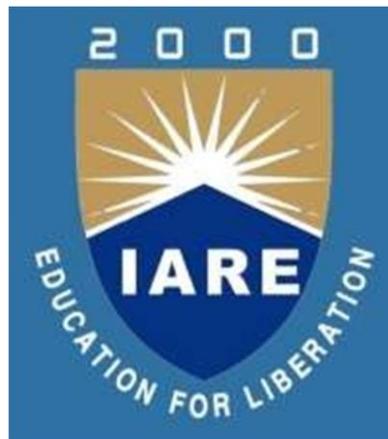# LECTURE NOTES

ON

FUNDUMENTALS OF ELECTRICAL ENGINEERING

I B.Tech I Semester

By

COMPUTER SCIENCE AND ENGINEERING

INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)
Dundigal, Hyderabad - 500 043

MODULE – I
INTRODUCTION TO ELECTRICAL CIRCUITS

Introduction

The interconnection of various electric elements in a prescribed manner comprises as an electric circuit in order to perform a desired function. The electric elements include controlled and uncontrolled source of energy, resistors, capacitors, inductors, etc. Analysis of electric circuits refers to computations required to determine the unknown quantities such as voltage, current and power associated with one or more elements in the circuit. To contribute to the solution of engineering problems one must acquire the basic knowledge of electric circuit analysis and laws. Many other systems, like mechanical, hydraulic, thermal, magnetic and power system are easy to analyze and model by a circuit. To learn how to analyze the models of these systems, first one needs to learn the techniques of circuit analysis. We shall discuss briefly some of the basic circuit elements and the laws that will help us to develop the background of subject. As we all know that modern life is overwhelmingly dependent on electricity, it is quite important for people to understand simple electrical circuits. Simple electrical circuits introduction is a good assistant for you to better know electrical circuits.

The Definition of Electrical Circuits

An electrical circuit is a closed loop of conductive material that allows electrons to flow through continuously without beginning or end. There is continuous electrical current goes from the supply to the load in an electrical circuit. People also say that a complete path, typically through conductors such as wires and through circuit elements, is called an electric circuit. An electrical circuit is an electrical device that provides a path for electrical current to flow. After you get the definition of the electrical circuit, now we are going to show you three simple electrical circuits.

Switch Circuit

A switch is a device for making and breaking the connection in an electric circuit. We operate switches for lights, fans, electric hair drier and more many times a day but we seldom try to see the connection made inside the switch circuit. The function of the switch is to connect or complete the circuit going to the load from the supply. It has moving contacts which are normally open. With a switch you can turn the device on or off, therefore, it is a very important component in an electrical circuit.

DC Lighting Circuit

As you can see from the picture below that the LED lamp uses DC supply battery. The battery is bipolar, one is anode and the other is cathode. Moreover, the anode is positive and the cathode is negative. Also, the lamp itself has two ends, one positive and the other is negative. Therefore, the anode of the battery is battery is connected to positive terminal of the lamp, meanwhile the cathode of the battery is connected to the negative terminal of the lamp. Once the above connection is complete, the LED lamp will light. Although it is simple electrical circuit, many people have no idea how to deal with the connection correctly.

Thermocouple Circuit

If you are looking to build a temperature-sensing device or you need to add sensing capabilities to a large system, you will have to familiarize yourself with thermocouples circuits and understand how to design them. A thermocouple is a device consisting of two dissimilar conductors that contact each other at one or more spots, and it is used to measure temperature. As you can see from the picture below that a thermocouple is made of two wires - iron and constantan, with a voltmeter. If the cold junction temperature is kept constant, then the EMF is proportional to the temperature of the hot junction.

Voltmeter will measure the EMF generated and this can be calibrated to measure the temperature. The temperature difference between the hot and cold junction will produce an EMF proportional to it. Because thermocouple junctions produce such low voltages, it is imperative that wire connections be very clean and tight for accurate and reliable operation. Despite these seemingly restrictive requirements, thermocouples remain one of the most robust and popular methods of industrial temperature measurement in modern use. A circuit diagram is like a map that shows electricity flows. This tutorial will show you a few of the common symbols and some of the professional terms to help you read circuit diagrams.

Learning to read electrical schematics is like learning to read maps. Electrical schematics show which electrical components used and how they are connected together. The electronic symbols consisted represent each of the components used. The symbols are connected with lines.

Recognizing Electrical Schematic Terms

Here are some of the standard and basic terms of circuit diagrams:

- Voltage: Voltage is the "pressure" or "force" of electricity, it is usually measured in volts (V) and the outlets in common house operate at 120V. Outlets of voltage may differ in other countries.
- Resistance: Resistance represents how easily electrons can flow through a certain material, and it is measured in Ohms (R or Ω). Current flow can move faster in conductors such as gold or copper, in this case we say the resistance is low. The movement of electrons is relatively slow in insulators such as plastic, wood, and air, in this case we say the resistance is high.
- Current: Current is the flow of electricity, or to be more specific, the flow of electrons. Current is measured in Amperes (Amps). The flow of current is only possible when a voltage supply is connected.
- DC (Direct Current): DC is the continuous current flow in one direction. DC can flow not just through conductors, but semi-conductors and insulators too.
- AC (Alternating Current): In AC, the current flow alternates between two directions according to a certain period, it often forms a sine wave. The frequency of AC is measured in Hertz (Hz), and it is usually 60 Hz.

Basic Elements & Introductory Concepts

- **Electrical Network:** A combination of various electric elements (Resistor, Inductor, Capacitor, Voltage source, Current source) connected in any manner what so ever is called an electrical network. We may classify circuit elements in two categories, passive and active elements.

- Passive Element: The element which receives energy (or absorbs energy) and then either converts it into heat (R) or stored it in an electric (C) or magnetic (L) field is called passive element.
- Active Element: The elements that supply energy to the circuit is called active element. Examples of active elements include voltage and current sources, generators, and electronic devices that require power supplies. A transistor is an active circuit element, meaning that it can amplify power of a signal. On the other hand, transformer is not an active element because it does not amplify the power level and power remains same both in primary and secondary sides. Transformer is an example of passive element.
- **Bilateral Element:** Conduction of current in both directions in an element (example: Resistance; Inductance; Capacitance) with same magnitude is termed as bilateral element.
- **Unilateral Element:** Conduction of current in one direction is termed as unilateral (example: Diode, Transistor) element.
- **Meaning of Response:** An application of input signal to the system will produce an output signal, the behavior of output signal with time is known as the response of the system**.**

BASIC CIRCUIT CONCEPTS

An electric circuit is formed by interconnecting components having different electric properties. It is therefore important, in the analysis of electric circuits, to know the properties of the involved components as well as the way the components are connected to form the circuit. In this introductory chapter some ideal electric components and simple connection styles are introduced. Without resort to advanced analysis techniques, we will attempt to solve simple problems involving circuits that contain a relatively small number of components connected in some relatively simple fashions. In particular we will derive a set of useful formulae for dealing with circuits that involve such simple connections as ``series'', ``parallel'', ``ladder'', ``star'' and ``delta''. This chapter serves as a review of the basic properties of electric circuits. In addition we will briefly introduce the PSPICE analysis programme and how it can be used to help analyze electric circuits.

