GANESH INSTITUTE OF ENGINEERING AND TECHNOLOGY (GIET),
Jagannath Prasad, Andharua, BHUBANESWAR

CIRCUIT NETWORK THEORY

(Th-02)

(As per the 2019-20 syllabus of the SCTE&VT, Bhubaneswar, Odisha)

Third Semester

Electrical Engg.

Prepared By: Er. Rashmi Ranjan Martha

CIRCUIT AND NETWORK THEORY

TH-2

CONTENT

SL NO	NAME OF THE TOPIC	EXPECTED MARKS
1	CIRCUIT ELEMENT AND ANALYSIS	10
2	NETWORK THEOREM	15
3	MAGNETIC CIRCUIT	10
4	COUPLED CIRCUIT	15
5	AC CIRCUIT AND RESONANCE	10
6	POLYPHASE CIRCUIT	10
7	FILTER	10
8	TWOPORT NETWORK	12
9	TRANSIENT	08

CIRCUIT ELEMENTS AND ANALYSIS

Chapter-1

1 . 1 Active, Passive, Unilateral & bilateral, Linear & Non linear elements

Active Elements and Passive Elements

We can classify the Network elements into either active or passive based on the ability of delivering power.

- **Active Elements:** deliver power to other elements, which are present in an electric circuit. Sometimes, they may absorb the power like passive elements. That means active elements have the capability of both delivering and absorbing power. **Examples**: Voltage sources and current sources.
- **Passive Elements** :can't deliver power (energy) to other elements, however they can absorb power. That means these elements either dissipate power in the form of heat or store energy in the form of either magnetic field or electric field. **Examples**: Resistors, Inductors, and capacitors.

Linear Elements and Non-Linear Elements

We can classify the network elements as linear or non-linear based on their characteristic to obey the property of linearity.

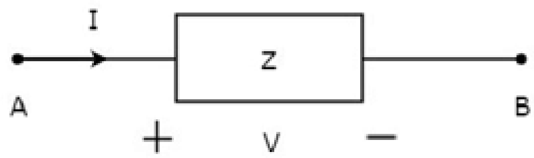
- **Linear Elements** are the elements that show a linear relationship between voltage and current. **Examples**: Resistors, Inductors, and capacitors.
- **Non-Linear Elements** are those that do not show a linear relation between voltage and current. **Examples**: Voltage sources and current sources.

Bilateral Elements and Unilateral Elements

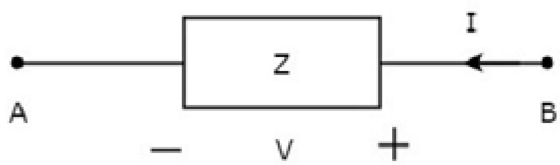
Network elements can also be classified as either bilateral or unilateral based on the direction of current flows through the network elements.

Bilateral Elements are the elements that allow the current in both directions and offer the same impedance in either direction of current flow. **Examples**: Resistors, Inductors and capacitors.

The concept of Bilateral elements is illustrated in the following figures.



In the above figure, the current (I) is flowing from terminals A to B through a passive element having impedance of Z Ω. It is the ratio of voltage (V) across that element between terminals A & B and current (I).



In the above figure, the current (I) is flowing from terminals B to A through a passive element having impedance of Z Ω. That means the current (–I) is flowing from terminals A to B. In this case too, we will get the same impedance value, since both the current and voltage having negative signs with respect to terminals A & B.

Unilateral Elements are those that allow the current in only one direction. Hence, they offer different impedances in both directions.

1 . 2 Mesh Analysis, Mesh Equations by inspection

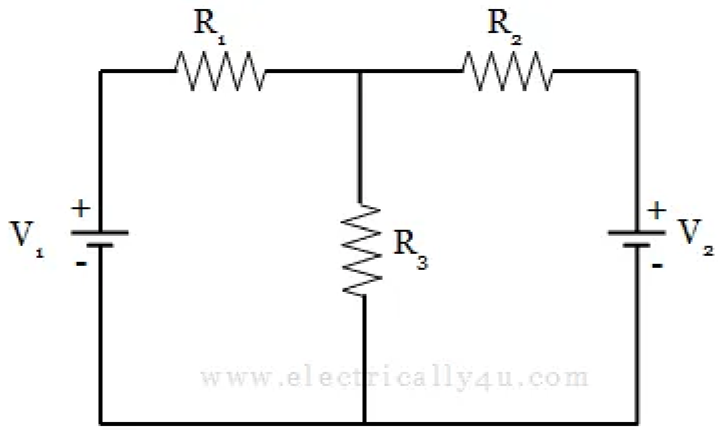
In Mesh analysis, we will consider the currents flowing through each mesh. Hence, Mesh analysis is also called as Mesh-current method.

A branch is a path that joins two nodes and it contains a circuit element. If a branch belongs to only one mesh, then the branch current will be equal to mesh current.

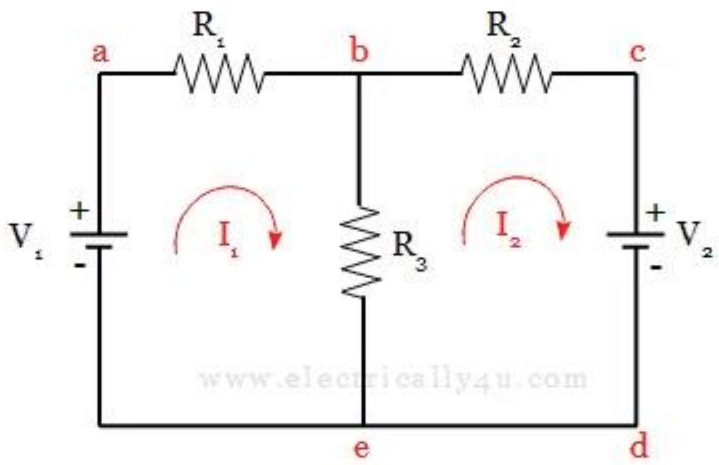
If a branch is common to two meshes, then the branch current will be equal to the sum (or difference) of two mesh currents, when they are in same (or opposite) direction.

Procedure of Mesh Analysis

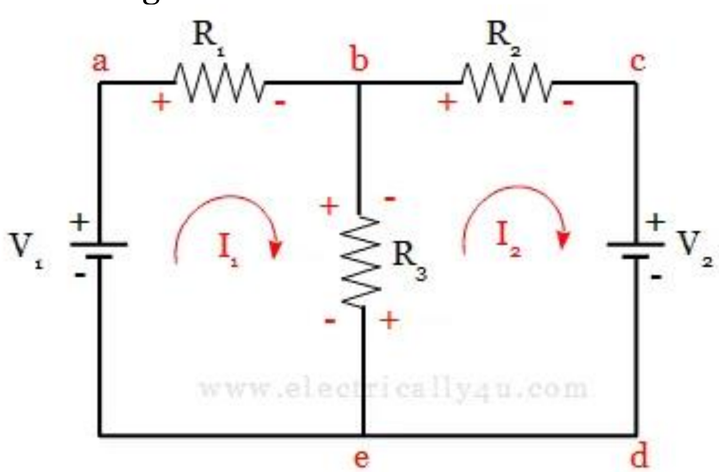
Consider a simple circuit as shown in the below figure.



1. Make sure that the circuit considered for analysis has only voltage sources. If there is any current source in the circuit, use the source transformation method and convert it into a voltage source.
2. Label the nodes with either numbers or alphabets.
3. Assign the mesh currents in each loop, such that all the current directions are in a clockwise direction.



4. Along the assumed direction of the current, mark the polarities of a voltage drop across each element. While doing, assign the correct polarity of the voltage source.



5. For each loop, write the current equation by applying KVL to each loop.
6. If two currents are flowing in the same branch, it is called a shared branch. In this circuit, mesh currents I_1 and I_2 are flowing through R_3(Branch '*be*').
7. Now, just look at the circuit. For loop1[*abea*], the current direction is assumed to be from '*b to e*', the current will be $I_1 - I_2$. But for loop2[*bcdeb*], the current direction is assumed to be from '*e to b*', so the current will be $I_2 - I_1$.

For the above circuit, apply KVL to loop1 and loop2 to get two equations

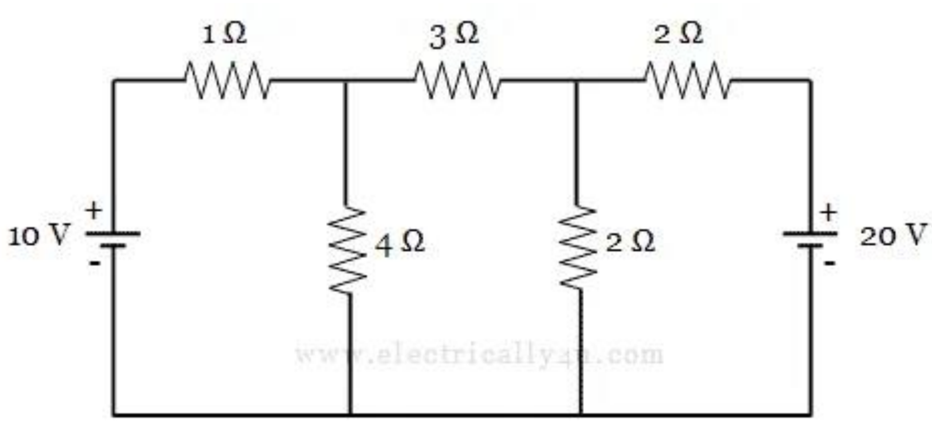
$$I_1 R_1 + (I_1 - I_2) R_3 = V_1$$

$$(I_2 - I_1) R_3 + I_2 R_2 = -V_2$$

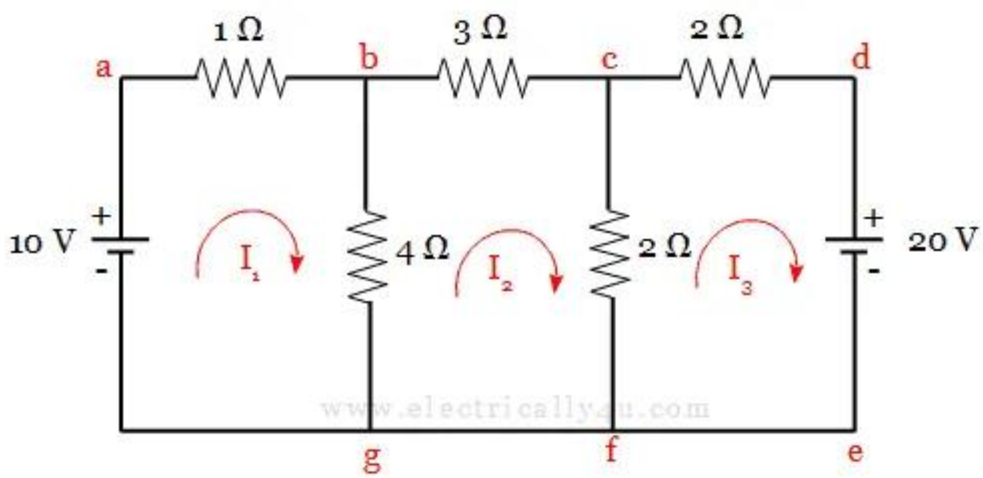
8. The above two equations are solved by using Cramer's rule, to obtain the solutions.
9. If the obtained solution is negative, then the actual direction of the current is opposite to that of the assumed direction.

Example

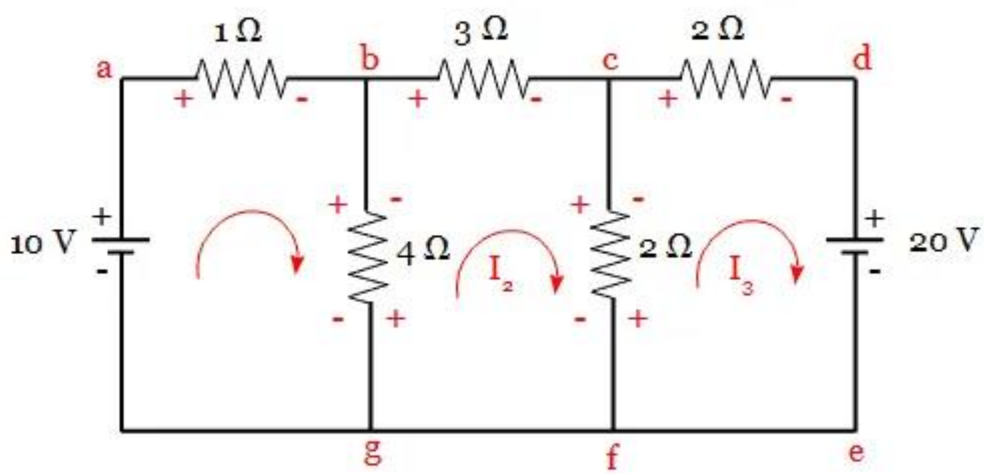
Determine the mesh currents in the circuit shown below.



The given circuit is redrawn by assigning the current and its direction in each loop and the nodes are labelled.



Along the direction of the current, the polarities of voltage drop are marked for each element. The circuit is redrawn with polarities as drawn below.



Now, from the above circuit diagram, we can observe that, there are three loops(*Loop1: abga, Loop2: bcfgb, Loop3: cdefc*). Each loop carries a current I_1, I_2 and I_3 respectively.

Apply KVL to loop1[abga], we get,

$$1I_1 + 4(I_1 - I_2) = 10$$

$$5I_1 - 4I_2 = 10 - -- > (1)$$

Apply KVL to loop2[bcfgb],

$$4(I_2 - I_1) + 3I_2 + 2(I_2 - I_3) = 0$$

$$-4I_1 + 9I_2 - 2I_3 = 0 - -- > (2)$$

Apply KVL to loop3[cdefc],

$$2(I_3 - I_2) + 2I_3 = -20$$

$$-2I_2 + 4I_3 = -20 - -- > (3)$$

The above three equations are written in matrix form, as shown below,

$$\begin{bmatrix} 5 & -4 & 0 \\ -4 & 9 & -2 \\ 0 & -2 & 4 \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \\ I_3 \end{bmatrix} = \begin{bmatrix} 10 \\ 0 \\ -20 \end{bmatrix}$$

Applying Cramer's rule, we get

$$\Delta = \begin{vmatrix} 5 & -4 & 0 \\ -4 & 9 & -2 \\ 0 & -2 & 4 \end{vmatrix} = 5(36-4) + 4(-16+0) + 0(8-0) = 160 - 64 + 0 = 96$$

$$\Delta_1 = \begin{vmatrix} 10 & -4 & 0 \\ 0 & 9 & -2 \\ -20 & -2 & 4 \end{vmatrix} = 10(36-4) + 4(0-40) + 0(0+180) = 320 - 160 + 0 = 160$$

$$\Delta_2 = \begin{vmatrix} 5 & 10 & 0 \\ -4 & 0 & -2 \\ 0 & -20 & 4 \end{vmatrix} = 5(0-40) - 10(-16-0) + 0(80-0) = -200 + 160 + 0 = -40$$

$$\Delta_3 = \begin{vmatrix} 5 & -4 & 10 \\ -4 & 9 & 0 \\ 0 & -2 & -20 \end{vmatrix} = 5(-180+0)+4(80-0)+10(8-0) = -900+320+80 = -500$$

Now, the value of current in each loop is determined

$$I_1 = \frac{\Delta_1}{\Delta} = \frac{160}{96} = 1.66A$$

$$I_2 = \frac{\Delta_2}{\Delta} = \frac{-40}{96} = -0.416A$$

$$I_3 = \frac{\Delta_3}{\Delta} = \frac{-500}{96} = -5.21A$$

The loop currents I_2 and I_3 have negative values, which means the actual current direction is opposite to the assumed direction.

Current through the branch 'bg' is $I_1 - I_2 = 1.664 + 0.416 = 2.08\ A$
Current through the branch 'cf' is $I_2 - I_3 = -0.416 + 5.21 = 4.794\ A$

1.3.Supermesh analysis

A supermesh forms when two meshes have a common current source (dependent or independent). Consider a circuit as shown below in Figure 1 in which the current source branch is common between meshes 1 and 2 so remove the current source branch and supermesh forms as shown in Figure 2.

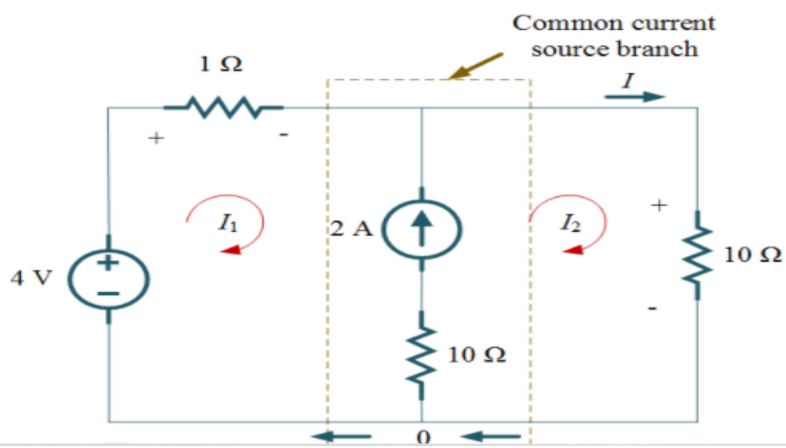


fig-1

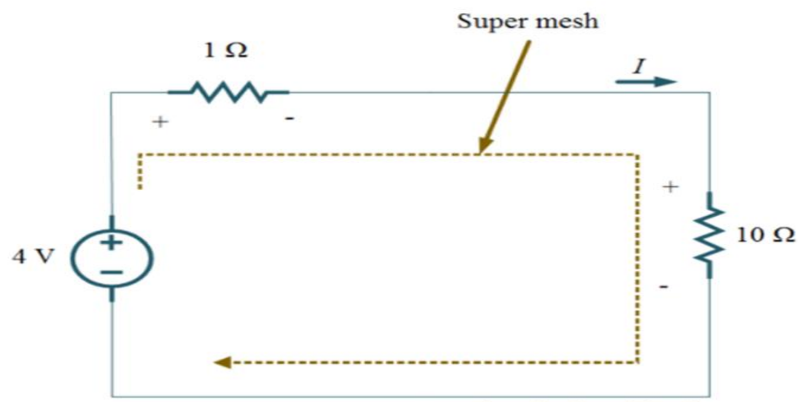


Figure 2

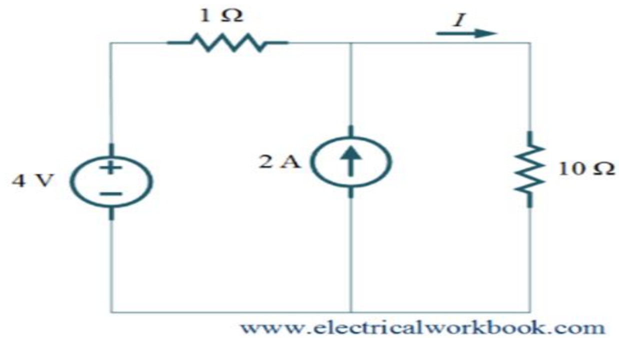
Procedure (steps) for applying mesh analysis:

Procedure (steps) for applying mesh analysis:

1. Identify the total number of meshes.
2. Assign the mesh currents and check for supermesh in the circuit.
3. If supermesh found, develop the KVL equation for it.
4. Solve the equations to find the mesh currents.

Example

Example 1. For the given network, find current I using Mesh analysis.



Solution:

As shown above, Figure is given in example 1, 2 A current source is connected between meshes 1 and 2 so this problem is based on supermesh.

Step 1: – The total number of meshes is 2.

Step 2: – Let us assign mesh currents I_1 and I_2 for meshes 1 and 2 respectively as shown in Figure 1. As shown in Figure 1, 2 A current source should be removed from the circuit because 2 A current source is connected between meshes 1 and 2.

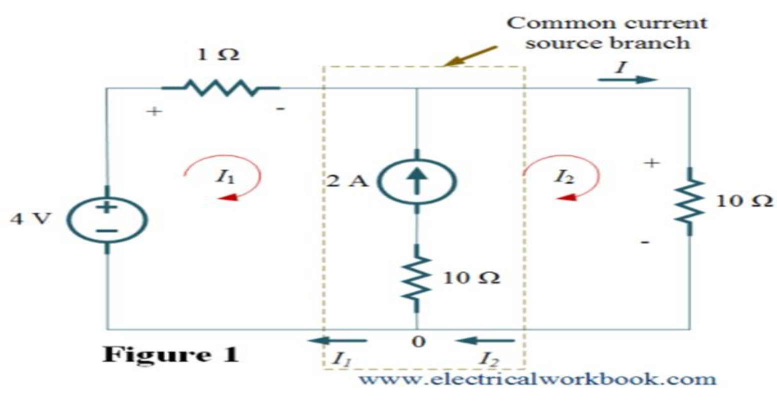


Figure 1

Step 3: – The reduced circuit having supermesh shown in Figure 2.

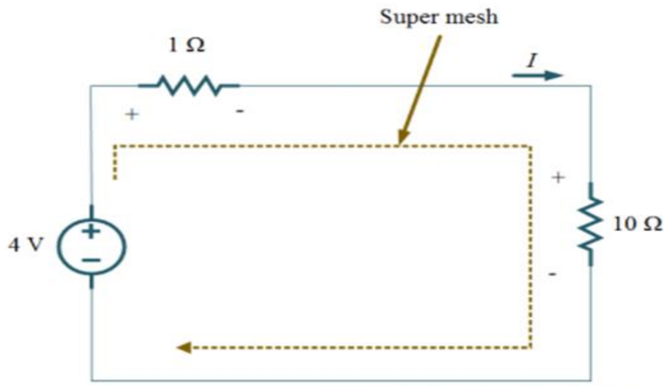


Figure 2

Apply KVL to supermesh

$$-4 + I_1 + I_2 + 2 = 0$$

$$I_1 + I_2 = 2 \qquad ....(1)$$

Apply KCL to node 0,

$$I_2 - I_1 = 2$$

$$I_2 = I_1 + 2 \qquad ....(2)$$

Put equation (2) in equation (1), we get

$$I_1 + I_1 + 2 = 2$$

$$2I_1 + 2 = 2$$

$$I_1 = 0 \text{ A} \qquad ....(3)$$

From Equation (1) ,

$$I_1 + I_2 = 2$$

Put equation (3) in equation (1), we get

$$0 + I_2 = 2$$

$$I_2 = 2$$

also

$$I = I_2$$

Therefore,

$$I = 2 \text{ A.}$$

1 . 4 Nodal Analysis, Nodal Equations by inspection

Nodal analysis is a method that provides a general procedure for analyzing circuits using node voltages as the circuit variables. Nodal Analysis is also called the Node-Voltage Method.

Some Features of Nodal Analysis are as

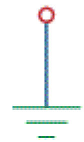
- Nodal Analysis is based on the application of the Kirchhoff's Current Law (KCL).
- Having 'n' nodes there will be 'n-1' simultaneous equations to solve.
- Solving 'n-1' equations all the nodes voltages can be obtained.
- The number of non reference nodes is equal to the number of Nodal equations that can be obtained.

Types of Nodes in Nodal Analysis

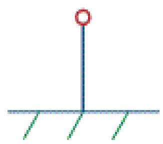
- Non Reference Node – It is a node which has a definite Node Voltage. e.g. Here Node 1 and Node 2 are the Non Reference nodes
- Reference Node – It is a node which acts a reference point to all the other node. It is also called the Datum Node.

Types of Reference Nodes

1. Chassis Ground – This type of reference node acts a common node for more than one circuits.



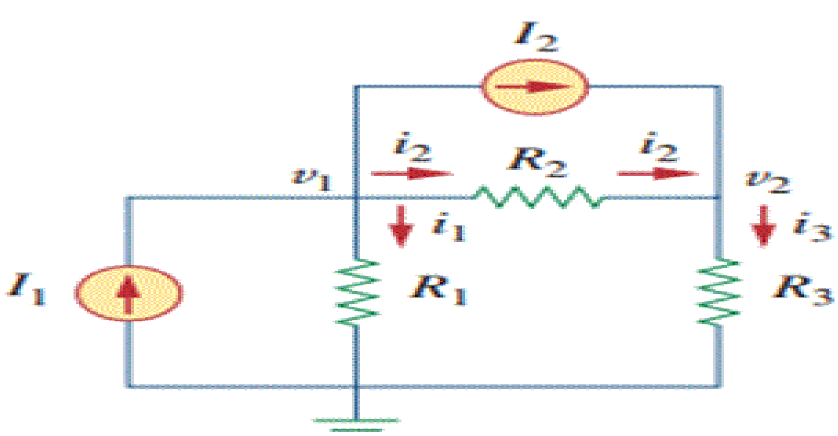
2. Earth Ground – When earth potential is used as a reference in any circuit then this type of reference node is called Earth Ground.



Solving of Circuit Using Nodal Analysis

Basic Steps Used in Nodal Analysis

1. Select a node as the reference node. Assign voltages V_1, V_2… V_{n-1} to the remaining nodes. The voltages are referenced with respect to the reference node.
2. Apply KCL to each of the non reference nodes.
3. Use Ohm's law to express the branch currents in terms of node voltages.



Node Always assumes that current flows from a higher potential to a lower potential in resistor. Hence, current is expressed as follows

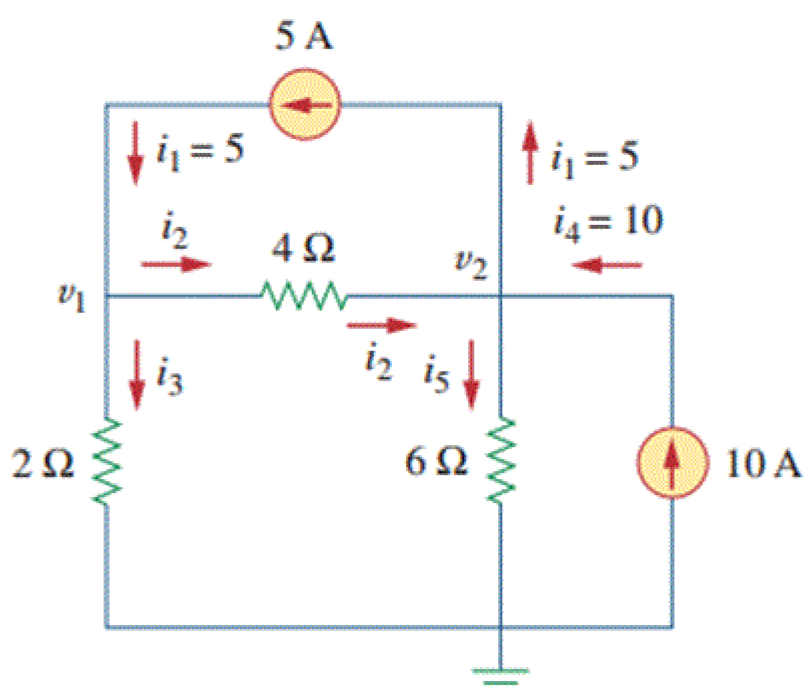
$$I = \frac{V_{high} - V_{low}}{R}$$

IV. After the application of Ohm's Law get the 'n-1' node equations in terms of node voltages and resistances.

V. Solve 'n-1' node equations for the values of node voltages and get the required node Voltages as result.

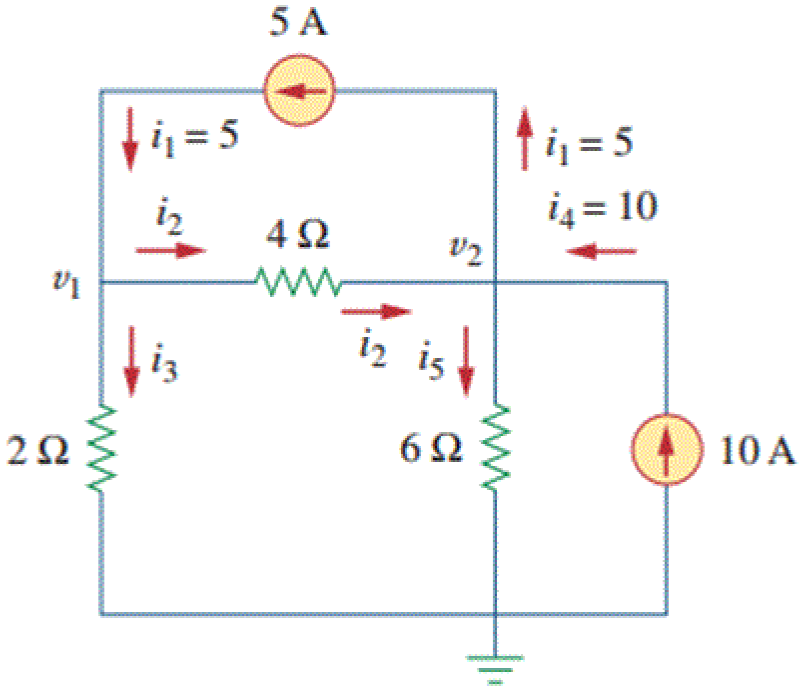
Nodal Analysis with Current Sources

Nodal analysis with current sources is very easy and it is discussed with a example below.
Example: Calculate Node Voltages in following circuit



In the following circuit we have 3 nodes from which one is reference node and other two are non reference nodes – Node 1 and Node 2.

Step I. Assign the nodes voltages as v_1 and $_2$ and also mark the directions of branch currents with respect to the reference nodes



Step II. Apply KCL to Nodes 1 and 2

KCL at Node 1

$$i_1 = i_2 + i_3 \cdots\cdots(1)$$

KCL at Node 2

$$i_2 + i_4 = i_1 + i_5 \cdots\cdots(2)$$

Step III. Apply Ohm's Law to KCL equations

- Ohm's law to KCL equation at Node 1

$$i_1 = i_2 + i_3 \Rightarrow 5 = \frac{v_1 - v_2}{4} + \frac{v_1 - 0}{2}$$

Simplifying the above equation we get,

$$3v_1 - v_2 = 20 \cdots\cdots(3)$$

- Now, Ohm's Law to KCL equation at Node 2

$$i_2 + i_4 = i_1 + i_5 \Rightarrow \frac{v_1 - v_2}{4} + 10 = 5 + \frac{v_2 - 0}{6}$$

Simplifying the above equation we get

$$-3v_1 + 5v_2 = 60 \cdots\cdots(4)$$

Step IV. Now solve the equations 3 and 4 to get the values of v_1 and v_2 as,

Using elimination method

$$3v_1 - v_2 = 20$$
$$-3v_1 + 5v_2 = 60$$
$$\Rightarrow 4v_2 = 80 \Rightarrow v_2 = 20\ Volts$$

And substituting value v_2 = 20 Volts in equation (3) we get-

$$3v_1 - 20 = 20 \Rightarrow v_1 = \frac{40}{3} = 13.333\ Volts$$

Hence node voltages are as v_1 = 13.33 Volts and v_2 = 20 Volts.

1 . 5 Super node Analysis.

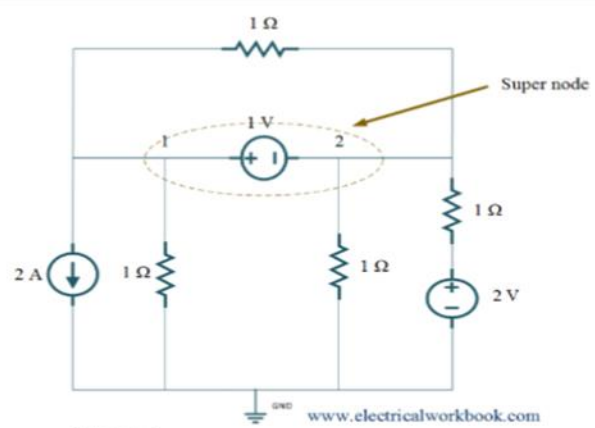


Figure 1

The two nonreference nodes form supernode if the voltage source (dependent or independent) is connected between two nonreference nodes. As shown below in Figure 1, 1 V voltage source is connected between nodes 1 and 2, so node 1 and node 2 forms supernode.

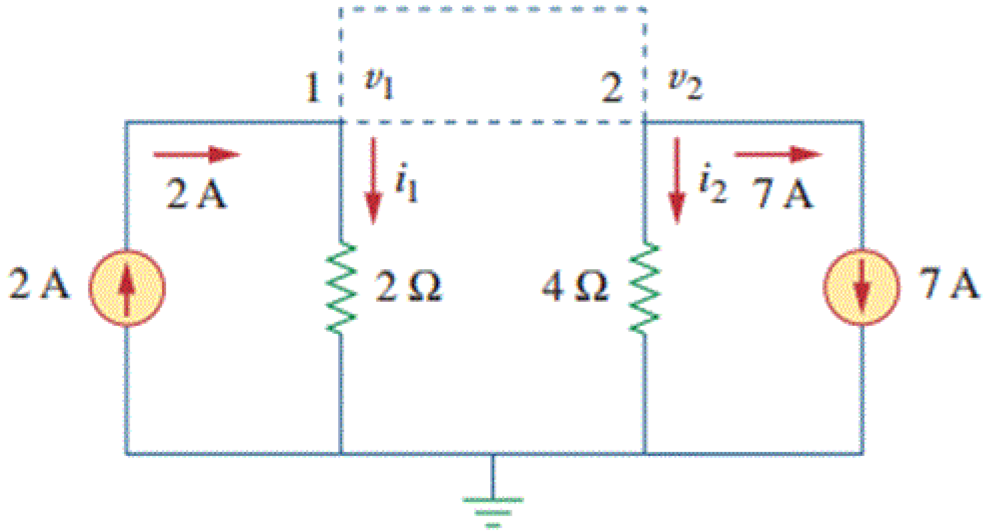
Procedure (steps) for applying Nodal Analysis: –

1. Identify the total number of nodes.
2. One node selected as reference node and it is assigned to have ground (zero) potential and the remaining nodes called as nonreference node and we assign voltage designations to nonreference nodes. And at last check for supernode.
3. Develop the KCL equations for each nonreference node.
4. Solve the equations to find the unknown node voltages.

Note:- Apply both KCL and KVL to determine the node voltages.

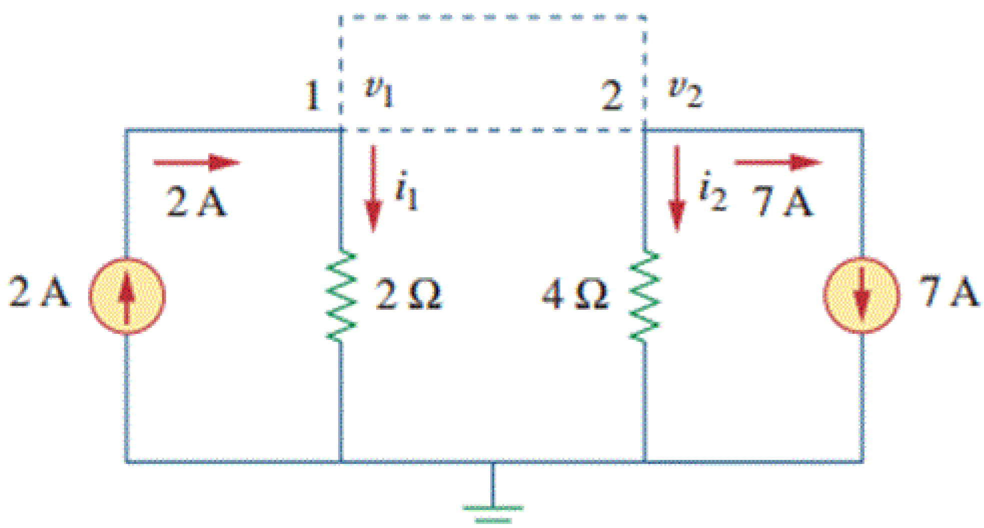
How Solve Any Circuit Containing Supernode

Let's take a example to understand how to **solve circuit containing Supernode** and applying KCL to the supernode as shown in figure gives,



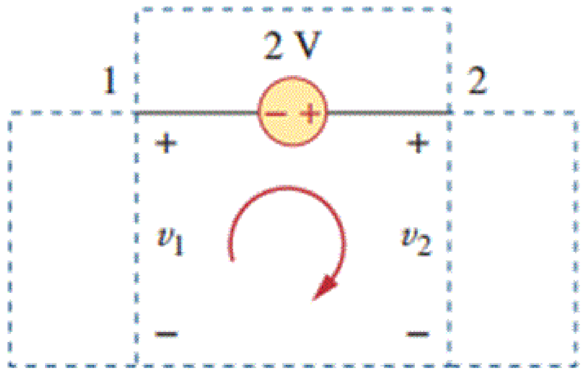
Here 2V voltage source is connected between Node-1 and Node-2 and it forms a Supernode with a 10Ω resistor in parallel.

Note – Any element connected in parallel with the voltage source forming Super node doesn't make any difference because $v_2 - v_1$ = 2V always whatever may be the value of resistor. Thus 10 Ω can be removed and circuit is redrawn and applying KCL to the supernode as shown in figure gives,



Expressing and in terms of the node voltages.

$$2 = \frac{v_1 - 0}{2} + \frac{v_2 - 0}{4} + 7 \Rightarrow 8 = 2v_1 + v_2 + 28 \Rightarrow v_2 = -2v_1 - 20 \cdots\cdots(5)$$



$$v_1 + 2 - v_2 = 0 \Rightarrow v_2 = v_1 + 2 \cdots\cdots(6)$$

From Equation 5 and 6 we can write as

$$v_2 = v_1 + 2 = -2v_1 - 20 \Rightarrow 3v_1 = -22$$
$$\Rightarrow v_1 = -7.333\ V \ \&\ v_2 = v_1 + 2 = -7.333 + 2 = -5.333\ V$$

Hence, $v_1 = -7.333V$ and $v_2 = -5.333V$ which is required answer.

