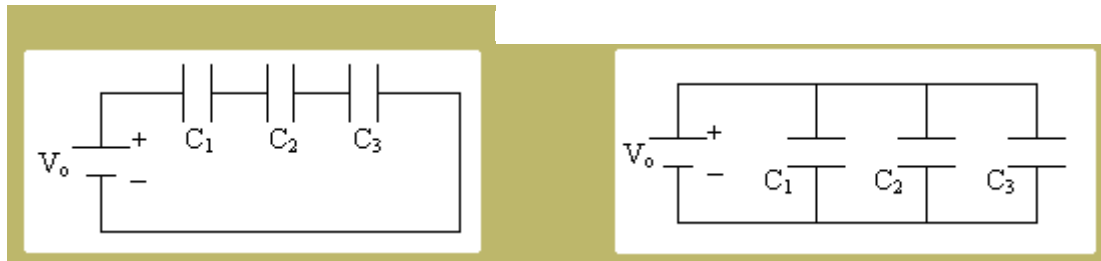
$$q(t) = CV(t) \quad (2)$$

When capacitors are connected together in series and then connected to a battery, as shown in Figure 1, the same amount of charge must build up on each of the capacitors. It can be shown that the behavior of these capacitors is as if there was one single capacitor with an effective capacitance, C_{eff} , given by the following

Capacitors in series
$$\frac{1}{C_{eff}} = \frac{1}{C_1} + \frac{1}{C_2} + \frac{1}{C_3} + \dots \quad (3)$$

It should be readily seen that the effective capacitance of capacitors in series is smaller than the capacitance of any of the contributing elements. However when capacitors are connected in parallel, as shown in Figure 2, the effective capacitance is given by the sum of each capacitance, or

Capacitors in parallel
$$C_{eff} = C_1 + C_2 + C_3 + \dots \quad (4)$$



Result - capacitance of a condenser is Measured without using R.L.C. bridge.

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).
- 8) Before the circuit connection it should be check out working condition of all the Component

Experiment no 10

Objective - Function of

- c. Low pass filter
- d. High pass filter
- e. Band pass filter

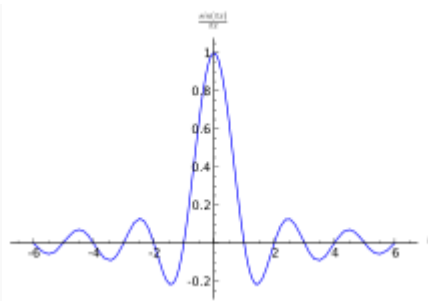
Apparatus required :- Op- Amp 741, Resistances, capacitor, Function generator.

Theory

A low-pass filter is a filter that passes low-frequency signals but attenuates (reduces the amplitude of) signals with frequencies higher than the cutoff frequency. The actual amount of attenuation for each frequency varies from filter to filter. It is sometimes called a high-cut filter, or treble cut filter when used in audio applications. A low-pass filter is the opposite of a high-pass filter, and a band-pass filter is a combination of a low-pass and a high-pass.

Low-pass filters exist in many different forms, including electronic circuits (such as a hiss filter used in audio), digital filters for smoothing sets of data, acoustic barriers, blurring of images, and so on. The moving average operation used in fields such as finance is a particular kind of low-pass filter, and can be analyzed with the same signal processing techniques as are used for other low-pass filters. Low-pass filters provide a smoother form of a signal, removing the short-term fluctuations, and leaving the longer-term trend

Ideal and real filters



The sinc function, the impulse response of an ideal low-pass filter.

An ideal low-pass filter completely eliminates all frequencies above the cutoff frequency while passing those below unchanged: its frequency response is a rectangular

function, and is a brick-wall filter. The transition region present in practical filters does not exist in an ideal filter. An ideal low-pass filter can be realized mathematically (theoretically) by multiplying a signal by the rectangular function in the frequency domain or, equivalently, convolution with its impulse response, a sinc function, in the time domain.

However, the ideal filter is impossible to realize without also having signals of infinite extent in time, and so generally needs to be approximated for real ongoing signals, because the sinc function's support region extends to all past and future times. The filter would therefore need to have infinite delay, or knowledge of the infinite future and past, in order to perform the convolution. It is effectively realizable for pre-recorded digital signals by assuming extensions of zero into the past and future, or more typically by making the signal repetitive and using Fourier analysis.