Direction and Polarity

Current direction indicates the direction of flow of positive charge, and voltage polarity indicates the relative potential between two points. Usually, ``+'' is assigned to a higher potential point and ``-'' to a lower potential point. However, during analysis, direction and polarity can be arbitrarily assigned on circuit diagrams. Actual direction and polarity will be governed by the sign of the value. Figure 1.1 shows some examples.
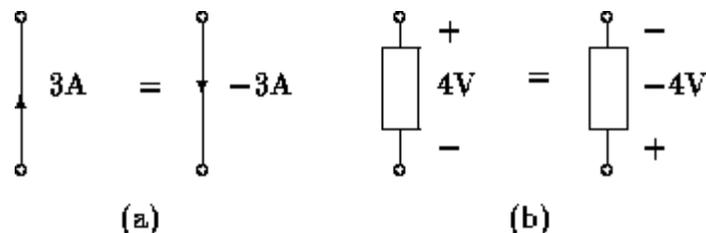

Figure 1.1: Equivalent assignments of (a) current direction; (b) voltage polarity

Ohm's Law | Equation Formula and Limitation of Ohm's Law

The most basic quantities of electricity are voltage, current and resistance or impedance. **Ohm's law** shows a simple relationship between these three quantities. This law is one of the most basic laws of electricity. This law helps to calculate the power, efficiency, current, voltage and resistance or impedance of any element of electrical circuit.

Statement of Ohm's Law

Whenever, we apply a potential difference i.e. voltage across a resistor of a closed electric circuit, current starts flowing through it. The statement of Ohm's law says that The current (I) is directly proportional to the applied voltage (V), provided temperature and all other factors remain constant. This equation presents the statement of Ohm's law. Here, we measure current in Ampere (or amps), voltage in unit of volt. The constant of proportionality R is the property of the conductor; we know it as resistance and measure it in ohm (Ω). Theoretically, the resistance has no dependence on the applied voltage, or on the flow of current. The value of R changes only if the conditions (like temperature, diameter and length etc.) of the resistor are changed by any means.

History of Ohm's Law

In the month of May 1827, Georg Simon Ohm published a book "Die Galvanische Kette, Mathematisch Bearbeitet". "Die Galvanische Kette, Mathematisch Bearbeitet" means "The Galvanic Circuit Investigated Mathematically". He presented the relationship between voltage (V), current (I), and resistance (R) based on his experimental data, in this book. Georg Simon Ohm had defined the fundamental interrelationship between current, voltage and resistance of a circuit which was later named Ohm's law. Because of this law and his excellence in the field of science and academics, he got the Copley Medal award in 1841. In 1872 the unit of electrical resistance was named 'OHM" in his honor. We can understand the physics behind Ohm's law well if we examine it from atomic level of a metal. A metal conductor contains plenty of free electrons. These free electrons randomly move in the conductor. When, we apply a voltage, across the conductor, the free electrons keep being accelerated towards higher potential end due to electrostatic force of the applied voltage. This means they acquire some kinetic energy as they move towards the + Ve end of the conductor. However, before they get very far they collide with an atom or ion, lose some of their kinetic energy and may bounce back. Again due to presence of static electric field the free electrons again being accelerated. This keeps happening. That means, even after application of external electric field, there will be still random motion in the free electrons of the conductor. Each free electron drifts towards +Ve end with its inherent random motion. As a result, the free electrons tend to "drift" towards the + Ve end, bouncing around from atom to atom on the way. This is how the materials resist a current. If we apply more voltage across the conductor, the more free electrons will move with more acceleration

which causes more drift velocity of the electrons. The drift velocity of the electrons is proportional to the applied static electric field. That more electrons pass through a cross section per unit time, which means more charge transfer per unit time. The rate of charge transfer per unit time is current.

$$I \propto V$$

Hence the current (I) we get is also proportional to the applied voltage (V).

Applications of Ohm's Law

The applications of ohm's law are that it helps us in determining either voltage, current or impedance or resistance of a linear electric circuit when the other two quantities are known to us. Apart from that, it makes power calculation a lot simpler, like when we know the value of the resistance for a particular circuit element, we need not know both the current and the voltage to calculate the power dissipation

$$V = RI \Rightarrow I = \frac{V}{R}$$

since, P = VI.

To replace either the voltage or current in the above expression to produce the result

$$Thus, \ P = VI = \frac{V^2}{R} = I^2 R \quad (Since \ V = IR)$$

We can see from the results, that the rate of energy loss varies with the square of the voltage or current. When we double the voltage applied to a circuit, obeying Ohm's law, the rate at which energy is supplied (or power) gets four times bigger. Similarly, the power dissipation at a circuit element is increased by 4 times when we make double the current through it.

Limitation of Ohm's Law

The limitations of Ohm's law are explained as follows:

1. This law cannot be applied to unilateral networks. A unilateral network has unilateral elements like diode, transistors, etc., which do not have same voltage current relation for both directions of current.
2. **Ohm's law** is also not applicable for non – linear elements.

Non-linear elements are those which do not have current exactly proportional to the applied voltage, that means the resistance value of those elements changes for different values of voltage and current. Examples of non – linear elements are thyristor, electric arc, etc.

Basic circuit elements

Definition of Resistor

A resistor offers resistance to the flow of current. The resistance is the measure of opposition to the flow of current in a resistor. More resistance means more opposition to current. The unit of resistance is ohm and it is represented as Ω. When one volt potential difference is applied across a resistor and for that one ampere of current flows through it, the resistance of the resistor is said to be one Ω. Resistor is one of the most essential passive elements in electrical and electronics engineering.