1 . 6 Source Transformation Technique

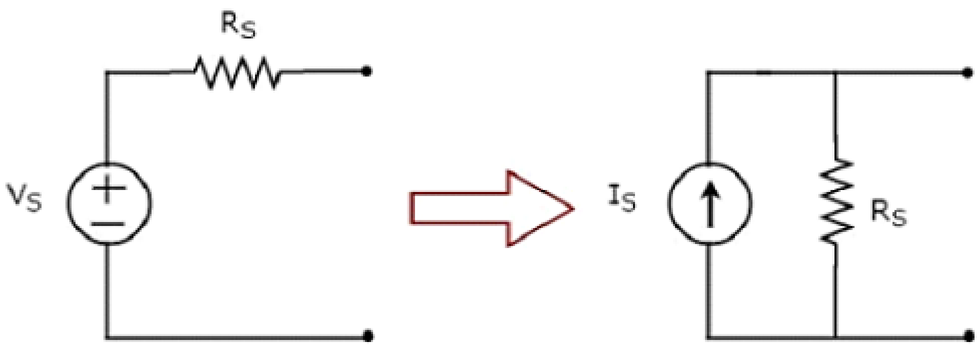
We know that there are two practical sources, namely, voltage source and current source. We can transform (convert) one source into the other based on the requirement, while solving network problems.

The technique of transforming one source into the other is called as source transformation technique. Following are the two possible source transformations −

- Practical voltage source into a practical current source
- Practical current source into a practical voltage source

Practical voltage source into a practical current source

The transformation of practical voltage source into a practical current source is shown in the following figure



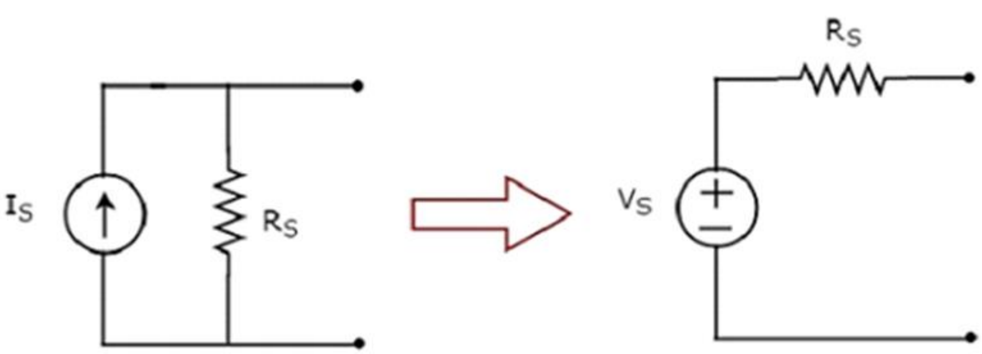
Practical voltage source consists of a voltage source (V_S) in series with a resistor (R_S). This can be converted into a practical current source as shown in the figure. It consists of a current source (I_S) in parallel with a resistor (R_S).

The value of IS will be equal to the ratio of V_S and R_S. Mathematically, it can be represented as

IS=VSRS

Practical current source into a practical voltage source

The transformation of practical current source into a practical voltage source is shown in the following figure.



Practical current source consists of a current source (I_S) in parallel with a resistor (R_S). This can be converted into a practical voltage source as shown in the figure. It consists of a voltage source (V_S) in series with a resistor (R_S).

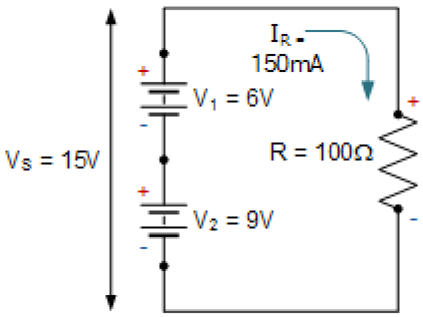
The value of V_S will be equal to the product of I_S and R_S. Mathematically, it can be represented as

VS=ISRS

1 . 7 Solve numerical problems (With Independent Sources Only)

Example No1

Two series aiding ideal voltage sources of 6 volts and 9 volts respectively are connected together to supply a load resistance of 100 Ohms. Calculate: the source voltage, V_S, the load current through the resistor, I_R and the total power, P dissipated by the resistor. Draw the circuit.



$$V_S = V_1 + V_2 = 6 + 9 = 15V$$

$$I_R = \frac{V_S}{R} = \frac{15V}{100\Omega} = 150mA$$

$$P_R = I^2R = 0.15^2 x\ 100 = 2.25W$$

Thus, V_S = 15V, I_R = 150mA or 0.15A, and P_R = 2.25W.

Voltage Source

Example No2

A battery supply consists of an ideal voltage source in series with an internal resistor. The voltage and current measured at the terminals of the battery were found to be V_{OUT1} = 130V at 10A, and V_{OUT2} = 100V at 25A. Calculate the

voltage rating of the ideal voltage source and the value of its internal resistance. Draw the I-V characteristics.

Firstly lets define in simple "**simultaneous equation form**", the two voltage and current outputs of the battery supply given as: V_{OUT1} and V_{OUT2}.

$$V_{OUT} = V_S - iR_S$$

$$V_{OUT1} = 130 = V_S + 10R_S$$
$$V_{OUT2} = 100 = V_S + 25R_S$$

As with have the voltages and currents in a simultaneous equation form, to find V_S we will first multiply V_{OUT1} by five, (5) and V_{OUT2} by two, (2) as shown to make the value of the two currents, (i) the same for both equations.

$$V_{OUT1} = 130 = V_S + 10R_S \;........\times 5$$
$$V_{OUT2} = 100 = V_S + 25R_S \;........\times 2$$

$$V_{OUT1} = 650 = 5V_S + 50R_S$$
$$V_{OUT2} = 200 = 2V_S + 50R_S$$

Having made the co-efficients for R_S the same by multiplying through with the previous constants, we now multiply the second equation V_{OUT2} by minus one, (-1) to allow for the subtraction of the two equations so that we can solve for V_S as shown.

$$V_{OUT1} = 650 = 5V_S + 50R_S$$
$$V_{OUT2} = 200 = 2V_S + 50R_S \;........\times -1$$

$$V_{OUT1} = 650 = 5V_S + 50R_S$$
$$V_{OUT2} = -200 = -2V_S - 50R_S$$

Rearrange to give:

$$650 - 200 = \left(5V_S - 2V_S\right) + \left(50R_S - 50R_S\right)$$
$$450 = 3V_S + 0$$

$$\therefore V_S = \frac{450}{3} = 150V$$

Knowing that the ideal voltage source, V_S is equal to 150 volts, we can use this value for equation V_{OUT1} (or V_{OUT2} if so wished) and solve to find the series resistance, R_S.

$$V_{OUT1} = 130 = V_S + 10R_S$$

$$V_S = 150V$$

$$130 = 150 + 10R_S$$

$$\therefore R_S = \frac{150 - 130}{10} = 2\Omega$$

Then for our simple example, the batteries internal voltage source is calculated as: V_S = 150 volts, and its internal resistance as: $R_S = 2\Omega$. The I-V characteristics of the battery are given as:

Short questions

1.what is active and passive element

An **active component** supplies energy to an electric circuit, and hence has the ability to electrically control the flow of charge. A **passive component** can only receive energy, which it can either dissipate or absorb.

2.what is unilateral and bilateral element

Unilateral elements in electrical circuit is defined as the element whose V-I characteristics changes on reversal of polarity of applied voltage. This essentially means that, the magnitude of the current passing through this type of element is affected due to change in the polarity of the applied voltage.

Bilateral elements are defined as the elements through which magnitude of current is independent of polarity of supply voltage. This means, the V-I characteristics of such type of element does not get affected by the polarity of voltage. A resistor, inductor, capacitors are example of bilateral circuit elements.

3.what is linear and non linear element

A *linear element* is one whose plot between, voltage across it and the current through it, comes out to be a straight line.

In other words, elements for which impedance remains constant for all values of voltage across them.
Examples :- resistors, inductors, capacitors etc.

4 whai is mesh?

A mesh network is a network in which devices -- or nodes -- are linked together, branching off other devices or nodes. These networks are set up to efficiently route data between devices and clients.

5 What is node?

The point through which an circuit element is connected to the circuit is called **node**. It is better to say, node is a point where, terminal of two or more circuit elements are connected together. Node is a junction point in the circuit.

Long questions

1.Explain super mesh and super node analysis?

2.Explain source transformation technique?

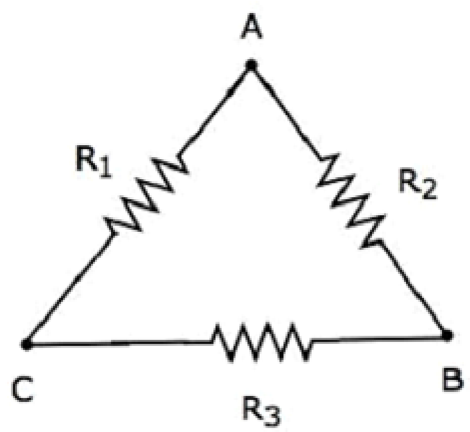
NETWORK THEOREMS:

Chapter-2

2.1 Star to delta and delta to star transformation

Delta Network

Consider the following delta network as shown in the following figure.



The following equations represent the equivalent resistance between two terminals of delta network, when the third terminal is kept open.

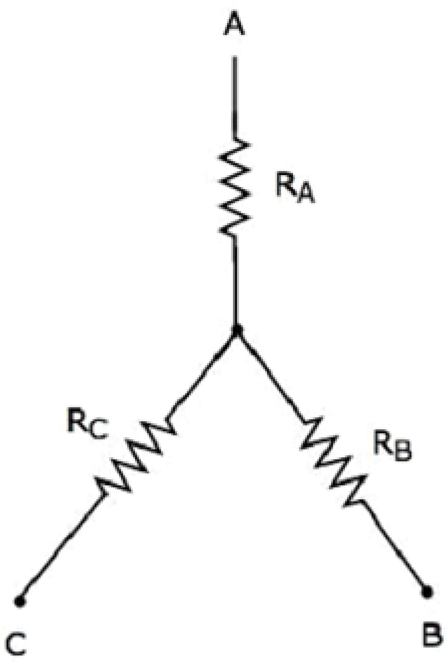
$$R_{AB}=(R_1+R_3)R_2/R_1+R_2+R_3$$

$$R_{BC}=(R_1+R_2)R_3/R_1+R_2+R_3$$

$$R_{CA}=(R_2+R_3)R_1/R_1+R_2+R_3$$

Star Network

The following figure shows the equivalent star network corresponding to the above delta network.



The following equations represent the **equivalent resistance** between two terminals of star network, when the third terminal is kept open.

$$R_{AB}=R_A+R_B$$

$$R_{BC}=R_B+R_C$$

$$R_{CA}=R_C+R_A$$

Star Network Resistances in terms of Delta Network Resistances

We will get the following equations by equating the right-hand side terms of the above equations for which the left-hand side terms are same.

$RA+RB=(R1+R3)R2/R1+R2+R3$ **Equation 1**
$3RB+RC=(R1+R2)R3/R1+R2+R3$ **Equation 2**
$RC+RA=(R2+R3)R1/R1+R2+R3$ **Equation 3**

By adding the above three equations, we will get

$$2(RA+RB+RC)=2(R1R2+R2R3+R3R1)/R1+R2+R3$$

$\Rightarrow RA+RB+RC=R1R2+R2R3+R3R1/R1+R2+R$ **Equation 4**

Subtract Equation 2 from Equation 4.

$$RA+RB+RC-(RB+RC)=R1R2+R2R3+R3R1/R1+R2+R3-(R1+R2)R3/R1+R2+R3$$
$$RA=R1R2/R1+R2+R3$$

By subtracting Equation 3 from Equation 4, we will get

$$R_B=\frac{R_2R_3}{R_1+R_2+R_3}$$

By subtracting Equation 1 from Equation 4, we will get

$$R_C=\frac{R_3R_1}{R_1+R_2+R_3}$$

By using the above relations, we can find the resistances of star network from the resistances of delta network. In this way, we can convert a delta network into a star network.

Star to Delta Conversion

In the previous chapter, we got the resistances of star network from delta network as

$R_A=\frac{R_1R_2}{R_1+R_2+R_3}$ Equation 1

$R_B=\frac{R_2R_3}{R_1+R_2+R_3}$ Equation 2

$R_C=\frac{R_3R_1}{R_1+R_2+R_3}$ Equation 3

Delta Network Resistances in terms of Star Network Resistances

Let us manipulate the above equations in order to get the resistances of delta network in terms of resistances of star network.

- Multiply each set of two equations and then add.

$$R_AR_B+R_BR_C+R_CR_A=\frac{R_1R_2^2R_3+R_2R_3^2R_1+R_3R_1^2R_2}{(R_1+R_2+R_3)^2}$$

$$\Rightarrow R_AR_B+R_BR_C+R_CR_A=\frac{R_1R_2R_3(R_1+R_2+R_3)}{(R_1+R_2+R_3)^2}$$

$\Rightarrow R_AR_B+R_BR_C+R_CR_A=\frac{R_1R_2R_3}{R_1+R_2+R_3}$ Equation 4

- By dividing Equation 4 with Equation 2, we will get

$$\frac{R_AR_B+R_BR_C+R_CR_A}{R_B}=R_1$$

$$\Rightarrow R_1=R_C+R_A+\frac{R_CR_A}{R_B}$$

- By dividing Equation 4 with Equation 3, we will get

$$R_2=R_A+R_B+\frac{R_AR_B}{R_C}$$

- By dividing Equation 4 with Equation 1, we will get

$$R_3=R_B+R_C+\frac{R_BR_C}{R_A}$$

By using the above relations, we can find the resistances of delta network from the resistances of star network. In this way, we can convert star network into delta network.

2.2 Super position Theorem

Superposition theorem is based on the concept of linearity between the response and excitation of an electrical circuit. It states that the response in a particular branch of a linear circuit when multiple independent sources are acting at the same time is equivalent to the sum of the responses due to each independent source acting at a time.

In this method, we will consider only one independent source at a time. So, we have to eliminate the remaining independent sources from the circuit. We can eliminate the voltage sources by shorting their two terminals and similarly, the current sources by opening their two terminals.

Therefore, we need to find the response in a particular branch 'n' times if there are 'n' independent sources. The response in a particular branch could be either current flowing through that branch or voltage across that branch.

Procedure of Superposition Theorem

Follow these steps in order to find the response in a particular branch using superposition theorem.

Step 1 − Find the response in a particular branch by considering one independent source and eliminating the remaining independent sources present in the network.

Step 2 − Repeat Step 1 for all independent sources present in the network.

Step 3 − Add all the responses in order to get the overall response in a particular branch when all independent sources are present in the network

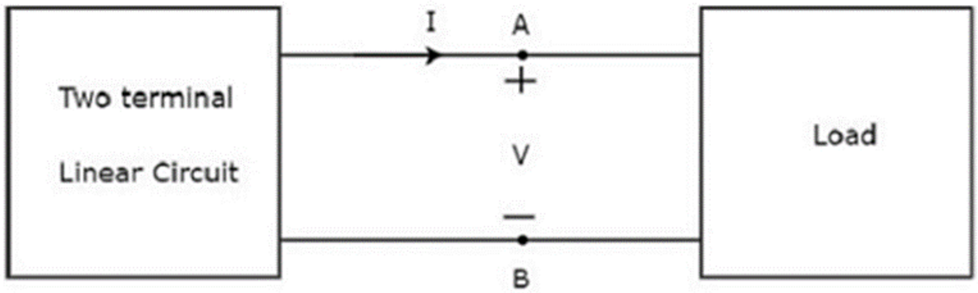
2.3 Thevenin's Theorem

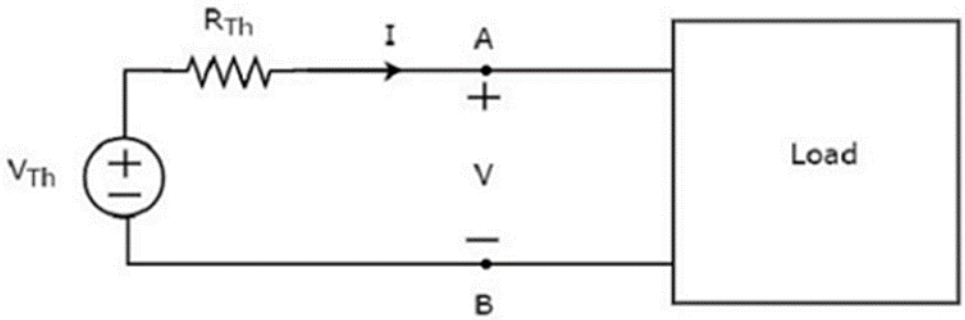
Thevenin's theorem states that any two terminal linear network or circuit can be represented with an equivalent network or circuit, which consists of a voltage source in series with a resistor. It is known as Thevenin's equivalent circuit. A linear circuit may contain independent sources, dependent sources, and resistors.

If the circuit contains multiple independent sources, dependent sources, and resistors, then the response in an element can be easily found by replacing the entire network to the left of that element with a Thevenin's equivalent circuit.

The response in an element can be the voltage across that element, current flowing through that element, or power dissipated across that element.

This concept is illustrated in following figures.





Thevenin's equivalent circuit resembles a practical voltage source. Hence, it has a voltage source in series with a resistor.

- The voltage source present in the Thevenin's equivalent circuit is called as Thevenin's equivalent voltage or simply Thevenin's voltage, V_{Th}.

The resistor present in the Thevenin's equivalent circuit is called as Thevenin's equivalent resistor or simply Thevenin's resistor, R_{Th}.

Methods of Finding Thevenin's Equivalent Circuit

There are three methods for finding a Thevenin's equivalent circuit. Based on the type of sources that are present in the network, we can choose one of these three methods. Now, let us discuss two methods one by one. We will discuss the third method in the next chapter.

Method 1

Follow these steps in order to find the Thevenin's equivalent circuit, when only the sources of independent type are present.

- Step 1 − Consider the circuit diagram by opening the terminals with respect to which the Thevenin's equivalent circuit is to be found.
- Step 2 − Find Thevenin's voltage V_{Th} across the open terminals of the above circuit.
- Step 3 − Find Thevenin's resistance R_{Th} across the open terminals of the above circuit by eliminating the independent sources present in it.

- Step 4 − Draw the Thevenin's equivalent circuit by connecting a Thevenin's voltage V_{Th} in series with a Thevenin's resistance R_{Th}.

Now, we can find the response in an element that lies to the right side of Thevenin's equivalent circuit.

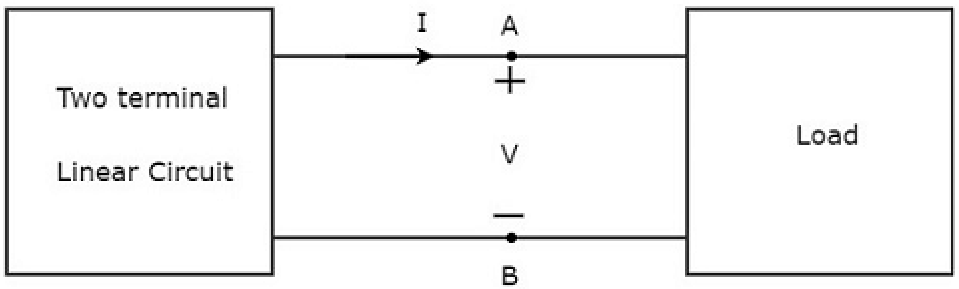
2.4 Norton's Theorem

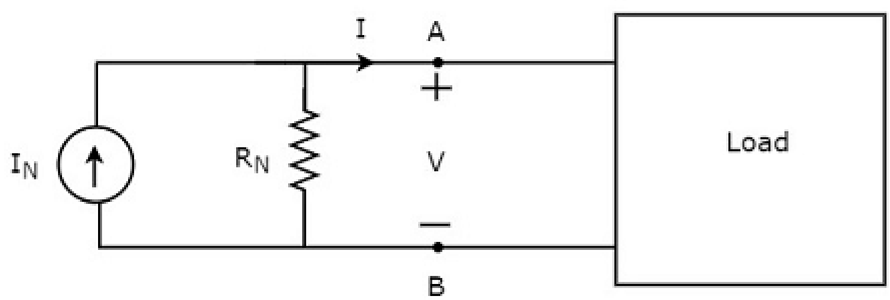
Norton's theorem is similar to Thevenin's theorem. It states that any two terminal linear network or circuit can be represented with an equivalent network or circuit, which consists of a current source in parallel with a resistor. It is known as Norton's equivalent circuit. A linear circuit may contain independent sources, dependent sources and resistors.

If a circuit has multiple independent sources, dependent sources, and resistors, then the response in an element can be easily found by replacing the entire network to the left of that element with a Norton's equivalent circuit.

The response in an element can be the voltage across that element, current flowing through that element or power dissipated across that element.

This concept is illustrated in following figures.





Norton's equivalent circuit resembles a practical current source. Hence, it is having a current source in parallel with a resistor.

- The current source present in the Norton's equivalent circuit is called as Norton's equivalent current or simply Norton's current I_N.
- The resistor present in the Norton's equivalent circuit is called as Norton's equivalent resistor or simply Norton's resistor R_N.

Methods of Finding Norton's Equivalent Circuit

There are three methods for finding a Norton's equivalent circuit. Based on the type of sources that are present in the network, we can choose one of these three methods. Now, let us discuss these three methods one by one.

Method 1

Follow these steps in order to find the Norton's equivalent circuit, when only the sources of independent type are present.

- Step 1 − Consider the circuit diagram by opening the terminals with respect to which, the Norton's equivalent circuit is to be found.
- Step 2 − Find the Norton's current I_N by shorting the two opened terminals of the above circuit.
- Step 3 − Find the Norton's resistance R_N across the open terminals of the circuit considered in Step1 by eliminating the independent sources present in it. Norton's resistance R_N will be same as that of Thevenin's resistance R_{Th}.
- Step 4 − Draw the Norton's equivalent circuit by connecting a Norton's current IN in parallel with Norton's resistance R_N.

Now, we can find the response in an element that lies to the right side of Norton's equivalent circuit.

Method 2

Follow these steps in order to find the Norton's equivalent circuit, when the sources of both independent type and dependent type are present.

- Step 1 − Consider the circuit diagram by opening the terminals with respect to which the Norton's equivalent circuit is to be found.
- Step 2 − Find the open circuit voltage V_{OC} across the open terminals of the above circuit.
- Step 3 − Find the Norton's current I_N by shorting the two opened terminals of the above circuit.
- Step 4 − Find Norton's resistance R_N by using the following formula.

$$R_N=V_{OC}I_N RN=VOCIN$$

- Step 5 − Draw the Norton's equivalent circuit by connecting a Norton's current I_N in parallel with Norton's resistance R_N.

Now, we can find the response in an element that lies to the right side of Norton's equivalent circuit.

Method 3

This is an alternate method for finding a Norton's equivalent circuit.

- Step 1 − Find a Thevenin's equivalent circuit between the desired two terminals. We know that it consists of a Thevenin's voltage source, V_{Th} and Thevenin's resistor, R_{Th}.
- Step 2 − Apply source transformation technique to the above Thevenin's equivalent circuit. We will get the Norton's equivalent circuit. Here,

Norton's current,

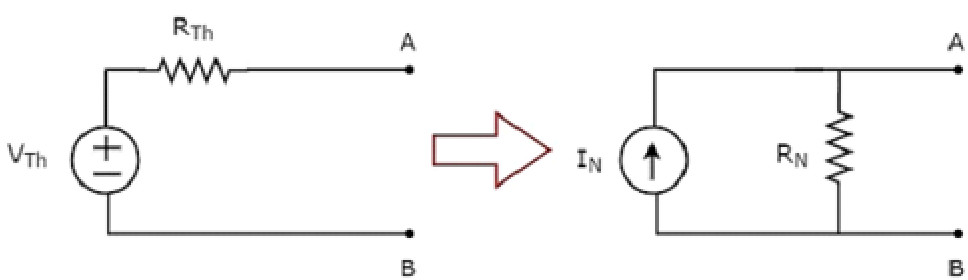
Norton's current,

$$I_N = \frac{V_{Th}}{R_{Th}}$$

Norton's resistance,

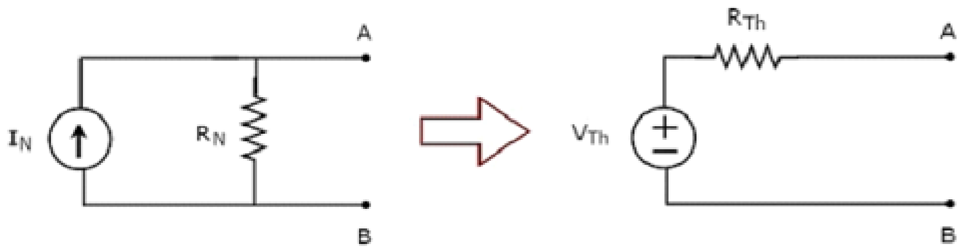
$$R_N = R_{Th}$$

This concept is illustrated in the following figure.



Now, we can find the response in an element by placing Norton's equivalent circuit to the left of that element.

Note − Similarly, we can find the Thevenin's equivalent circuit by finding a Norton's equivalent circuit first and then apply source transformation technique to it. This concept is illustrated in the following figure.



This is the Method 3 for finding a Thevenin's equivalent circuit.

2.5 Maximum power Transfer Theorem.

Maximum power transfer theorem states that the DC voltage source will deliver maximum power to the variable load resistor only when the load resistance is equal to the source resistance.

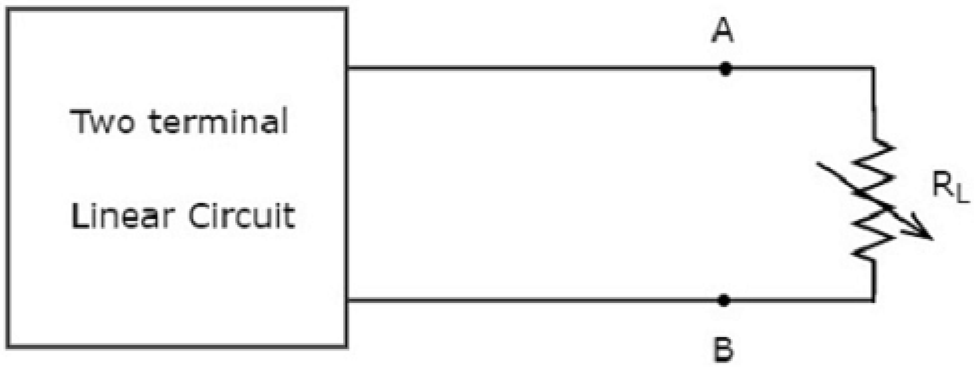
Similarly, Maximum power transfer theorem states that the AC voltage source will deliver maximum power to the variable complex load only when the load impedance is equal to the complex conjugate of source impedance.

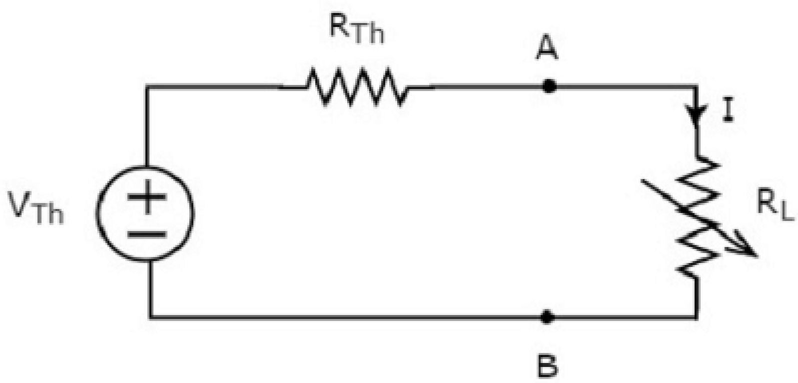
In this chapter, let us discuss about the maximum power transfer theorem for DC circuits.

Proof of Maximum Power Transfer Theorem

Replace any two terminal linear network or circuit to the left side of variable load resistor having resistance of R_L ohms with a Thevenin's equivalent circuit. We know that Thevenin's equivalent circuit resembles a practical voltage source.

This concept is illustrated in following figures.





The amount of power dissipated across the load resistor is

- Substitute $I = \frac{V_{Th}}{2R_{Th}}$ in the above equation.

$$P_S = 2(\frac{V_{Th}}{2R_{Th}})^2 R_{Th}$$

$$\Rightarrow P_S = 2(\frac{{V_{Th}}^2}{4{R_{Th}}^2}) R_{Th}$$

$$\Rightarrow P_S = \frac{{V_{Th}}^2}{2R_{Th}}$$

- Substitute the values of $P_{L,Max}$ and P_S in Equation 2.

$$\eta_{Max} = \frac{(\frac{{V_{Th}}^2}{4R_{Th}})}{(\frac{{V_{Th}}^2}{2R_{Th}})}$$

$$\Rightarrow \eta_{Max} = \frac{1}{2}$$

Efficiency of Maximum Power Transfer

We can calculate the efficiency of maximum power transfer, η_{Max} using following formula.

$$\eta_{Max} = \frac{P_{L,Max}}{P_S} \qquad \textbf{Equation 2}$$

Where,

- $P_{L,Max}$ is the maximum amount of power transferred to the load.
- P_S is the amount of power generated by the source.

The **amount of power generated** by the source is

$$P_S = I^2 R_{Th} + I^2 R_L$$

$$\Rightarrow P_S = 2I^2 R_{Th},\ since\ R_L = R_{Th}$$

- Substitute $I = \frac{V_{Th}}{2R_{Th}}$ in the above equation.

$$\Rightarrow (R_{Th} + R_L)^2 - 2R_L(R_{Th} + R_L) = 0$$

$$\Rightarrow (R_{Th} + R_L)(R_{Th} + R_L - 2R_L) = 0$$

$$\Rightarrow (R_{Th} - R_L) = 0$$

$$\Rightarrow R_{Th} = R_L \; or \; R_L = R_{Th}$$

Therefore, the **condition for maximum power** dissipation across the load is $R_L = R_{Th}$.

That means, if the value of load resistance is equal to the value of source resistance i.e., Thevenin's resistance, then the power dissipated across the load will be of maximum value.

The value of Maximum Power Transfer

Substitute $R_L = R_{Th} \,\&\, P_L = P_{L,Max}$ in Equation 1.

$$P_{L,Max} = {V_{Th}}^2 \{\frac{R_{Th}}{(R_{Th} + R_{Th})^2}\}$$

$$P_{L,Max} = {V_{Th}}^2 \{\frac{R_{Th}}{4{R_{Th}}^2}\}$$

We can represent the efficiency of maximum power transfer in terms of **percentage** as follows −

$$\%\eta_{Max} = \eta_{Max} \times 100\%$$

$$\Rightarrow \%\eta_{Max} = (\frac{1}{2}) \times 100\%$$

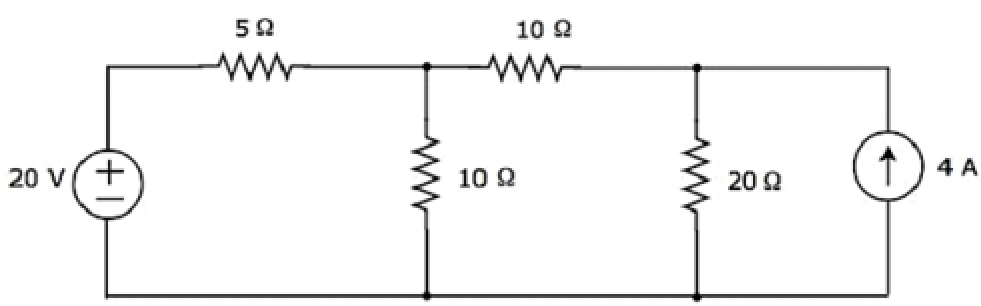
$$\Rightarrow \%\eta_{Max} = 50\%$$

Therefore, the efficiency of maximum power transfer is **50 %**.

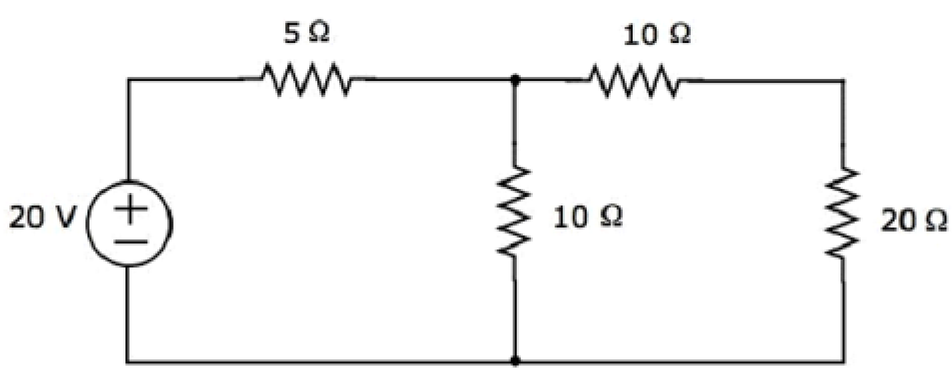
2.6 Solve numerical problems (With Independent Sources Only

Example

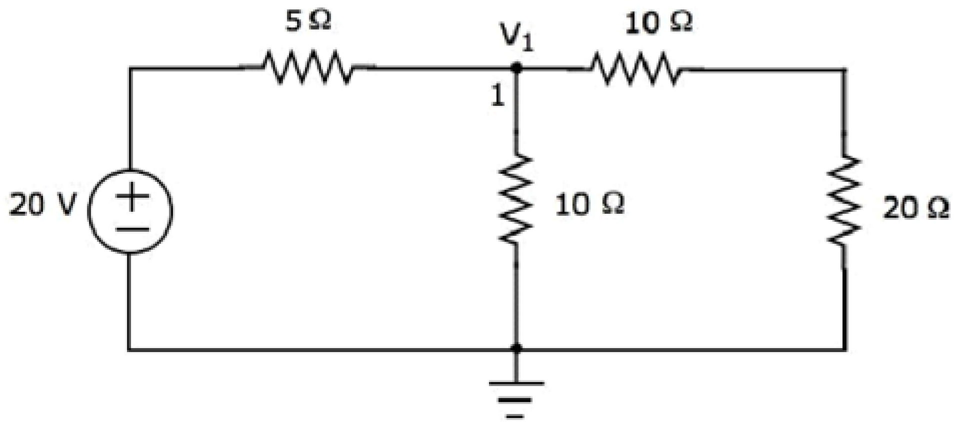
Find the current flowing through 20 Ω resistor of the following circuit using superposition theorem.



Step 1 − Let us find the current flowing through 20 Ω resistor by considering only 20 V voltage source. In this case, we can eliminate the 4 A current source by making open circuit of it. The modified circuit diagram is shown in the following figure.



There is only one principal node except Ground in the above circuit. So, we can use nodal analysis method. The node voltage V_1 is labelled in the following figure. Here, V_1 is the voltage from node 1 with respect to ground.



The nodal equation at node 1 is

$$\frac{V_1 - 20}{5} + \frac{V_1}{10} + \frac{V_1}{10 + 20} = 0$$

$$\Rightarrow \frac{6V_1 - 120 + 3V_1 + V_1}{30} = 0$$

$$\Rightarrow 10V_1 = 120$$

$$\Rightarrow V_1 = 12V$$

The **current flowing through 20 Ω resistor** can be found by doing the following simplification.

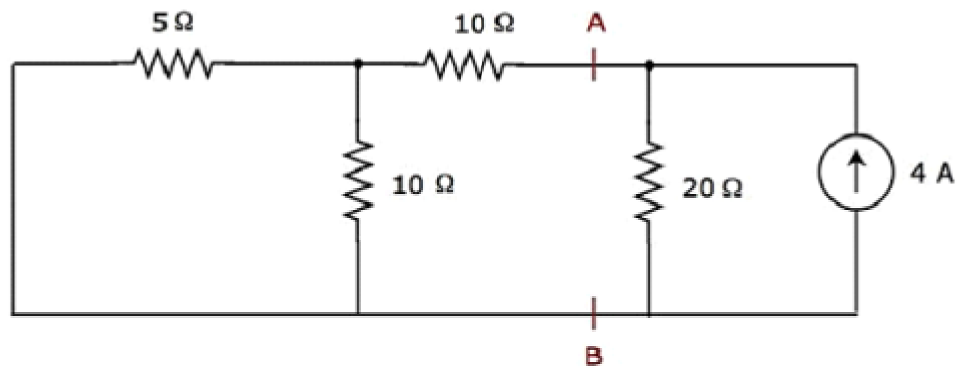
$$I_1 = \frac{V_1}{10 + 20}$$

Substitute the value of V_1 in the above equation.

$$I_1 = \frac{12}{10 + 20} = \frac{12}{30} = 0.4A$$

Therefore, the current flowing through 20 Ω resistor is 0.4 A, when only 20 V voltage source is considered.

Step 2 − Let us find the current flowing through 20 Ω resistor by considering only 4 A current source. In this case, we can eliminate the 20 V voltage source by making short-circuit of it. The modified circuit diagram is shown in the following figure.



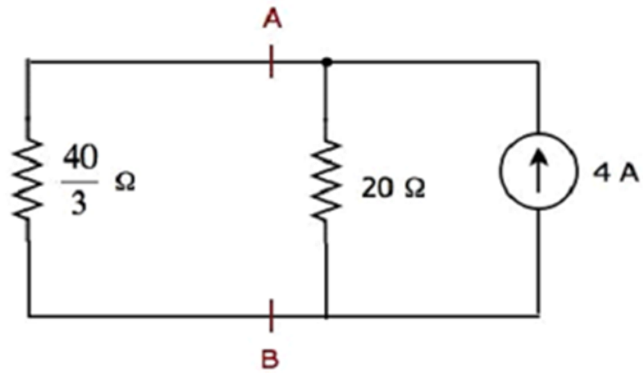
In the above circuit, there are three resistors to the left of terminals A & B. We can replace these resistors with a single equivalent resistor. Here, 5 Ω & 10 Ω resistors are connected in parallel and the entire combination is in series with 10 Ω resistor.

The equivalent resistance to the left of terminals A & B will be

The **equivalent resistance** to the left of terminals A & B will be

$$R_{AB} = (\frac{5 \times 10}{5 + 10}) + 10 = \frac{10}{3} + 10 = \frac{40}{3}\Omega$$

The simplified circuit diagram is shown in the following figure.