Real filters for real-time applications approximate the ideal filter by truncating and windowing the infinite impulse response to make a finite impulse response; applying that filter requires delaying the signal for a moderate period of time, allowing the computation to "see" a little bit into the future. This delay is manifested as phase shift. Greater accuracy in approximation requires a longer delay.

An ideal low-pass filter results in ringing artifacts via the Gibbs phenomenon. These can be reduced or worsened by choice of windowing function, and the design and choice of real filters involves understanding and minimizing these artifacts. For example, "simple truncation [of sinc] causes severe ringing artifacts," in signal reconstruction, and to reduce these artifacts one uses window functions "which drop off more smoothly at the edges."^[2]

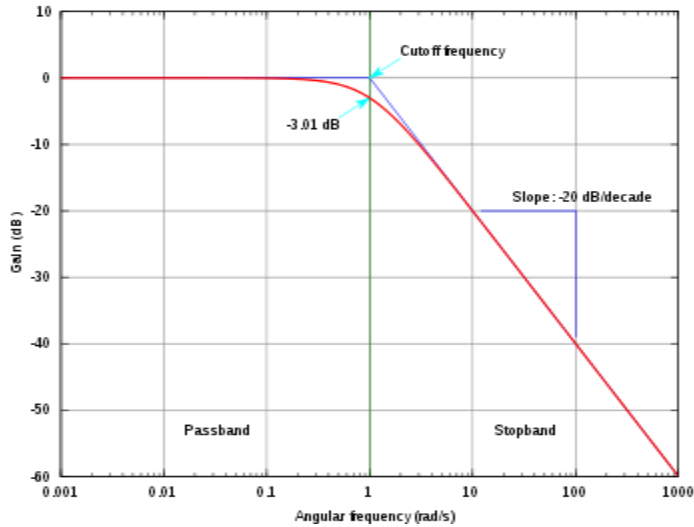
The Whittaker–Shannon interpolation formula describes how to use a perfect low-pass filter to reconstruct a continuous signal from a sampled digital signal. Real digital-to-analog converters use real filter approximations

Function of filters

There are many different types of filter circuits, with different responses to changing frequency. The frequency response of a filter is generally represented using a Bode plot, and the filter is characterized by its cutoff frequency and rate of frequency rolloff. In all cases, at the cutoff frequency, the filter attenuates the input power by half or 3 dB. So the order of the filter determines the amount of additional attenuation for frequencies higher than the cutoff frequency.

- A first-order filter, for example, will reduce the signal amplitude by half (so power reduces by a factor of 4), or 6 dB, every time the frequency doubles (goes up one octave); more precisely, the power rolloff approaches 20 dB per decade in the limit of high frequency. The magnitude Bode plot for a first-order filter looks like a horizontal line below the cutoff frequency, and a diagonal line above the cutoff frequency. There is also a "knee curve" at the boundary between the two, which smoothly transitions between the two straight line regions. If the transfer function of a first-order low-pass filter has a zero as well as a pole, the Bode plot will flatten out again, at some maximum attenuation of high frequencies; such an effect is caused for example by a little bit of the input leaking around the one-pole filter; this one-pole-one-zero filter is still a first-order low-pass. See Pole-zero plot and RC circuit.
- A second-order filter attenuates higher frequencies more steeply. The Bode plot for this type of filter resembles that of a first-order filter, except that it falls off more quickly. For example, a second-order Butterworth filter will reduce the signal amplitude to one fourth its original level every time the frequency doubles (so power decreases by 12 dB per octave, or 40 dB per decade). Other all-pole second-order filters may roll off at different rates initially depending on their Q factor, but approach the same final rate of 12 dB per octave; as with the first-order filters, zeroes in the transfer function can change the high-frequency asymptote. See RLC circuit.
- Third- and higher-order filters are defined similarly. In general, the final rate of power rolloff for an order-n all-pole filter is $6n$ dB per octave (i.e., $20n$ dB per decade).