Resistor

It is some time required to introduce electrical resistance in different circuit to limit the current through it. Resistor is an element of circuit which does the same. Such as series connected resistor limits the current flowing through the light emitting diode (LED). In addition to that resistors serve many other purposes in electrical and electronic applications. The most essential requirement of a resistor is that its value of electrical resistance should not vary with temperature for a wide range. That means resistance variation with temperature must be as minimal as possible for a wide range of temperature. In other word the temperature coefficient of resistance of must be minimum for the materials by which a resistor is made of. Power Rating of Resistor

When current passes through a resistor there would be I2R loss and hence as per Joules law of heating there must be temperature rise in the resistor. A resistor must be operated within a temperature limit so that there should not be any permanent damage due high temperature. The power rating of resistor is defined as the maximum power that a resistor can dissipate in form of heat to maintain the temperature within maximum allowable limit. How much power a resistor will dissipate depends upon material, dimensions, voltage rating, maximum temperature limit of the resistor and ambient temperature.

Voltage Rating of Resistor

This rating is defined as the maximum voltage that can be applied across a resistor due to which power dissipation will be within its allowable limit. Actually voltage rating of resistor is related to the power rating. As we know that power rating of resistor is expressed as

$$P = \frac{V^2}{R} \Rightarrow V = \sqrt{PR}$$

Where, V is the applied voltage across the resistor and R is the resistance value of the resistor in ohms. From above equation it is clear that for limiting P, V must be limited for a particular resistor of resistance R. This V is voltage rating of resistor of power rating P watts and resistance R Ω.

Resistor Color Code

Suppose we have a carbon composition resistor which has four color bands among which first band is blue second band is yellow, third band is red and fourth band is golden. So from the above rule the first digit of the number will be 6 (as Blue ⇒ 6), the second digit of the number will be 4 (as Yellow ⇒ 4) and the multiplier of this two digit number will be 102 (as Red ⇒ 2). Hence, electrical resistance value of the resistor will be 64 × 102 Ω. The tolerance of that value may be ± 5 % as the color of fourth band is golden.

Resistance

When voltage is applied to a piece of metal wire, as shown in figure 1.2 (a), the current I flowing through the wire is proportional to the voltage V across two points in the wire. This property is known as Ohm's law, which reads

$$V = IR \quad \text{or} \quad I = GV$$

where R is called resistance, and G is called conductance. The resistance R and the conductance G of the same piece of wire is related by R = 1/G. Resistance is measured in ohms (Ω) and conductance in siemens (S or ℧).
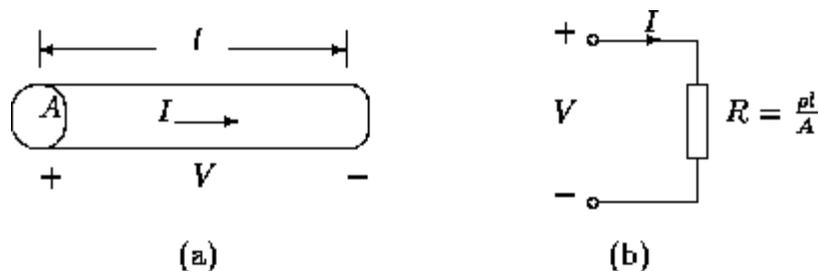

Figure 1.2: Ohm's law. (a) Metal wire; (b) circuit symbol

Any apparatus/device that has this property is called a resistor. Study of the physics of resistance shows that it is proportional to the length of the metal wire, l, and inversely proportional to the cross-sectional area, A,

i.e.,
$$R = \frac{\rho l}{A}$$

where the proportionality constant ρ is known as the resistivity of the metal. We may calculate the power required to pass current I through a resistor of resistance R using the previously derived formula, i.e.,

$$P = VI$$

$$P = \frac{V^2}{R} = GV^2 = I^2R \geq 0 \quad \text{for all } V, I$$

Using the Ohm's law equation, we get

The last inequality defines a property called passivity.

INDUCTOR:-

Definition of Inductance

If a changing flux is linked with a coil of a conductor there would be an emf induced in it. The property of the coil of inducing emf due to the changing flux linked with it is known as **inductance of the coil**. Due to this property all electrical coil can be referred as **inductor**. In other way, an inductor can be defined as an energy storage device which stores energy in form of magnetic field.

Theory of Inductor

A current through a conductor produces a magnetic field surround it. The strength of this field depends upon the value of current passing through the conductor. The direction of the magnetic field is found using the right hand grip rule, which shown. The flux pattern for this magnetic field would be number of concentric circle perpendicular to the detection of current.

Now if we wound the conductor in form of a coil or solenoid, it can be assumed that there will be concentric circular flux lines for each individual turn of the coil as shown. But it is not possible practically, as if concentric circular flux lines for each individual turn exist, they will intersect each other. However, since lines of flux cannot intersect, the flux lines for individual turn will distort to form complete flux loops around the whole coil as shown. This flux pattern of a current carrying coil is similar to a flux pattern of a bar magnet as shown.

Now if the current through the coil is changed, the magnetic flux produced by it will also be changed at same rate. As the flux is already surrounds the coil, this changing flux obviously links the coil. Now according to Faraday's law of electromagnetic induction, if changing flux links with a coil, there would be an induced emf in it. Again as per Lenz's law this induced emf opposes every cause of producing it. Hence, the induced emf is in opposite of the applied voltage across the coil.

Types of Inductor

There are many types of inductors; all differ in size, core material, type of windings, etc. so they are used in wide range of applications. The maximum capacity of the inductor gets specified by the type of core material and the number of turns on coil. Depending on the value, inductors typically exist in two forms, fixed and variable. The number of turns of the fixed coil remains the same. This type is like resistors in shape and they can be distinguished by the fact that the first color band in fixed inductor is always silver. They are usually used in electronic equipment as in radios, communication apparatus, electronic testing instruments, etc. The number of turns of the coil in variable inductors, changes depending on the design of the inductor. Some of them are designed to have taps to change the number of turns. The other design is fabricated to have a many fixed inductors for which, it can be switched into parallel or series combinations. They often get used in modern electronic equipment. Core or heart of inductor is the main part of the inductor. Some types of inductor depending on the material of the core will be discussed. Ferromagnetic Core Inductor or Iron-core Inductors

This type uses ferromagnetic materials such as ferrite or iron in manufacturing the inductor for increasing the inductance. Due to the high magnetic permeability of these materials, inductance can be increased in response of increasing the magnetic field. At high frequencies it suffers from core loses, energy loses, that happens in ferromagnetic cores.

Air Core Inductor

Air cored inductor is the type where no solid core exists inside the coils. In addition, the coils that wound on nonmagnetic materials such as ceramic and plastic, are also considered as air cored. This type does not use magnetic materials in its construction. The main advantage of this form of inductors is that, at high magnetic field strength, they have a minimal signal loss. On the other hand, they need a bigger number of turns to get the same inductance that the solid cored inductors would produce. They are free of core losses because they are not depending on a solid core.