We can find the current flowing through 20 Ω resistor, by using **current division principle**.

$$I_2 = I_S(\frac{R_1}{R_1 + R_2})$$

Substitute $I_S = 4A,\ R_1 = \frac{40}{3}\Omega$ and $R_2 = 20\Omega$ in the above equation.

$$I_2 = 4(\frac{\frac{40}{3}}{\frac{40}{3} + 20}) = 4(\frac{40}{100}) = 1.6A$$

Therefore, the current flowing through 20 Ω resistor is 1.6 A, when only 4 A current source is considered.

Step 3 − We will get the current flowing through 20 Ω resistor of the given circuit by doing the addition of two currents that we got in step 1 and step 2. Mathematically, it can be written as

$$I = I_1 + I_2$$

Substitute, the values of I_1 and I_2 in the above equation.

$$I = 0.4 + 1.6 = 2A$$

Therefore, the current flowing through 20 Ω resistor of given circuit is 2 A.

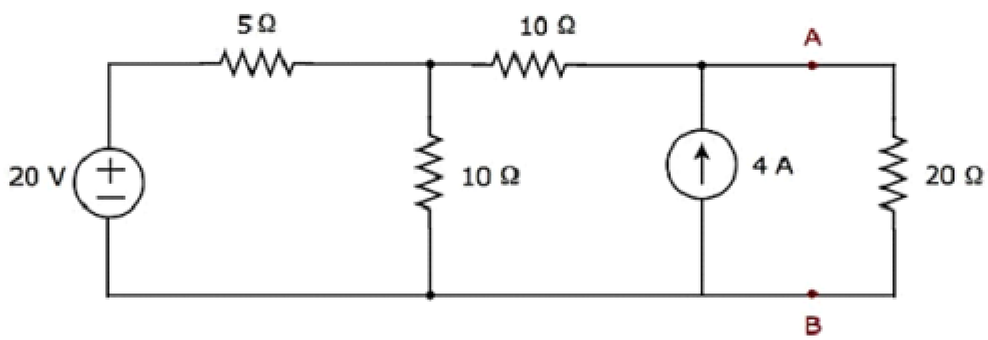
Note − We can't apply superposition theorem directly in order to find the amount of power delivered to any resistor that is present in a linear circuit, just by doing the addition of powers delivered to that resistor due to each independent source. Rather, we can calculate either total current flowing through or voltage across that resistor by using superposition theorem and from that, we can calculate the amount of power delivered to that resistor using

and from that, we can calculate the amount of power delivered to that resistor using I^2R

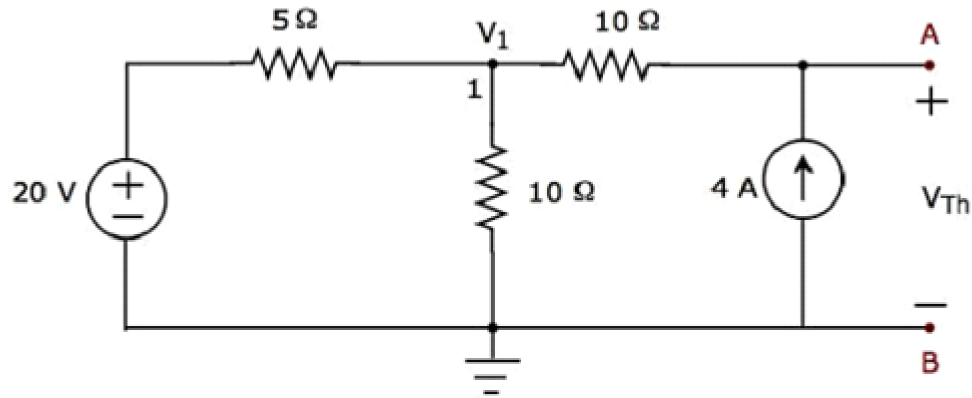
or $\frac{V^2}{R}$.

Example

Find the current flowing through 20 Ω resistor by first finding a Thevenin's equivalent circuit to the left of terminals A and B.



Step1 − In order to find the Thevenin's equivalent circuit to the left side of terminals A & B, we should remove the 20 Ω resistor from the network by opening the terminals A & B. The modified circuit diagram is shown in the following figure.



Step 2 − Calculation of Thevenin's voltage V_{Th}.

There is only one principal node except Ground in the above circuit. So, we can use nodal analysis method. The node voltage V_1 and Thevenin's voltage V_{Th} are labelled in the above figure. Here, V_1 is the voltage from node 1 with respect to Ground and V_{Th} is the voltage across 4 A current source.

- The nodal equation at node 1 is

- The **nodal equation** at node 1 is

$$\frac{V_1 - 20}{5} + \frac{V_1}{10} - 4 = 0$$

$$\Rightarrow \frac{2V_1 - 40 + V_1 - 40}{10} = 0$$

$$\Rightarrow 3V_1 - 80 = 0$$

$$\Rightarrow V_1 = \frac{80}{3} V$$

- The voltage across series branch 10 Ω resistor is

$$V_{10\Omega} = (-4)(10) = -40V$$

- There are two meshes in the above circuit. The **KVL equation** around second mesh is

$$V_1 - V_{10\Omega} - V_{Th} = 0$$

- Substitute the values of V_1 and $V_{10\Omega}$ in the above equation.

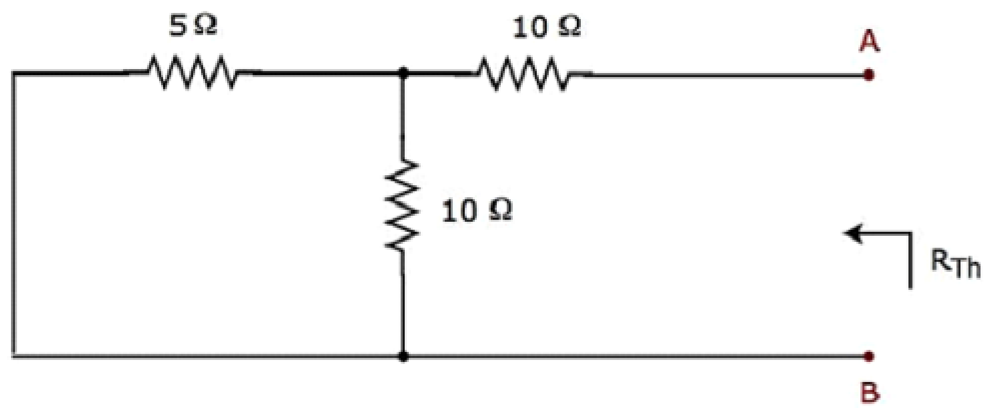
$$\frac{80}{3} - (-40) - V_{Th} = 0$$

$$V_{Th} = \frac{80 + 120}{3} = \frac{200}{3} V$$

- Therefore, the Thevenin's voltage is $V_{Th} = \frac{200}{3} V$

Step 3 − Calculation of Thevenin's resistance R_{Th}.

Short circuit the voltage source and open circuit the current source of the above circuit in order to calculate the Thevenin's resistance R_{Th} across the terminals A & B. The modified circuit diagram is shown in the following figure.

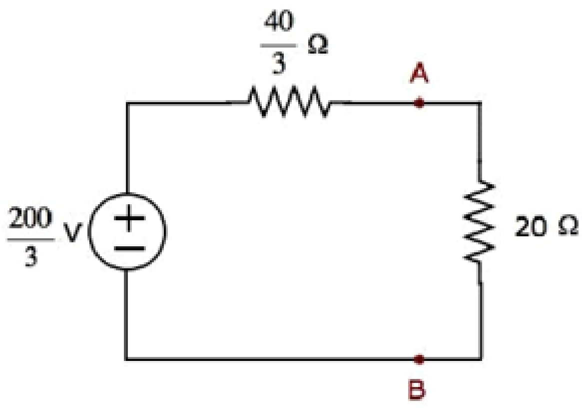


The Thevenin's resistance across terminals A & B will be

$$R_{Th} = (\frac{5 \times 10}{5 + 10}) + 10 = \frac{10}{3} + 10 = \frac{40}{3}\Omega$$

Therefore, the Thevenin's resistance is $\mathbf{R_{Th} = \frac{40}{3}\Omega}$.

Step 4 − The Thevenin's equivalent circuit is placed to the left of terminals A & B in the given circuit. This circuit diagram is shown in the following figure.



The current flowing through the 20 Ω resistor can be found by substituting the values of V_{Th}, R_{Th} and R in the following equation.

$$l = \frac{V_{Th}}{R_{Th} + R}$$

$$l = \frac{\frac{200}{3}}{\frac{40}{3} + 20} = \frac{200}{100} = 2A$$

Therefore, the current flowing through the 20 Ω resistor is **2 A**.

Method 2

Follow these steps in order to find the Thevenin's equivalent circuit, when the sources of both independent type and dependent type are present.

- **Step 1 −** Consider the circuit diagram by opening the terminals with respect to which, the Thevenin's equivalent circuit is to be found.
- **Step 2 −** Find Thevenin's voltage $\mathbf{V_{Th}}$ across the open terminals of the above circuit.
- **Step 3 −** Find the short circuit current $\mathbf{I_{SC}}$ by shorting the two opened terminals of the above circuit.
- **Step 4 −** Find Thevenin's resistance $\mathbf{R_{Th}}$ by using the following formula.

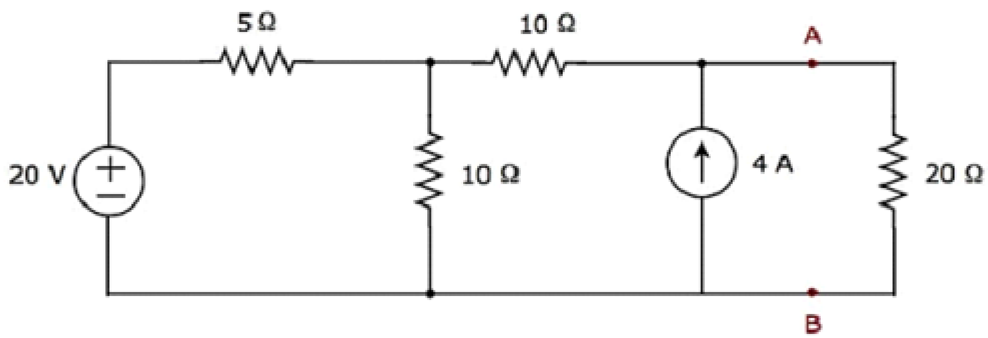
$$R_{Th} = \frac{V_{Th}}{I_{SC}}$$

Step 5 − Draw the Thevenin's equivalent circuit by connecting a Thevenin's voltage V_{Th} in series with a Thevenin's resistance R_{Th}.

Now, we can find the response in an element that lies to the right side of the Thevenin's equivalent circuit.

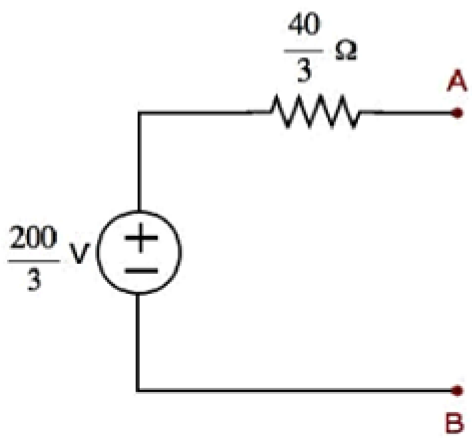
Example

Find the current flowing through 20 Ω resistor by first finding a Norton's equivalent circuit to the left of terminals A and B.



Let us solve this problem using Method 3.

Step 1 − In previous chapter, we calculated the Thevenin's equivalent circuit to the left side of terminals A & B. We can use this circuit now. It is shown in the following figure.



Here, Thevenin's voltage, $V_{Th} = \frac{200}{3} V$ and Thevenin's resistance, $R_{Th} = \frac{40}{3} \Omega$

Step 2 − Apply source transformation technique to the above Thevenin's equivalent circuit. Substitute the values of V_{Th} and R_{Th} in the following formula of Norton's current

$$I_N = \frac{V_{Th}}{R_{Th}}$$

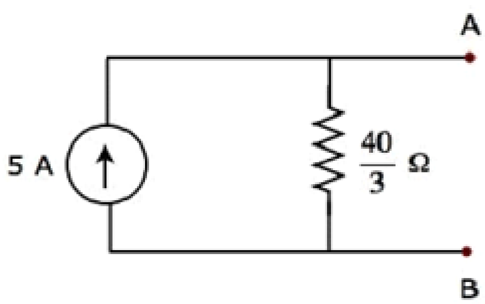
$$I_N = \frac{\frac{200}{3}}{\frac{40}{3}} = 5A$$

Therefore, Norton's current I_N is **5 A**.

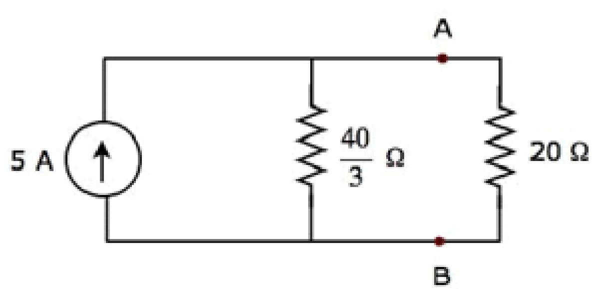
We know that Norton's resistance, R_N is same as that of Thevenin's resistance R_{Th}.

$$\mathbf{R_N = \frac{40}{3}\Omega}$$

The Norton's equivalent circuit corresponding to the above Thevenin's equivalent circuit is shown in the following figure.



Now, place the Norton's equivalent circuit to the left of the terminals A & B of the given circuit.



By using current division principle, the current flowing through the 20 Ω resistor will be

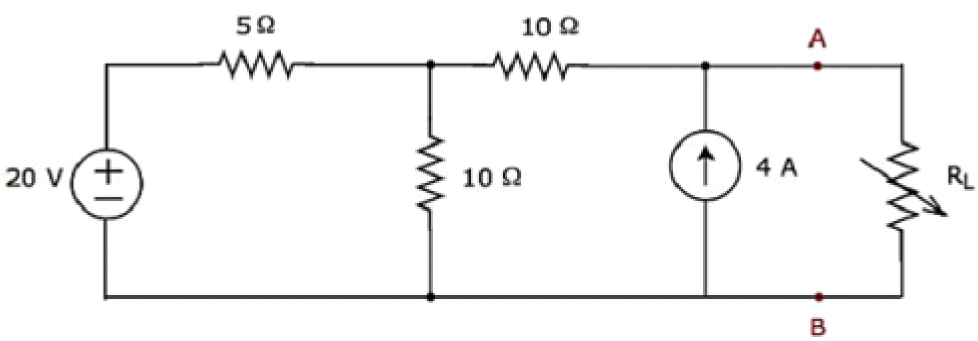
$$I_{20\Omega} = 5(\frac{\frac{40}{3}}{\frac{40}{3}+20})$$

$$I_{20\Omega} = 5(\frac{40}{100}) = 2A$$

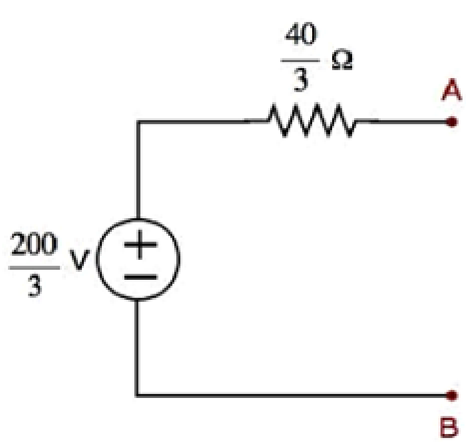
Therefore, the current flowing through the 20 Ω resistor is **2 A**.

Example

Find the maximum power that can be delivered to the load resistor R_L of the circuit shown in the following figure.

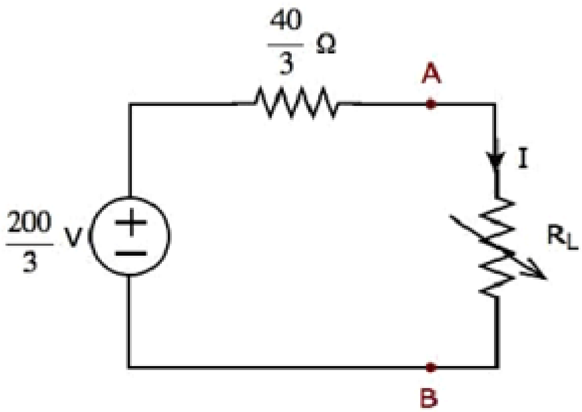


Step 1 − In Thevenin's Theorem chapter, we calculated the Thevenin's equivalent circuit to the left side of terminals A & B. We can use this circuit now. It is shown in the following figure.



Here, Thevenin's voltage $V_{Th} = \frac{200}{3} V$ and Thevenin's resistance $R_{Th} = \frac{40}{3} \Omega$

Step 2 − Replace the part of the circuit, which is left side of terminals A & B of the given circuit with the above Thevenin's equivalent circuit. The resultant circuit diagram is shown in the following figure.



Step 3 − We can find the maximum power that will be delivered to the load resistor, R_L by using the following formula.

PL,Max=VTh24RTh

$$P_{L,Max} = \frac{{V_{Th}}^2}{4R_{Th}}$$

Substitute $V_{Th} = \frac{200}{3} V$ and $R_{Th} = \frac{40}{3} \Omega$ in the above formula.

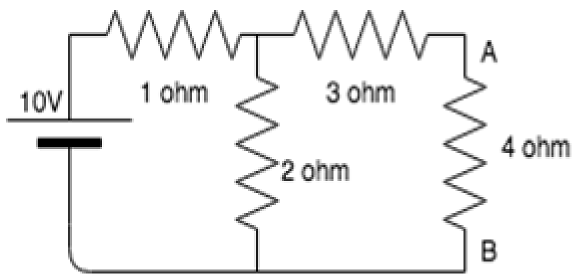
$$P_{L,Max} = \frac{(\frac{200}{3})^2}{4(\frac{40}{3})}$$

$$P_{L,Max} = \frac{250}{3} W$$

Therefore, the **maximum power** that will be delivered to the load resistor RL of the given circuit is $\frac{250}{3}$ **W**

SHORT QUESTIONS

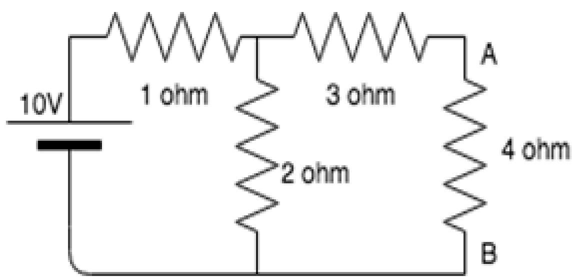
1. Calculate the Thevenin resistance across the terminal AB for the following circuit.



Answer: b

Explanation: Thevenin resistance is found by opening the circuit between the specified terminal and shorting all voltage sources. When the 10V source is shorted, we get: Rth=(1||2)+3=3.67 ohm.

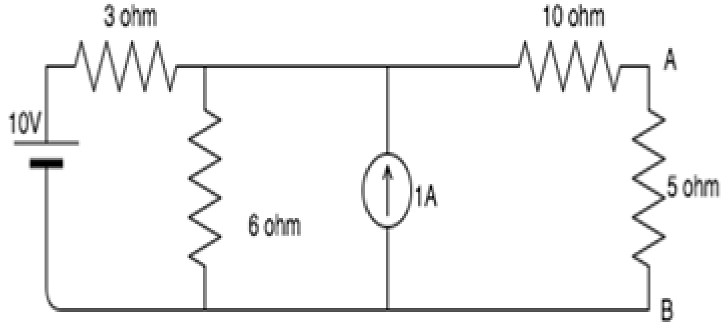
2. Calculate V_{th} for the given circuit.



Answer: c

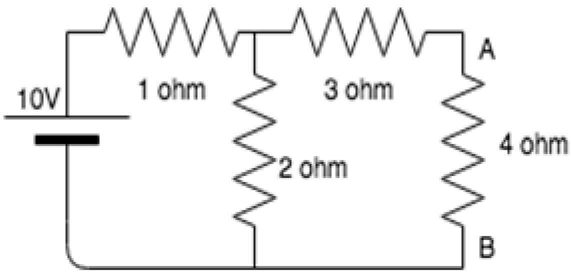
Explanation: 4 ohm is removed and then v across 2 ohm is calculated by voltage divider 2*10/(2+1) = 6.67V. Voltage between A and B i.e. V_{th} is equal to voltage across 4 ohm resistance since no current flow through 3 ohm resistance. So, V_{th} = 6.67V.

3. Calculate the Norton resistance for the following circuit if 5 ohm is the load resistance.



Answer: c
Explanation: Shorting all voltage sources and opening all current sources we have: RN=(3||6)+10 = 12 ohm.

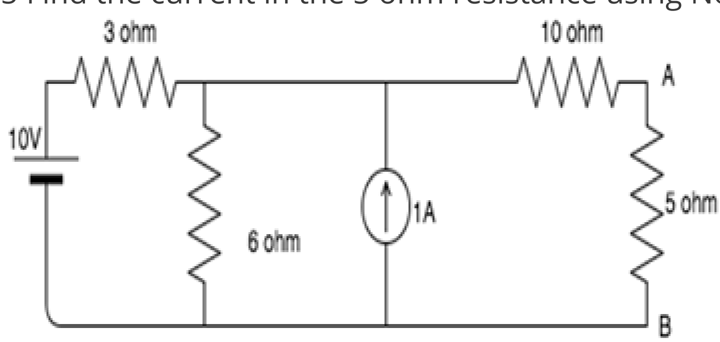
4 Calculate the current across the 4 ohm resistor.



a)0.86A
b)1.23A
c)2.22A
d) 0.67A

Answer: a
Explanation: Thevenin resistance is found by opening the circuit between the specified terminal and shorting all voltage sources.
When the 10V source is shorted, we get: Rth=(1||2)+3=3.67 ohm.
Vth is calculated by opening the specified terminal.
Using voltage divider, Vth= 2*10/(2+1)=6.67V.
On drawing the Thevenin equivalent circuit, we get Rth, 4 ohm and Vth in series.
Applying Ohm's law, I=Vth/(4+Rth) = 0.86A.

5 Find the current in the 5 ohm resistance using Norton's theorem.



a) 1A
b) 1.5A
c) 0.25A
d) 0.5A
View Answer
Answer: d
Explanation: Shorting all voltage sources and opening all current sources we have:

R_N=(3||6)+10 = 12 ohm.
Since the 5 ohm is the load resistance, we short it and find the resistance through the short.
If we apply source transformation between the 6 ohm resistor and the 1A source, we get a 6V source in series with a 6 ohm resistor. Now we have two meshes. Let us consider I1 flowing in the first mesh and I2 flowing in the second mesh.
The mesh equations are:
9I1-6I2=4
-6I1+16I2=6
On solving these equations simultaneously, we get I2=0.72A, which is the short circuit current.
Connecting the current source in parallel to R_N which is in turn connected in parallel to the load resistance=5ohm, we get Norton's equivalent circuit.
Using current divider: I = 0.72*12/(12+5) = 0.5 A.

LONG QUESTIONS

1.EXPLAIN SUPERPOSITION THEOREM?

2 Explain Norton ,Thevenin and maximum power transfer theorem?

CHAPTER-3

MAGNETIC CIRCUIT

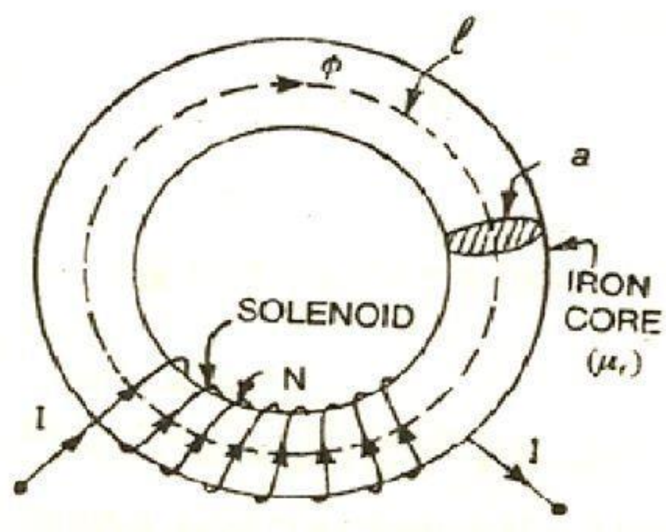
3.1.INTRODUCTION

The closed path followed by magnetic lines of forces is called **the** magnetic circuit**. In the** magnetic circuit, magnetic flux or magnetic lines of force starts from a point and ends at the same point after completing its path.

Flux is generated by magnets, it can be a permanent magnet or electromagnets.

A magnetic circuit is made up of magnetic materials having high permeability such as iron, soft steel, etc. Magnetic circuits are used in various devices like electric motor, transformers, relays, generators galvanometer, etc.

Consider a solenoid having N turns wound on an iron core. The magnetic flux of ø Weber sets up in the core when the current of I ampere is passed through a solenoid.



Let, l = mean length of the magnetic circuit
A = cross-sectional area of the core
μr = relative permeability of the core
Now the flux density in the core material

$$B = \frac{\varphi}{a} \quad (Weber/m^2)$$

Magnetising force in the core

$$H = B/\mu_0\mu_r$$

$$H = {}^{\varphi}/_{a}\,\mu_0\mu_r \text{ AT/m (Ampere turns/meter)}$$

According to work law, the work done in moving a unit pole once round the magnetic circuit is equal to the ampere-turns enclosed by the magnetic circuit.

$$Hl = NI$$

$$H = \frac{\varphi}{a\mu_0\mu_r} x\, l$$

$$H = NI$$

$$\varphi = \frac{NI}{l/a\mu_0\mu_r}$$

The above equation explains the following points:

1. Directly proportional to the number of turns (N) and current (I).
   It shows that the flux increase if the number of turns or current increases and decreases when either of the two quantity decreases. NI is the magnetomotive force (MMF).
2. Inversely proportional to $l/a\ \mu_0\mu_r$, where ($l/a\ \mu_0\mu_r$) is known as reluctance. The lower the reluctance, the higher will be the flux and vice- verse.

3.2.(Magnetising force,Intensity,MMF,Flux and their relation)

Magnetising force:

The magnetic force is a consequence of the electromagnetic force, one of the four fundamental forces of nature, and is caused by the **motion of charges**. Two objects containing charge with the same direction of motion have a magnetic attraction force between them. Similarly, objects with charge moving in opposite directions have a repulsive force between them.

Consider two objects. The magnitude of the magnetic force between them depends on **how much charge** is in **how much motion** in each of the two objects and how far apart they are. The direction of the force depends on the relative directions of motion of the charge in each case.

The usual way to go about finding the magnetic force is framed in terms of a fixed amount of charge q*q*q moving at constant velocity v*v*v in a uniform magnetic field B*B*B. If we don't know the magnitude of the magnetic field directly then we can still use this method because it is often possible to calculate the magnetic field based on the distance to a known current.

The magnetic force is described by the Lorentz Force law**:**

$$\vec{F} = q\vec{v} \times \vec{B}$$

In this form it is written using the vector cross product. We can write the magnitude of the magnetic force by expanding the cross product. Written in terms of the angle θ ($< 180^\circ$) between the velocity vector and the magnetic field vector:

$$\boxed{F = qvB\sin\theta}$$

The direction of the force can be found using the *right-hand-slap rule*. This rule describes the direction of the force as the direction of a 'slap' of an open hand. As with the right-hand-grip rule, the fingers point in the direction of the magnetic field. The thumb points in the direction that **positive** charge is moving. If the moving charge is negative (for example, electrons) then you need to reverse the direction of your thumb because the force will be in the opposite direction. Alternatively, you can use your left hand for moving **negative** charge.

[Explain]

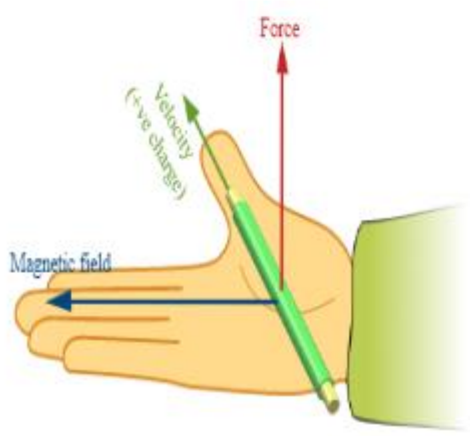


Figure 1: *Using the right-hand-slap rule for the force due to a positive charge moving in a magnetic field.*

Using the right-hand-slap rule for the force due to a positive charge moving in a magnetic field.

Figure 1: *Using the right-hand-slap rule for the force due to a positive charge moving in a magnetic field.*

Sometimes we want to find the force on a wire carrying a current III in a magnetic field. This can be done by rearranging our previous expression. If we recall that velocity is a distance / time then if a wire has length LLL we can write

qv = qL/t

and since current is the amount of charge flowing per second,

qv = IL

and therefore

$$F = BIL\sin\theta$$

Magnetic Intensity (H):

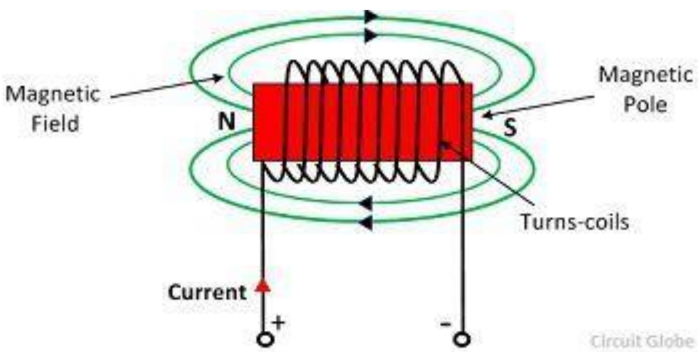
The ability of a magnetic field to magnetize a material medium is called its magnetic intensity H. Its magnitude is measured by the number of ampere-turns flowing round unit length of a solenoid, required to produce that magnetic field.

When a material medium is placed in a magnetic field, it gets magnetized. The magnetic moment per unit volume of the material is called the intensity of magnetization M (or simply magnetization).

M=Magnetic moment/Volume
S.I. unit of magnetization is (Am^{-1}). Lines representing intensity of magnetisation are called lines of magnetisation. For a uniformly magnetised material, each dipole will point in the same direction and M will be constant throughout.

Magnetomotive Force

Definition: The current flowing in an electric circuit is due to the existence of electromotive force similarly **magnetomotive force** (MMF) is required to drive the magnetic flux in the magnetic circuit. The magnetic pressure, which sets up the magnetic flux in a magnetic circuit is called Magnetomotive Force. The SI unit of MMF is Ampere-turn (AT), and their CGS unit is G (gilbert). The MMF for the inductive coil shown in the figure below is expressed as



$$F = NI$$

Where, N – numbers of turns of the inductive coil
I – current

The strength of the MMF is equivalent to the product of the current around the turns and the number of turns of the coil. As per work law, the MMF is defined as the work done in moving the unit magnetic pole (1weber) once around the magnetic circuit.

The MMF is also known as the magnetic potential. It is the property of a material to give rise to the magnetic field. The magnetomotive force is the product of the magnetic flux and the magnetic reluctance. The reluctance is the opposition offers by the magnetic field to set up the magnetic flux on it. The MMF regarding reluctance and magnetic flux is given as

$F = \emptyset R$ Where R – reluctance
Φ – magnetic flux

The magnetomotive force can measure regarding magnetic field intensity and the length of the substance. The magnetic field strength is the force act on the unit pole placed on the magnetic field. MMF regarding field intensity is expressed as $F = Hl$

Where H is the magnetic field strength, and l is the length of the substance.

Magnetic Flux

Magnetic flux is a measurement of the total magnetic field which passes through a given area. It is a useful tool for helping describe the effects of the magnetic force on something occupying a given area. The measurement of magnetic flux is tied to the particular area chosen. We can choose to make the area any size we want and orient it in any way relative to the magnetic field.

If we use the field-line picture of a magnetic field then every field line passing through the given area contributes some magnetic flux. The

angle at which the field line intersects the area is also important. A field line passing through at a glancing angle will only contribute a small component of the field to the magnetic flux. When calculating the magnetic flux we include only the **component** of the magnetic field vector which is **normal** to our test area.

If we choose a simple flat surface with area A*A*A as our test area and there is an angle \theta*θ*theta between the normal to the surface and a magnetic field vector (magnitude B) then the magnetic flux is,

[Explain]

$\Phi = BA\cos\theta$

In the case that the surface is perpendicular to the field then the angle is zero and the magnetic flux is simply B A*BA*B, A. Figure 1 shows an example of a flat test area at two different angles to a magnetic field and the resulting magnetic flux

Relation Between flux and mmf.

MMF is the driver that gives rise to the flux in a magnetic circuit. It is measured in ampere-turns (A-t) and is analogous to voltage in an electric circuit.

Flux density is a measure of the total magnetic “strength” or density pf lines of force set up in a magnetic circuit. It is analagous to current set up in an electric circuit. It is measured in teslas.

The flux density B is related to the MMF H by the equation B = H / Rm where Rm is the magnetic reluctance. This is analagous to the resistance in an electrical circuit. Reluctance is a function of the materials in the magnetic circuit, their cross sectional areas and the magnetic path length.

3 . 3 (Permeability, reluctance and permeance)

Magnetic permeability

The permeability or magnetic permeability is defined as the ability of a material to allow the magnetic lines of force to pass through it. It helps the development of the magnetic field in a magnetic circuit.
The SI unit of permeability is Henry/meter (H/m).

$$\text{Mathematically, } \mu = \mu_0 \mu_r \text{ H/m}$$

Where, μ_0 = permeability of free space (vacuum) = $4\pi * 10^{-7}$ Henry/meter

μ_r = relative permeability of a magnetic material

It is the ratio of magnetic flux density (B) to magnetizing force (H).

$$\mu = \frac{B}{H}$$

Relative Permeability

Relative Permeability is defined as the degree to which the material is a better conductor of magnetic flux as compared to free space.

It is denoted by .μr

Magnetic reluctance

Magnetic reluctance (also known as reluctance, magnetic resistance, or a magnetic insulator) is defined as the opposition offered by a magnetic circuit to the production of magnetic flux. It is the property of the material that opposes the creation of magnetic flux in a magnetic circuit.

The magnetic reluctance in a magnetic circuit is analogous to the resistance in an electric circuit as it opposes the production of magnetic flux in a magnetic circuit but it does not give rise to the dissipation of energy rather it stores magnetic energy.

Reluctance is directly proportional to the length of the magnetic circuit and inversely proportional to the area of the cross-section of the magnetic path. It is a scalar quantity and denoted by S. Note that a scalar quantity is one which is fully described by a magnitude (or numerical value) only. No direction is required to define the scalar quantity.

Mathematically it can be expressed as

where, l = length of the magnetic path in meters

= permeability of free space (vacuum) = Henry/meter
= relative permeability of a magnetic material

= Cross sectional area in square meters ( )

In AC as well as DC magnetic fields, the reluctance is the ratio of the magnetomotive force (m.m.f) to the magnetic flux in a magnetic circuit. In a pulsating AC or DC field, the reluctance is also pulsating.

Permeance
Permeance is defined as a measure of the ease with which magnetic flux can be admitted through a material or magnetic circuit. Permeance is the reciprocal of reluctance. Permeance is directly proportional to the magnetic flux and is denoted by the letter P.

From the above equation we can say that the quantity of magnetic flux for a number of ampere-turns is depended on permeance.

In terms of magnetic permeability, permeance is given by

$$P = \frac{\mu_0 \mu_r A}{l} = \frac{\mu A}{l}$$

Where,

- μ_0 = Permeability of free space (vacuum) = $4\pi * 10^{-7}$ Henry/meter
- μ_r = Relative permeability of a magnetic material
- l = Length of the magnetic path in meter
- A = Cross sectional area in square meters (m^2)

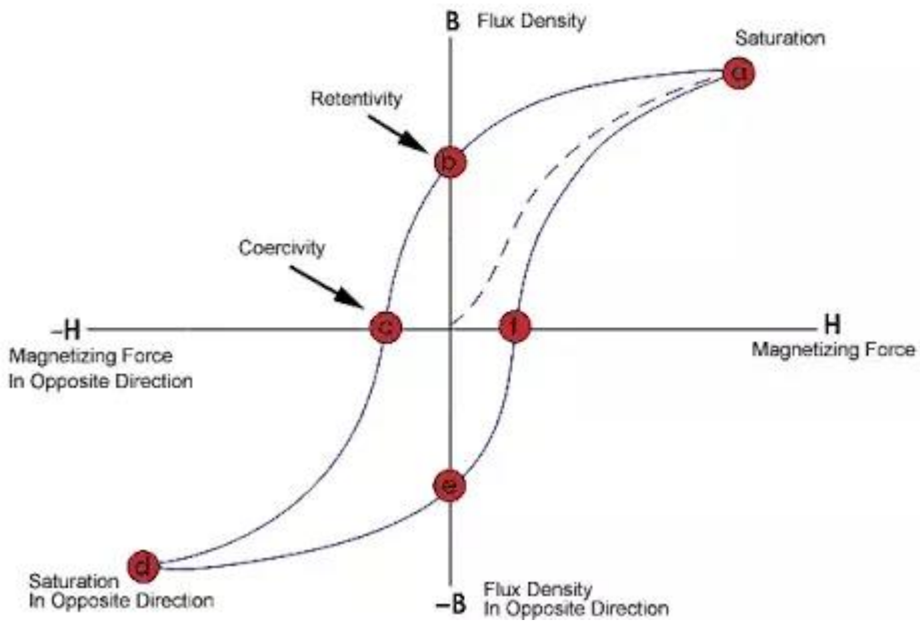
3 . 4 Analogy between electric and Magnetic Circuits

The Difference Between Both the circuits are explained below in the tabulated form.