Waveform



The gain-magnitude frequency response of a first-order (one-pole) low-pass filter. Power gain is shown in decibels (i.e., a 3 dB decline reflects an additional half-power attenuation). Angular frequency is shown on a logarithmic scale in units of radians per second.

On any Butterworth filter, if one extends the horizontal line to the right and the diagonal line to the upper-left (the asymptotes of the function), they will intersect at exactly the "cutoff frequency". The frequency response at the cutoff frequency in a first-order filter is 3 dB below the horizontal line. The various types of filters – Butterworth filter, Chebyshev filter, Bessel filter, etc. – all have different-looking "knee curves". Many second-order filters are designed to have "peaking" or resonance, causing their frequency response at the cutoff frequency to be above the horizontal line. See electronic filter for other types.

High pass filter

A high-pass filter, or HPF, is an LTI filter that passes high frequencies well but attenuates (i.e., reduces the amplitude of) frequencies lower than the filter's cutoff frequency. The actual amount of attenuation for each frequency is a design parameter of the filter. It is sometimes called a low-cut filter or bass-cut filter.^[1]

function of high pass filter

The simple first-order electronic high-pass filter shown in Figure 1 is implemented by placing an input voltage across the series combination of a capacitor and a resistor and using the voltage across the resistor as an output. The product of the resistance and capacitance ($R \times C$) is the time constant (τ); it is inversely proportional to the cutoff frequency f_c , at which the output power is half the input power. That is,

$$f_c = \frac{1}{2\pi\tau} = \frac{1}{2\pi RC},$$

where f_c is in hertz, τ is in seconds, R is in ohms, and C is in farads.

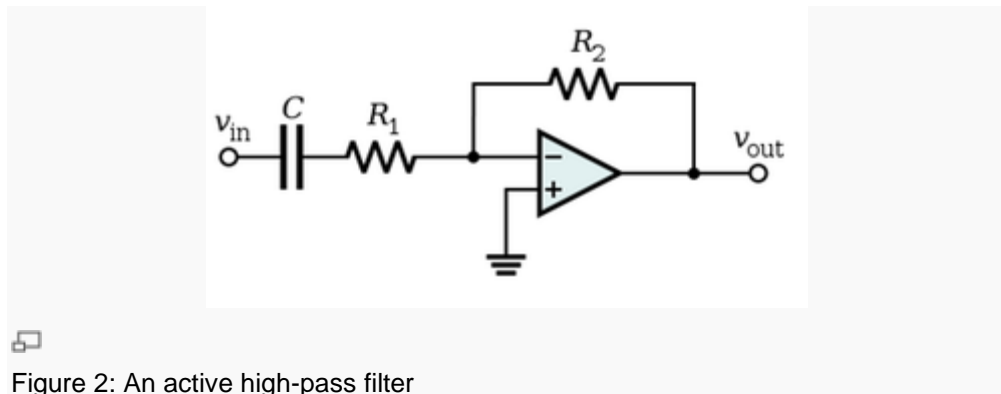


Figure 2: An active high-pass filter

Figure 2 shows an active electronic implementation of a first-order high-pass filter using an operational amplifier. In this case, the filter has a passband gain of $-R_2/R_1$ and has a corner frequency of

$$f_c = \frac{1}{2\pi\tau} = \frac{1}{2\pi R_1 C},$$

Because this filter is active, it may have non-unity passband gain. That is, high-frequency signals are inverted and amplified by R_2/R_1 .

band-pass filter

A band-pass filter is a device that passes frequencies within a certain range and rejects (attenuates) frequencies outside that range. An example of an analogue electronic band-pass filter is an RLC circuit (a resistor–inductor–capacitor circuit). These filters can also be created by combining a low-pass filter with a high-pass filter.^[1]