Toroidal Core Inductor

Toroidal Inductor constructs of a circular ring-formed magnetic core that characterized by it is magnetic with high permeability material like iron powder, for which the wire wounded to get inductor. It works pretty well in AC electronic circuits' application. The advantage of this type is that, due to its symmetry, it has a minimum loss in magnetic flux; therefore it radiates less electromagnetic interference near circuits or devices. Electromagnetic interference is very important in electronics that require high frequency and low power.

This form gets typified by its stacks made with thin steel sheets, on top of each other designed to be parallel to the magnetic field covered with insulating paint on the surface; commonly on oxide finish. It aims to block the eddy currents between steel sheets of stacks so the current keeps flowing through its sheet and minimizing loop area for which it leads to great decrease in the loss of energy. Laminated core inductor is also a low frequency inductor. It is more suitable and used in transformer applications.

Powdered Iron Core

Its core gets constructed by using magnetic materials that get characterized by its distributed air gaps. This gives the advantage to the core to store a high level of energy comparing to other types. In addition, very good inductance stability is gained with low losses in eddy current and hysteresis. Moreover, it has the lowest cost alternative.

Choke

The main purpose of it is to block high frequencies and pass low frequencies. It exists in two types; RF chokes and power chokes.

Applications of Inductors

In general there are a lot of applications due to a big variety of inductors. Here are some of them. Generally the inductors are very suitable for radio frequency, suppressing noise, signals, isolation and for high power applications. More applications summarized here:

- Energy Storage
- Sensors
- Transformers
- Filters

- Motors

Introduction to Capacitors

Just like the Resistor, the **Capacitor**, sometimes referred to as a condenser, is a simple passive device that is used to "store electricity" on its plates. The capacitor is a component which has the ability or "capacity" to store energy in the form of an electrical charge producing a potential difference (*Static Voltage*) across its plates, much like a small rechargeable battery. There are many different kinds of capacitors available from very small capacitor beads used in resonance circuits to large power factor correction capacitors, but they all do the same thing, they store charge. In its basic form, a capacitor consists of two or more parallel conductive (metal) plates which are not connected or touching each other, but are electrically separated either by air or by some form of a good insulating material such as waxed paper, mica, ceramic, plastic or some form of a liquid gel as used in electrolytic capacitors. The insulating layer between a capacitors plates is commonly called the **Dielectric**.

A Typical Capacitor

Due to this insulating layer, DC current can not flow through the capacitor as it blocks it allowing instead a voltage to be present across the plates in the form of an electrical charge. The conductive metal plates of a capacitor can be either square, circular or rectangular, or they can be of a cylindrical or spherical shape with the general shape, size and construction of a parallel plate capacitor depending on its application and voltage rating. When used in a direct current or DC circuit, a capacitor charges up to its supply voltage but blocks the flow of current through it because the dielectric of a capacitor is non-conductive and basically an insulator. However, when a capacitor is connected to an alternating current or AC circuit, the flow of the current appears to pass straight through the capacitor with little or no resistance. There are two types of electrical charge, positive charge in the form of Protons and negative charge in the form of Electrons. When a DC voltage is placed across a capacitor, the positive (+ve) charge quickly accumulates on one plate while a corresponding and opposite negative (-ve) charge accumulates on the other plate. For every particle of +ve charge that arrives at one plate a charge of the same sign will depart from the -ve plate. Then the plates remain charge neutral and a potential difference due to this charge is established between the two plates. Once the capacitor reaches its steady state condition an electrical current is unable to flow through the capacitor itself and around the circuit due to the insulating properties of the dielectric used to separate the plates. The flow of electrons onto the plates is known as the capacitors Charging Current which continues to flow until the voltage across both plates (and hence the capacitor) is equal to the applied voltage Vc. At this point the capacitor is said to be "fully charged" with electrons. The strength or rate of this charging current is at its maximum value when the plates are

fully discharged (initial condition) and slowly reduces in value to zero as the plates charge up to a potential difference across the capacitors plates equal to the source voltage. The amount of potential difference present across the capacitor depends upon how much charge was deposited onto the plates by the work being done by the source voltage and also by how much capacitance the capacitor has and this is illustrated below.

The parallel plate capacitor is the simplest form of capacitor. It can be constructed using two metal or metallised foil plates at a distance parallel to each other, with its capacitance value in Farads, being fixed by the surface area of the conductive plates and the distance of separation between them. Altering any two of these values alters the the value of its capacitance and this forms the basis of operation of the variable capacitors. Also, because capacitors store the energy of the electrons in the form of an electrical charge on the plates the larger the plates and/or smaller their separation the greater will be the charge that the capacitor holds for any given voltage across its plates. In other words, larger plates, smaller distance, more capacitance.

By applying a voltage to a capacitor and measuring the charge on the plates, the ratio of the charge Q to the voltage V will give the capacitance value of the capacitor and is therefore given as: C = Q/V this equation can also be re-arranged to give the more familiar formula for the quantity of charge on the plates as: Q = C x V. Although we have said that the charge is stored on the plates of a capacitor, it is more correct to say that the energy within the charge is stored in an "electrostatic field" between the two plates. When an electric current flows into the capacitor, charging it up, the electrostatic field becomes more stronger as it stores more energy. Likewise, as the current flows out of the capacitor, discharging it, the potential difference between the two plates decreases and the electrostatic field decreases as the energy moves out of the plates. The property of a capacitor to store charge on its plates in the form of an electrostatic field is called the Capacitance of the capacitor. Not only that, but capacitance is also the property of a capacitor which resists the change of voltage across it.

The Capacitance of a Capacitor

Capacitance is the electrical property of a capacitor and is the measure of a capacitors ability to store an electrical charge onto its two plates with the unit of capacitance being the **Farad** (abbreviated to F) named after the British physicist Michael Faraday. Capacitance is defined as being that a capacitor has the capacitance of **One Farad** when a charge of **One Coulomb** is stored on the plates by a voltage of **One volt**. Note that capacitance, C is always positive in value and has no negative units. However, the Farad is a very large unit of measurement to use on its own so sub-multiples of the Farad are generally used such as micro-farads, nano-farads and pico-farads, for example.