BASIS	MAGNETIC CIRCUIT	ELECTRIC CIRCUIT
Definition	The closed path for magnetic flux is called magnetic circuit.	The closed path for electric current is called electric circuit.
Relation Between Flux and Current	Flux = mmf/reluctance	Current = emf/ resistance
Units	Flux φ is measured in weber (wb)	Current I is measured in amperes
MMF and EMF	Magnetomotive force is the driving force and is measured in Ampere turns (AT) Mmf =∫ H.dl	Electromotive force is the driving force and measured in volts (V) Emf = ∫ E.dl
Reluctance and Resistance	Reluctance opposes the flow of magnetic flux S = l/aμ and measured in (AT/wb)	Resistance opposes the flow of current R = ρ. l/a and measured in (Ώ)
Relation between Permeance and Conduction	Permeance = 1/reluctance	Conduction = 1/ resistance
Analogy	Permeability	Conductivity
Analogy	Reluctivity	Resistivity
Density	Flux density B = φ/a (wb/m2)	Current density J = I/a (A/m2)
Intensity	Magnetic intensity H = NI/l	Electric density E = V/d

BASIS	MAGNETIC CIRCUIT	ELECTRIC CIRCUIT
Drops	Mmf drop = φS	Voltage drop = IR
Flux and Electrons	In magnetic circuit molecular poles are aligned. The flux does not flow, but sets up in the magnetic circuit.	In electric circuit electric current flows in the form of electrons.
Examples	For magnetic flux, there is no perfect insulator. It can set up even in the non magnetic materials like air, rubber, glass etc.	For electric circuit there are a large number of perfect insulators like glass, air, rubber, PVC and synthetic resin which do not allow it to flow through them.
Variation of Reluctance and Resistance	The reluctance (S) of a magnetic circuit is not constant rather it varies with the value of B.	The resistance (R) of an electric circuit is almost constant as its value depends upon the value of ρ. The value of ρ and R can change slightly if the change in temperature takes place
Energy in the circuit	Once the magnetic flux sets up in a magnetic circuit, no energy is expanded. Only a small amount of energy is required at the initial stage to create flux in the circuit	Energy is expanding continuously, as long as the current flows through the electrical circuit. This energy is dissipated in the form of heat.
Applicable Laws	Khirchhoff flux and mmf law is followed	Khirchhoff voltage and current law is followed. (KVL and KCL)
Magnetic and Electric lines	Magnetic lines of flux starts from North pole and ends at South pole.	Electric lines or current starts from positive charge and ends on negative charge.

3 . 5 B-H Curve



You must be talking about the hysteresis curve. It basically represents the variation of the magnetic field inside a ferromagnetic with the external magnetizing field. You'll notice that in this curve as you increase the external magnetizing field , the magnetic field inside the ferromagnetic material also increases and vice versa. But even when the external magnetizing field is zero (After magnetization) , there will still be some residual magnetic field in the material. This residual magnetic field is called the “retentivity” of the material. To remove this residual magnetism , a magnetizing field has to be applied in the negative direction. The minimum magnetizing field required in the negative direction to destroy residual magnetism is called the “coercivity” of the material. The area under the curve represents energy loss per unit volume per cycle. What is this energy loss? It is the energy expended to demagnetize the material.

3 . 6 Series & parallel magnetic circuit

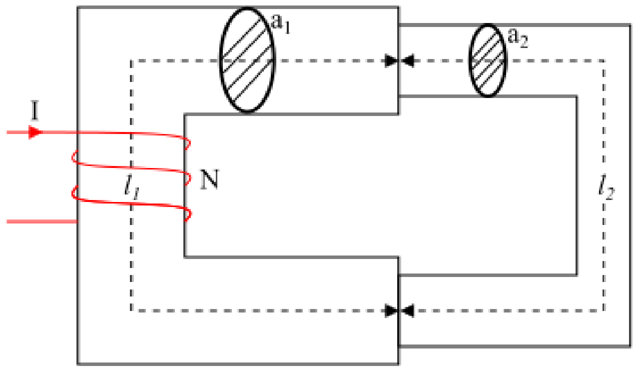
There are two types of magnetic circuits −

- Series Magnetic Circuit
- Parallel Magnetic Circuit

Series Magnetic Circuit

When the same magnetic flux ψ flows through each part of the magnetic circuit, then the circuit is called as *series magnetic circuit*.

Consider a *composite series magnetic circuit* (a series magnetic circuit that has parts of different dimensions and materials is called a composite series magnetic circuit) consisting of two different magnetic materials of different relative permeability. Each part of this series magnetic circuit will offer reluctance to the magnetic flux ψ. Since the different parts of the magnetic circuit are in series, the total reluctance is equal to the sum of reluctances of individual parts.



Referring the figure of series magnetic circuit, we have,

$$\text{Total Reluctance}, S_T = \frac{l_1}{\mu_0 \mu_{r1} a_1} + \frac{l_2}{\mu_0 \mu_{r2} a_2}$$

$$\text{Total MMF} = \text{Magnetic flux} \times \text{Total Reluctance}$$

$$\Longrightarrow \text{Total MMF} = \psi\left(\frac{l_1}{\mu_0 \mu_{r1} a_1} + \frac{l_2}{\mu_0 \mu_{r2} a_2}\right)$$

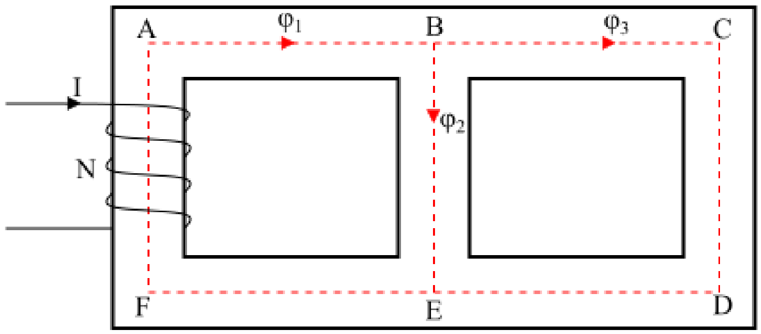
$$\Longrightarrow \text{Total MMF} = \left(\frac{B_1}{\mu_0 \mu_{r2}}\right) \times l_1 + \left(\frac{B_2}{\mu_0 \mu_{r2}}\right) \times l_2$$

$$\Longrightarrow \text{Total MMF} = H_1 \times l_1 + H_2 \times l_2$$

Therefore, total MMF required to set up the magnetic flux in a series magnetic circuit is the sum of MMF required by individual parts of the circuit.

Parallel Magnetic Circuit

A magnetic circuit which has more one path for the magnetic flux is called as parallel magnetic circuit.



Consider a coil of N turns wound on limb AF carries an electric current of I amperes. The magnetic flux φ1φ1 set up by the coil divides at B into two paths viz. −

- The magnetic φ2φ2 passes along the path BE.
- The magnetic flux φ3φ3 passes along the path BCDE.

Therefore, the total flux is,

$$\varphi 1=\varphi 2+\varphi 3$$

The path BE and BCDE are in parallel and hence form a parallel magnetic circuit. In a parallel magnetic circuit, the MMF required for the whole parallel magnetic circuit is equal to MMF required for any one of the parallel paths.

Let

$$S1=ReluctanceofmagneticpathABEF$$

$$S2=Reluctanceofmagneticpath$$

$$S3=ReluctanceofmagneticpathBCDE$$

Therefore,

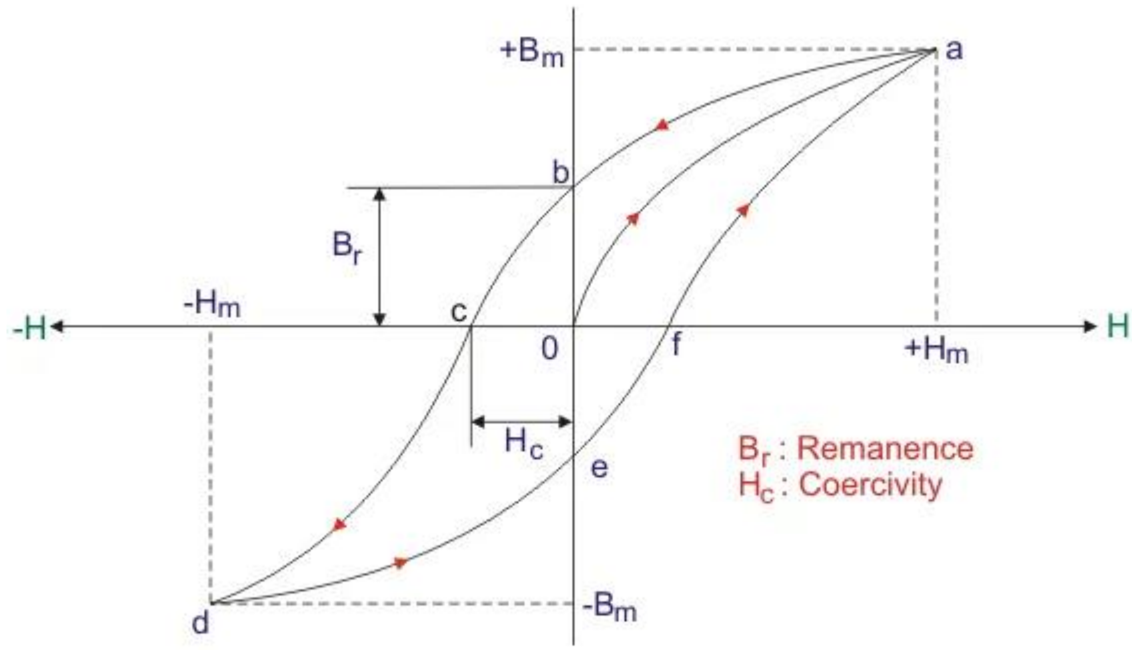
$$TotalMMF=MMFforpathABEF+MMFforpathBEorBCDE$$

$$\Longrightarrow TotalMMF=\varphi 1S1+\psi 2S2+\varphi 3S3$$

3 . 7 Hysteresis loop

A hysteresis loop (also known as a hysteresis curve) is a four-quadrant graph that shows the relationship between the induced magnetic flux density B and the magnetizing force H. It is often referred to as the B-H loop. From hysteresis loops, we can determine a number of magnetic properties about a material. Such as the retentivity, residual magnetism (or residual flux), coercive force, permeability, and the reluctance.

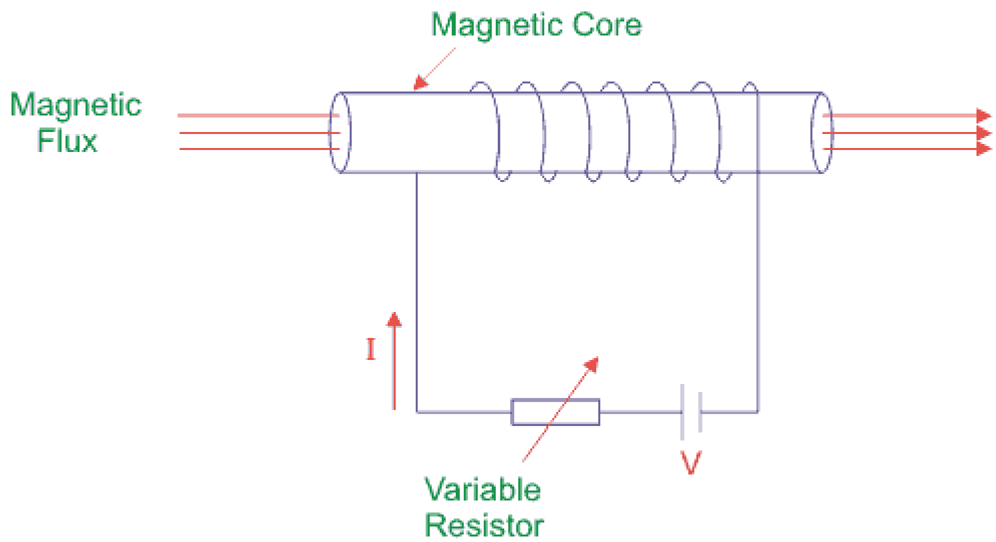
An example hysteresis loop is shown below.



To understand a hysteresis loop, let's suppose we take a magnetic material to use as a core around which insulated wire is wound.

The coils are connected to a DC supply through a variable resistor to vary the current "I". We know that current I is directly proportional to the value of magnetizing force (H) as

Where N = number of turns of coil and l is the effective length of the coil. The magnetic flux density of this core is B which is directly proportional to magnetizing force H.



Now, we should be familiar with some important terms related to a **hysteresis loop**.

Definition of Hysteresis

Hysteresis of a magnetic material is a property by virtue of which the flux density (B) of this material lags behind the magnetizing force (H).

Short questions:

1.Define magnetic reluctance?

Magnetic reluctance (also known as reluctance, magnetic resistance, or a magnetic insulator) is defined as the opposition offered by a magnetic circuit to the production of magnetic flux. It is the property of the material that opposes the creation of magnetic flux in a magnetic circuit.

2.Mention some magnetic materials?

- iron. Iron is an extremely well-known ferromagnetic metal. ...
- Nickel. Nickel is another popular magnetic metal with ferromagnetic properties. ...
- Cobalt. Cobalt is an important ferromagnetic metal. ...
- Steel. ...
- Stainless Steel. ...
- Rare Earth Metals. ...
- Aluminium. ...
- Gold.

3. Why An air gap is usually inserted in magnetic circuits?
   An air gap is usually inserted in magnetic circuits to prevent saturation
4. Why hystresis loss takes place in amagnetic material?
   Ans-Hystresis loss takes place in a magnetic material due to its high retentivity

Long questions:

1.Explain B_H curve?

2.Explain hystresis loop?

3.Explain difference between electric circuit and magnetic circuit?

COUPLED CIRCUITS

CHAPTER-4

4 . 1 Self Inductance and Mutual Inductance

An electric circuit is said to be a **coupled circuit**, when there exists a mutual inductance between the coils (or inductors) present in that circuit. Coil is nothing but the series combination of resistor and inductor. In the absence of resistor, coil becomes inductor. Sometimes, the terms coil and inductor are interchangeably used.

- ***Self-inductance:*** Self inductance is defined as the phenomenon in which a change in electric current in a circuit produces an induced electro-motive-force in the same circuit.
- ***Self inductance unit:*** The self-inductance of a coil is said to be one henry if a current change of one ampere per second through a circuit produces an electro-motive force of one volt in the circuit.

- When current passes along a wire, and especially when it passes through a coil or inductor, a magnetic field is induced. This extends outwards from the wire or inductor and could couple with other circuits. However it also couples with the circuit from which it is set up.

- The magnetic field can be envisaged as concentric loops of magnetic flux that surround the wire, and larger ones that join up with others from other loops of the coil enabling self-coupling within the coil.

- When the current in the coil changes, this causes a voltage to be induced the different loops of the coil - the result of self-inductance.

Self induction

- In terms of quantifying the effect of the inductance, the basic formula below quantifies the effect.

  - VL=−NdϕdtVL=-Ndϕdt

- Where:
  VL= induced voltage in volts
  N= number of turns in the coil
  dφ/dt = rate of change of magnetic flux in webers / second

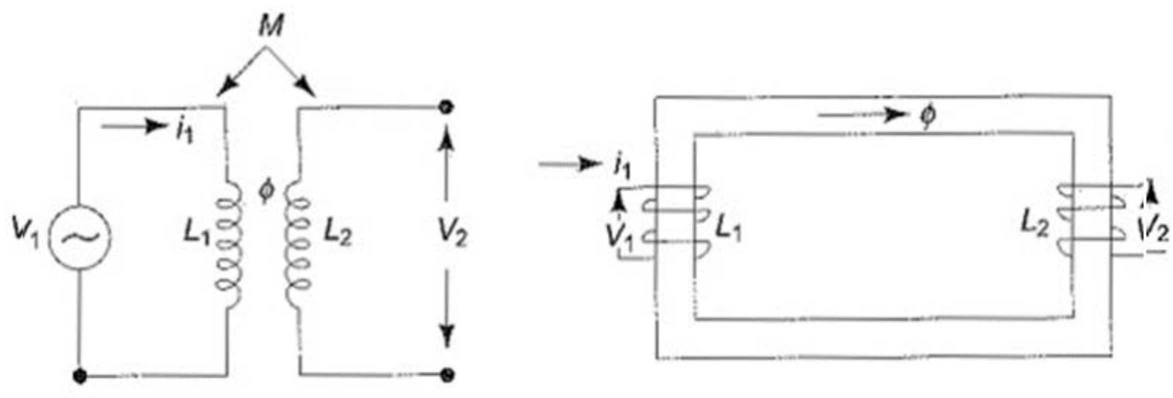
- The induced voltage in an inductor may also be expressed in terms of the inductance (in henries) and the rate of change of current.

  - VL=−LdidtVL=-Ldidt

-

- Self induction is the way in which single coils and chokes operate. A choke is used in radio frequency circuits because it opposes any change, i.e. the radio frequency signal, but allows any steady, i.e. DC current to flow.

- Mutual Inductance of Coupled Circuits – A voltage is induced in a coil when there is a time rate of change of current through it. The inductance parameter L, is defined in terms of the voltage across it and the time rate of change of current through it υ(t) = L di(t)/dt, where υ(t) is the voltage across the coil, I(t) is the current through the coil and L is the inductance of the coil. Strictly speaking, this definition is of self-inductance and this is considered as a circuit element with a pair of terminals. Whereas a circuit element “mutual inductor” does not exist.
- Mutual inductance is a property associated with two or more coils or inductors which are in close proximity and the presence of common magnetic flux which links the coils. A transformer is such a device whose operation is based on mutual inductance.



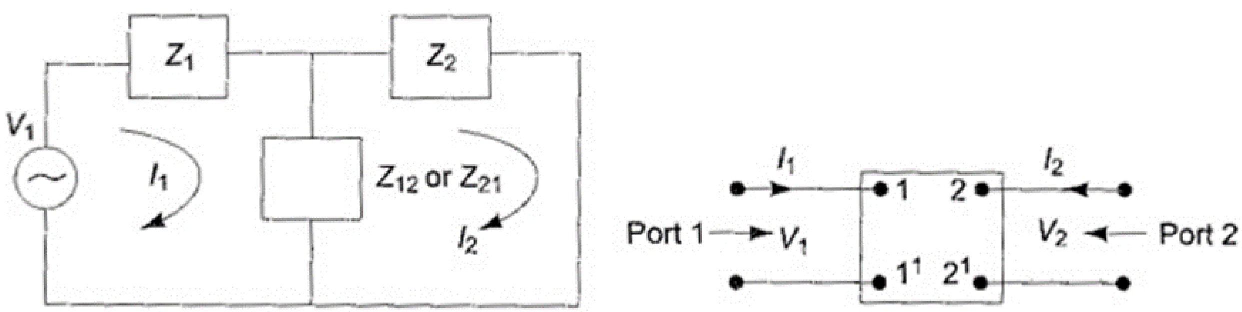
(a) (b)

4.2 Conductivity Coupled Circuit and Mutual Impedance:

A conductively Coupled Circuits which does not involve magnetic coupling is shown in Fig. .a

In the circuit shown the impedance Z_{12} or Z_{21} common to loop 1 and loop 2 is called **mutual impedance**. It may consists of a pure resistance, a pure inductance, a pure capacitance or a combination of any of these elements.

The general definition of mutual impedance is explained with the help of Fig. (b)



The network in the box may be of any configuration of circuit elements with two ports having two pairs of terminals 1-1′ and 2-2′ available for measurement. The mutual impedance between port 1 and 2 can be measured at 1-1′ or 2-2′. If it is measured at 2-2′.

It can be defined as the voltage developed (V_2) at 2-2′ per unit current (I_1) at port 1-1′. If the box contains linear bilateral elements, then the mutual impedance measured at 2-2′ is same as the impedance measured at 1 -1′ and is defined as the voltage developed (V_1) at 1-1′ per unit current (I_2) at port 2-2′.

4.3.Dot Convention in Coupled Circuits:

Dot Convention in Coupled Circuits is used to establish the choice of correct sign for the mutually induced voltages in coupled circuits.

Circular dot marks and/or special symbols are placed at one end of each of two coils which are mutually coupled to simplify the diagrammatic representation of the windings around its core.

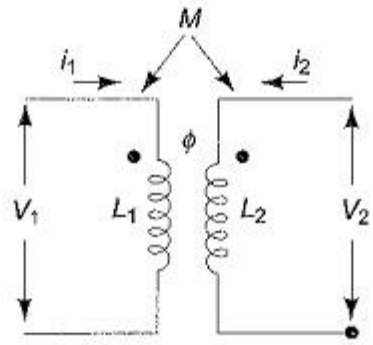


Fig. 10.5

Let us consider Fig. 10.5, which shows a pair of linear, time invariant, coupled inductors with self inductances L_1 and L_2 and a mutual inductance M. If these inductions form a portion of a network, currents i_1 and i_2 are shown, each arbitrarily assumed entering at the dotted terminals, and voltages υ_1 and υ_2 are developed across the inductors. The voltage across L_1 is, thus composed of two parts and is given by

$$v_1(t) = L_1 \frac{di_1(t)}{dt} \pm M \frac{di_2(t)}{dt}$$

The first term on the RHS of the above equation is the self induced voltage due to i_1, and the second term represents the mutually induced voltage due to i_2.

Similarly,

$$v_2(t) = L_2 \frac{di_2(t)}{dt} \pm M \frac{di_1(t)}{dt}$$

Although the self-induced voltages are designated with positive sign, mutually induced voltages can be either positive or negative depending on the direction of the winding of the coil and can be decided by the presence of the dots placed at one end of each of the two coils. The convention is as follows.

If two terminals belonging to different coils in a coupled circuit are marked identically with dots then for the same direction of current relative to like terminals, the magnetic flux of self and mutual induction in each coil add together. The physical basis of the Dot Convention in Coupled Circuits can be verified by examining Fig. 10.6. Two coils ab and cd are shown wound on a common iron core.

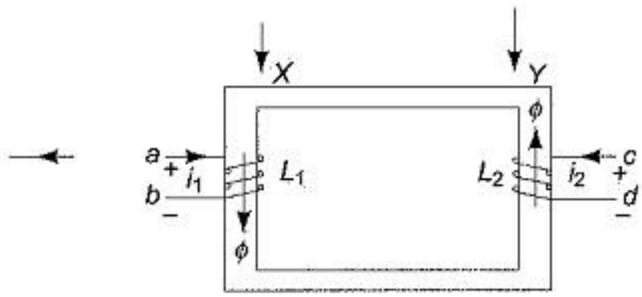


Fig. 10.6

It is evident from Fig.10.6 that the direction of the winding of the coil ab is clock-wise around the core as viewed at X, and that of cd is anti-clockwise as viewed at Y. Let the direction of current ii in the first coil be from a to b, and increasing with time. The flux produced by i_1 in the core has a direction which may be found by right hand rule, and which is downwards in the left limb of the core. The flux also increases with time in the direction shown at X. Now suppose that the current i_2 in the second coil is from c to d, and increasing with time.

The application of the right hand rule indicates that the flux produced by i_2 in the core has an upward direction in the right limb of the core. The flux also increases with time in the direction shown at Y. The assumed currents i_1 and i_2 produce flux in the core that are additive. The terminals a and c of the two coils attain similar polarities simultaneously. The two simultaneously positive terminals are identified by two dots by the side of the terminals as shown in Fig. 10.7.

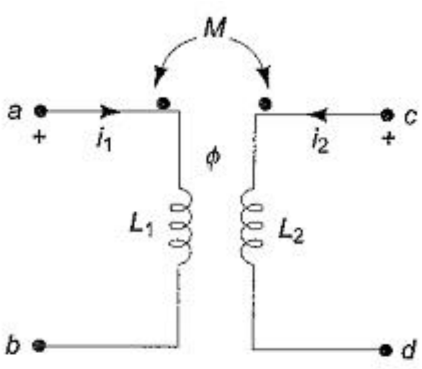


Fig. 10.7

The other possible location of the dots is the other ends of the coil to get additive fluxes in the core, i.e. at b and d. It can be concluded that the mutually induced voltage is positive when currents i_1 and i_2 both enter (or leave) the windings by the dotted terminals. If the current in one winding enters at the dot-marked terminals and the current in the other winding leaves at the dot-marked terminal, the voltages due to self and mutual induction in any coil have opposite signs.

4.4 Coefficient of coupling

The amount of coupling between the inductively coupled coils is expressed in terms of the coefficient of coupling, which is defined as

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

where

- M = mutual inductance between the coils
- L_1 = self inductance of the first coil, and
- L_2 = self inductance of the second coil

Coefficient of coupling is always less than unity, and has a maximum value of 1 (or 100%). This case, for which K = 1, is called perfect coupling, when the entire flux of one coil links the other. The greater the coefficient of coupling between the two coils, the greater the mutual inductance between them, and vice-versa. It can be expressed as the fraction of the magnetic flux produced by the current in one coil that links the other coil.

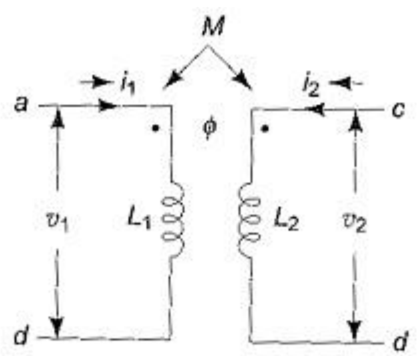


Fig. 10.10

r a pair of mutually coupled circuits shown in Fig. 10.10, let us assume initially that i_1, i_2 are zero at t = 0.

$$v_1(t) = L_1 \frac{di_1(t)}{dt} + M \frac{di_2(t)}{dt}$$

$$v_2(t) = L_2 \frac{di_2(t)}{dt} + M \frac{di_1(t)}{dt}$$

Initial energy in the coupled circuit at t = 0 is also zero. The net energy input to the system shown in Fig. 10.10 at time t is given by

$$W(t) = \int_0^t [v_1(t)\, i_1(t) + v_2(t)\, i_2(t)]\, dt$$

Substituting the values of $\upsilon_1(t)$ and $\upsilon_2(t)$ in the above equation yields

$$W(t) = \int_0^t \left[L_1 i_1(t) \frac{di_1(t)}{dt} + L_2 i_2(t) \frac{di_2(t)}{dt} + M(i_1(t)) \frac{di_2(t)}{dt} + i_2(t) \frac{di_1(t)}{dt} \right] dt$$

From which we get

$$W(t) = \frac{1}{2} L_1[i_1(t)]^2 + \frac{1}{2} L_2[i_2(t)]^2 + M[i_1(t)i_2(t)]$$

If one current enters a dot-marked terminal while the other leaves a dot marked terminal, the above equation becomes

$$W(t) = \frac{1}{2} L_1[i_1(t)]^2 + \frac{1}{2} L_2[i_2(t)]^2 - M[i_1(t)i_2(t)]$$

According to the definition of passivity, the net electrical energy input to the system is non-negative. W(t) represents the energy stored within a passive network, it cannot be negative.

$$W(t) \geq 0 \text{ for all values of } i_1,\, i_2;\, L_1,\, L_2 \text{ or } M$$

The statement can be proved in the following way. If i_1 and i_2 are both positive or negative, W(t) is positive. The other condition where the energy equation could be negative is

$$W(t) = \frac{1}{2} L_1[i_1(t)]^2 + \frac{1}{2} L_2[i_2(t)]^2 - M[i_1(t)\, i_2(t)]$$

The above equation can be rearranged as

$$W(t) = \frac{1}{2}\left(\sqrt{L_1} i_1 - \frac{M}{\sqrt{L_1}} i_2 \right)^2 + \frac{1}{2}\left(L_2 - \frac{M^2}{L_1} \right) i_2^2$$

The first term in the parenthesis of the right side of the above equation is positive for all values of i_1 and i_2, and, thus, the last term cannot be negative; hence

$$L_2 - \frac{M^2}{L_1} \geq 0$$

$$\frac{L_1 L_2 - M^2}{L_1} \geq 0$$

$$L_1 L_2 - M^2 \geq 0$$

$$\sqrt{L_1 L_2} \geq M$$

$$M \leq \sqrt{L_1 L_2}$$

Obviously the maximum value of the mutual inductance is $\sqrt{L_1 L_2}$. Thus, we define the coefficient of coupling for the coupled circuit as

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

The coefficient, K, is a non negative number and is independent of the reference directions of the currents in the coils. If the two coils are a great distance apart in space, the mutual inductance is very small, and K is also very small. For iron-core coupled circuits, the value of K may be as high as 0.99, for air-core coupled circuits, K varies between 0.4 to 0.8.

4.5.**Series Connection of Coupled Inductors**:

Let there be two inductors connected in series, with self inductances L_1 and L_2 and mutual inductance of M. Two kinds of Series Connection of Coupled Inductors are possible; series aiding as in Fig. 10.19(a), and series opposition as in Fig. 10.19(b).

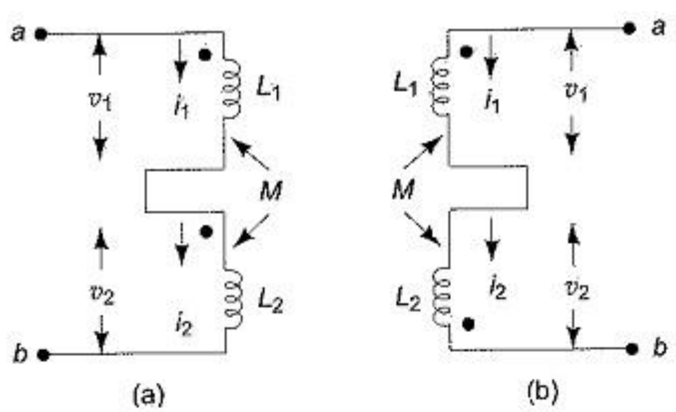


Fig. 10.19

In the case of series aiding connection, the currents in both inductors at any instant of time are in the same direction relative to like terminals as shown in Fig. 10.19(a). For this reason, the magnetic fluxes of self induction and of mutual induction linking with each element add together.

In the case of series opposition connection, the currents in the two inductors at any instant of time are in opposite direction relative to like terminals as shown in Fig. 10.19(b). The inductance of an element is given by L = Φ/i, where Φ is the flux produced by the inductor.

where

$$\phi_1 = L_1 i_1 + M i_2$$
$$\phi_2 = L_2 i_2 + M i_1$$
$$\phi = Li = L_1 i_1 + M i_2 + L_2 i_2 + M i_1$$

Since

$$i_1 = i_2 = i$$
$$L = L_1 + L_2 + 2M$$

Similarly, for the series opposition

$$\phi = \phi_1 + \phi_2$$

where

$$\phi_1 = L_1 i_1 - M i_2$$
$$\phi_2 = L_2 i_2 - M i_1$$
$$\phi = Li = L_1 i_1 - M i_2 + L_2 i_2 - M i_1$$

Since

$$i_1 = i_2 = i$$
$$L = L_1 + L_2 - 2M$$

In general, the inductance of Series Connection of Coupled Inductors is given by $L = L_1 + L_2 \pm 2M$. Positive sign is applied to the series aiding connection, and negative sign to the series opposition

Parallel Connection of Coupled Coils:

Parallel Connection of Coupled Coils – Consider two inductors with self inductances L_1 and L_2 connected parallel which are mutually coupled with mutual inductance M as shown in Fig. 10.20.

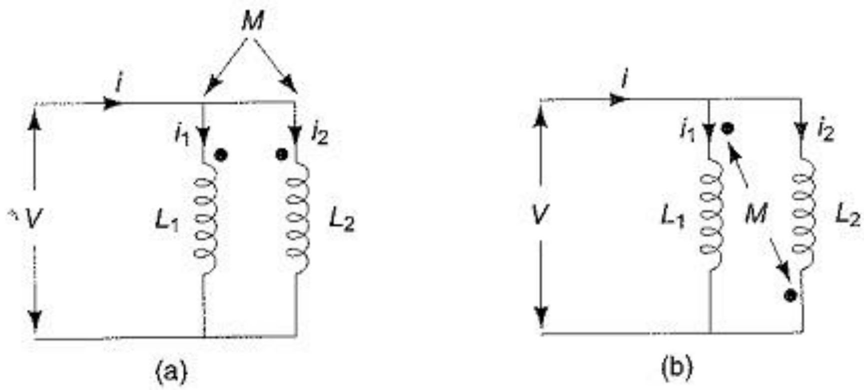


Fig. 10.20

Let us consider Fig. 10.20(a) where the self induced emf n each coil assists the mutually induced emf as shown by the dot convention.

$$i = i_1 + i_2$$

$$\frac{di}{dt} = \frac{di_1}{dt} + \frac{di_2}{dt} \qquad (10.3)$$

The voltage across the parallel branch is given by

$$v = L_1 \frac{di_1}{dt} + M\frac{di_2}{dt} \text{ or } L_2 \frac{di_2}{dt} + M\frac{di_1}{dt}$$

The voltage across the parallel branch is given by

$$v = L_1 \frac{di_1}{dt} + M\frac{di_2}{dt} \text{ or } L_2 \frac{di_2}{dt} + M\frac{di_1}{dt}$$

also

$$L_1 \frac{di_1}{dt} + M\frac{di_2}{dt} = L_2 \frac{di_2}{dt} + M\frac{di_1}{dt}$$

$$\frac{di_1}{dt} (L_1 - M) = \frac{di_2}{dt} (L_2 - M)$$

$$\frac{di_1}{dt} = \frac{di_2}{dt} \frac{(L_2 - M)}{(L_1 - M)} \qquad (10.4)$$

Substituting Eq. 10.4 in Eq. 10.3, we get

$$\frac{di}{dt} = \frac{di_2}{dt}\frac{(L_2 - M)}{(L_1 - M)} + \frac{di_2}{dt} = \frac{di_2}{dt}\left[\frac{(L_2 - M)}{L_1 - M} + 1\right] \qquad (10.5)$$

If L_{eq} is the equivalent inductance of the Parallel Connection of Coupled Coils in Fig 10.20 (a) then υ is given by

$$v = L_{eq}\frac{di}{dt}$$

$$L_{eq}\frac{di}{dt} = L_1\frac{di_1}{dt} + M\frac{di_2}{dt}$$

$$\frac{di}{dt} = \frac{1}{L_{eq}}\left[L_1\frac{di_1}{dt} + M\frac{di_2}{dt}\right]$$

Substituting Eq. 10.4 in the above equation we get

$$\frac{di}{dt} = \frac{1}{L_{eq}}\left[L_1\frac{di_2(L_2 - M)}{dt(L_2 - M)} + M\frac{di_2}{dt}\right]$$

$$= \frac{1}{L_{eq}}\left[L_1\frac{(L_2 - M)}{(L_1 - M)} + M\right]\frac{di_2}{dt} \qquad (10.6)$$

Equating Eq. 10.6 and Eq. 10.5, we get

$$\frac{L_2 - M}{L_2 - M} + 1 = \frac{1}{L_{eq}}\left[L_1\left(\frac{L_2 - M}{L_1 - M}\right) + M\right]$$

Rearranging and simplifying the above equation results in

$$L_{eq} = \frac{L_1 L_2 - M^2}{L_1 + L_2 - 2M}$$

If the voltage induced due to mutual inductance oppose the self induced emf in each coil as shown by the dot convention in Fig. 10.20(b), the equivalent inductance is given by

$$L_{eq} = \frac{L_1 L_2 - M^2}{L_1 + L_2 - 2M}$$

4 . 6 Solve numerical problems

For the three coupled coils in Fig. 13.72, calculate the total inductance.

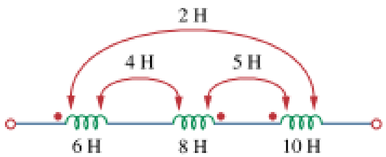


Figure 13.72
For Prob. 13.1.

Chapter 13, Solution 1.

For coil 1, $L_1 - M_{12} + M_{13} = 6 - 4 + 2 = 4$

For coil 2, $L_2 - M_{21} - M_{23} = 8 - 4 - 5 = -1$

For coil 3, $L_3 + M_{31} - M_{32} = 10 + 2 - 5 = 7$

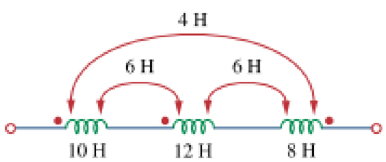
$L_T = 4 - 1 + 7 = 10\text{H}$

or $L_T = L_1 + L_2 + L_3 - 2M_{12} - 2M_{23} + 2M_{12}$

$L_T = 6 + 8 + 10 = \underline{\mathbf{10H}}$

Chapter 13, Problem 2.

Determine the inductance of the three series-connected inductors of Fig. 13.73.



Solution 2. L = L1 + L2 + L3 + 2M12 – 2M23 –2M31

= 10 + 12 +8 + 2x6 – 2x6 –2x4 = 22H

, Problem 3. Two coils connected in series-aiding fashion have a total inductance of 250 mH. When connected in a series-opposing configuration, the coils have a total inductance of 150 mH. If the inductance of one coil (L1) is three times the other,

find L1, L2, and M. What is the coupling coefficient?

, Solution 3. L1 + L2 + 2M = 250 mH (1)

L1 + L2 – 2M = 150 mH (2)

Adding (1) and (2), 2L1 + 2L2 = 400 mH

But, L1 = 3L2,, or 8L2 + 400, and L2 = 50 mH L1 = 3L2 = 150 mH

From (2), 150 + 50 – 2M = 150 leads to M = 25 mH

k = M/ L L 25 / 50x150 1 2 = = 0.2887

short questions

1.what is dot convention

Dot convention is a technique, which gives the details about voltage polarity at the dotted terminal. This information is useful, while writing KVL equations.

- If the current enters at the dotted terminal of one coil (or inductor), then it induces a voltage at another coil (or inductor), which is having **positive polarity** at the dotted terminal.
- If the current leaves from the dotted terminal of one coil (or inductor), then it induces a voltage at another coil (or inductor), which is having **negative polarity** at the dotted terminal.

2.whai is self inductance

Self inductance is an effect that is noticed when a single coil experiences the effect of inductance.

Under the effects of self inductance and changes in current induce an EMF or electro-motive force in that same wire or coil, producing what is often termed a back-EMF.

As the effect is noticed in the same wire or coil that generated the magnetic field, the effect is known as self inductance.

3.what is mutual inductance

When two coils are brought in proximity with each other the magnetic field in one of the coils tend to link with the other. This further leads to the generation of voltage in the second coil. This property of a coil which affects or changes the current and voltage in a secondary coil is called mutual inductance.