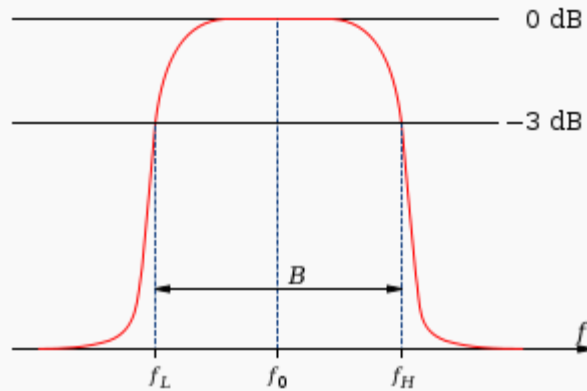
Function of Band pass

Band pass is an adjective that describes a type of filter or filtering process; it is frequently confused with passband, which refers to the actual portion of affected spectrum. The two words are both compound words that follow the English rules of formation: the primary meaning is the latter part of the compound, while the modifier is the first part. Hence, one may correctly say 'A dual bandpass filter has two passbands'. A bandpass signal is a signal containing a band of frequencies away from zero frequency, such as a signal that comes out of a bandpass filter.^[2]

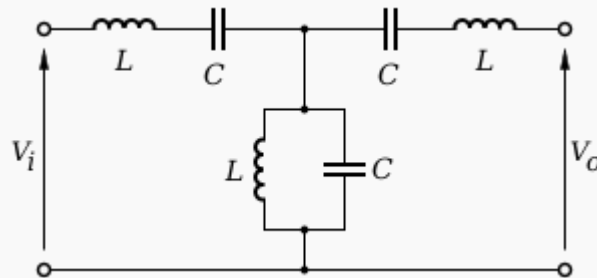
An ideal bandpass filter would have a completely flat passband (e.g. with no gain/attenuation throughout) and would completely attenuate all frequencies outside the passband. Additionally, the transition out of the passband would be instantaneous in frequency. In practice, no bandpass filter is ideal. The filter does not attenuate all frequencies outside the desired frequency range completely; in particular, there is a region just outside the intended passband where frequencies are attenuated, but not rejected. This is known as the filter roll-off, and it is usually expressed in dB of attenuation per octave or decade of frequency. Generally, the design of a filter seeks to make the roll-off as narrow as possible, thus allowing the filter to perform as close as possible to its intended design. Often, this is achieved at the expense of pass-band or stop-band ripple.

The bandwidth of the filter is simply the difference between the upper and lower cutoff frequencies. The shape factor is the ratio of bandwidths measured using two different attenuation values to determine the cutoff frequency, e.g., a shape factor of 2:1 at 30/3 dB means the bandwidth measured between frequencies at 30 dB attenuation is twice that measured between frequencies at 3 dB attenuation.

Waveform



Bandwidth measured at half-power points (gain -3 dB, $\sqrt{2}/2$, or about 0.707 relative to peak) on a diagram showing magnitude transfer function versus frequency for a band-pass filter



A medium-complexity example of a band-Pass filter

Result -Function of Low pass filter High pass filter Band pass filter are studied.

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).
- 9) Before the circuit connection it should be check out working condition of all the Component.

Experiment – 11 and 12

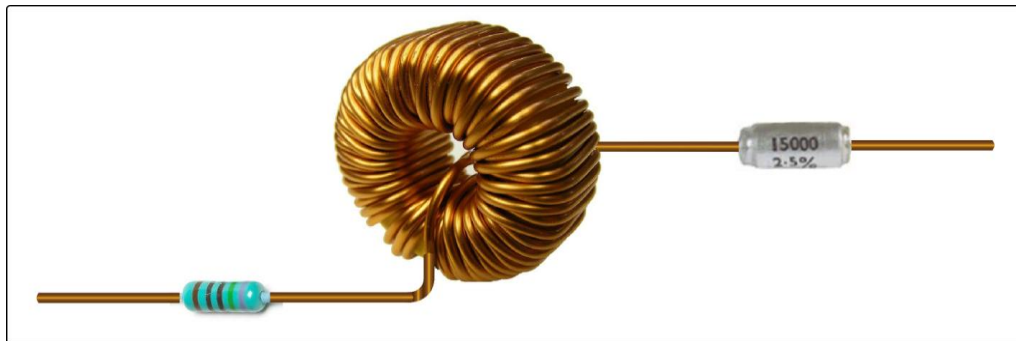
Objective - Find different electrical parameter in R.L.:R.C.:R.L.C. series circuits and draw the

2. 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, 1□ 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 resistance is 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, f_r . Frequencies are measured in units of hertz. In this article, however, angular frequency, ω , 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^[1]