Standard Units of Capacitance

    Microfarad (μF) $1\mu F = 1/1,000,000 = 0.000001 = 10^{-6}$ F

    Nanofarad (nF) $1nF = 1/1,000,000,000 = 0.000000001 = 10^{-9}$ F

    Picofarad (pF) $1pF = 1/1,000,000,000,000 = 0.000000000001 = 10^{-12}$ F

Then using the information above we can construct a simple table to help us convert between pico-Farad (pF), to nano-Farad (nF), to micro-Farad (μF) and to Farads (F) as shown.

Capacitance of a Parallel Plate Capacitor

The capacitance of a parallel plate capacitor is proportional to the area, A in metres^2 of the smallest of the two plates and inversely proportional to the distance or separation, d(i.e. the dielectric thickness) given in metres between these two conductive plates. The generalised equation for the capacitance of a parallel plate capacitor is given as: $C = \varepsilon(A/d)$ where ε represents the absolute permittivity of the dielectric material being used. The permittivity of a vacuum, ε_o also known as the "permittivity of free space" has the value of the constant 8.84 x 10^{-12} Farads per metre. To make the maths a little easier, this dielectric constant of free space, ε_o, which can be written as: $1/(4\pi \times 9\times10^9)$, may also have the units of picofarads (pF) per metre as the constant giving: 8.84 for the value of free space. Note though that the resulting capacitance value will be in picofarads and not in farads.

Generally, the conductive plates of a capacitor are separated by some kind of insulating material or gel rather than a perfect vacuum. When calculating the capacitance of a capacitor, we can consider the permittivity of air, and especially of dry air, as being the same value as a vacuum as they are very close.

Capacitance Example No1

A capacitor is constructed from two conductive metal plates 30cm x 50cm which are spaced 6mm apart from each other, and uses dry air as its only dielectric material. Calculate the capacitance of the capacitor.

$$\text{Using:} \quad C = \varepsilon_0 \frac{A}{d}$$

$$\text{where:} \quad \varepsilon_0 = 8.84\times10^{-12}$$

$$A = 0.3\times 0.5\, m^2 \quad \text{and} \quad d = 6\times10^{-3}\, m$$

$$C = \frac{8.84\times10^{-12} \times (0.3\times0.5)}{6\times10^{-3}} = 0.221 nF$$

Then the value of the capacitor consisting of two plates separated by air is calculated as 221pF or 0.221nF

The Dielectric of a Capacitor

As well as the overall size of the conductive plates and their distance or spacing apart from each other, another factor which affects the overall capacitance of the device is the type of dielectric material being used. In other words the "Permittivity" (ε) of the dielectric. The conductive plates of a capacitor are generally made of a metal foil or a metal film allowing for the flow of electrons and charge, but the dielectric material used is always an insulator. The various insulating materials used as the dielectric in a capacitor differ in their ability to block or pass an electrical charge. This dielectric material can be made from a number of insulating materials or combinations of these materials with the most common types used being: air, paper, polyester, polypropylene, Mylar, ceramic, glass, oil, or a variety of other materials. The factor by which the dielectric material, or insulator, increases the capacitance of the capacitor compared to air is known as the **Dielectric Constant**, **k** and a dielectric material with a high dielectric constant is a better insulator than a dielectric material with a lower dielectric constant. Dielectric constant is a dimensionless quantity since it is relative to free space. The actual permittivity or "complex permittivity" of the dielectric material between the plates is then the product of the permittivity of free space (ε_o) and the relative permittivity (ε_r) of the material being used as the dielectric and is given as:

Complex Permittivity

$$\varepsilon = \varepsilon_0 \times \varepsilon_r$$

In other words, if we take the permittivity of free space, ε_o as our base level and make it equal to one, when the vacuum of free space is replaced by some other type of insulating material, their permittivity of its dielectric is referenced to the base dielectric of free space giving a multiplication factor known as "relative permittivity", ε_r. So the value of the complex permittivity, ε will always be equal to the relative permittivity times one. Typical units of dielectric permittivity, ε or dielectric constant for common materials are: Pure Vacuum = 1.0000, Air = 1.0006, Paper = 2.5 to 3.5, Glass = 3 to 10, Mica = 5 to 7, Wood = 3 to 8 and Metal Oxide Powders = 6 to 20 etc. This then gives us a final equation for the capacitance of a capacitor as:

$$\text{Capacitance, } C = \frac{\varepsilon_o \varepsilon_r A}{d} \text{ Farads}$$

One method used to increase the overall capacitance of a capacitor while keeping its size small is to "interleave" more plates together within a single capacitor body. Instead of just one set of parallel plates,

a capacitor can have many individual plates connected together thereby increasing the surface area, A of the plates.

For a standard parallel plate capacitor as shown above, the capacitor has two plates, labelled A and B. Therefore as the number of capacitor plates is two, we can say that n = 2, where "n" represents the number of plates.

Then our equation above for a single parallel plate capacitor should really be:

$$\text{Capacitance, C} = \frac{\varepsilon_0 \varepsilon_r (n-1) A}{d} \text{ Farads}$$

However, the capacitor may have two parallel plates but only one side of each plate is in contact with the dielectric in the middle as the other side of each plate forms the outside of the capacitor. If we take the two halves of the plates and join them together we effectively only have "one" whole plate in contact with the dielectric.

As for a single parallel plate capacitor, n – 1 = 2 – 1 which equals 1 as $C = (\varepsilon_o.\varepsilon_r \times 1 \times A)/d$ is exactly the same as saying: $C = (\varepsilon_o.\varepsilon_r.A)/d$ which is the standard equation above.

Voltage Rating of a Capacitor

All capacitors have a maximum voltage rating and when selecting a capacitor consideration must be given to the amount of voltage to be applied across the capacitor. The maximum amount of voltage that can be applied to the capacitor without damage to its dielectric material is generally given in the data sheets as: WV, (working voltage) or as WV DC, (DC working voltage).

If the voltage applied across the capacitor becomes too great, the dielectric will break down (known as electrical breakdown) and arcing will occur between the capacitor plates resulting in a short-circuit. The working voltage of the capacitor depends on the type of dielectric material being used and its thickness.The DC working voltage of a capacitor is just that, the maximum DC voltage and NOT the maximum AC voltage as a capacitor with a DC voltage rating of 100 volts DC cannot be safely subjected to an alternating voltage of 100 volts. Since an alternating voltage has an r.m.s. value of 100 volts but a peak value of over 141 volts!.

Then a capacitor which is required to operate at 100 volts AC should have a working voltage of at least 200 volts. In practice, a capacitor should be selected so that its working voltage either DC or AC should be at least 50 percent greater than the highest effective voltage to be applied to it. Another factor which affects the operation of a capacitor is **Dielectric Leakage**. Dielectric leakage occurs in a capacitor as the result of an unwanted leakage current which flows through the dielectric material.