Long questions

1.what is series connection of coupled circuit?

2.explain parallel connection coupled circuit?

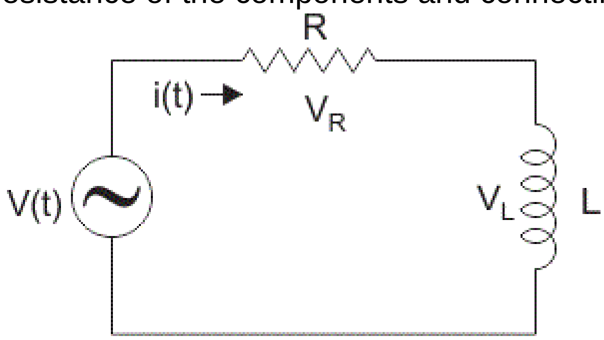
AC CIRCUIT AND RESONANCE:

CHAPTER-5

5.1 A.C. through R-L, R-C & R-L-C Circuit

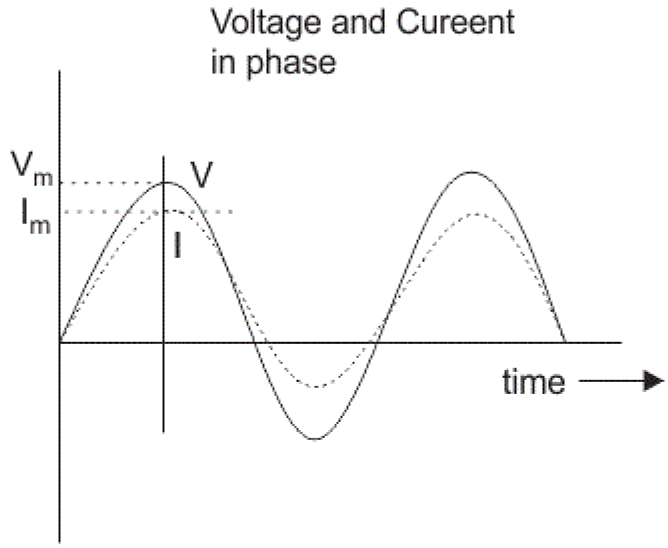
An RL circuit (also known as an RL filter or RL network) is defined as an electrical circuit consisting of the passive circuit elements of a resistor (R) and an inductor (L) connected together, driven by a voltage source or current source.

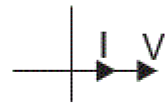
Due to the presence of a resistor in the ideal form of the circuit, an RL circuit will consume energy, akin to an RC circuit or RLC circuit.
This is unlike the ideal form of an LC circuit, which will consume no energy due to the absence of a resistor. Although this is only in the ideal form of the circuit, and in practice, even an LC circuit will consume some energy because of the non-zero resistance of the components and connecting wires.



Consider a simple RL circuit in which resistor, R and inductor, L are connected in series with a voltage supply of V volts. Let us think the current flowing in the circuit is I (amp) and current through resistor and inductor is I_R and I_L respectively. Since both resistance and inductor are connected in series, so the current in both the elements and the circuit remains the same. i.e $I_R = I_L = I$. Let V_R and V_l be the voltage drop across resistor and inductor.
Before drawing the **phasor diagram of series RL circuit**, one should know the relationship between voltage and current in case of resistor and inductor.

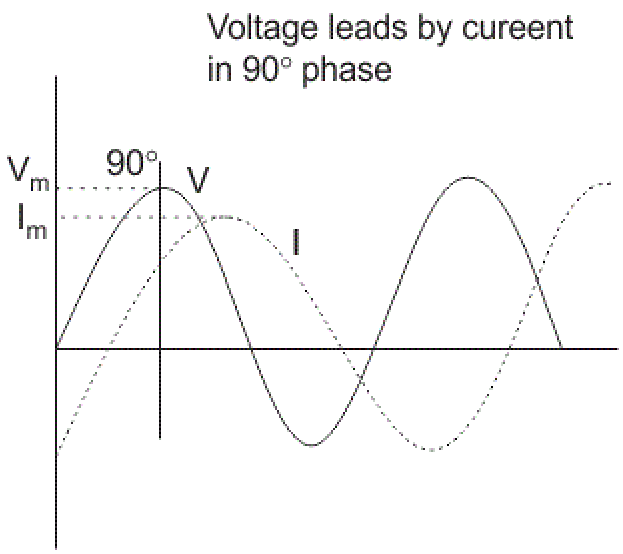
1. Resistor
   In case of resistor, the voltage and the current are in same phase or we can say that the phase angle difference between voltage and current is zero.

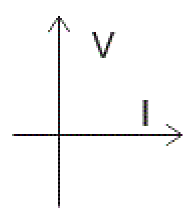




2Inductor

In inductor, the voltage and the current are not in phase. The voltage leads that of current by 90^o or in other words, voltage attains its maximum and zero value 90^o before the current attains it.





RL Circuit

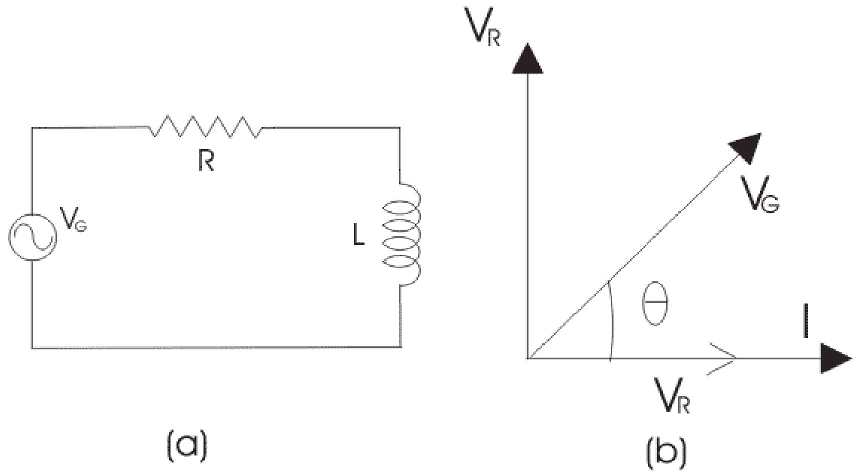
For drawing the phasor diagram of series RL circuit; follow the following steps:

Step- I. In case of series RL circuit, resistor and inductor are connected in series, so current flowing in both the elements are same i.e $I_R = I_L = I$. So, take current phasor as reference and draw it on horizontal axis as shown in diagram.

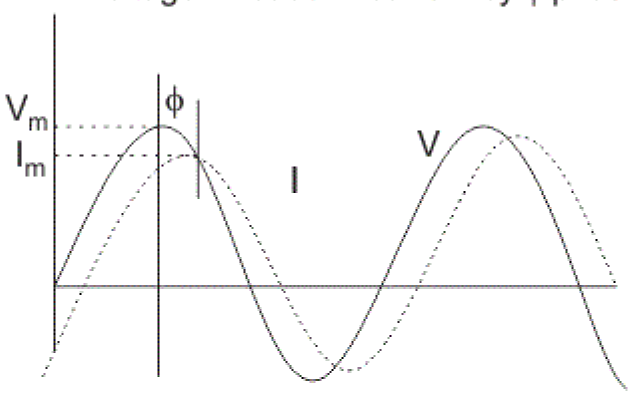
Step- II. In case of resistor, both voltage and current are in same phase. So draw the voltage phasor, V_R along same axis or direction as that of current phasor. i.e V_R is in phase with I.

Step- III. We know that in inductor, voltage leads current by 90^o, so draw V_L (voltage drop across inductor) perpendicular to current phasor.
Step- IV. Now we have two voltages V_R and V_L. Draw the resultant vector(V_G) of these two voltages. Such as,

and from right angle triangle we get, phase angle



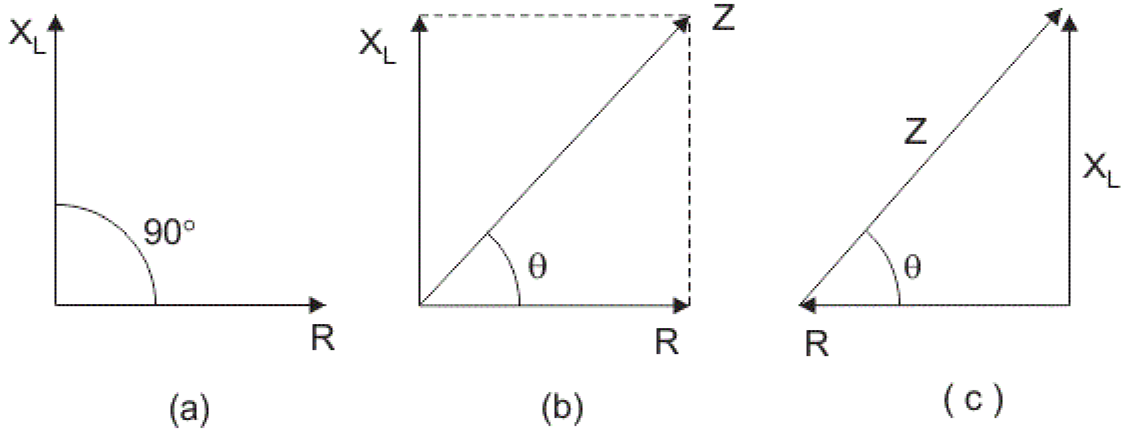
Voltage in leads in current by ϕ phase



CONCLUSION: In case of pure resistive circuit, the phase angle between voltage and current is zero and in case of pure inductive circuit, phase angle is 90^o but when we combine both resistance and inductor, the phase angle of a series RL circuit is between 0^o to 90^o.

Applying Kirchhoff voltage law (i.e sum of voltage drop must be equal to apply voltage) to this circuit we get,
Impedance of Series RL Circuit
The impedance of series RL circuit opposes the flow of alternating current. The impedance of series RL Circuit is nothing but the combine effect of resistance (R) and inductive reactance (X_L) of the circuit as a whole. The impedance Z in ohms is given by,

$Z = (R^2 + X_L^2)^{0.5}$ and from right angle triangle, phase angle $\theta = \tan^{-1}(X_L/R)$.



RC Circuit

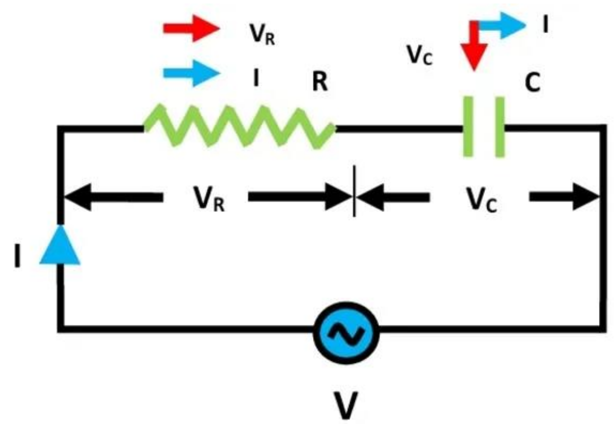
An RC circuit (also known as an RC filter or RC network) stands for a resistor-capacitor circuit. An RC circuit is defined as an electrical circuit composed of the passive circuit components of a resistor (R) and capacitor (C), driven by a voltage source or current source.

Due to the presence of a resistor in the ideal form of the circuit, an RC circuit will consume energy, akin to an RL circuit or RLC circuit.

This is unlike the ideal form of an LC circuit, which will consume no energy due to the absence of a resistor. Although this is only in the ideal form of the circuit, and in practice, even an LC circuit will consume some energy because of the non-zero resistance of the components and connecting wires.

Series RC Circuit

In an RC series circuit, a pure resistor having resistance R in ohms and a pure capacitor of capacitance C in Farads are connected in series.



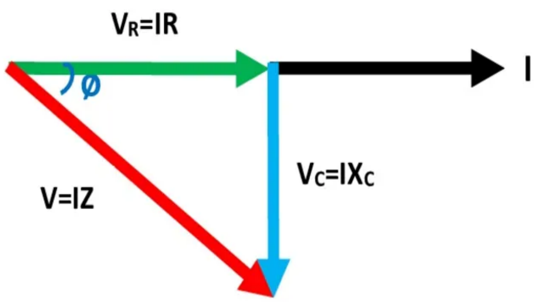
Here is the RMS value of the current in the circuit.

is the voltage across the resistor R.

is the voltage across the capacitor C.

is the RMS value of the supply voltage.

The figure shows a vector diagram of the series RC circuit.



Since in a series circuit current $'I'$ is the same so it is taken as a reference.

$V_R = IR$ is drawn in phase with current $'I'$ because in a pure resistor the voltage and current are in phase with each other.

$V_C = IX_C$ is drawn lagging with current $'I'$ by 90^0 because in a pure capacitor voltage and current are 90^0 out of each other i.e. voltage lags current by 90^0 or current leads the voltage by 90^0.

Now V is the vector sum of V_R and V_C.

$$therefore,\ V^2 = {V_R}^2 + {V_C}^2$$

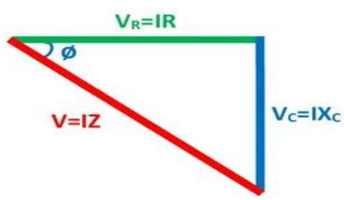
$$V = \sqrt{{V_R}^2 + {V_C}^2} \ = \sqrt{IR^2 + I{X_C}^2}$$
$$= I\sqrt{R^2 + {X_C}^2} \ = IZ$$

The impedance of an R-C series circuit is

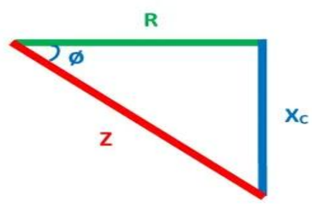
$$Z = \sqrt{R^2 + {X_C}^2}$$

$$where,\ X_C = \frac{1}{\omega C} = \frac{1}{2\pi f C}$$

The voltage and impedance triangle are shown in figure.



Voltage Triangle



Impedance Triangle

As seen, the vector V lags I by an angle ø where

$$tan\phi = \frac{IX_C}{IR}$$

$$\phi = tan^{-1}\frac{X_C}{R}$$

Thus in an R-C series circuit current $'I'$ leads the supply voltage $'V'$ by an angle

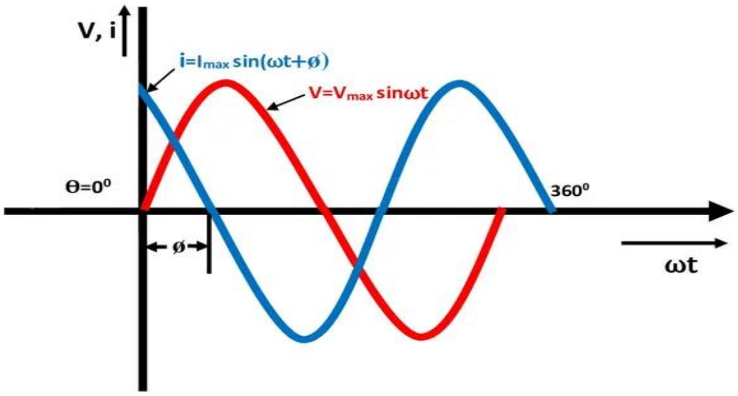
$$\phi = tan^{-1}\frac{X_C}{R}$$

$$i.e.\ \ if\ V = V_m sin\omega t$$

$$i = I_m sin(\omega t + \phi)$$

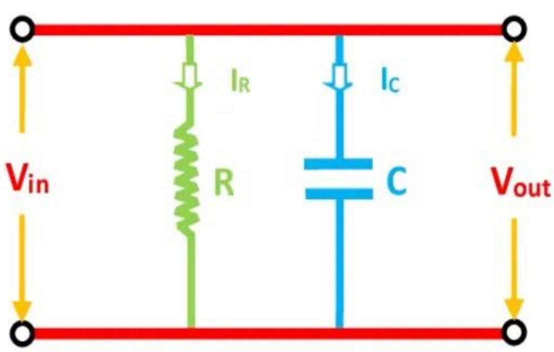
$$where,\ I_m = \frac{V_m}{Z}$$

The voltage and current waveforms of the R-C series circuit are shown in fig.



PARALLEL R-C CIRCUIT

In a parallel R-C circuit a pure resistor having resistance   in ohms and a pure capacitor of capacitance   in Farads are connected in parallel.



Voltage drops in a parallel RC circuit are the same hence the applied voltage is equal to the voltage across the resistor and voltage across the capacitor. Current in a parallel R-C circuit is the sum of the current through the resistor and capacitor.

$$V = V_R = V_C$$

$$I = I_R + I_C$$

For the resistor, current through it given by ohm's law:

$$I_R = \frac{V_{in}}{R}$$

The voltage-current relationship for the capacitor is:

$$I_C = C\frac{dV_{in}}{dt}$$

Applying KCL (Kirchhoff's Current Law) to parallel R-C circuit

$$I_R + I_C = 0$$

$$\frac{v}{R} + C\frac{dV}{dt} = 0$$

Above equation is the first-order differential equation of an R-C circuit.

In an **RLC circuit**,

the most fundamental elements of a resistor, inductor, and capacitor are connected across a voltage supply. All of these elements are linear and passive in nature. Passive components are ones that consume energy rather than producing it; linear elements are those which have a linear relationship between voltage and current.

There are number of ways of connecting these elements across voltage supply, but the most common method is to connect these elements either in series or in parallel. The **RLC circuit** exhibits the property of resonance in same way as LC circuit exhibits, but in this circuit the oscillation dies out quickly as compared to LC circuit due to the presence of resistor in the circuit.

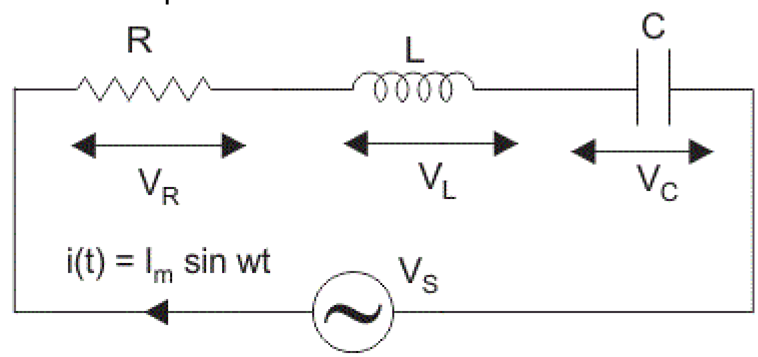
Series RLC Circuit

When a resistor, inductor and capacitor are connected in series with the voltage supply, the circuit so formed is called series RLC circuit.

Since all these components are connected in series, the current in each element remains the same,

$I_R = I_L = I_C = I(t)\ where\ I(t) = I_M \sin \omega t$

Let V_R be the voltage across resistor, R.
V_L be the voltage across inductor, L.
V_C be the voltage across capacitor, C.
X_L be the inductive reactance.
X_C be the capacitive reactance.



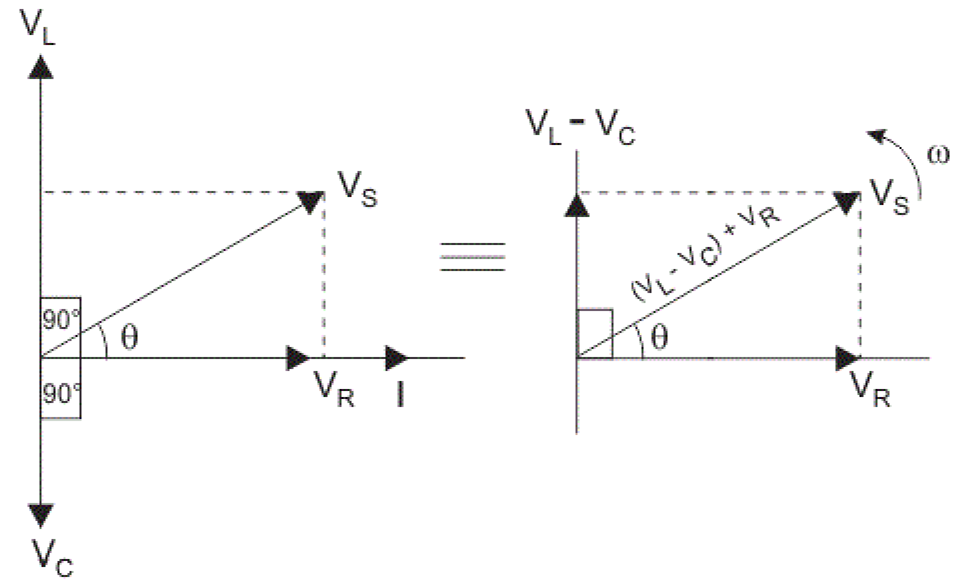
The total voltage in the RLC circuit is not equal to the algebraic sum of voltages across the resistor, the inductor, and the capacitor; but it is a vector sum because, in the case of the resistor the voltage is in-phase with the current, for inductor the voltage leads the current by 90^o and for capacitor, the voltage lags behind the current by 90^o

So, voltages in each component are not in phase with each other; so they cannot be added arithmetically. The figure below shows the phasor diagram of the series RLC circuit. For drawing the phasor diagram for RLC series circuit, the current is taken as reference because, in series circuit the current in each element remains the same and the corresponding voltage vectors for each component are drawn in reference to common current vector.

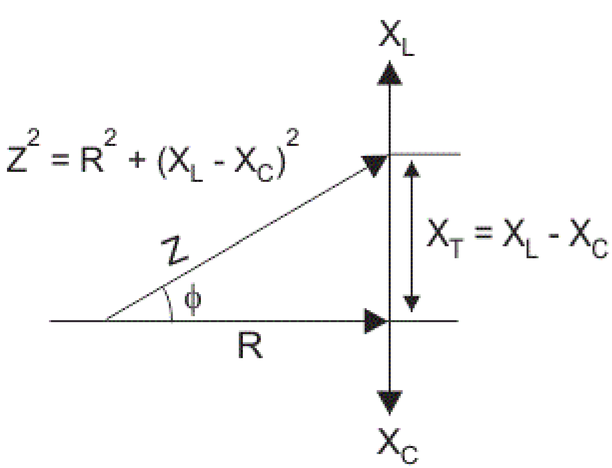
$$V_S^2 = V_R^2 + (V_L - V_C)^2 (if V_L > V_C)$$

$$V_S^2 = V_R^2 + (V_L - V_C)^2 (if V_L < V_C)$$

$$Where\ V_R = IR, V_L = IX_L, V_C = IX_C$$



The Impedance for a Series RLC Circuit



The impedance Z of a series RLC circuit is defined as opposition to the flow of current due circuit resistance R, inductive reactance, X_L and capacitive reactance, X_C. If the inductive reactance is greater than the capacitive reactance i.e $X_L > X_C$, then the RLC circuit has lagging phase angle and if the capacitive reactance is greater than the inductive reactance i.e $X_C > X_L$ then, the RLC circuit have leading phase angle and if both inductive and capacitive are same
i.e $X_L = X_C$ then circuit will behave as purely resistive circuit.

We know that

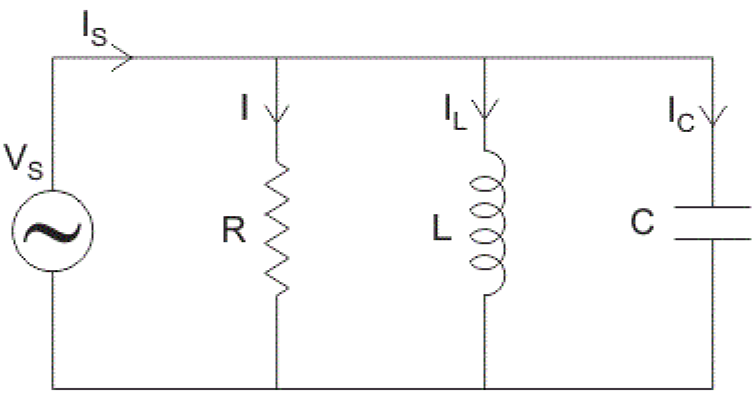
Where,

Substituting the values

$$V_S = I\sqrt{R^2 + (X_L - X_C)^2} \; or \; impedance \; Z = \sqrt{R^2 + (X_L - X_C)^2}$$

In parallel RLC Circuit the resistor, inductor and capacitor are connected in parallel across a voltage supply. The parallel RLC circuit is exactly opposite to the series RLC circuit. The applied voltage remains the same across all components and the supply current gets divided.

The total current drawn from the supply is not equal to mathematical sum of the current flowing in the individual component, but it is equal to its vector sum of all the currents, as the current flowing in resistor, inductor and capacitor are not in the same phase with each other; so they cannot be added arithmetically.



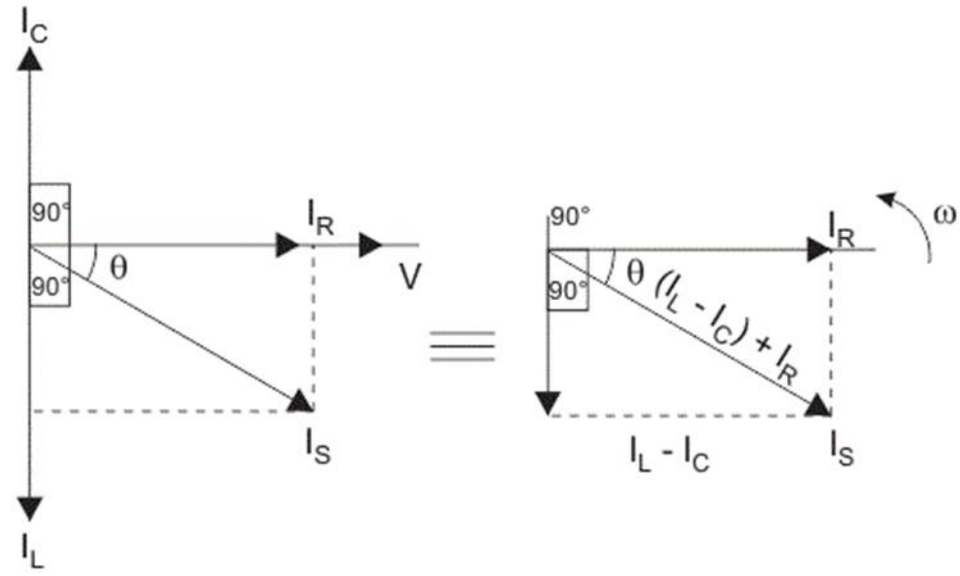
Phasor diagram of parallel RLC circuit, I_R is the current flowing in the resistor, R in amps.

I_C is the current flowing in the capacitor, C in amps.

I_L is the current flowing in the inductor, L in amps.

I_s is the supply current in amps.

In the parallel RLC circuit, all the components are connected in parallel; so the voltage across each element is same. Therefore, for drawing phasor diagram, take voltage as reference vector and all the other currents i.e I_R, I_C, I_L are drawn relative to this voltage vector. The current through each element can be found using Kirchhoff's Current Law, which states that the sum of currents entering a junction or node is equal to the sum of current leaving that node.



$$I_S^2 = I_R^2 + (I_L - I_C)^2$$

$$Now,\ I_R = \frac{V}{R}, I_C = \frac{V}{X_C}\ and\ I_L = \frac{V}{X_L}$$

$$I_S = \sqrt{\frac{V^2}{R^2} + \left(\frac{V}{X_L} - \frac{V}{X_C}\right)^2}$$

$$So,\ admitance,\ \frac{1}{Z} = \frac{I_S}{V} = Y = \sqrt{\frac{1}{R^2} + \left(\frac{1}{X_L} - \frac{1}{X_C}\right)^2}$$

As shown above in the equation of impedance, Z of a parallel RLC circuit; each element has reciprocal of impedance (1 / Z) i.e. admittance, Y. So in parallel RLC circuit, it is convenient to use admittance instead of impedance

5.2 Solution of problems of A.C. through R-L, R-C & R-L-C series Circuit by complex algebra method.

Problem: An AC series *RL* circuit is made up of a resistor that has a resistance value of 150 Ω and an inductor that has an inductive reactance value of 100 Ω. Calculate the impedance and the phase angle theta (θ) of the circuit.

Solution:

$$\begin{aligned} Z &= \sqrt{R^2 + X_L^2} \\ &= \sqrt{150^2 + 100^2} \\ &= \sqrt{32{,}500} \\ &= 180\ \Omega \end{aligned}$$

$$\begin{aligned} \theta &= \tan^{-1}\left(\frac{X_L}{R}\right) \\ &= \tan^{-1}\left(\frac{100}{150}\right) \\ &= \tan^{-1}(0.667) \\ &= 33.7^\circ \end{aligned}$$

or

$$\begin{aligned} Z &= R + jX_L \\ &= 150 + j100 \\ &= 180\ \Omega \angle 33.7^\circ \end{aligned}$$

Once the impedance of a circuit is found it is possible to find the current by using Ohm's law and substituting Z for R as follows:

$$I = \frac{E_T}{Z}$$

5.3. Solution of problems of A.C. through R-L, R-C & R-L-C parallel & Composite Circuit

$+ 172 \angle 38.43° = 220 + j\,0$ V]

EXAMPLE 3.2. *In a series R-L circuit, R = 10 Ω, L = 20 mH. The circuit current being 10 sin 314 t A, find the total voltage drop (v) across the elements R and L. Also obtain the phase angle between i and v.*

SOLUTION. The phase angle θ between v and i is given by

$$\theta = \tan^{-1}\frac{\omega L}{R} = \tan^{-1}\frac{2\pi \times 50 \times 20 \times 10^{-3}}{10} = 32.13°.$$

$[\because\ i = 10 \sin 314\,t \quad \therefore\ \omega = 314 = 2\pi f \text{ i.e., } f = 50 \text{ Hz}]$

$$v = I_m \sqrt{R^2 + (\omega L)^2} \sin(\omega t + \theta)$$
$$= 10\sqrt{10^2 + (314 \times 20 \times 10^{-3})^2} \sin(314\,t + \theta)$$
$$= 118.08 \sin(314\,t + 32.13°) \text{ V}$$

[here, i lags v by 32.13

EXAMPLE 3.3. *In a series R-L circuit, the current and voltage are given as i = 1 cos (314 t − 20°), v = 10 cos (314 t + 10°). Find the values of R and L.*

SOLUTION. Let v be the reference vector, i lags v by $(20 + 10) = 30°$.

Also, $\dfrac{V_m}{I_m} = \dfrac{10}{1} = 10.$

Hence, $\dfrac{10}{1} = \sqrt{R^2 + (314.L)^2}$ or, $R^2 + (314.L)^2 = 100$ ...(1)

and $\tan 30° = 0.577 = \dfrac{314\,L}{R}$ ...(2)

Solving (1) and (2), $L = 15.9$ **mH** ; $R = 8.66\ \Omega$

EXAMPLE 3.4. *In a series RC circuit, the values of R = 10 Ω and C = 25 nF. A sinusoidal voltage of 50 MHz is applied and the maximum voltage across the capacitance is 2.5 V. Find the maximum voltage across the series combination.*

SOLUTION. $\omega = 2\pi f = 2 \cdot \pi \cdot 50 \times 10^6$ rad/sec $= 314 \times 10^6$ rad/sec.

The current through the capacitor being given by

$I = \omega CV$, in this problem,

$$I_{max} = \omega C V_{max} = 314 \times 10^6 \times 25 \times 10^{-9} \times 2.5 = 19.625 \text{ A}$$

Again, $V_{max} = I_{max} Z = I_{max}\sqrt{R^2 + \left(\dfrac{1}{\omega C}\right)^2}$

$$= 19.625\sqrt{10^2 + \left(\frac{1}{2 \cdot \pi \cdot 50 \times 10^6 \times 25 \times 10^{-9}}\right)^2}$$

$$= \mathbf{196.26\ V}$$

5.4 Power factor & power triangle

Power Factor:

Power Factor is defined as the cosine of an angle between the current and voltage. Otherwise, the ratio between the real power to reactive power is called power factor.

$$Power\ factor = cos\theta$$

Unit: Since power factor is a ratio quantity, that's why it does not have any units.

In generally power is the ability to do the work. In the electrical domain, electrical power is the amount of electrical energy that can be transferred to some other form (heat, light etc.) per unit time.

Mathematically it is the product of voltage drop across the element and current flowing through the element is called power.

Considering first the DC circuits, having only DC voltage sources (any current source or voltage source), the inductors behave as short circuit and capacitors behave like an open circuit in steady state.

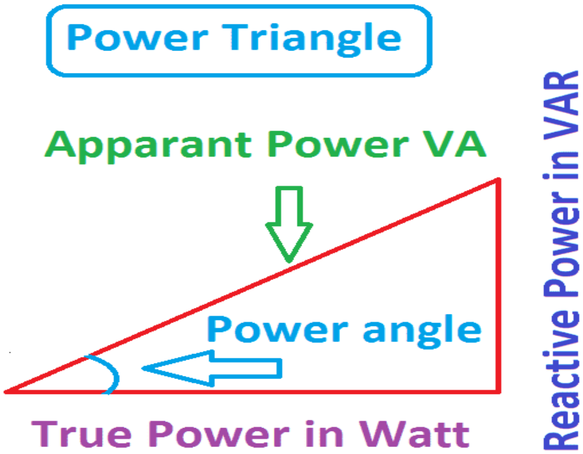
Hence the entire circuit behaves like a resistive circuit and the entire electrical power is dissipated in the form of heat. Here the voltage and current are in the same phase and the total electrical power is given by.

$$Power = Voltage * Current$$

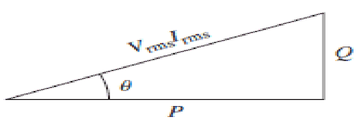
But in AC power system due to the alternating nature (the frequency), inductor and capacitor provide a certain impedance to the circuit. An inductive circuit store the energy in the form of a magnetic field.

The capacitor stores the energy in the form of the electric field. But in resister works same like DC circuits. Due to the inductive and capacitive action, the power factor presents.

Power Triangle:



The relationships between real power P, reactive power Q, apparent power VI and the power angle θ can be represented by the power triangle. The power triangle is shown in Figure for an inductive load and capacitive load in which case θ and Q are positive and Q are negative.



Generally, The power factor is equal to the real or true power P in watts (W) divided by the apparent power |S| in volt-ampere (VA) from the Power triangle:

$$PF = \frac{P(W)}{|S(VA)|}$$

$$PF - power\ factor.$$

$$P \quad - real\ power\ in\ watts\ (W).$$

$$|S| \quad - apparent\ power - the\ magnitude\ of\ the\ complex\ power\ in\ volt\ amps\ (VA).$$

Generally, Power factor tells about the phase relationship between the voltage and current. Under ideal conditions current and voltage are "in phase" and the power factor is "100%." If inductive loads (motors) are present, power factor less than 100% (typically 80 to 90%) can occur.

Low power factor, electrically speaking, causes heavier current to flow in power distribution lines in order to deliver a given number of kilowatts to an electrical load. Due to this, we have to spend unnecessary energy cost as power loss and installation cost as increased in equipment capacity.

.5.5 Expression for active, reactive, apparent power

Active Power

Definition: The power which is actually consumed or utilised in an AC Circuit is called True power or Active power or Real power. It is measured in kilowatt (kW) or MW. It is the actual outcomes of the electrical system which runs the electric circuits or load.

Reactive Power

Definition: The power which flows back and forth that means it moves in both the directions in the circuit or reacts upon itself, is called Reactive Power. The reactive power is measured in kilo volt-ampere reactive (kVAR) or MVAR.

Apparent Power

Definition: The product of root mean square (RMS) value of voltage and current is known as Apparent Power. This power is measured in kVA or MVA.

It has been seen that power is consumed only in resistance. A pure inductor and a pure capacitor do not consume any power since in a half cycle whatever power is received from the source by these components, the same power is returned to the source. This power which returns and flows in both the direction in the circuit, is called Reactive power. This reactive power does not perform any useful work in the circuit.

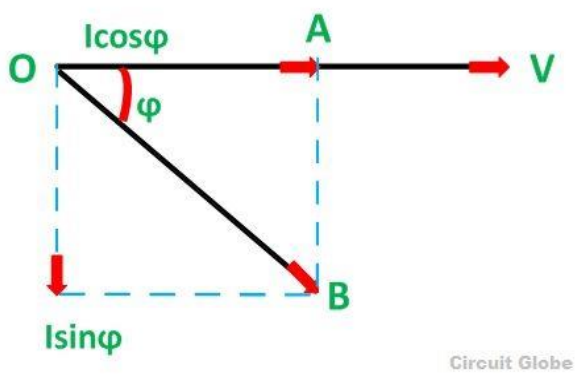
In a purely resistive circuit, the current is in phase with the applied voltage, whereas in a purely inductive and capacitive circuit the current is 90 degrees out of phase, i.e., if the inductive load is connected in the circuit the current lags voltage by 90 degrees and if the capacitive load is connected the current leads the voltage by 90 degrees.

Hence, from all the above discussion, it is concluded that the *current in phase with the voltage produces true or active power*, whereas, the *current 90 degrees out of phase with the voltage contributes to reactive power* in the circuit.

Therefore,

- True power = voltage x current in phase with the voltage
- Reactive power = voltage x current out of phase with the voltage

The phasor diagram for an inductive circuit is shown below:



aking voltage V as reference, the current I lags behind the voltage V by an angle ϕ. The current I is divided into two components:

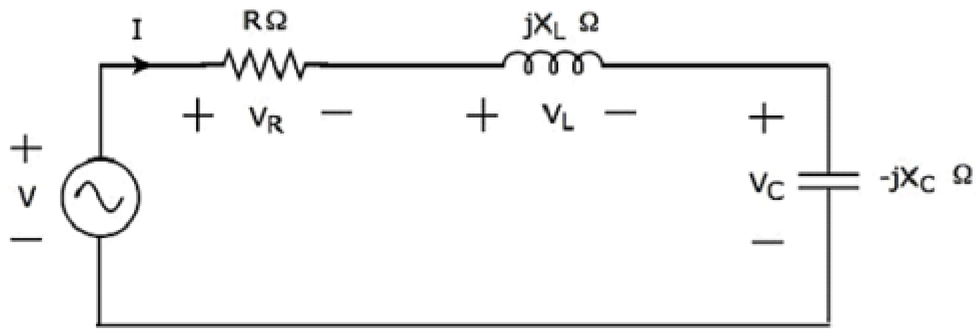
- I Cos ϕ in phase with the voltage V
- I Sin ϕ which is 90 degrees out of phase with the voltage V

Therefore, the following expression shown below gives the active, reactive and apparent power respectively.