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 α by

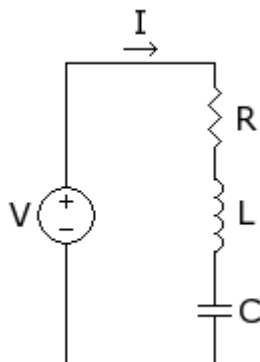
Bandwidth

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. Both band-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,

For the case where the source is an unchanging voltage, differentiating and dividing by L leads to the second order differential equation:

This can usefully be expressed in a more generally applicable form:

and are both in units of angular frequency. is called the neper frequency, or attenuation, and is a measure of how fast the transient response of the circuit will die away after the stimulus has been removed. Neper occurs in the name because the units can also be considered to be nepers per second, neper being a unit of attenuation. is the angular resonance frequency and is discussed later.^[2]

Transient response

Plot showing underdamped and overdamped responses of a series RLC circuit. The critical damping plot is the bold red curve. The plots are normalised for L=1, C=1 and

The differential equation for the circuit solves in three different ways depending on the value of ζ . These are underdamped ($\zeta < 1$), overdamped ($\zeta > 1$) and critically damped ($\zeta = 1$). The differential equation has the characteristic equation,^[6]

The roots of the equation in s are, The general solution of the differential equation is an exponential in either root or a linear superposition of both,

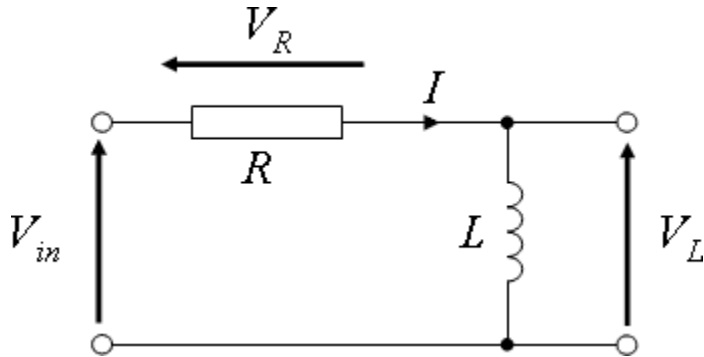
The coefficients A_1 and A_2 are determined by the boundary conditions of the specific problem being analysed. That is, they are set by the values of the currents and voltages in the circuit at the onset of the transient and the presumed value they will settle to after infinite time.

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.

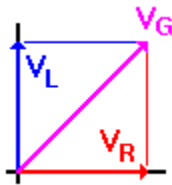
The fundamental passive linear circuit elements are the resistor (R), capacitor (C) and inductor (L). These circuit elements can be combined to form an electrical circuit in four distinct ways: the RC circuit, the RL circuit, the LC circuit and the RLC circuit with the abbreviations indicating which components are used. These circuits exhibit important

types of behaviour that are fundamental to analogue electronics. In particular, they are able 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 LC Series Circuit



The vectors for this example circuit are shown to the right. This time the composite phase angle 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 circuit parameters.