Generally, it is assumed that the resistance of the dielectric is extremely high and a good insulator blocking the flow of DC current through the capacitor (as in a perfect capacitor) from one plate to the other. However, if the dielectric material becomes damaged due excessive voltage or over temperature, the leakage current through the dielectric will become extremely high resulting in a rapid loss of charge on the plates and an overheating of the capacitor eventually resulting in premature failure of the capacitor.

16

Then never use a capacitor in a circuit with higher voltages than the capacitor is rated for otherwise it may become hot and explode.

Active elements- voltage source and current source exercise problems

An active element is one that is capable of continuously supplying energy to a circuit, such as a battery, a generator, an operational amplifier, etc. A passive element on the other hand are physical elements such as resistors, capacitors, inductors, etc, which cannot generate electrical energy by themselves but only consume it. The types of active circuit elements that are most important to us are those that supply electrical energy to the circuits or network connected to them. These are called "electrical sources" with the two types of electrical sources being the *voltage source* and the *current source*. The current source is usually less common in circuits than the voltage source, but both are used and can be regarded as complements of each other.

An electrical supply or simply, "a source", is a device that supplies electrical power to a circuit in the form of a voltage source or a current source. Both types of electrical sources can be classed as a direct (DC) or alternating (AC) source in which a constant voltage is called a DC voltage and one that varies sinusoidally with time is called an AC voltage. So for example, batteries are DC sources and the 230V wall socket or mains outlet in your home is an AC source. We said earlier that electrical sources supply energy, but one of the interesting characteristic of an electrical source, is that they are also capable of converting non-electrical energy into electrical energy and vice versa. For example, a battery converts chemical energy into electrical energy, while an electrical machine such as a DC generator or an AC alternator converts mechanical energy into electrical energy.

Renewable technologies can convert energy from the sun, the wind, and waves into electrical or thermal energy. But as well as converting energy from one source to another, electrical sources can both deliver or absorb energy allowing it to flow in both directions. Another important characteristic of an electrical source and one which defines its operation, are its I-V characteristics. The I-V characteristic of an electrical source can give us a very nice pictorial description of the source, either as a voltage source and a current source as shown.

Electrical Sources

Electrical sources, both as a voltage source or a current source can be classed as being either *independent* (ideal) or *dependent*, (controlled) that is whose value depends upon a voltage or current elsewhere within the circuit, which itself can be either constant or time-varying. When dealing with circuit laws and analysis, electrical sources are often viewed as being "ideal", that is the source is ideal because it could theoretically deliver an infinite amount of energy without loss thereby having characteristics represented by a straight line. However, in real or practical sources there is always a resistance either connected in parallel for a current source, or series for a voltage source associated with the source affecting its output.

The Voltage Source

A voltage source, such as a battery or generator, provides a potential difference (voltage) between two points within an electrical circuit allowing current to flowing around it. Remember that voltage can exist without current. A battery is the most common voltage source for a circuit with the voltage that appears across the positive and negative terminals of the source being called the terminal voltage.

Ideal Voltage Source

An ideal voltage source is defined as a two terminal active element that is capable of supplying and maintaining the same voltage, (v) across its terminals regardless of the current, (i) flowing through it. In other words, an ideal voltage source will supply a constant voltage at all times regardless of the value of the current being supplied producing an I-V characteristic represented by a straight line.

Then an ideal voltage source is known as an **Independent Voltage Source** as its voltage does not depend on either the value of the current flowing through the source or its direction but is determined solely by the value of the source alone. So for example, an automobile battery has a 12V terminal voltage that remains constant as long as the current through it does not become to high, delivering power to the car in one direction and absorbing power in the other direction as it charges.

On the other hand, a **Dependent Voltage Source** or controlled voltage source, provides a voltage supply whose magnitude depends on either the voltage across or current flowing through some other circuit element. A dependent voltage source is indicated with a diamond shape and are used as equivalent electrical sources for many electronic devices, such as transistors and operational amplifiers.

Connecting Voltage Sources Together

Ideal voltage sources can be connected together in both parallel or series the same as for any circuit element. Series voltages add together while parallel voltages have the same value. Note that unequal ideal voltage sources cannot be connected directly together in parallel.

Voltage Source in Parallel

While not best practice for circuit analysis, ideal voltage sources can be connected in parallel provided they are of the same voltage value. Here in this example, two 10 volt voltage source are combined to produce 10 volts between terminals A and B. Ideally, there would be just one single voltage source of 10 volts given between terminals A and B.

What is not allowed or is not best practice, is connecting together ideal voltage sources that have different voltage values as shown, or are short-circuited by an external closed loop or branch.

Badly Connected Voltage Sources

However, when dealing with circuit analysis, voltage sources of different values can be used providing there are other circuit elements in between them to comply with Kirchoff's Voltage Law, KVL.

Unlike parallel connected voltage sources, ideal voltage sources of different values can be connected together in series to form a single voltage source whose output will be the algebraic addition or subtraction of the voltages used. Their connection can be as: series-aiding or series-opposing voltages as shown.

Voltage Source in Series

Series aiding voltage sources are series connected sources with their polarities connected so that the plus terminal of one is connected to the negative terminal of the next allowing current to flow in the same direction. In the example above, the two voltages of 10V and 5V of the first circuit can be added, for a V_S of $10 + 5 = 15V$. So the voltage across terminals A and B is 15 volts.

Series opposing voltage sources are series connected sources which have their polarities connected so that the plus terminal or the negative terminals are connected together as shown in the second circuit above. The net result is that the voltages are subtracted from each other. Then the two voltages of 10V and 5V of the second circuit are subtracted with the smaller voltage subtracted from the larger voltage. Resulting in a V_S of $10 - 5 = 5V$.

The polarity across terminals A and B is determined by the larger polarity of the voltage sources, in this example terminal A is positive and terminal B is negative resulting in +5 volts. If the series-opposing voltages are equal, the net voltage across A and B will be zero as one voltage balances out the other. Also any currents (I) will also be zero, as without any voltage source, current can not flow.

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

Practical Voltage Source

We have seen that an ideal voltage source can provide a voltage supply that is independent of the current flowing through it, that is, it maintains the same voltage value always. This idea may work well for circuit analysis techniques, but in the real world voltage sources behave a little differently as for a practical voltage source, its terminal voltage will actually decrease with an increase in load curren. As the terminal voltage of an ideal voltage source does not vary with increases in the load current, this implies that an ideal voltage source has zero internal resistance, $R_S = 0$. In other words, it is a resistorless voltage source. In reality all voltage sources have a very small internal resistance which reduces their terminal voltage as they supply higher load currents. For non-ideal or practical voltage sources such as batteries, their internal resistance (R_S) produces the same effect as a resistance connected in series with an ideal voltage source as these two series connected elements carry the same current as shown.