- Active power P = V x I cosϕ = V I cosϕ
- Reactive power P_r or Q = V x I sinϕ = V I sinϕ
- Apparent power P_a or S = V x I = VI

- 5.6 Derive the resonant frequency of series resonance and parallel resonance circuit

- **Resonance** occurs in electric circuits due to the presence of energy storing elements like inductor and capacitor. It is the fundamental concept based on which, the radio and TV receivers are designed in such a way that they should be able to select only the desired station frequency.
- There are **two types** of resonances, namely series resonance and parallel resonance. These are classified based on the network elements that are connected in series or parallel. In this chapter, let us discuss about series resonance.
- ## Series Resonance Circuit Diagram
- If the resonance occurs in series RLC circuit, then it is called as Series Resonance. Consider the following series RLC circuit, which is represented in phasor domain.



-
- Here, the passive elements such as resistor, inductor and capacitor are connected in series. This entire combination is in series with the input sinusoidal voltage source.
- Apply KVL around the loop.

$$V - V_R - V_L - V_C = 0$$

$$\Rightarrow V - IR - I(jX_L) - I(-jX_C) = 0$$

$$\Rightarrow V = IR + I(jX_L) + I(-jX_C)$$

$$\Rightarrow V = I[R + j(X_L - X_C)] \quad \textbf{Equation 1}$$

The above equation is in the form of ***V* = *IZ***.

Therefore, the **impedance Z** of series RLC circuit will be

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

Resonant Frequency

The frequency at which resonance occurs is called as resonant frequency f_r. In series RLC circuit resonance occurs, when the imaginary term of impedance *Z* is zero, i.e., the value of XL−XC should be equal to zero.

$$\Rightarrow X_L = X_C$$

Substitute $X_L = 2\pi fL$ and $X_C = \frac{1}{2\pi fC}$ in the above equation.

$$2\pi fL = \frac{1}{2\pi fC}$$

$$\Rightarrow f^2 = \frac{1}{(2\pi)^2 LC}$$

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

Therefore, the **resonant frequency f_r** of series RLC circuit is

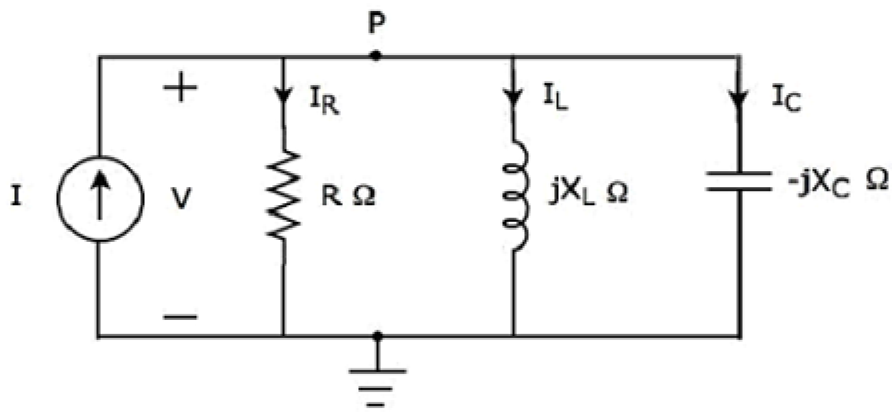
$$f_r = \frac{1}{(2\pi)\sqrt{LC}}$$

Where, *L* is the inductance of an inductor and *C* is the capacitance of a capacitor.

The resonant frequency f_r of series RLC circuit depends only on the inductance *L* and capacitance *C*. But, it is independent of resistance *R*.

Parallel Resonance Circuit Diagram

If the resonance occurs in parallel RLC circuit, then it is called as Parallel Resonance. Consider the following parallel RLC circuit, which is represented in phasor domain.



Here, the passive elements such as resistor, inductor and capacitor are connected in parallel. This entire combination is in parallel with the input sinusoidal current source.

Write nodal equation at node P.

Write **nodal equation** at node P.

$$-I + I_R + I_L + I_C = 0$$

$$\Rightarrow -I + \frac{V}{R} + \frac{V}{jX_L} + \frac{V}{-jX_C} = 0$$

$$\Rightarrow I = \frac{V}{R} - \frac{jV}{X_L} + \frac{jV}{X_C}$$

$$\Rightarrow I = V[\frac{1}{R} + j(\frac{1}{X_C} - \frac{1}{X_L})]$$ **Equation 1**

The above equation is in the form of ***I* = *VY***.

Therefore, the **admittance Y** of parallel RLC circuit will be

$$Y = \frac{1}{R} + j(\frac{1}{X_C} - \frac{1}{X_L})$$

Resonant Frequency

We know that the resonant frequency, f_r is the frequency at which, resonance occurs. In parallel RLC circuit resonance occurs, when the imaginary term of admittance, Y is zero. i.e., the value of the value of $\frac{1}{X_C} - \frac{1}{X_L}$ should be equal to zero

$$\Rightarrow \frac{1}{X_C} = \frac{1}{X_L}$$

$$\Rightarrow X_L = X_C$$

The above resonance condition is same as that of series RLC circuit. So, the **resonant frequency, f_r** will be same in both series RLC circuit and parallel RLC circuit.

Therefore, the **resonant frequency, f_r** of parallel RLC circuit is

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

Where,

- L is the inductance of an inductor.
- C is the capacitance of a capacitor.

The **resonant frequency, f_r** of parallel RLC circuit depends only on the inductance ***L*** and capacitance ***C***. But, it is independent of resistance ***R***.

5.7 Define Bandwidth, Selectivity & Q-factor in series circuit.

Bandwidth of RLC Circuit:

The bandwidth of any system is the range of frequencies for which the current or output voltage is equal to 70.7% of its value at the resonant frequency, and it is denoted by BW. Figure 8.9 shows the response of a series Bandwidth of RLC Circuit.

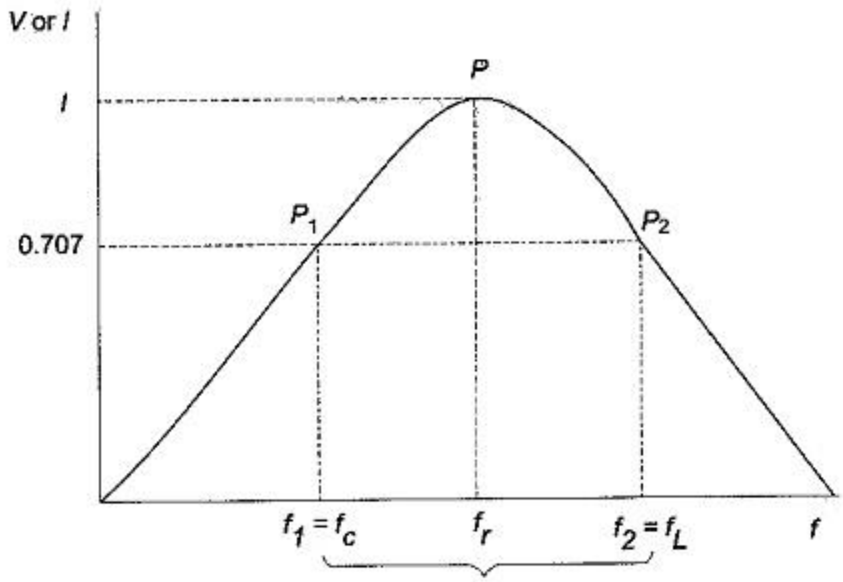


Fig. 8.9

Here the frequency f_1 is the frequency at which the current is 0.707 times the current at resonant value, and it is called the lower cut-off frequency. The frequency f_2 is the frequency at which the current is 0.707 times the current at resonant value (i.e. maximum value), and is called the upper cut-off frequency. The bandwidth, or BW, is defined as the frequency difference between f_2 and f_1.

$$\therefore \qquad BW = f_2 - f_1$$

The unit of BW is hertz (Hz).

If the current at P_1 is $0.707I_{max}$, the impedance of the Bandwidth of RLC Circuit at this point is √2 R, and hence

$$\frac{1}{\omega_1 C} - \omega_1 L = R \qquad (8.1)$$

Similarly,

$$\omega_2 L - \frac{1}{\omega_2 C} = R \qquad (8.2)$$

If we equate both the above equations, we get

$$\frac{1}{\omega_1 C} - \omega_1 L = \omega_2 L - \frac{1}{\omega_2 C}$$

$$L(\omega_1 + \omega_2) = \frac{1}{C}\left(\frac{\omega_1 + \omega_2}{\omega_1 \omega_2}\right) \qquad (8.3)$$

From Eq. 8.3, we get

$$\omega_1 \omega_2 = \frac{1}{LC}$$

we have

$$\omega_r^2 = \frac{1}{LC}$$

$$\omega_r^2 = \omega_1 \omega_2 \qquad (8.4)$$

If we add Eqs 8.1 and 8.2, we get

Since

$$C = \frac{1}{\omega_r^2 L}$$

and

$$\omega_1 \omega_2 = \omega_r^2$$

$$(\omega_2 - \omega_1)L + \frac{\omega_r^2 L(\omega_2 - \omega_1)}{\omega_r^2} = 2R \qquad (8.6)$$

From Eq. 8.6, we have

$$\omega_2 - \omega_1 = \frac{R}{L} \qquad (8.7)$$

$$f_2 - f_1 = \frac{R}{2\pi L} \qquad (8.8)$$

$$BW = \frac{R}{2\pi L}$$

or

From Eq. 8.9, we have

$$f_2 - f_1 = \frac{R}{2\pi L}$$

$$f_r - f_1 = \frac{R}{4\pi L}$$

$$f_2 - f_r = \frac{R}{4\pi L}$$

The lower frequency limit

$$f_1 = f_r - \frac{R}{4\pi L} \qquad (8.9)$$

The upper frequency limit

$$f_2 = f_r + \frac{R}{4\pi L} \qquad (8.10)$$

If we divide the equation on both sides by f_r, we get

$$\frac{f_2 - f_1}{f_r} = \frac{R}{2\pi f_r L} \qquad (8.11)$$

Here an important property of a coil is defined. It is the ratio of the reactance of the coil to its resistance. This ratio is defined as the Q of the coil. Q is known as a figure of merit, it is also called quality factor and is an indication of the quality of a coil.

$$Q = \frac{X_L}{R} = \frac{2\pi f_r L}{R} \qquad (8.12)$$

If we substitute Eq. (8.11) in Eq. (8.12), we get

$$\frac{f_2 - f_1}{f_r} = \frac{1}{Q} \qquad (8.13)$$

The upper and lower cut-off frequencies are sometimes called the half-power frequencies. At these frequencies the power from the source is half of the power delivered at the resonant frequency.

At resonant frequency, the power is

$$P_{\max} = I_{\max}^2 R$$

At frequency f_1, the power is

$$P_1 = \left(\frac{I_{max}}{\sqrt{2}}\right)^2 R = \frac{I_{max}^2 R}{2}$$

Similarly, at frequency f_2, the power is

$$P_2 = \left(\frac{I_{max}}{\sqrt{2}}\right)^2 R$$
$$= \frac{I_{max}^2 R}{2}$$

The response curve in Fig. 8.9 is also called the selectivity curve of the Bandwidth of RLC Circuit. Selectivity indicates how well a resonant circuit responds to a certain frequency and eliminates all other frequencies. The narrower the bandwidth, the greater the selectivity.

Short questions

1. Define resonance. What is the condition for resonance for an RLC series circuit?

Ans- A circuit is said to be in resonance when the applied voltage and current are in phase. For an RLC series circuit, at resonance the inductive and capacitive reactance are equal.

2. . How the RLC series circuit behaves for the frequencies above and below the resonant frequencies.

Ans- For frequencies below resonant frequency, the capacitive reactance is more than the inductive reactance. Therefore the equivalent reactance is equal to capacitive and the circuit behaves like a RC circuit.

For frequencies above resonant frequency, the inductive reactance is more than the capacitive reactance. Therefore the equivalent reactance is equal to inductive and the circuit behaves like a RL circuit.

3. . Derive the expression for resonant frequency

Ans- At resonance condition, the inductive and capacitive reactances are equal.

$$X_L = X_C$$

$$2\pi fL = \frac{1}{2\pi fC}$$

$$f^2 = \frac{1}{4\pi^2 LC}$$

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

.

4. 4. Define resonant frequency

Ans- The frequency at resonance is called as resonant frequency.

It is also defined as the geometric mean of two half power frequencies is called resonant frequency.

$$fo = \sqrt{f_1 f_2}$$

5. Define Q factor

Ans- It is the ratio between capacitor voltage or inductor voltage at resonance to supply voltage is called as Q-factor or quality factor.

$$Q\ factor = \frac{\text{capacitor voltage or inductor voltage}}{\text{supply voltage}}$$

It is also defined as

$$Q\ factor = 2\pi \times \frac{\text{Maximum energy stored}}{\text{Energy dissipated per cycle}}$$

$$Q\ factor = \frac{\omega oL}{R} = \frac{1}{\omega oRC} = \frac{1}{R}\sqrt{\frac{L}{C}} = \frac{fo}{B.W}$$

6. Define Bandwidth.

Ans- It is defined as the width of the resonant curve upto frequency at which the power in the circuit is half of its maximum value. The difference between two half power frequencies is also called as band width.

7. Define selectivity.

Ans- It is the ratio of bandwidth to resonant frequency.

$$\text{Selectivity} = \frac{Band\,width}{\text{Re}\,sonant\,frequency} = \frac{f_2 - f_1}{f_0}$$

Selectivity of a resonant circuit is its ability to discriminate between signals of desired and undesired frequencies.

Long question

1.Deduce expression for active, reactive, apparent power.

2. Derive the resonant frequency of series resonance and parallel resonance circuit

Polyphase circuit

Chapter-6

6.1. Concept of poly-phase system and phase sequence

Polyphase System is a combination of two or more than two voltages having same magnitude and frequency but displaced from each other by an equal electrical angle. As poly means, many (more than one) and phase means windings or circuits. Each of them has a single alternating voltage of the same magnitude and frequency.

The angular displacement between the adjacent voltages is called a phase difference and depends upon the number of phases.

$$\text{Phase Difference} = \frac{360 \text{ electrical degrees}}{\text{Number of phases}}$$

However, the above equation does not hold good for the two-phase system where the voltages are displaced by an angle of 90 degrees electrical.

Thus, in other words, a polyphase system can be defined as an AC system having a group of (two or more than two) equal voltages of same frequency arranged to have an equal phase difference between the adjacent EMFs.

Single-phase systems are employed for the operation of almost all the domestic and commercial applications.

For examples – Fans, Televisions, Refrigerators, Washing machines, Mixer-grinder, Computers, Exhaust Fans, Lamps, Electric Toasters, Electric Irons, etc. But the single-phase system has its limitations in the field of generation, transmission, distribution and industrial applications.

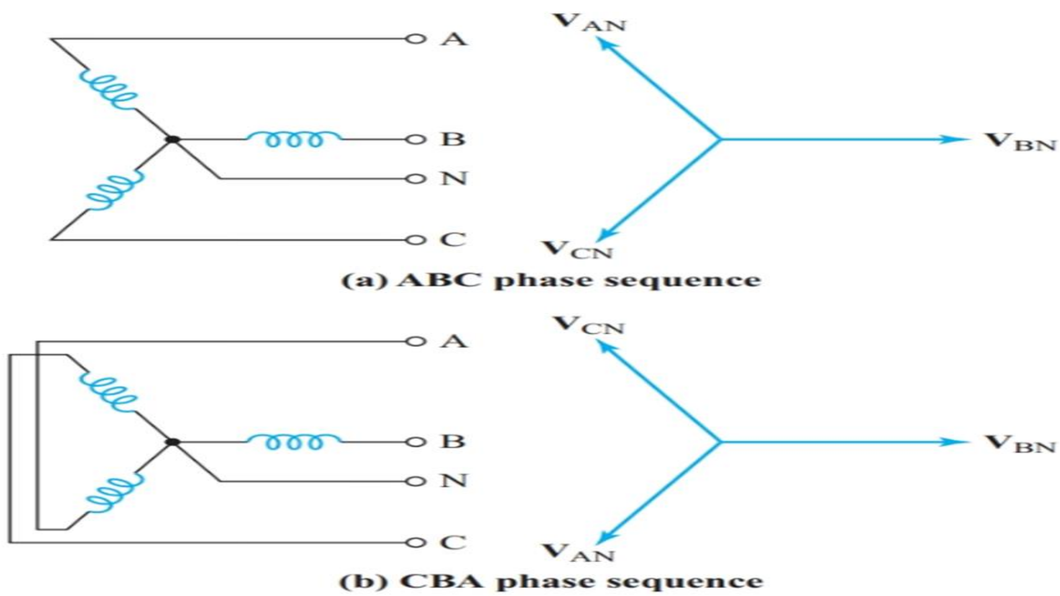
Thus, because of such limitations, the single-phase system is replaced by Polyphase System.

In a three-phase service supplied by an electrical utility company, all three line voltages have the same magnitude and are displaced from one another by 120°. We can measure the magnitude of these voltages quite readily. However, the magnitudes do not tell us whether V_{BN} leads V_{AN} by 120° or lags behind it by 120°. The phase sequence (or phase rotation) of a three-phase system governs the direction of rotation of three-phase motors and the division of the current among the three lines feeding an unbalanced load.

Figure (a) shows that V_{BN} lags behind V_{AN} by 120° and V_{CN} lags behind V_{BN} by 120°. Therefore, the instantaneous voltage to line A passes through its positive peak value first, then the instantaneous voltage to line B passes through its positive peak value, followed by the instantaneous voltage to line C. Thus, this system has an ABC phase sequence.

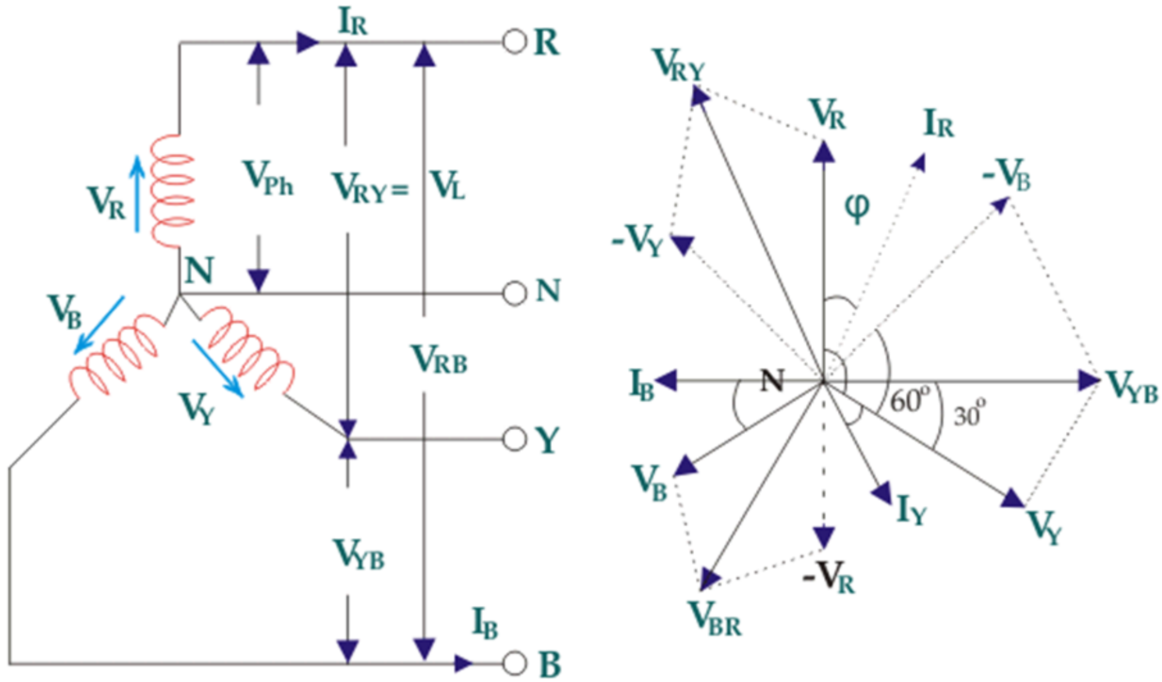
If we reverse the leads to two of the generator coils, as in Figure (b), V_{BN} now leads V_{AN} by 120° and V_{CN} leads V_{BN} by 120°. Thus the phase sequence has been reversed and the system now has a CBA phase sequence.

In a three-phase service supplied by an electrical utility company, all three line voltages have the same magnitude and are displaced from one another by 120°. We can measure the magnitude of these voltages quite readily. However, the magnitudes do not tell us whether V_{BN} leads V_{AN} by 120° or lags behind it by 120°. The phase sequence (or phase rotation) of a three-phase system governs the direction of rotation of three-phase motors and the division of the current among the three lines feeding an unbalanced load.



6.2 Relation between phase and line quantities in star & delta connection

derive the relations between line and phase currents and voltages of a star connected system, we have first to draw a balanced star connected system.



Suppose due to load impedance the current lags the applied voltage in each phase of the system by an angle ϕ. As we have considered that the system is perfectly balanced, the magnitude of the current and voltage of each phase is the same. Let us say, the magnitude of the voltage across the red phase i.e. magnitude of the voltage between neutral point (N) and red phase terminal (R) is VR. Similarly, the magnitude of the voltage across the yellow phase is VY and the magnitude of the voltage across the blue phase is VB. In the balanced star system, the magnitude of phase voltage in each phase is Vph.

∴ VR = VY = VB = Vph

We know in the star connection, line current is same as phase current. The magnitude of this current is same in all three phases and say it is I_L.

∴ $I_R = I_Y = I_B = I_L$, Where, I_R is line current of R phase, I_Y is line current of Y phase and I_B is line current of B phase. Again, phase current, I_{ph} of each phase is same as line current I_L in star connected system.

∴ $I_R = I_Y = I_B = I_L = I_{ph}$.

Now, let us say, the voltage across R and Y terminal of the star connected circuit is V_{RY}.

The voltage across Y and B terminal of the star connected circuit is V_{YB}<!–

The voltage across B and R terminal of the star connected circuit is VBR.

From the diagram, it is found that

$$V_{RY} = V_R + (-V_Y)$$

$V_{RY} = V_R + (-V_Y)$
Similarly, $V_{YB} = V_Y + (-V_B)$
And, $V_{BR} = V_B + (-V_R)$
Now, as angle between V_R and V_Y is 120°(electrical), the angle between V_R and $-V_Y$ is 180° – 120° = 60°(electrical).

Thus, for the star-connected system line voltage = √3 × phase voltage.
Line current = Phase current
As, the angle between voltage and current per phase is φ, the electric power per phase is

So the total power of three phase system is

$$3 \times \frac{V_L}{\sqrt{3}} I_L \cos\phi = \sqrt{3} V_L I_L \cos\phi$$

In star connection, the phase current and line current are same but voltages are changed by a factor of √3 as,

Il = Ip

Vl = √3Vp

But in delta connection the phase and line voltages are same but currents are changed by the factor of √3 as,

Vl = Vp

Il = √3Ip

6.3 Power equation in 3-phase balanced circuit.

Balanced loads, in a 3φ system, have identical impedance in each secondary winding (Figure a). The impedance of each winding in a delta load is shown as Z_Δ (Figure 12a), and the impedence in a wye load is shown as Z_y (Figure b). For either the delta or wye connection, the lines A, B, and C supply a 3φ system of voltages

In a balanced delta load, the line voltage (V_L) is equal to the phase voltage ($V_ø$), and the line current (I_L) is equal to the square root of three times the phase current ($\sqrt{3}I_ø$).

The below Equation is a mathematical representation of V_L in a balanced delta load.

$$\mathbf{V_L = V_ø}$$

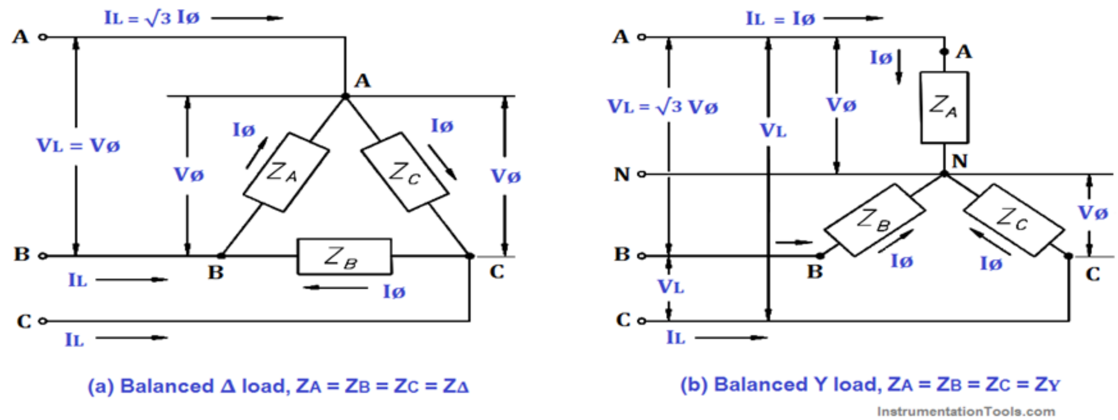
The below Equation is a mathematical representation of I_L in a balanced delta load.

$$I_L = \sqrt{3} I_ø$$

In a balanced wye load, the line voltage (V_L) is equal to the square root of three times phase voltage ($\sqrt{3} V_ø$), and line current (I_L) is equal to the phase current ($I_ø$).

The below Equation is a mathematical representation of V_L in a balanced wye load.

$$V_L = \sqrt{3} V_ø$$



The below Equation is a mathematical representation of I_L in a balanced wye load.

$$I_L = I_ø$$

Because the impedance of each phase of a balanced delta or wye load has equal current, phase power is one third of the total power.

The below Equation is the mathematical representation for phase power ($P_ø$) in a balanced delta or wye load.

$$P_ø = V_ø I_ø \cos\theta$$

Total power (P_T) is equal to three times the single-phase power.

The below Equation is the mathematical representation for total power in a balanced delta or wye load.

$$P_T = 3 V_ø I_ø \cos\theta$$

In a delta-connected load,

$$V_L = V_\phi \quad \text{and} \quad I_\phi = \frac{\sqrt{3}\, I_L}{3}$$

so:

$$P_T = \sqrt{3}\, V_L I_L \cos\theta$$

In a wye-connected load,

$$I_L = I_\phi \quad \text{and} \quad V_\phi = \frac{\sqrt{3}\, V_L}{3}$$

so:

$$P_T = \sqrt{3}\, V_L I_L \cos\theta$$

As you can see, the total power formulas for delta- and wye-connected loads are identical.

6.4 Solve numerical problems

Example 1:

Each phase of a delta- connected 3φ AC generator supplies a full load current of 200 A at 440 volts with a 0.6 lagging power factor, as shown in Figure 14.

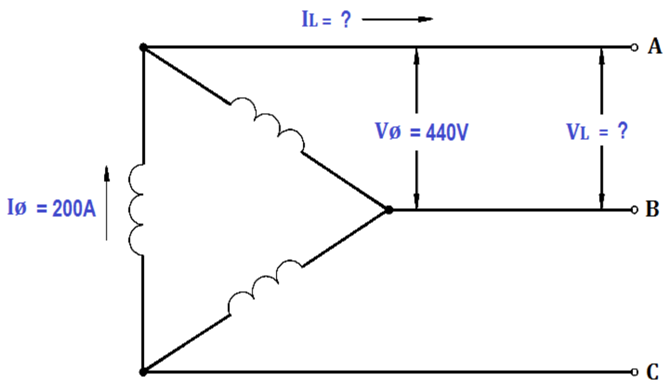


Figure 14 : Three-Phase Delta Generator

Find:

1. V_L
2. I_L
3. P_T
4. Q_T
5. S_T

Solution :

1. Calculate V_L

$\mathbf{V_L = V_ø}$

$V_L = 440V$

2.Calculate I_L

$I_L = \sqrt{3} I_ø$

$I_L = 1.73 \times 200$

$I_L = 346$ amps

Calculate P_T

$P_T = \sqrt{3}\ V_L\ I_L\ \cos\theta$

$P_T = 1.73 \times 440 \times 346 \times 0.6$

$P_T = 158.2$ kW

4. Calculate Q_T

$Q_T = \sqrt{3}\ V_L\ I_L\ \sin\theta$

$Q_T = 1.73 \times 440 \times 346 \times 0.8$

$Q_T = 210.7$ KVR

5. Calculate S_T

$S_T = \sqrt{3}\ V_L\ I_L$

$S_T = 263.4$ KVA

Example 2:

Each phase of a wye- connected 3φ AC generator supplies a 100 A current at a phase voltage of 240V and a power factor of 0.9 lagging, as shown in Figure 15.

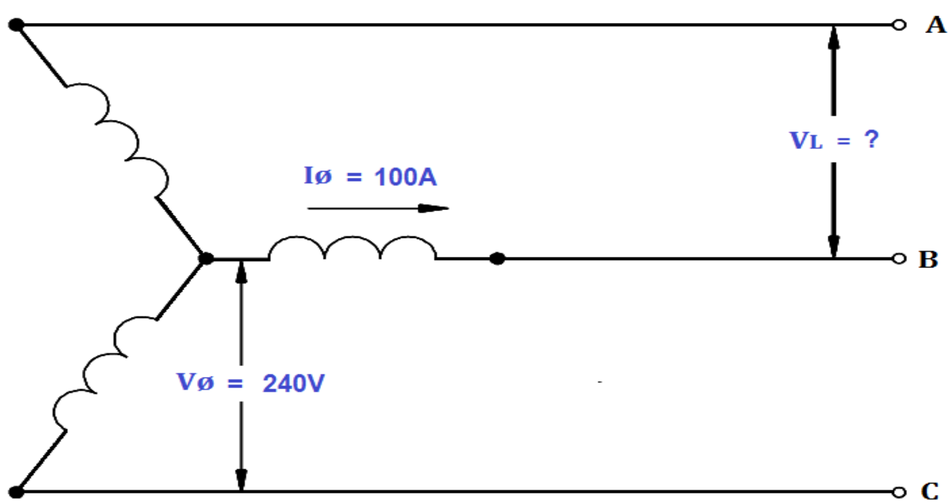


Figure 15 : Three-Phase Wye Generator

Find:

1. V_L
2. P_T
3. Q_T
4. S_T

Solution :

1. Calculate V_L

$V_L = \sqrt{3} V_{ø}$

$V_L = 1.73 \times 240$

$V_L = 415.2$ volts

2. Calculate P_T

$P_T = \sqrt{3}\ V_L\ I_L \cos\theta$

$P_T = 1.73 \times 415.2 \times 100 \times 0.9$

$P_T = 64.6$ kW

3. Calculate Q_T

$Q_T = \sqrt{3}\ V_L\ I_L \sin\theta$

$Q_T = 1.73 \times 415.2 \times 100 \times 0.436$

$Q_T = 31.3$ KVAR

4. Calculate S_T

$S_T = \sqrt{3}\ V_L\ I_L$

$S_T = 1.73 \times 415.2 \times 100$

$S_T = 71.8$ KVA

6.5 Measurement of 3-phase power by two wattmeter method.

n the three-phase power systems, one, two, or three wattmeters can be used to measure the total power. A wattmeter may be considered to be a voltmeter and an ammeter combined in the same box, which has a deflection proportional to $V_{rms}I_{rms} \cos \theta$, where θ is the angle between the voltage and current. Hence, a wattmeter has two voltage and two current terminals, which have + or − polarity signs. Three power measurement methods utilizing the wattmeters are described next, and are applied to the balanced three-phase ac load.

1 Two-Wattmeter Method

This method can be used in a three-phase three-wire balanced or unbalanced load system that may be connected Δ or Y. To perform the measurement, two wattmeters are connected as shown in Fig. 3-23.

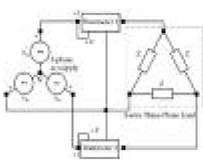


Figure . Two-wattmeter method in star- or delta-connected load.

In the balanced loads, the sum of the two wattmeter readings gives the total power. This can be proven in a star-connected load mathematically using the power reading of each meter as

Equation 3.57

$$\begin{aligned} P_1 &= V_{12}I_1 \cos(30^\circ + \theta) = V_{line}I_{line} \cos(30^\circ + \theta) \\ P_2 &= V_{32}I_3 \cos(30^\circ - \theta) = V_{line}I_{line} \cos(30^\circ - \theta) \\ P_{total} &= P_1 + P_2 = \sqrt{3}V_{line}I_{line} \cos \theta \end{aligned}$$

If the difference of the readings is computed,

Equation 3.58

$$\begin{aligned} P_2 - P_1 &= V_{line}I_{line} \cos(30^\circ - \theta) - V_{line}I_{line} \cos(30^\circ + \theta) \\ &= V_{line}I_{line} \sin \theta \end{aligned}$$

$1/\sqrt{3}$ which is times the total three-phase reactive power. This means that the two-wattmeter method can also indicate the total reactive power in the three-phase loads and also the power factor (see Fig. 3-24).

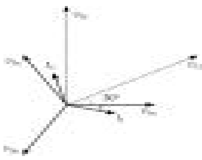


Figure. Three-phase voltage phasors used in the two-wattmeter method.

2 Three-Wattmeter Method

This method is used in a three-phase four-wire balanced or unbalanced load. The connections are made with one meter in each line as shown in Fig. 3-25. In this configuration, the total active power supplied to the load is equal to the sum of the three wattmeter readings.

Equation 3.59

$$P_{\text{total}} = P_1 + P_2 + P_3$$

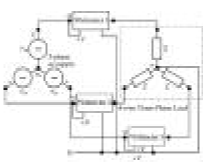


Figure. The wattmeter connections in the three-phase four-wire loads.

3 One-Wattmeter Method

This method is suitable only in three-phase four-wire balanced loads. The connection of the wattmeter is similar to the drawing given in Fig. 3-25. The total power is equal to three times the reading of only one wattmeter that is connected between one phase and the neutral terminal.

6.6 .Solve numerical problems.

1.Two wattmeters are connected to measure the input power to a balanced 3-phase load by the two-wattmeter method. If the instrument readings are 8kW and 4kW, determine (a) the total power input and (b) the load power factor.

(a)Total input power,

$P = P1 + P2 = 8 + 4 =$ **12kW**

(b) $\tan \varphi = \sqrt{3}(P1 - P2)/(P1 + P2)$

$= \sqrt{3}\ (8 - 4) / (8 + 4)$

$= \sqrt{3}\ (4/12)$

$= \sqrt{3}(1/3)$

$= 1/\sqrt{3}$

Hence $\varphi = \tan^{-1} 1\ \sqrt{3} \quad = 30^\circ$

Power factor= $\cos \varphi = \cos 30^\circ =$ **0.866**

2.Two wattmeters connected to a 3-phase motor indicate the total power input to be 12kW. The power factor is 0.6. Determine the readings of each wattmeter.

If the two wattmeters indicate $P1$ and $P2$ respectively

Then $P1 + P2 = 12kW$ ---(1)

$\tan \varphi = \sqrt{3}(P1 - P2)/(P1 + P2)$

And power factor=0.6= $\cos \varphi$.

Angle $\varphi = \cos^{-1} 0.6 = 53.13^\circ$ and

$\tan 53.13^\circ = 1.3333$.

Hence

$1.3333 = \sqrt{3}(P1 - P2)/12$

From which,

$P1 - P2 \quad = 12(1.3333)/\sqrt{3}$

i.e. $P1 - P2 = 9.237kW$ ----(2)

Adding Equations (1) and (2) gives:

$2P1 = 21.237$

i.e $P1 = 21.237/2$

= 10.62kW Hence wattmeter 1 reads 10.62kW From Equation (1), wattmeter 2 reads

(12−10.62)=1.38kW

3. **Three loads, each of resistance 30, are connected in star to a 415 V, 3-phase supply. Determine**

   (a) the system phase voltage, (b) the phase current and (c) the line current.

A '415 V, 3-phase supply' means that 415 V is the line voltage, VL

(a) For a star connection, $V_L = \sqrt{3} V_p$ Hence phase voltage, $V_p = V_L/\sqrt{3}$

$= 415/\sqrt{3}$

= **239.6 V or 240 V**

correct to 3 significant figures

(b) Phase current, $I_p = V_p/R_p$

$= 240/30$

= **8 A**

(c) For a star connection, $I_p = I_L$ Hence the line current, I_L = **8 A**

Short question s

1.What are the Advantages of 3 phase system?

i. Most of the electric power is generated and distributed in three-phase.

ii. The instantaneous power in a three-phase system can be constant.

iii. The amount of power, the three-phase system is more economical that the single-phase.

iv. In fact, the amount of wire required for a three-phase system is less than that required for an equivalent single-phase system.

2.Define phase, line & neutral?

Phase

Describes or pertains to one element or device in a load, line, or source. It is simply a "branch" of the circuit and could look something like this.

Line

refers to the "transmission line" or wires that connect the source (supply) to the load. It may be modeled as a small impedance (actually 3 of them), or even by just a connecting line.

Neutral

the 4th wire in the 3-phase system. It's where the phases of a Y connection come together. 3. **Deifne Phase**

3.Define line voltage and line current?

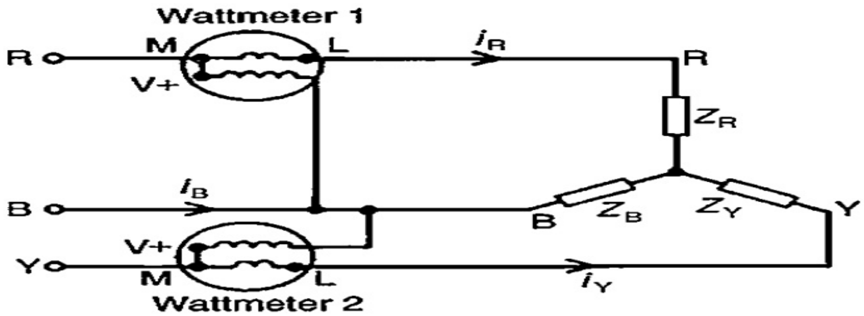
Line Currents

the currents flowing in each of the lines (Ia, Ib, and Ic). This definition does not change with connection type.