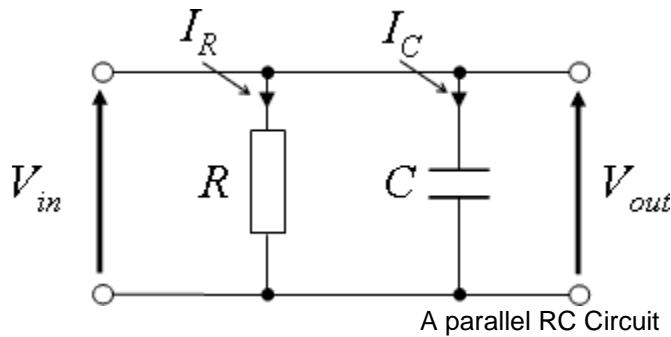
$$X_L = 2\pi fL$$

$$I = E/Z$$

$$V_L = I \times X_L$$

$$V_R = I \times R$$

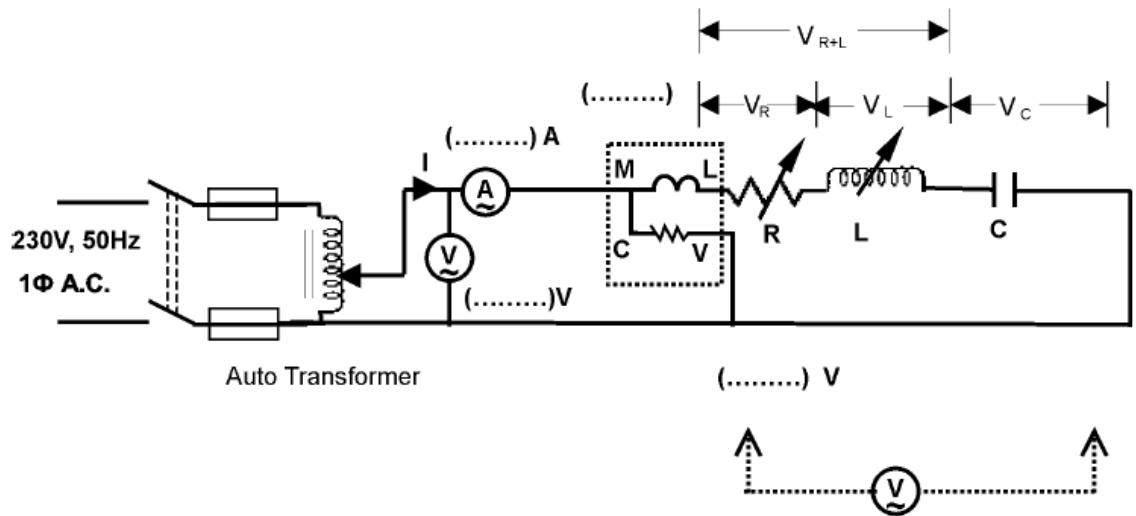
RC Circuits



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 the source 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

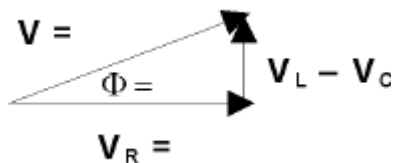
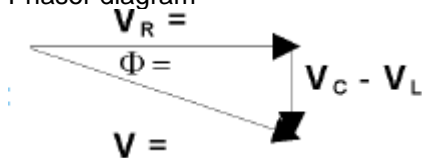
Resistance of inductor = r = Ω

Table for measured and calculated values of V, I etc

Sr. No.	V (V)	I (A)	VR (V)	VL (V)	VR+L (V)	VC (V)	Z= $\frac{V}{I}$ Ω	R= $\frac{VR}{I}$ Ω	ZL= $\frac{VR+L}{I}$ Ω	XC= $\frac{VC}{I}$ Ω	XL Ω	Φ from phasor diagram
1												
2												
3												
4												
5												
6												

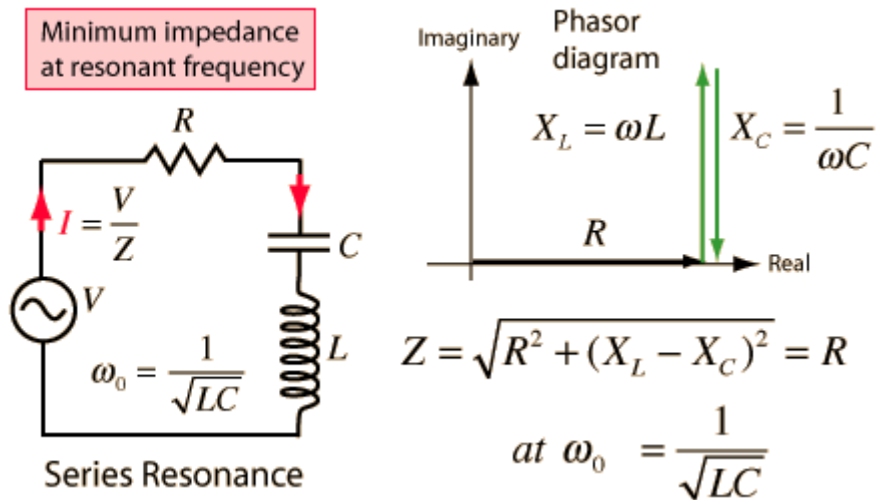
Power consumed by circuit = Watt

Phasor diagram



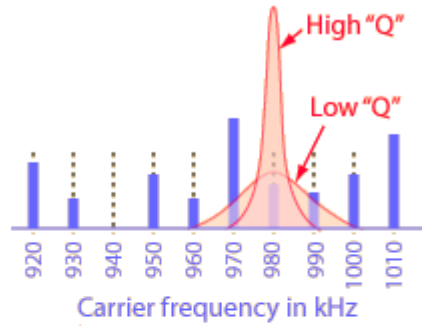
Series Resonance

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.