Ideal and Practical Voltage Source

You may have noticed that a practical voltage source closely resembles that of a Thevenin's equivalent circuit as Thevenin's theorem states that "any linear network containing resistances and sources of emf and current may be replaced by a single voltage source, V_S in series with a single resistance, R_S". Note that if the series source resistance is low, the voltage source is ideal. When the source resistance is infinite, the voltage source is open-circuited.

In the case of all real or practical voltage sources, this internal resistance, R_S no matter how small has an effect on the I-V characteristic of the source as the terminal voltage falls off with an increase in load current. This is because the same load current flows through R_S. Ohms law tells us that when a current, (i) flows through a resistance, a voltage drop is produce across the same resistance. The value of this voltage drop is given as $i*R_S$. Then V_{OUT} will equal the ideal voltage source, V_S minus the $i*R_S$ voltage drop across the resistor. Remember that in the case of an ideal source voltage, R_S is equal to zero as there is no internal resistance, therefore the terminal voltage is same as V_S. Then the voltage sum around the loop given by Kirchoff's voltage law, KVL is: $V_{OUT} = V_S - i*R_S$. This equation can be plotted to give the I-V characteristics of the actual output voltage. It will give a straight line with a slope $-R_S$ which intersects the vertical voltage axis at the same point as V_S when the current i = 0 as shown.

Practical Voltage Source Characteristics

Therefore, all ideal voltage sources will have a straight line I-V characteristic but non-ideal or real practical voltage sources will not but instead will have an I-V characteristic that is slightly angled down by an amount equal to $i*R_S$ where R_S is the internal source resistance (or impedance). The I-V characteristics of a real battery provides a very close approximation of an ideal voltage source since the source resistance R_S is usually quite small. The decrease in the angle of the slope of the I-V characteristics as the current increases is known as regulation. Voltage regulation is an important measure of the quality of a practical voltage source as it measures the variation in terminal voltage between no load, that is when $I_L = 0$, (an open-circuit) and full load, that is when I_L is at maximum, (a short-circuit).

Dependent Voltage Source

Unlike an ideal voltage source which produces a constant voltage across its terminals regardless of what is connected to it, a controlled or dependent voltage source changes its terminal voltage depending upon the voltage across, or the current through, some other element connected to the circuit, and as such it is sometimes difficult to specify the value of a dependent voltage source, unless you know the actual value of the voltage or current on which it depends.

Dependent voltage sources behave similar to the electrical sources we have looked at so far, both practical and ideal (independent) the difference this time is that a dependent voltage source can be controlled by an input current or voltage. A voltage source that depends on a voltage input is generally referred to as a Voltage Controlled Voltage Source or VCVS. A voltage source that depends on a current input is referred too as a Current Controlled Voltage Source or CCVS.

Ideal dependent sources are commonly used in the analysing the input/output characteristics or the gain of circuit elements such as operational amplifiers, transistors and integrated circuits. Generally, an ideal voltage dependent source, either voltage or current controlled is designated by a diamond-shaped symbol as shown.

Dependent Voltage Source Symbols
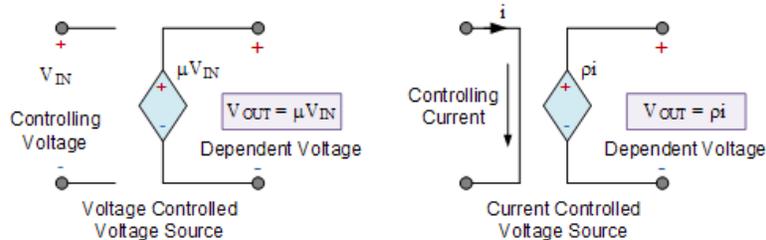
Figure dependent sources

An ideal dependent voltage-controlled voltage source, VCVS, maintains an output voltage equal to some multiplying constant (basically an amplification factor) times the controlling voltage present elsewhere in the circuit. As the multiplying constant is, well, a constant, the controlling voltage, V_{IN} will determine the magnitude of the output voltage, V_{OUT}. In other words, the output voltage "depends" on the value of input voltage making it a dependent voltage source and in many ways, an ideal transformer can be thought of as a VCVS device with the amplification factor being its turns ratio. Then the VCVS output voltage is determined by the following equation: $V_{OUT} = \mu V_{IN}$.

Note that the multiplying constant μ is dimensionless as it is purely a scaling factor because $\mu = V_{OUT}/V_{IN}$, so its units will be volts/volts. An ideal dependent current-controlled voltage source, CCVS, maintains an output voltage equal to some multiplying constant (rho) times a controlling current input generated elsewhere within the connected circuit. Then the output voltage "depends" on the value of the input current, again making it a dependent voltage source. As a controlling current, I_{IN} determines the magnitude of the output voltage, V_{OUT} times the magnification constant ρ (rho), this allows us to model a current-controlled voltage source as a trans-resistance amplifier as the multiplying constant, ρ gives us the following equation: $V_{OUT} = \rho I_{IN}$. This multiplying constant ρ (rho) has the units of Ohm's because $\rho = V_{OUT}/I_{IN}$, and its units will therefore be volts/amperes.

Current source:

As its name implies, a *current source* is a circuit element that maintains a constant current flow regardless of the voltage developed across its terminals as this voltage is determined by other circuit elements. That is, an ideal constant current source continually provides a specified amount of current regardless of the impedance that it is driving and as such, an ideal current source could, in theory, supply an infinite amount of energy. So just as a voltage source may be rated, for example, as 5 volts or 10 volts, etc, a current source will also have a current rating, for example, 3 amperes or 15 amperes, etc.

Ideal constant current sources are represented in a similar manner to voltage sources, but this time the current source symbol is that of a circle with an arrow inside to indicates the direction of the flow of the

current. The direction of the current will correspond to the polarity of the corresponding voltage, flowing out from the positive terminal. The letter "i" is used to indicate that it is a current source as shown.