Line Voltages

the voltages between any two of the lines (Vab, Vbc, and Vca). These may also be referred to as the line-to-line voltages. This definition does not change with connection type.

4.Draw two wattmeter methods for measurement of power in 3 phase systems?



Long questions

1. Write the relationship of line and phase voltage and current in star?

2.Write the relationship of line and phase voltage and current in delta?

3.Write 3 phase power equation?

4.Write the power factor calculation of two wattmeter method?

FILTER

CHAPTER-7

7.1 Define filter

Filters as the name suggests, they filter the frequency components. That means, they allow certain frequency components and / or reject some other frequency components.

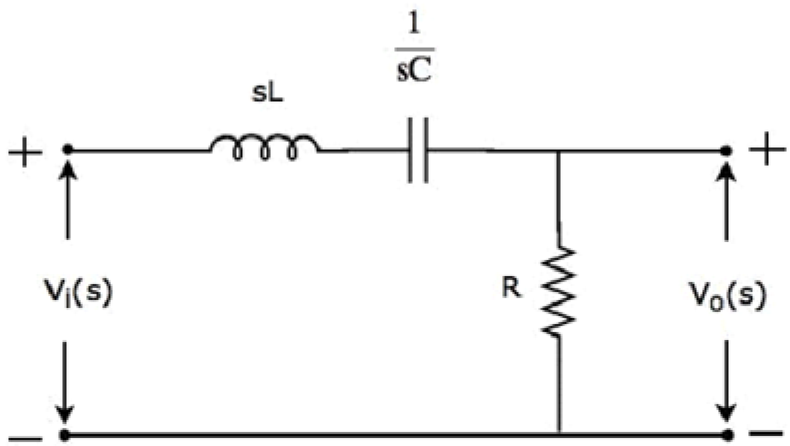
In this chapter, let us discuss about the passive filters. Those are the electric circuits or networks having passive elements like resistor, inductor and capacitor.

7.2 Classification of pass Band, stop Band and cut-off frequency.

Band Pass Filter

Band pass filter as the name suggests, it allows (passes) only one band of frequencies. In general, this frequency band lies in between low frequency range and high frequency range. That means, this filter rejects (blocks) both low and high frequency components.

The s-domain circuit diagram (network) of Band pass filter is shown in the following figure.



It consists of three passive elements inductor, capacitor and resistor, which are connected in series. Input voltage is applied across this entire combination and the output is considered as the voltage across resistor

Here, $V_i(s)$ and $V_o(s)$ are the Laplace transforms of input voltage, $v_i(t)$ and output voltage, $v_o(t)$ respectively.

The **transfer function** of the above network is

$$H(s) = \frac{V_o(s)}{V_i(s)} = \frac{R}{R + \frac{1}{sC} + sL}$$

$$\Rightarrow H(s) = \frac{sCR}{s^2LC + sCR + 1}$$

Substitute $s = j\omega$ in the above equation.

$$H(j\omega) = \frac{j\omega CR}{1 - \omega^2 LC + j\omega CR}$$

Magnitude of transfer function is

$$|H(j\omega)| = \frac{\omega CR}{\sqrt{(1-\omega^2 LC)^2 + (\omega CR)^2}}$$

- At $\omega = 0$, the magnitude of transfer function is equal to 0.
- At $\omega = \frac{1}{\sqrt{LC}}$, the magnitude of transfer function is equal to 1.
- At $\omega = \infty$, the magnitude of transfer function is equal to 0.

Therefore, the magnitude of transfer function of **Band pass filter** will vary from 0 to 1 & 1 to 0 as ω varies from 0 to ∞.

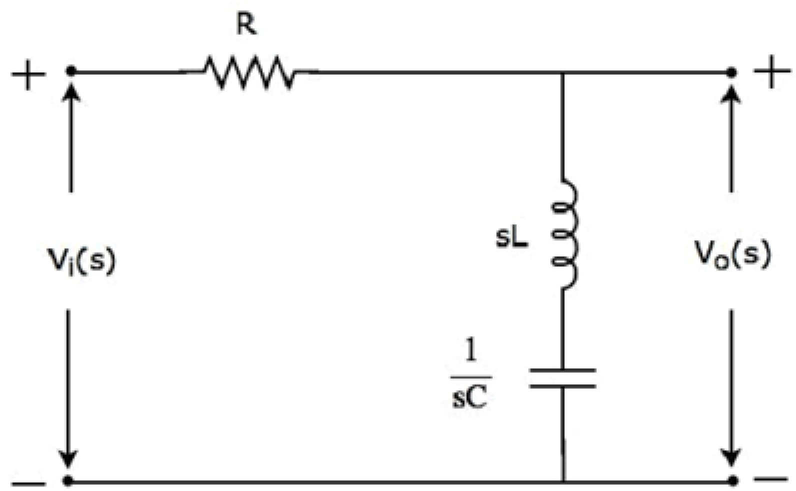
- At ω = 0, the magnitude of transfer function is equal to 0.
- At ω=1LC√ω=1LC, the magnitude of transfer function is equal to 1.
- At ω = ∞, the magnitude of transfer function is equal to 0.

Therefore, the magnitude of transfer function of Band pass filter will vary from 0 to 1 & 1 to 0 as ω varies from 0 to ∞.

Band Stop Filter

Band stop filter as the name suggests, it rejects (blocks) only one band of frequencies. In general, this frequency band lies in between low frequency range and high frequency range. That means, this filter allows (passes) both low and high frequency components.

The s-domain (network) of circuit diagramand stop filter is shown in the following figure.



It consists of three passive elements resistor, inductor and capacitor, which are connected in series. Input voltage is applied across this entire combination and the output is considered as the voltage across the combination of inductor and capacitor.

Here, Vi(s)Vi(s) and Vo(s)Vo(s) are the Laplace transforms of input voltage, vi(t)vi(t) and output voltage, vo(t)vo(t) respectively.

The **transfer function** of the above network is

$$H(s) = \frac{V_o(s)}{V_i(s)} = \frac{sL + \frac{1}{sC}}{R + sL + \frac{1}{sC}}$$

$$\Rightarrow H(s) = \frac{s^2LC + 1}{s^2LC + sCR + 1}$$

Substitute, $s = j\omega$ in the above equation.

$$H(j\omega) = \frac{1 - \omega^2 LC}{1 - \omega^2 LC + j\omega CR}$$

Magnitude of transfer function is

$$|H(j\omega)| = \frac{1 - \omega^2 LC}{\sqrt{(1 - \omega^2 LC)^2 + (\omega CR)^2}}$$

- At $\omega = 0$, the magnitude of transfer function is equal to 1.
- At $\omega = \frac{1}{\sqrt{LC}}$, the magnitude of transfer function is equal to 0.
- At $\omega = \infty$, the magnitude of transfer function is equal to 1.

Therefore, the magnitude of transfer function of Band stop filter will vary from 1 to 0 & 0 to 1 as ω varies from 0 to ∞.

Cutoff Frequency

Cutoff frequency (also known as corner frequency, or break frequency) is defined as a boundary in a system's frequency response at which energy flowing through the system begins to be attenuated (reflected or reduced) rather than passing through

The cutoff frequency or corner frequency in electronics is the frequency either above or below which the power output of a circuit, such as a line, amplifier, or electronic filter (e.g. a high pass filter) has fallen to a given proportion of the power in the passband.

Most frequently this proportion is one-half the passband power, also referred to as the 3 dB point since a fall of 3 dB corresponds approximately to half power. As a voltage ratio, this is a fall to approximately 0.707.

For any filtering circuits such as RC circuits, the cutoff frequency is a very important characteristic. At this point, the amount of attenuation due to the filter starts to increase swiftly.
To indicate how long the amplifier gain can remain constant to frequency, we need to define a range of frequencies. Along with that range, the gain should not deviate more than 70.7% of the maximum gain that has been defined as a reference at mid-frequency. In the below-shown curve, f1 and f2 indicate lower cut off and upper cut off frequencies.

7.3 Classification of filters. 9.4 Constant – K low pass filter

Filters are mainly classified into four types based on the band of frequencies that are allowing and / or the band of frequencies that are rejecting. Following are the types of filters.

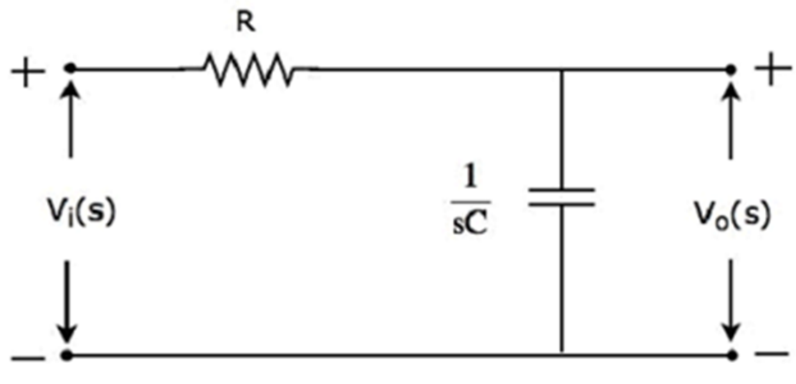
- Low Pass Filter
- High Pass Filter
- Band Pass Filter
- Band Stop Filter

7.4 Constant – K low pass filter.

Low Pass Filter

Low pass filter as the name suggests, it allows (passes) only low frequency components. That means, it rejects (blocks) all other high frequency components.

The s-domain circuit diagram (network) of Low Pass Filter is shown in the following figure.



It consists of two passive elements resistor and capacitor, which are connected in series. Input voltage is applied across this entire combination and the output is considered as the voltage across capacitor.

Here, $V_i(s)$ and $V_o(s)$ are the Laplace transforms of input voltage, $v_i(t)$ and output voltage, $v_o(t)$ respectively.

The **transfer function** of the above network is

$$H(s) = \frac{V_o(s)}{V_i(s)} = \frac{\frac{1}{sC}}{R + \frac{1}{sC}}$$

$$\Rightarrow H(s) = \frac{1}{1 + sCR}$$

Substitute, $s = j\omega$ in the above equation.

$$H(j\omega) = \frac{1}{1 + j\omega CR}$$

Magnitude of transfer function is

$$|H(j\omega)| = \frac{1}{\sqrt{(1 + (\omega CR)^2}}$$

- At $\omega = 0$, the magnitude of transfer function is equal to 1.
- At ω=1CRω=1CR, the magnitude of transfer function is equal to 0.707.
- At $\omega = \infty$, the magnitude of transfer function is equal to 0.

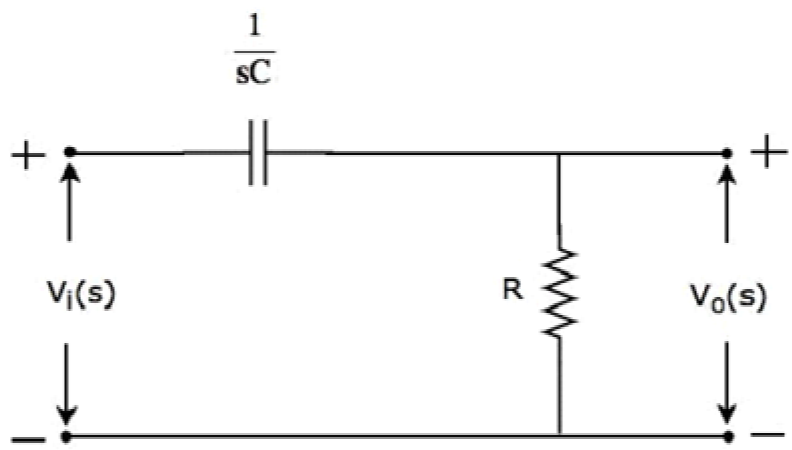
Therefore, the magnitude of transfer function of Low pass filter will vary from 1 to 0 as ω varies from 0 to ∞.

7.5 Constant – K high pass filter

High Pass Filter

High pass filter as the name suggests, it allows (passes) only high frequency components. That means, it rejects (blocks) all low frequency components.

The s-domain circuit diagram (network) of High pass filter is shown in the following figure.



It consists of two passive elements capacitor and resistor, which are connected in series. Input voltage is applied across this entire combination and the output is considered as the voltage across resistor.

Here, $V_i(s)$ and $V_o(s)$ are the Laplace transforms of input voltage, $v_i(t)$ and output voltage, $v_o(t)$ respectively.

The **transfer function** of the above network is

$$H(s) = \frac{V_o(s)}{V_i(s)} = \frac{R}{R + \frac{1}{sC}}$$

$$\Rightarrow H(s) = \frac{sCR}{1 + sCR}$$

Substitute, $s = j\omega$ in the above equation.

$$H(j\omega) = \frac{j\omega CR}{1 + j\omega CR}$$

Magnitude of transfer function is

$$|H(j\omega)| = \frac{\omega CR}{\sqrt{(1 + (\omega CR)^2}}$$

- At $\omega = 0$, the magnitude of transfer function is equal to 0.

- At ω=1CRω=1CR, the magnitude of transfer function is equal to 0.707.
- At $\omega = \infty$, the magnitude of transfer function is equal to 1.

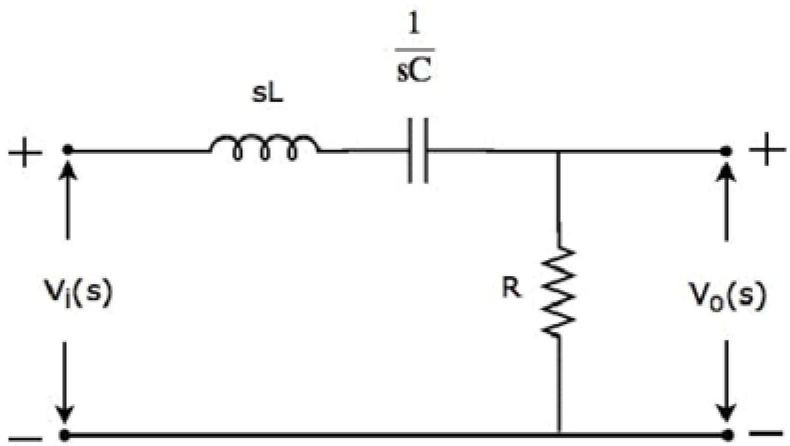
Therefore, the magnitude of transfer function of High pass filter will vary from 0 to 1 as ω varies from 0 to ∞.

7.6 Constant – K Band pass filter.

Band Pass Filter

Band pass filter as the name suggests, it allows (passes) only one band of frequencies. In general, this frequency band lies in between low frequency range and high frequency range. That means, this filter rejects (blocks) both low and high frequency components.

The s-domain circuit diagram (network) of Band pass filter is shown in the following figure.



It consists of three passive elements inductor, capacitor and resistor, which are connected in **series**. Input voltage is applied across this entire combination and the output is considered as the voltage across resistor.

Here, $V_i(s)$ and $V_o(s)$ are the Laplace transforms of input voltage, $v_i(t)$ and output voltage, $v_o(t)$ respectively.

The **transfer function** of the above network is

$$H(s) = \frac{V_o(s)}{V_i(s)} = \frac{R}{R + \frac{1}{sC} + sL}$$

$$\Rightarrow H(s) = \frac{sCR}{s^2LC + sCR + 1}$$

Substitute $s = j\omega$ in the above equation.

$$H(j\omega) = \frac{j\omega CR}{1 - \omega^2 LC + j\omega CR}$$

Magnitude of transfer function is

$$|H(j\omega)| = \frac{\omega CR}{\sqrt{(1-\omega^2 LC)^2 + (\omega CR)^2}}$$

- At $\omega = 0$, the magnitude of transfer function is equal to 0.
- At ω=1LC√ω=1LC, the magnitude of transfer function is equal to 1.
- At $\omega = \infty$, the magnitude of transfer function is equal to 0.

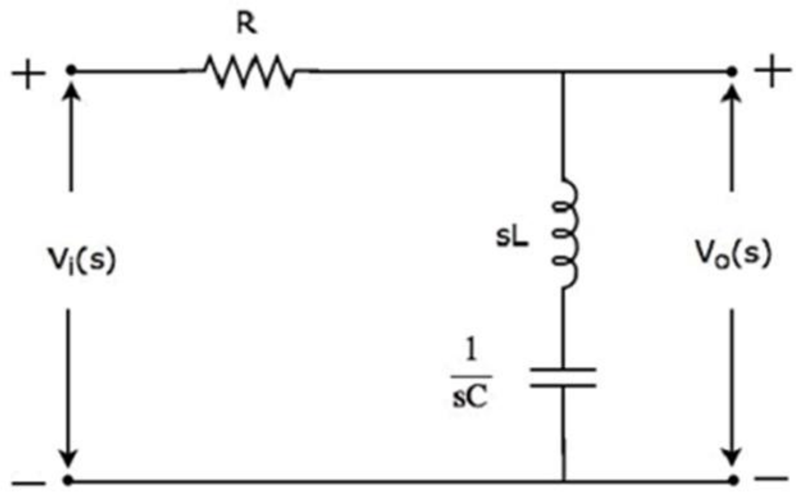
Therefore, the magnitude of transfer function of Band pass filter will vary from 0 to 1 & 1 to 0 as ω varies from 0 to ∞.

7.7 Constant – K Band elimination filter

Band elimination Filter

Band stop filter as the name suggests, it rejects (blocks) only one band of frequencies. In general, this frequency band lies in between low frequency range and high frequency range. That means, this filter allows (passes) both low and high frequency components.

The s-domain (network) of circuit diagram and stop filter is shown in the following figure.



It consists of three passive elements resistor, inductor and capacitor, which are connected in series. Input voltage is applied across this entire combination and the output is considered as the voltage across the combination of inductor and capacitor.

Here, Vi(s)Vi(s) and Vo(s)Vo(s) are the Laplace transforms of input voltage, vi(t)vi(t) and output voltage, vo(t)vo(t) respectively.

Here, $V_i(s)$ and $V_o(s)$ are the Laplce transforms of input voltage, $v_i(t)$ and output voltage, $v_o(t)$ respectively.

The **transfer function** of the above network is

$$H(s) = \frac{V_o(s)}{V_i(s)} = \frac{sL + \frac{1}{sC}}{R + sL + \frac{1}{sC}}$$

$$\Rightarrow H(s) = \frac{s^2LC + 1}{s^2LC + sCR + 1}$$

Substitute, $s = j\omega$ in the above equation.

$$H(j\omega) = \frac{1 - \omega^2 LC}{1 - \omega^2 LC + j\omega CR}$$

Magnitude of transfer function is

$$|H(j\omega)| = \frac{1 - \omega^2 LC}{\sqrt{(1 - \omega^2 LC)^2 + (\omega CR)^2}}$$

- At $\omega = 0$, the magnitude of transfer function is equal to 1.
- At ω=1LC√ω=1LC, the magnitude of transfer function is equal to 0.
- At $\omega = \infty$, the magnitude of transfer function is equal to 1.

Therefore, the magnitude of transfer function of Band stop filter will vary from 1 to 0 & 0 to 1 as ω varies from 0 to ∞.

7.8 Solve Numerical problems

1390

Example 19.1. *In a simple T section, a low pass filter has a design impedance R_0. Find $Z_{0\pi}$ at 0.9 f_C.*

Solution. $\frac{f}{f_C} = 0.9$ (given)

However, for *LPF*, $Z_{0\pi} = \frac{R_0}{\sqrt{1-\left(\frac{f}{f_C}\right)^2}} = \frac{R_0}{\sqrt{[1-(0.9)^2]}} = 2.3\ R_0.$

Example 19.2. *Design a constant K-low pass filter having cut-off frequency 2.5 kHz and design resistance $R_0 = 700\ \Omega$. Also find the frequency at which this filter produces attenuation of 19.1 dB. Find its characteristic impedances and phase constant at pass band and stop or attenuation band.*

Solution. Here from the given data,

$$f_c = 2.5\text{ kHz} = 2500\text{ Hz}\ ; \quad R_o = 700\ \Omega$$

Attenuation $\alpha = 19.1\text{ dB} = \frac{19.1}{8.686} = 2.199$ nepers

The design elements of *LPF* are *L* and *C* given by the relations

$$L = \frac{R_o}{\pi f_c} = \frac{700}{\pi \times 2500}\text{ H} = 89.127\text{ mH}$$

and

$$C = \frac{1}{\pi R_o f_c} = \frac{1}{\pi \times 700 \times 2500}\text{ F} = 0.182\ \mu\text{F}$$

Attenuation in attenuation band of a low pass filter is given by the relation

$$\alpha = 2\cosh^{-1}\left(\frac{f}{f_c}\right)$$

where, *f* be the frequency at which this filter produces attenuation 19.1 dB or 2.199 nepers.

Hence, $\frac{f}{f_c} = \cosh\left(\frac{\alpha}{2}\right)$

or $\frac{f}{2500} = \cosh\left(\frac{2.199}{2}\right) = 1.6678509$

or $f = 2500 \times 1.6678509\text{ Hz} = 4.170\text{ kHz}.$

The characteristic impedances of a *T*-type and π-type *LPF* are given by the relations,

$$Z_{oT} = R_o\sqrt{1-\left(\frac{f}{f_c}\right)^2}$$

and

$$Z_{o\pi} = \frac{R_o}{\sqrt{1-\left(\frac{f}{f_c}\right)^2}}$$

Short questions

1. Give the specifications of (i) Low pass filter (ii) High pass filter

Ans.:

1. **Low Pass Filter:**

For a low pass filter, its specifications in terms of its frequency response is given below:

$$\begin{cases} 1-\delta^{-} \leq |H(\omega)| \leq 1+\delta^{+}, 0 \leq \omega \leq \omega_p \\ 0 \leq |H(\omega)| \leq \delta_s \qquad , \omega_s \leq \omega \leq \pi \end{cases}$$

2. **High Pass Filter:**

For a high pass filter, its specifications in terms of its frequency response is given below:

2.What is cut off frequency?

- Cutoff frequency (also known as corner frequency, or break frequency) is defined as a boundary in a system's frequency response at which energy flowing through the system begins to be attenuated (reflected or reduced) rather than passing through.

- The cutoff frequency or corner frequency in electronics is the frequency either above or below which the power output of a circuit, such as a line, amplifier, or electronic filter (e.g. a high pass filter) has fallen to a given proportion of the power in the passband.

3. What is Bandwidth?

n signal processing, bandwidth is defined as the difference between upper cutoff frequency and low cutoff frequency. The frequency f2 lies along with a high-frequency range and f1 in the low-frequency range. We can also name these two frequencies as Half–Power frequencies since voltage gain drops to 70.7 % of the maximum value. This represents the power level of one–half of the power at reference frequency in the mid-range frequency. Since the change is not noticeable, the audio amplifier has a flat response from f1 to f2.

Long questions

1.Explain low pass and band pass filter?

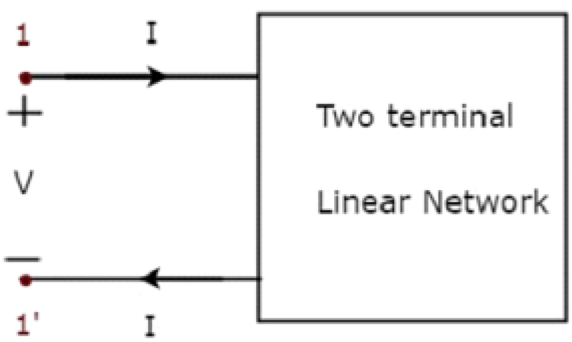
TWO-PORT NETWORK:

Chapter-8

8.1 Open circuit impedance (z) parameters

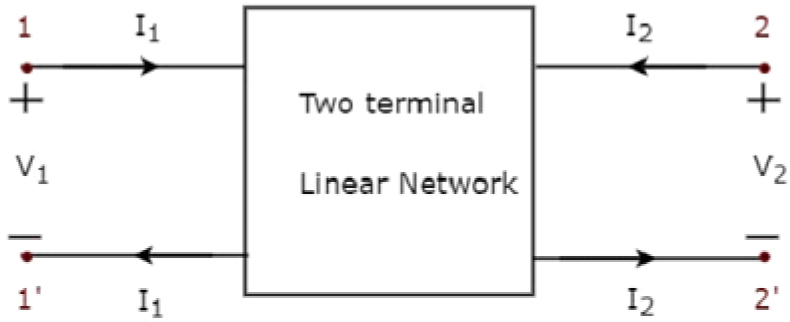
In general, it is easy to analyze any electrical network, if it is represented with an equivalent model, which gives the relation between input and output variables. For this, we can use **two port network** representations. As the name suggests, two port networks contain two ports. Among which, one port is used as an input port and the other port is used as an output port. The first and second ports are called as port1 and port2 respectively.

One port network is a two terminal electrical network in which, current enters through one terminal and leaves through another terminal. Resistors, inductors and capacitors are the examples of one port network because each one has two terminals. One port network representation is shown in the following figure.



Here, the pair of terminals, 1 & 1’ represents a port. In this case, we are having only one port since it is a one port network.

Similarly, **two port network** is a pair of two terminal electrical network in which, current enters through one terminal and leaves through another terminal of each port. Two port network representation is shown in the following figure.



Here, one pair of terminals, 1 & 1’ represents one port, which is called as **port1** and the other pair of terminals, 2 & 2’ represents another port, which is called as **port2**.

There are **four variables** V_1, V_2, I_1 and I_2 in a two port network as shown in the figure. Out of which, we can choose two variables as independent and another two variables as dependent. So, we will get six possible pairs of equations. These equations represent the dependent variables in terms of independent variables. The coefficients of independent variables are called as **parameters**. So, each pair of equations will give a set of four parameters.

Z parameters

We will get the following set of two equations by considering the variables V_1 & V_2 as dependent and I_1 & I_2 as independent. The coefficients of independent variables, I_1 and I_2 are called as **Z parameters**.

$$V1=Z11I1+Z12$$

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

The **Z parameters** are

$$Z_{11}=V_1/I_1, when I_2=0$$

$$Z_{12}=V_1/I_2, when I_1=0$$

$$Z_{21}=V_2/I_1, when I_2=0$$

$$Z_{22}=V_2/I_2, when I_1=0$$

Z parameters are called as impedance parameters because these are simply the ratios of voltages and currents. Units of Z parameters are Ohm (Ω).

We can calculate two Z parameters, Z_{11} and Z_{21}, by doing open circuit of port2. Similarly, we can calculate the other two Z parameters, Z_{12} and Z_{22} by doing open circuit of port1. Hence, the Z parameters are also called as open-circuit impedance parameters.

8.2 Short circuit admittance (y) parameters

Y parameters

We will get the following set of two equations by considering the variables I_1 & I_2 as dependent and V_1 & V_2 as independent. The coefficients of independent variables, V_1 and V_2 are called as Y parameters.

$$I1=Y11V1+Y12V2$$

$$I2=Y21V1+Y22V2$$

The **Y parameters** are

$$Y11=I1/V1,whenV2=0$$

$$Y12=I1/V2,whenV1=0$$

$$Y21=I2/V1,whenV2=0$$

$$Y22=I2/V2,whenV1=0$$

Y parameters are called as admittance parameters because these are simply, the ratios of currents and voltages. Units of Y parameters are mho.

We can calculate two Y parameters, Y_{11} and Y_{21} by doing short circuit of port2. Similarly, we can calculate the other two Y parameters, Y_{12} and Y_{22} by doing short circuit of port1. Hence, the Y parameters are also called as short-circuit admittance parameters.

8.3 Transmission (ABCD) parameters

We will get the following set of two equations by considering the variables V_1 & I_1 as dependent and V_2 & I_2 as independent. The coefficients of V_2 and $-I_2$ are called as T parameters.

$$V1=AV2-BI2$$

$$I1=CV2-DI2$$

The T parameters are

$$A=V1V2,whenI2=0$$

$$B=-V1I2,whenV2=0$$

$$C=I1V2,whenI2=0$$

$$D=-I1I2,whenV2=0$$

T parameters are called as transmission parameters or ABCD parameters. The parameters, A and D do not have any units, since those are dimension less. The units of parameters, B and C are ohm and mho respectively.

We can calculate two parameters, A and C by doing open circuit of port2. Similarly, we can calculate the other two parameters, B and D by doing short circuit of port2.

T ' parameters

We will get the following set of two equations by considering the variables V_2 & I_2 as dependent and V_1 & I_1 as independent. The coefficients of V_1 and $-I_1$ are called as T' parameters.

$$V2=A'V1-B'I1$$

$$I2=C'V1-D'I1$$

The T' parameters are

$$A'=V2V1, when I1=0$$

$$B'=-V2I1, when V1=0$$

$$C'=I2V1, when I1=0$$

$$D'=-I2I1, when V1=0$$

T' parameters are called as inverse transmission parameters or A'B'C'D' parameters. The parameters A' and D' do not have any units, since those are dimension less. The units of parameters, B' and C', are Ohm and Mho respectively.

We can calculate two parameters, A' and C', by doing an open circuit of port1. Similarly, we can calculate the other two parameters, B' and D', by doing a short circuit of port1.

8.4 Hybrid (h) parameters

h-parameters

We will get the following set of two equations by considering the variables V_1 & I_2 as dependent and I_1 & V_2 as independent. The coefficients of independent variables, I_1 and V_2, are called as h-parameters.

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

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

The h-parameters are

$$h_{11}=V_1I_1, when V_2=0$$

$$h_{12} = \frac{V_1}{V_2}, when\ I_1 = 0$$

$$h_{21} = \frac{I_2}{I_1}, when\ V_2 = 0$$

$$h_{22} = \frac{I_2}{V_2}, when\ I_1 = 0$$

h-parameters are called as hybrid parameters. The parameters, h_{12} and h_{21}, do not have any units, since those are dimension-less. The units of parameters, h_{11} and h_{22}, are Ohm and Mho respectively.

We can calculate two parameters, h_{11} and h_{21} by doing short circuit of port2. Similarly, we can calculate the other two parameters, h_{12} and h_{22} by doing open circuit of port1.

The h-parameters or hybrid parameters are useful in transistor modelling circuits (networks).

8.5 Inter relationships of different parameters

Z parameters to Y parameters

Here, we have to represent Y parameters in terms of Z parameters. So, in this case Y parameters are the desired parameters and Z parameters are the given parameters.

Step 1 − We know that the following set of two equations, which represents a two port network in terms of Y parameters.

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

We can represent the above two equations in **matrix** form as

$$\begin{bmatrix} I_1 \\ I_2 \end{bmatrix} = \begin{bmatrix} Y_{11} & Y_{12} \\ Y_{21} & Y_{22} \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix}$$ **Equation 1**

Step 2 − We know that the following set of two equations, which represents a two port network in terms of **Z parameters**.

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

We can represent the above two equations in **matrix** form as

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

Step 3 − We can modify it as

$$\begin{bmatrix} I_1 \\ I_2 \end{bmatrix} = \begin{bmatrix} Z_{11} & Z_{12} \\ Z_{21} & Z_{22} \end{bmatrix}^{-1} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix}$$ **Equation 2**

Step 4 − By equating Equation 1 and Equation 2, we will get

$$\begin{bmatrix} Y_{11} & Y_{12} \\ Y_{21} & Y_{22} \end{bmatrix} = \begin{bmatrix} Z_{11} & Z_{12} \\ Z_{21} & Z_{22} \end{bmatrix}^{-1}$$
$$\Rightarrow \begin{bmatrix} Y_{11} & Y_{12} \\ Y_{21} & Y_{22} \end{bmatrix} = \frac{\begin{bmatrix} Z_{22} & -Z_{12} \\ -Z_{21} & Z_{11} \end{bmatrix}}{\Delta Z}$$

Where,

$$\Delta Z = Z_{11}Z_{22} - Z_{12}Z_{21}$$

So, just by doing the inverse of Z parameters matrix, we will get Y parameters matrix.

Z parameters to T parameters

Here, we have to represent T parameters in terms of Z parameters. So, in this case T parameters are the desired parameters and Z parameters are the given parameters.

Step 1 − We know that, the following set of two equations, which represents a two port network in terms of T parameters.

$V1=AV2-BI2$
$I1=CV2-DI2$

Step 2 − We know that the following set of two equations, which represents a two port network in terms of Z parameters.

$V1=Z11I1+Z12I2$
$V2=Z21I1+Z22I2$

Step 3 − We can modify the above equation as

$\Rightarrow V2-Z22I2=Z21I1$
$\Rightarrow I1=(1Z21)V2-(Z22Z21)I2$
Step 4 − The above equation is in the form of $I1=CV2-DI2$. Here,
$C=1/Z21$
$D=Z22/Z21$

Step 5 − Substitute I_1 value of Step 3 in V_1 equation of Step 2.

$$V_1 = Z_{11}\{(\frac{1}{Z_{12}})V_2 - (\frac{Z_{22}}{Z_{21}})I_2\} + Z_{12}I_2$$

$$\Rightarrow V_1 = (\frac{Z_{11}}{Z_{21}})V_2 - (\frac{Z_{11}Z_{22} - Z_{12}Z_{21}}{Z_{21}})I_2$$

Step 6 − The above equation is in the form of $V_1 = AV_2 - BI_2$. Here,

$$A = \frac{Z_{11}}{Z_{21}}$$

$$B = \frac{Z_{11}Z_{22} - Z_{12}Z_{21}}{Z_{21}}$$

Step 7 − Therefore, the **T parameters matrix** is

$$\begin{bmatrix} A & B \\ C & D \end{bmatrix} = \begin{bmatrix} \frac{Z_{11}}{Z_{21}} & \frac{Z_{11}Z_{22}-Z_{12}Z_{21}}{Z_{21}} \\ \frac{1}{Z_{21}} & \frac{Z_{22}}{Z_{21}} \end{bmatrix}$$

Y parameters to Z parameters

Here, we have to represent Z parameters in terms of Y parameters. So, in this case Z parameters are the desired parameters and Y parameters are the given parameters.

Step 1 − We know that, the following matrix equation of two port network regarding Z parameters as

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} Z_{11} & Z_{12} \\ Z_{21} & Z_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix} \qquad \textbf{Equation 3}$$

Step 2 − We know that, the following matrix equation of two port network regarding Y parameters as

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

Step 3 − We can modify it as

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} Y_{11} & Y_{12} \\ Y_{21} & Y_{22} \end{bmatrix}^{-1} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix} \qquad \textbf{Equation 4}$$

Step 4 − By equating Equation 3 and Equation 4, we will get

$$\begin{bmatrix} Z_{11} & Z_{12} \\ Z_{21} & Z_{22} \end{bmatrix} = \begin{bmatrix} Y_{11} & Y_{12} \\ Y_{21} & Y_{22} \end{bmatrix}^{-1}$$

$$\Rightarrow \begin{bmatrix} Z_{11} & Z_{12} \\ Z_{21} & Z_{22} \end{bmatrix} = \frac{\begin{bmatrix} Y_{22} & -Y_{12} \\ -Y_{21} & Y_{11} \end{bmatrix}}{\Delta Y}$$

Where,

$\Delta Y = Y_{11}Y_{22} - Y_{12}Y_{21}$

So, just by doing the inverse of Y parameters matrix, we will get the Z parameters matrix.

Y parameters to T parameters

Here, we have to represent T parameters in terms of Y parameters. So, in this case, T parameters are the desired parameters and Y parameters are the given parameters.

Step 1 − We know that, the following set of two equations, which represents a two port network in terms of T parameters.

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} Z_{11} & Z_{12} \\ Z_{21} & Z_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix}$$ **Equation 3**

Step 2 − We know that, the following matrix equation of two port network regarding Y parameters as

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

Step 3 − We can modify it as

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} Y_{11} & Y_{12} \\ Y_{21} & Y_{22} \end{bmatrix}^{-1} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix}$$ **Equation 4**

Step 4 − By equating Equation 3 and Equation 4, we will get

$$\begin{bmatrix} Z_{11} & Z_{12} \\ Z_{21} & Z_{22} \end{bmatrix} = \begin{bmatrix} Y_{11} & Y_{12} \\ Y_{21} & Y_{22} \end{bmatrix}^{-1}$$

$$\Rightarrow \begin{bmatrix} Z_{11} & Z_{12} \\ Z_{21} & Z_{22} \end{bmatrix} = \frac{\begin{bmatrix} Y_{22} & -Y_{12} \\ -Y_{21} & Y_{11} \end{bmatrix}}{\Delta Y}$$

21

Step 5 − Substitute V_1 value of Step 3 in I_1 equation of Step 2.

$$I_1 = Y_{11}\{(\frac{-Y_{22}}{Y_{21}})V_2 - (\frac{-1}{Y_{21}})I_2\} + Y_{12}V_2$$

$$\Rightarrow I_1 = (\frac{Y_{12}Y_{21} - Y_{11}Y_{22}}{Y_{21}})V_2 - (\frac{-Y_{11}}{Y_{21}})I_2$$

Step 6 − The above equation is in the form of $I_1 = CV_2 - DI_2$. Here,

$$C = \frac{Y_{12}Y_{21} - Y_{11}Y_{22}}{Y_{21}}$$

$$D = \frac{-Y_{11}}{Y_{21}}$$

Step 7 − Therefore, the **T parameters matrix** is

$$\begin{bmatrix} A & B \\ C & D \end{bmatrix} = \begin{bmatrix} \frac{-Y_{22}}{Y_{21}} & \frac{-1}{Y_{21}} \\ \frac{Y_{12}Y_{21} - Y_{11}Y_{22}}{Y_{21}} & \frac{-Y_{11}}{Y_{21}} \end{bmatrix}$$

T parameters to h-parameters

Here, we have to represent h-parameters in terms of T parameters. So, in this case hparameters are the desired parameters and T parameters are the given parameters.

Step 1 − We know that, the following h-parameters of a two port network.

$$h_{11} = \frac{V_1}{I_1}, \; when \; V_2 = 0$$

$$h_{12} = \frac{V_1}{V_2}, \; when \; I_1 = 0$$

$$h_{21} = \frac{I_2}{I_1}, \; when \; V_2 = 0$$

$$h_{22} = \frac{I_2}{V_2}, \; when \; I_1 = 0$$

Step 2 − We know that the following set of two equations of two port network regarding **T parameters**.