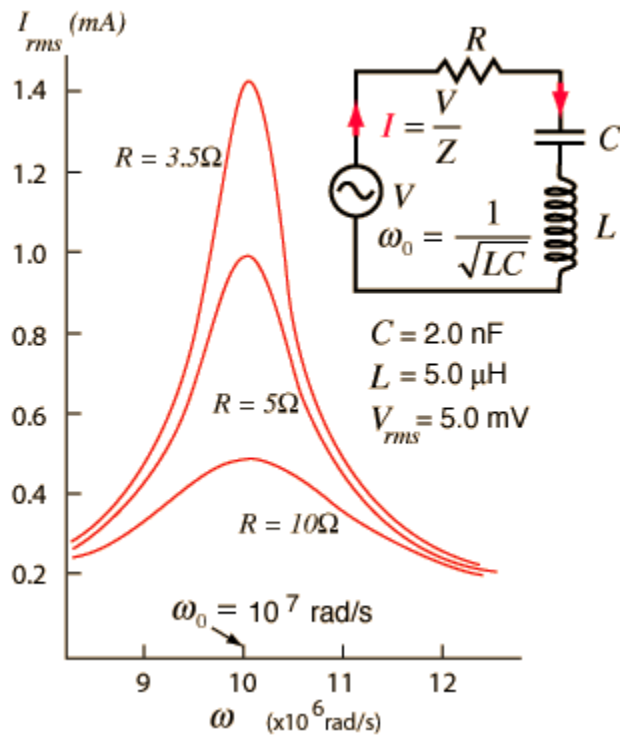


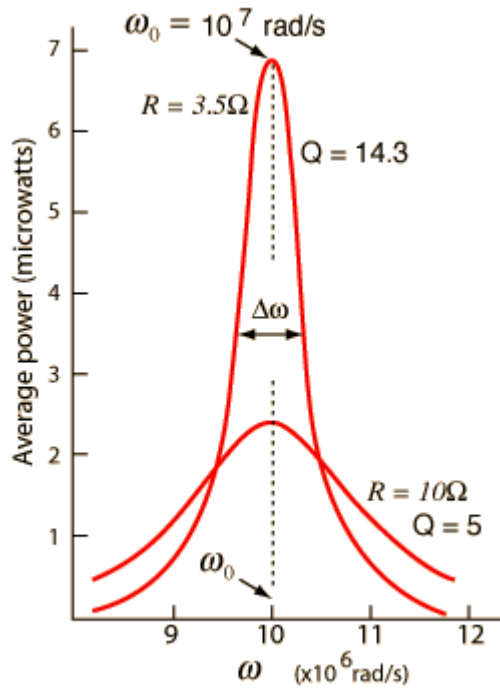
10 kHz bandwidth from
540-1600 kHz for
106 possible bands

AM Radio

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.





Using the same circuit parameters, the illustration at left shows the power dissipated in the circuit as a function of frequency. Since this power depends upon the square of the current, these resonant curves appear steeper and narrower than the resonance peaks for current above.

The quality factor Q is defined by

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

where $\Delta\omega$ is the width of the resonant power curve at half maximum.

Since that width turns out to be $\Delta\omega = R/L$, 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 the range 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 of the 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 involving them can be expressed in terms of the frequency.