$V_1 = AV_2 - BI_2$ **Equation 5**

$I_1 = CV_2 - DI_2$ **Equation 6**

Step 3 − Substitute $V_2 = 0$ in the above equations in order to find the two h-parameters, h_{11} and h_{21}.

$$\Rightarrow V_1 = -BI_2$$

$$\Rightarrow I_1 = -DI_2$$

Substitute, V_1 and I_1 values in h-parameter, h_{11}.

$$-BI_2$$

Step 2 − We know that the following set of two equations of two port network regarding **T parameters**.

$V_1 = AV_2 - BI_2$ **Equation 5**

$I_1 = CV_2 - DI_2$ **Equation 6**

Step 3 − Substitute $V_2 = 0$ in the above equations in order to find the two h-parameters, h_{11} and h_{21}.

$$\Rightarrow V_1 = -BI_2$$

$$\Rightarrow I_1 = -DI_2$$

Substitute, V_1 and I_1 values in h-parameter, h_{11}.

$$h_{11} = \frac{-BI_2}{-DI_2}$$

$$\Rightarrow h_{11} = \frac{B}{D}$$

Substitute I_1 value in h-parameter h_{21}.

$$h_{21} = \frac{I_2}{-DI_2}$$

$$\Rightarrow h_{21} = -\frac{1}{D}$$

Step 4 − Substitute $I_1 = 0$ in the second equation of step 2 in order to find the h-parameter h_{22}.

$$0 = CV_2 - DI_2$$

$$\Rightarrow CV_2 = DI_2$$

$$\Rightarrow \frac{I_2}{V_2} = \frac{C}{D}$$

$$\Rightarrow h_{22} = \frac{C}{D}$$

Step 5 − Substitute $I_2 = (\frac{C}{D})V_2$ in the first equation of step 2 in order to find the h-parameter, h_{12}.

$$V_1 = AV_2 - B(\frac{C}{D})V_2$$

$$\Rightarrow V_1 = (\frac{AD - BC}{D})V_2$$

$$\Rightarrow \frac{V_1}{V_2} = \frac{AD - BC}{D}$$

$$\Rightarrow h_{12} = \frac{AD - BC}{D}$$

Step 6 − Therefore, the h-parameters matrix is

$$\begin{bmatrix} h_{11} & h_{12} \\ h_{21} & h_{22} \end{bmatrix} = \begin{bmatrix} \frac{B}{D} & \frac{AD-BC}{D} \\ -\frac{1}{D} & \frac{C}{D} \end{bmatrix}$$

h-parameters to Z parameters

Here, we have to represent Z parameters in terms of h-parameters. So, in this case Z parameters are the desired parameters and h-parameters are the given parameters.

Step 1 − We know that, the following set of two equations of two port network regarding **Z parameters**.

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

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

Step 2 − We know that, the following set of two equations of two-port network regarding **h-parameters**.

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

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

Step 3 − We can modify the above equation as

$$\Rightarrow I_2 - h_{21}I_1 = h_{22}V_2$$

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

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

Step 3 − We can modify the above equation as

$$\Rightarrow I_2 - h_{21}I_1 = h_{22}V_2$$

$$\Rightarrow V_2 = \frac{I_2 - h_{21}I_1}{h_{22}}$$

$$\Rightarrow V_2 = (\frac{-h_{21}}{h_{22}})I_1 + (\frac{1}{h_{22}})I_2$$

The above equation is in the form of $V_2 = Z_{21}I_1 + Z_{22}I_2. Here,$

$$Z_{21} = \frac{-h_{21}}{h_{22}}$$

$$Z_{22} = \frac{1}{h_{22}}$$

Step 4 − Substitute V_2 value in first equation of step 2.

$$V_1 = h_{11}I_1 + h_{21}\{(\frac{-h_{21}}{h_{22}})I_1 + (\frac{1}{h_{22}})I_2\}$$

$$\Rightarrow V_1 = (\frac{h_{11}h_{22} - h_{12}h_{21}}{h_{22}})I_1 + (\frac{h_{12}}{h_{22}})I_2$$

The above equation is in the form of $V_1 = Z_{11}I_1 + Z_{12}I_2$. Here,

$$Z_{11} = \frac{h_{11}h_{22} - h_{12}h_{21}}{h_{22}}$$

$$Z_{12} = \frac{h_{12}}{h_{22}}$$

Step 5 – Therefore, the Z parameters matrix is

$$\begin{bmatrix} Z_{11} & Z_{12} \\ Z_{21} & Z_{22} \end{bmatrix} = \begin{bmatrix} \frac{h_{11}h_{22} - h_{12}h_{21}}{h_{22}} & \frac{h_{12}}{h_{22}} \\ \frac{-h_{21}}{h_{22}} & \frac{1}{h_{22}} \end{bmatrix}$$

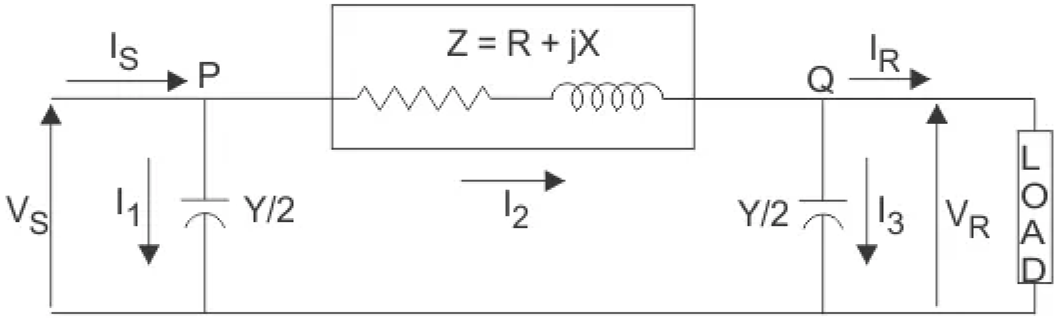
In this way, we can convert one set of parameters into other set of parameters.

8.6 T and π representation.

Nominal Π Representation of a Medium Transmission Line

In case of a nominal Π representation (i.e. nominal pi model), the lumped series impedance is placed at the middle of the circuit whereas the shunt admittances are at the ends. As we can see from the diagram of the Π network below, the total lumped shunt admittance is divided into 2 equal halves, and each half with value Y/2 is placed at both the sending and the receiving end while the entire circuit impedance is between the two.

The shape of the circuit so formed resembles that of a symbol **Π**, and for this reason, it is known as the nominal Π representation of a **medium transmission line**. It is mainly used for determining the general circuit parameters and performing load flow analysis.



Nominal network of medium transmission line

As we can see here, V_S and V_R is the supply and receiving end voltages respectively, and I_s is the current flowing through the supply end. I_R is the current flowing through the receiving end of the circuit. I_1 and I_3 are the values of currents flowing through the admittances. And I_2 is the current through the impedance Z

Now applying KCL, at node P, we get.

$$I_S = I_1 + I_2 \cdots\cdots\cdots (1)$$

Similarly applying KCL, to node Q.

$$I_2 = I_3 + I_R \cdots\cdots\cdots (2)$$

Now substituting equation (2) to equation (1)

$$I_S = I_1 + I_3 + I_R$$

$$= \frac{Y}{2}V_S + \frac{Y}{2}V_R + I_R \text{--------}(3)$$

Now by applying KVL to the circuit,

$$V_S = V_R + ZI_2$$

$$= V_R + Z(V_R\frac{Y}{2} + I_R)$$

$$= (Z\frac{Y}{2} + 1)\, V_R + ZI_R \text{----------}(4)$$

Now substituting equation (4) to equation (3), we get.

$$I_s = \frac{Y}{2}[(\frac{Y}{2}Z + 1)V_R + ZI_R] + \frac{Y}{2}V_R + I_R$$

$$= Y(\frac{Y}{4}Z + 1)V_R + (\frac{Y}{2}Z + 1)I_R \text{------------}(5)$$

Comparing equation (4) and (5) with the standard ABCD parameter equations

$$V_S = AV_R + BI_R$$

$$I_S = CV_R + DI_R$$

We derive the ABCD parameters of a medium transmission line as:

$$A = (\frac{Y}{2}Z + 1)$$

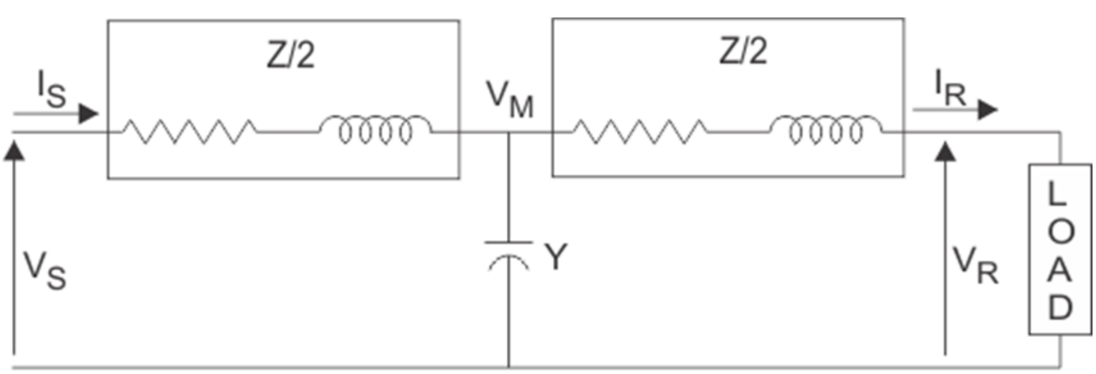
$$B = Z\ \Omega$$

$$C = Y(\frac{Y}{4}Z + 1)$$

$$D = (\frac{Y}{2}Z + 1)$$

Nominal T Representation of a Medium Transmission Line

In the nominal T model of a medium transmission line the lumped shunt admittance is placed in the middle, while the net series impedance is divided into two equal halves and placed on either side of the shunt admittance. The circuit so formed resembles the symbol of a capital T, and hence is known as the nominal T network of a medium length transmission line and is shown in the diagram below.



Nominal T representation of a medium transmission line

Here also V_s and V_r is the supply and receiving end voltages respectively, and
I_s is the current flowing through the supply end.
I_r is the current flowing through the receiving end of the circuit.
Let M be a node at the midpoint of the circuit, and the drop at M, be given by V_m.
Applying KVL to the above network we get,

$$\frac{V_S - V_M}{Z/2} = Y V_M + \frac{V_m - V_R}{Z/2}$$

$$\text{Or } V_M = \frac{2(V_S + V_R)}{YZ + 4} \text{-----------------(6)}$$

And the receiving end current

$$\text{Or } I_R = \frac{2(V_M - V_R)}{Z/2} \text{-----------------(7)}$$

Now substituting V_M from equation (6) to (7) we get,

$$\text{Or } I_R = \frac{[(2V_S + V_R)/YZ+4] - V_R}{Z/2}$$

Rearranging the above equation:

$$V_S = (\frac{Y}{2}Z + 1)V_R + Z(\frac{Y}{4}Z + 1)I_R \text{ ----------------(8)}$$

Now the sending end current is,

$$I_s = YV_M + I_R \cdots\cdots\cdot(9)$$

Substituting the value of V_M to equation (9) we get,

$$\text{Or } I_S = Y V_R + (\frac{Y}{2}Z + 1)I_R \text{ ----------------(10)}$$

Again comparing equation (8) and (10) with the standard ABCD parameter equations,

$$V_S = AV_R + BI_R$$

$$I_S = CV_R + DI_R$$

The parameters of the **T** network of a **medium transmission line** are

$$A = (\frac{Y}{2}Z + 1)$$

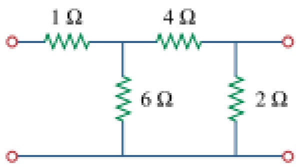
$$B = Z(\frac{Y}{4}Z + 1)\ \Omega$$

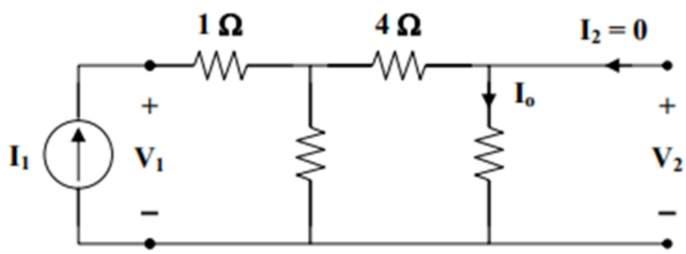
$$C = Y \text{ mho}$$

$$D = (\frac{Y}{2}Z + 1)$$

8.7 Solve numerical problems

Obtain the z parameters for the network in Fig. 19.65.





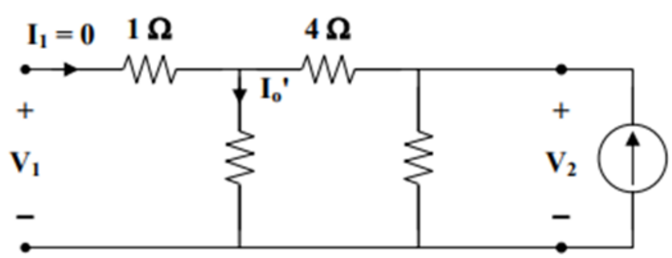
(a)

$$\mathbf{z}_{11} = \frac{\mathbf{V}_1}{\mathbf{I}_1} = 1 + 6 \,||\, (4+2) = 4\,\Omega$$

$$\mathbf{I}_o = \frac{1}{2}\mathbf{I}_1, \qquad \mathbf{V}_2 = 2\mathbf{I}_o = \mathbf{I}_1$$

$$\mathbf{z}_{21} = \frac{\mathbf{V}_2}{\mathbf{I}_1} = 1\,\Omega$$

To get $\mathbf{z}_{22}$ and $\mathbf{z}_{12}$, consider the circuit in Fig. (b).

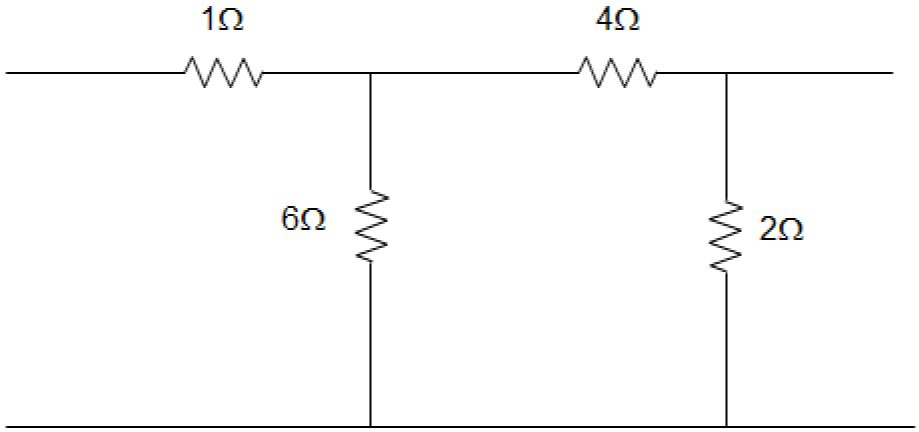


(b)

$$\mathbf{z}_{22} = \frac{\mathbf{V}_2}{\mathbf{I}_2} = 2 \,||\, (4+6) = 1.667\,\Omega$$

Short questions

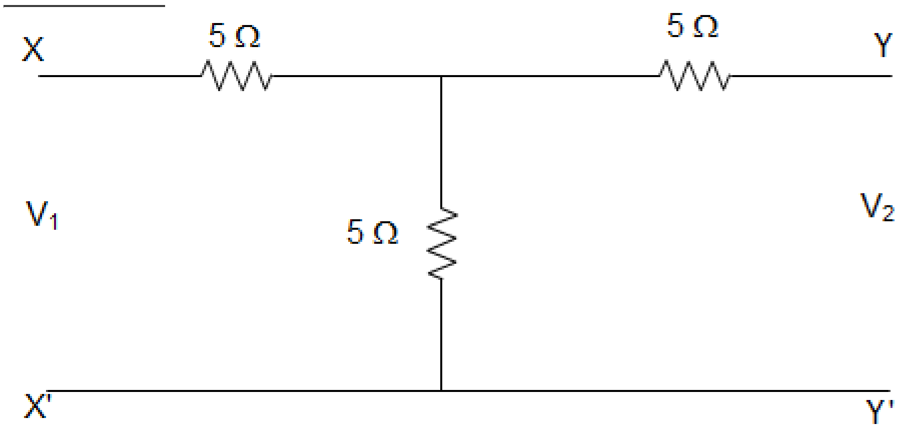
1. 1. For the circuit given below, the value of the z_{12} parameter is ___________



a) z_{12} = 1 Ω
b) z_{12} = 4 Ω
c) z_{12} = 1.667 Ω
d) z_{12} = 2.33 Ω

Answer: a
Explanation: z_{11} = V1I1 = 1 + 6 || (4+2) = 4Ω
I_0 = 12I1
$V_2 = 2I_0 = I_1$
z_{21} = V2I1 = 1Ω
z_{22} = V2I2 = 2 || (4+6) = 1.667Ω
So, I'_0 = 22+10I2=16I2
$V_1 = 6I'_0 = I_2$
z_{12} = V1I2 = 1Ω
Hence, [z] = [4:1; 1:1.667] Ω.

3. In the circuit given below, the value of the hybrid parameter h_{21} is __________



a) 10 Ω
b) 0.5 Ω
c) 5 Ω
d) 2.5 Ω

Answer: b

Explanation: Hybrid parameter h_{21} is given by, h_{21} = I2I1, when $V_2 = 0$.

Therefore short circuiting the terminal Y-Y', and applying Kirchhoff's law, we get,

$-5 I_2 – (I_2 – I_1)5 = 0$

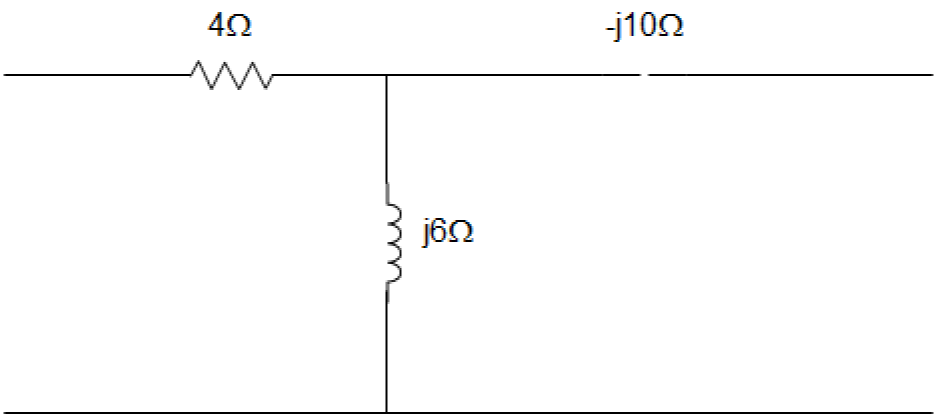
Or, $-I_2 = I_2 – I_1$

Or, $-2I_2 = -I_1$

∴ I2I1=12

Hence h_{21} = 0.5 Ω.

3. For the circuit given below, the value of z_{22} parameter is ____________



a) z_{22} = 4 + j6 Ω

b) z_{22} = j6 Ω

c) z_{22} = -j4 Ω

d) z_{22} = -j6 + 4 Ω

View Answer

Answer: c

Explanation: z_{12} = j6 = z_{21}

$z_{11} – z_{12}$ = 4

Or, $z_{11} = z_{12}$ + 4 = 4 + j6 Ω

And $z_{22} – z_{12}$ = -j10

Or, $z_{22} = z_{12}$ + -j10 = -j4 Ω

∴ [z] = [4+j6:j6; j6:-j4] Ω.

4.what is h parameter ?

Hybrid parameters (also known as h parameters) are known as 'hybrid' parameters as they use Z parameters, Y parameters, voltage ratio, and current ratios to represent the relationship between voltage and current in a two port network. H parameters are useful in describing the input-output characteristics of circuits where it is hard to measure Z or Y parameters (such as in a transistor).

H parameters encapsulate all the important linear characteristics of the circuit, so they are very useful for simulation purposes. The relationship between voltages and current in h parameters can be represented as:

Long questions

Explain the inter relation between parameters?

TRANSIENTS

CHAPTER-9

9.1 Steady state & transient state response.

if the output of an electric circuit for an input varies with respect to time, then it is called as time response. The time response consists of following two parts.

- Transient Response
- Steady state Response

In this chapter, first let us discuss about these two responses and then observe these two responses in a series RL circuit, when it is excited by a DC voltage source.

Transient Response

After applying an input to an electric circuit, the output takes certain time to reach steady state. So, the output will be in transient state till it goes to a steady state. Therefore, the response of the electric circuit during the transient state is known as transient response.

The transient response will be zero for large values of ‘t’. Ideally, this value of ‘t’ should be infinity. But, practically five time constants are sufficient.

Presence or Absence of Transients

Transients occur in the response due to sudden change in the sources that are applied to the electric circuit and / or due to switching action. There are two possible switching actions. Those are opening switch and closing switch.

- The transient part will not present in the response of an electrical circuit or network, if it contains only resistances. Because resistor is having the ability to adjust any amount of voltage and current.
- The transient part occurs in the response of an electrical circuit or network due to the presence of energy storing elements such as inductor and capacitor. Because they can't change the energy stored in those elements instantly.

9.2 Response to R-L, R-C & RLC circuit under DC condition.

1. So far we have considered dc resistive network in which currents and voltages were independent of time. More specifically, Voltage (cause input) and current (effect output) responses displayed simultaneously except for a constant multiplicative factor (VR). Two basic passive elements namely, inductor and capacitor are introduced in the dc network. Automatically, the question will arise whether or not the methods developed in lesson-3 to lesson-8 for resistive circuit analysis are still valid. The voltage/current relationship for these two passive elements are defined by the derivative (voltage across the inductor

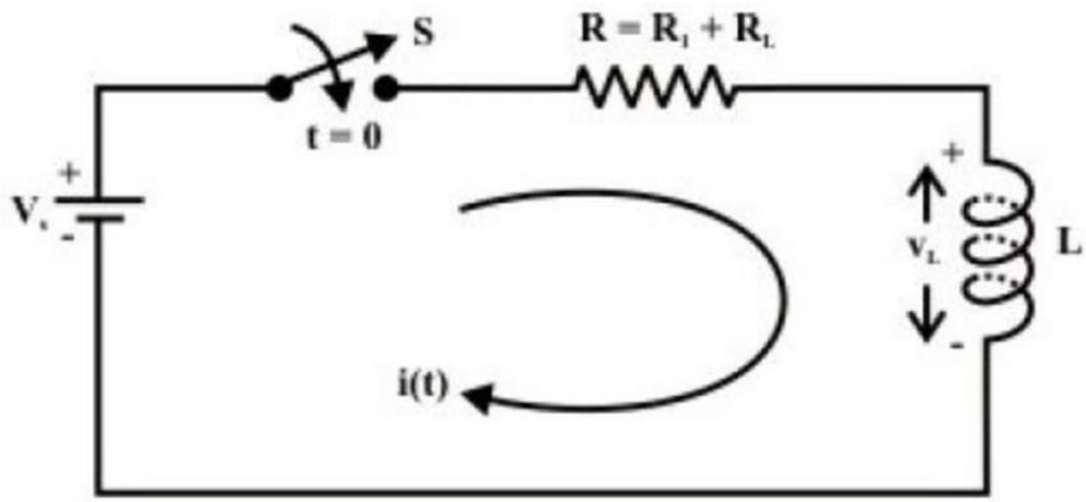
$$v_L(t) = L\frac{di_L(t)}{dt}$$

where =current flowing through the inductor ; current through the capacitor

$$i_C(t) = C\frac{dv_C(t)}{dt},$$

voltage across the capacitor) or in integral form as ()C

$$i_L(t) = \frac{1}{L}\int_0^t v_L(t)dt + i_L(0) \quad or \quad v_C(t) = \frac{1}{C}\int_0^t i(t)dt + v_C(0)$$



Our problem is to study the growth of current in the circuit through two stages, namely; (i) dc transient response (ii) steady state response of the system

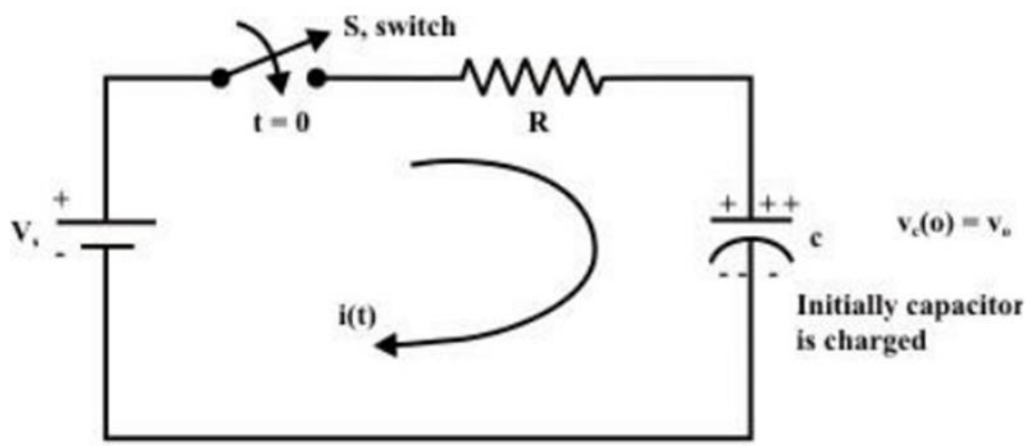
D.C Transients: The behavior of the current and the voltage in the circuit switch is closed until it reaches its final value is called dc transient response of the concerned circuit. The response of a circuit (containing resistances, inductances, capacitors and switches) due to sudden application of voltage or current is called transient response. The most common instance of a transient response in a circuit occurs when a switch is turned on or off –a rather common event in an electric circuit.

TRANSIENT RESPONSE OF RC CIRCUITS

Ideal and real capacitors: An ideal capacitor has an infinite dielectric resistance and plates (made of metals) that have zero resistance. However, an ideal capacitor does not exist as all dielectrics have some leakage current and all capacitor plates have some resistance. A capacitor's of how much charge (current) it will allow to leak through the dielectric medium. Ideally, a charged

capacitor is not supposed to allow leaking any current through the dielectric medium and also assumed not to dissipate any power loss in capacitor plates resistance. Under this situation, the model as shown in fig. represents the ideal capacitor. However, all real or practical capacitor leaks current to some extend due to leakage resistance of dielectric medium. This leakage resistance can be visualized as a resistance connected in parallel with the capacitor and power loss in capacitor plates can be realized with a resistance connected in series with capacitor. The model of a real capacitor is shown in fig.

Let us consider a simple series RC−circuit shown in fig. is connected through a switch 'S' to a constant voltage source



The switch 'S' is closed at time 't=0' It is assumed that the capacitor is initially charged with a voltage and the current flowing through the circuit at any instant of time '' after closing the switch is

RANSIENT RESPONSE OF RLC CIRCUITS

In the preceding lesson, our discussion focused extensively on dc circuits having resistances with either inductor () or capacitor () (i.e., single storage element) but not both. Dynamic response of such first order system has been studied and discussed in detail. The presence of resistance, inductance, and capacitance in the dc circuit introduces at least a second order differential equation or by two simultaneous coupled linear first order differential equations. We shall see in next section that the complexity of analysis of second order circuits increases significantly when compared with that encountered with first order circuits. Initial conditions for the circuit variables and their derivatives play an important role and this is very crucial to analyze a second order dynamic system.

Response of a series R-L-C circuit

Consider a series RLcircuit as shown in fig.11.1, and it is excited with a dc voltage source C−−sV.

Applying around the closed path for ,

$$L\frac{di(t)}{dt}+Ri(t)+v_c(t)=V_s$$

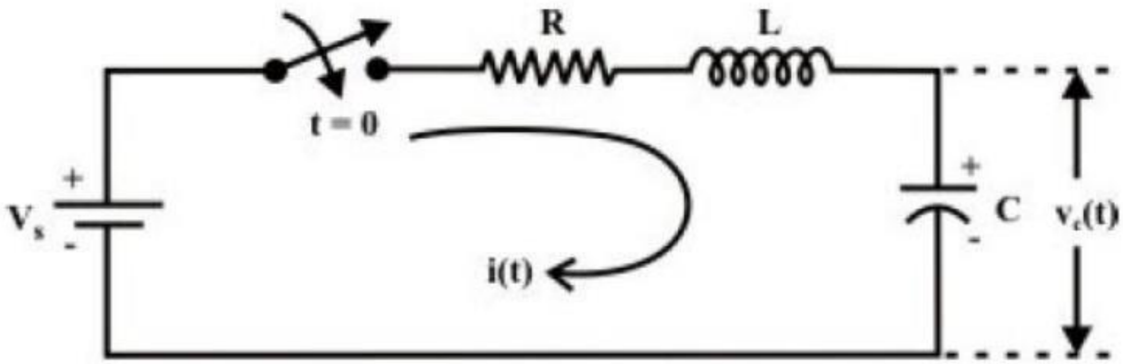


Fig. 11.1: A Simple R-L-C circuit excited with a dc voltage source

The current through the capacitor can be written as Substituting the current ‘’expression in eq.(11.1) and rearranging the terms,

$$i(t)=C\frac{dv_c(t)}{dt}$$

$$LC\frac{d^2v_c(t)}{dt^2}+RC\frac{dv_c(t)}{dt}+v_c(t)=V_s$$

The above equation is a 2nd-order linear differential equation and the parameters associated with the differential equation are constant with time. The complete solution of the above differential equation has two components; the transient response and the steady state response. Mathematically, one can write the complete solution as

$$v_c(t)=v_{cn}(t)+v_{cf}(t) = \left(A_1 e^{\alpha_1 t} + A_2 e^{\alpha_2 t}\right)+A$$

$$LC\frac{d^2v_c(t)}{dt^2}+RC\frac{dv_c(t)}{dt}+v_c(t) = 0 \Rightarrow \frac{d^2v_c(t)}{dt^2}+\frac{R}{L}\frac{dv_c(t)}{dt}+\frac{1}{LC}v_c(t) = 0$$

$$a\,\frac{d^2v_c(t)}{dt^2}+b\,\frac{dv_c(t)}{dt}+c\,v_c(t) = 0 \text{ (where } a=1,\ b=\frac{R}{L} \text{ and } c=\frac{1}{LC})$$

Since the system is linear, the nature of steady state response is same as that of forcing function (input voltage) and it is given by a constant value. Now, the first part of the total response is completely dies out with time while and it is defined as a transient or natural response of the system. The natural or transient response (see Appendix in Lesson-10) of second order differential equation can be obtained from the homogeneous equation (i.e., from force free system) that is expressed by

$$\alpha^2+\frac{R}{L}\alpha+\frac{1}{LC}=0 \quad \Rightarrow a\,\alpha^2+b\,\alpha+c=0 \text{ (where } a=1,\ b=\frac{R}{L} \text{ and } c=\frac{1}{LC})$$

and solving the roots of this equation (11.5) on that associated with transient part of the complete solution (eq.11.3) and they are given below.

1. The roots of the characteristic equation are classified in three groups depending upon the values of the parameters ,,RLand of the circuit
2.
3. Case-A (overdamped response): That the roots are distinct with negative real parts. Under this situation, the natural or transient part of the complete solution is written as

$$\alpha_1=\left(-\frac{R}{2L}+\sqrt{\left(\frac{R}{2L}\right)^2-\frac{1}{LC}}\right)=\left(-\frac{b}{2a}+\frac{1}{a}\sqrt{\left(\frac{b}{2}\right)^2-ac}\right);$$

$$\alpha_2=\left(-\frac{R}{2L}-\sqrt{\left(\frac{R}{2L}\right)^2-\frac{1}{LC}}\right)=\left(-\frac{b}{2a}-\frac{1}{a}\sqrt{\left(\frac{b}{2}\right)^2-ac}\right)$$

where, $b=\frac{R}{L}$ and $c=\frac{1}{LC}$.

$$\dot{v}_{cn}(t) = A_1 e^{\alpha_1 t} + A_2 e^{\alpha_2 t}$$

and each term of the above expression decays exponentially and ultimately reduces to zero as and it is termed as overdamped response of input free system. A system that is overdamped responds slowly to any change in excitation. It may be noted that the exponential term t→∞11tAeαtakes longer time to decay its value to zero than the term21tAeα. One can introduce a factorξ that provides an information about the speed of system response and it is defined by damping ratio

$$(\xi) = \frac{Actual\ \ damping}{critical\ \ damping} = \frac{b}{2\sqrt{ac}} = \frac{R/L}{2/\sqrt{LC}} > 1$$

7.3 Solve numerical problems

Example 7.2 A capacitor in an RC circuit with R = 25 Ω and C = 50 μF is being charged with initial zero voltage. What is the time taken for the capacitor voltage to reach 40 % of its steady state value?

Solution With R = 25 Ω and C = 50 μF, τ = RC = 1.25 x 10^{-3} s; hence 1/RC = 800 s^{-1}.

Taking the capacitor steady state voltage as E, $v_C(t) = E(1 - e^{-\frac{1}{RC}t})$

Let t_1 be the time at which the capacitor voltage becomes 0.4 E. Then

$0.4\,E = E\,(1- e^{-800\,t_1})$ i.e. $0.4 = 1 - e^{-800\,t_1}$

$e^{-800\,t_1} = 0.6$ i.e. $-800\,t_1 = \ln 0.6 = -0.5108$

Therefore, $t_1 = \dfrac{0.5108}{800}\,s = 0.6385 \times 10^{-3}\ s = 0.6385\ ms$

Example 7.3 In an RC circuit, having a time constant of 2.5 ms, the capacitor discharges with initial voltage of 80 V. (a) Find the time at which the capacitor voltage reaches 55 V, 30 V and 10 V (b) Calculate the capacitor voltage at time 1.2 ms, 3 ms and 8 ms.

Solution (a) Time constant RC = 2.5 ms; Thus $\dfrac{1}{RC} = \dfrac{1000}{2.5} = 400\ s^{-1}$

During discharge, capacitor voltage is given by $v_C(t) = 80\,e^{-400\,t}$ V

Let t_1, t_2 and t_3 be the time at which capacitor voltage becomes 55 V, 30 V and 10 V.

$55 = 80\,e^{-400t_1}$; $-400\,t_1 = \ln\dfrac{55}{80} = -0.3747$; Thus $t_1 = 0.93765$ ms

$30 = 80\,e^{-400t_2}$; $-400\,t_2 = \ln\dfrac{30}{80} = -0.9808$; Thus $t_2 = 2.452$ ms

$10 = 80\,e^{-400t_3}$; $-400\,t_3 = \ln\dfrac{10}{80} = -2.0794$; Thus $t_3 = 5.1985$ ms

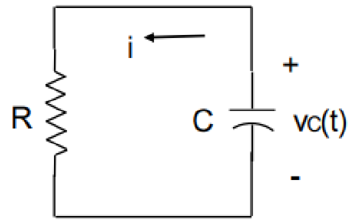
$$10 = 80\ e^{-400t_3};\ -400\ t_3 = \ln \frac{10}{80} = -2.0794;\ \text{Thus}\ t_3 = 5.1985\ \text{ms}$$

(b) $v_C(1.2\text{x}\ 10^{-3}) = 80\ e^{-400\ t} = 80\ e^{-0.48} = 49.5027\ \text{V}$

$v_C(3\text{x}\ 10^{-3}) = 80\ e^{-400\ t} = 80\ e^{-1.2} = 24.0955\ \text{V}$

$v_C\ (8\text{x}\ 10^{-3}) = 80\ e^{-400\ t} = 80\ e^{-3.2} = 3.261\ \text{V}$

Example 7.4 Consider the circuit shown below.



Given

$v_C(t) = 56\ e^{-250\ t}\ \text{V}$ for $t > 0$

$i(t) = 7\ e^{-250\ t}\ \text{mA}$ for $t > 0$

(a) Find the values of R and C. (b) Determine the time constant.

(c) At what time the voltage $v_C(t)$ will reach half of its initial value?

(c) At what time the voltage $v_C(t)$ will reach half of its initial value?

Solution (a) Given that $v_C(t) = 56\ e^{-250\ t}$ V. Therefore $\tau = RC = \frac{1}{250}$ s

$$\text{Resistance } R = \frac{v_C(t)}{i(t)} = 8000\ \Omega;\ \text{Thus capacitance } C = \frac{1}{250\ \text{X}\ 8000}\ \text{F} = 0.5\ \mu\text{F}$$

(b) Time constant $= RC = 4 \times 10^{-3}$ s = 4 ms

(c) Let t_1 be the time taken for the voltage to reach half of its initial value of 56 V.

Then, $56\, e^{-250 t_1} = 28$; i.e. $e^{-250 t_1} = 0.5$ i.e. $-250\ t_1 = \ln 0.5 = -0.6931$;

$$\text{Time } t_1 = \frac{0.6931}{250} \text{s} = 2.7724 \times 10^{-3}\ \text{s} = 2.7724\ \text{ms}$$

Short questions

1.Define response.?

The current flowing through or voltage across branches in the circuit is called response.

2. Define steady state response?.

Ans-The behavior of voltage or current does not change with time is called steady state response

3. Define transient response.?

   Ans-The voltage or current are changed from one transient state to another transient state is called transient response.

4. Why transient occurs in electric circuits?

Ans-The inductance will not allow the sudden change in current and the capacitance will not allow sudden change in voltage. Hence inductive and capacitive circuits (or in general reactive circuits0transient occurs during switching operation.

5.Define time constant of RL circuit.

Ans-The time constant of RL circuit is defined as the time taken by the current through the inductance to reach 63.21%of its final steady state value.

6.Define time constant of RC circuit.

Ans-The time constant of RC circuit is defined as the time taken by the voltage across the capacitance to reach 63.21%of its final steady state value.

Long questions

1.Write the conditions for response of an RLC series network?

2.Sketch the transient current and voltages of RL circuit.?

3.Sketch the transient current and voltages of RC circuit.?