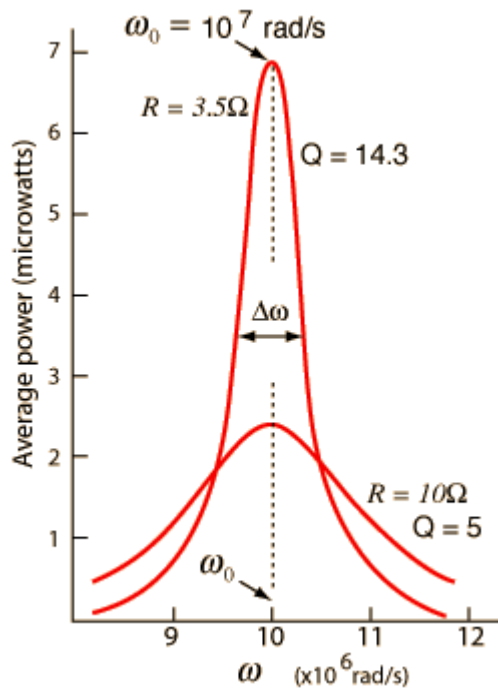
$$(X_L - X_C)^2 = \left(\omega L - \frac{1}{\omega C} \right)^2 = \frac{L^2}{\omega^2} (\omega^2 - \omega_0^2)^2$$

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 were used in the example on the Q factor of the series 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).

10) Before the circuit connection it should be check out working condition of all the Component

Experiment 13 and 14

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.

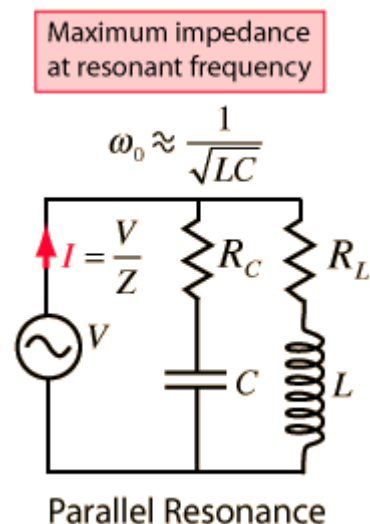
In simple reactive circuits with little or no resistance, the effects of radically altered impedance will manifest at the resonance frequency predicted by the equation given earlier. In a parallel (tank) LC circuit, this means infinite impedance at resonance. In a series LC circuit, it means zero impedance at resonance:

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

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

Different possible definitions of the resonant frequency for a parallel resonant circuit:

1. The frequency at which $\omega L = 1/\omega C$, i.e., the resonant frequency of the equivalent series RLC circuit. This is satisfactory if the resistances are small.
2. The frequency at which the parallel impedance is a maximum.
3. The frequency at which the current is in phase with the voltage, unity power factor.



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 resonant frequency:

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

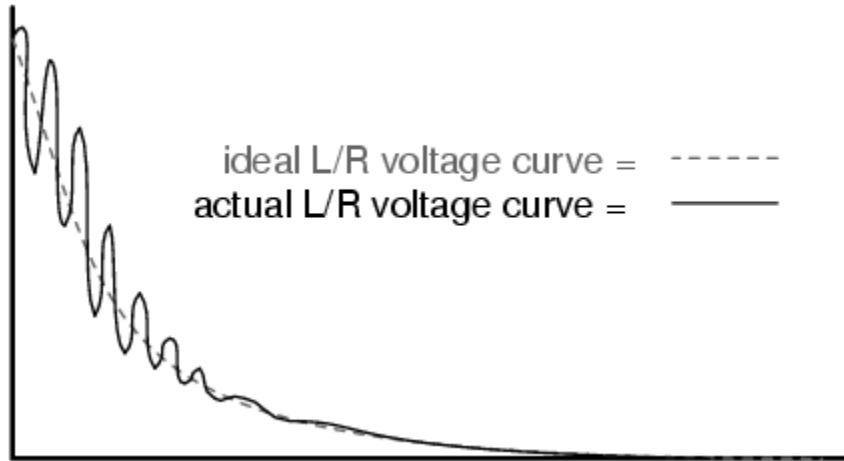
The above resonant frequency expression is obtained by taking the impedance expressions for the parallel RLC circuit and setting the expression for X_{eq} equal to zero to force the phase to zero. After about a page of algebra, the above expression emerges. Note that for small values of the resistances, this approaches the series resonant frequency.

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 antiresonance, 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 phase angle. Using values from the last example circuit:

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

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

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

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

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 (see Technical Bulletin #10). 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:

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

Result – objective is reached .

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).

11) Before the circuit connection it should be check out working condition of all

the Component