Ideal Current Source
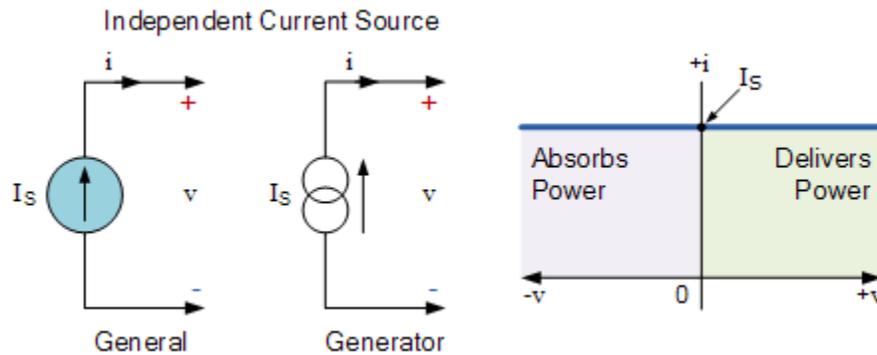
Figure current sources

Then an ideal current source is called a "constant current source" as it provides a constant steady state current independent of the load connected to it producing an I-V characteristic represented by a straight line. As with voltage sources, the current source can be either independent (ideal) or dependent (controlled) by a voltage or current elsewhere in the circuit, which itself can be constant or time-varying.

Ideal independent current sources are typically used to solve circuit theorems and for circuit analysis techniques for circuits that containing real active elements. The simplest form of a current source is a resistor in series with a voltage source creating currents ranging from a few milli-amperes to many hundreds of amperes. Remember that a zero-value current source is an open circuit as R = 0. The concept of a current source is that of a two-terminal element that allows the flow of current indicated by the direction of the arrow. Then a current source has a value, i, in units of amperes, (A) which are typically abbreviated to amps. The physical relationship between a current source and voltage variables around a network is given by Ohm's law as these voltage and current variables will have specified values.

It may be difficult to specify the magnitude and polarity of voltage of an ideal current source as a function of the current especially if there are other voltage or current sources in the connected circuit. Then we may know the current supplied by the current source but not the voltage across it unless the power supplied by the current source is given, as P = V*I. However, if the current source is the only source within the circuit, then the polarity of voltage across the source will be easier to establish. If however there is more than one source, then the terminal voltage will be dependent upon the network in which the source is connected.

Connecting Current Sources Together

Just like voltage sources, ideal current sources can also be connected together to increase (or decrease) the available current. But there are rules on how two or more independent current sources with different values can be connected, either in series or parallel.

Current Source in Parallel

Connecting two or more current sources in parallel is equivalent to one current source whose total current output is given as the algebraic addition of the individual source currents. Here in this example, two 5 amp current sources are combined to produce 10 amps as IT = I1 + I2. Current sources of different values may be connected together in parallel. For example, one of 5 amps and one of 3 amps would combined to give a single current source of 8 amperes as the arrows representing the current source both point in the same direction. Then as the two currents add together, their connection is said to be: parallel-aiding. While not best practice for circuit analysis, parallel-opposing connections use current sources that are connected in opposite directions to form a single current source whose value is the algebraic subtraction of the individual sources.

Parallel Opposing Current Sources

Here, as the two current sources are connected in opposite directions (indicated by their arrows), the two currents subtract from each other as the two provide a closed-loop path for a circulating current complying with Kirchoff's Current Law, KCL. So for example, two current sources of 5 amps each would result in zero output as 5A -5A = 0A. Likewise, if the two currents are of different values, 5A and 3A, then the output will be the subtracted value with the smaller current subtracted from the larger current. Resulting in a IT of 5 - 3 = 2A.

We have seen that ideal current sources can be connected together in parallel to form parallel-aiding or parallel-opposing current sources. What is not allowed or is not best practice for circuit analysis, is connecting together ideal current sources in series combinations.

Current Sources in Series

Current sources are not allowed to be connected together in series, either of the same value or ones with different values. Here in this example, two current sources of 5 amps each are connected together in series, but what is the resulting current value. Is it equal to one source of 5 amps, or is it equal to the addition of the two sources, that is 10 amps. Then series connected current sources add an unknown factor into circuit analysis, which is not good.

Also, another reason why series connected sources are not allowed for circuit analysis techniques is that they may not supply the same current in the same direction. Series-aiding or series-opposing currents do not exist for ideal current sources.

Current Source Example No1

Two current sources of 250 milli-amps and 150 milli-amps respectively are connected together in a parallel-aiding configuration to supply a connected load of 20 ohms. Calculate the voltage drop across the load and the power dissipated. Draw the circuit. Then, IT = 0.4A or 400mA, VR = 8V, and PR = 3.2W

Practical Current Source

We have seen that an ideal constant current source can supply the same amount of current indefinitely regardless of the voltage across its terminals, thus making it an independent source. This therefore implies that the current source has an infinite internal resistance, $(R = \infty)$. This idea works well for circuit analysis techniques, but in the real world current sources behave a little differently as practical current sources

always have an internal resistance, no matter how large (usually in the mega-ohms range), causing the generated source to vary somewhat with the load.

A practical or non-ideal current source can be represented as an ideal source with an internal resistance connected across it. The internal resistance (RP) produces the same effect as a resistance connected in parallel (shunt) with the current source as shown. Remember that circuit elements in parallel have exactly the same voltage drop across them.

Ideal and Practical Current Source

You may have noticed that a practical current source closely resembles that of a Norton's equivalent circuit as Norton's theorem states that "any linear dc network can be replaced by an equivalent circuit consisting of a constant-current source, IS in parallel with a resistor, RP". Note that if this parallel resistance is very low, RP = 0, the current source is short-circuited. When the parallel resistance is very high or infinite, RP ≈ ∞, the current source can be modelled as ideal.

An ideal current source plots a horizontal line on the I-V characteristic as shown previously above. However as practical current sources have an internal source resistance, this takes some of the current so the characteristic of this practical source is not flat and horizontal but will reduce as the current is now splitting into two parts, with one part of the current flowing into the parallel resistance, RP and the other part of the current flowing straight to the output terminals.

Ohms law tells us that when a current, (i) flows through a resistance, (R) a voltage drop is produce across the same resistance. The value of this voltage drop will be given as i*RP. Then VOUT will be equal to the voltage drop across the resistor with no load attached. We remember that for an ideal source current, RP is infinite as there is no internal resistance, therefore the terminal voltage will be zero as there is no voltage drop.

The sum of the current around the loop given by Kirchoff's current law, KCL is: IOUT = IS - VS/RP. This equation can be plotted to give the I-V characteristics of the output current. It is given as a straight line with a slope –RP which intersects the vertical voltage axis at the same point as IS when the source is ideal as shown.

Practical Current Source Characteristic

Therefore, all ideal current sources will have a straight line I-V characteristic but non-ideal or real practical current sources will have an I-V characteristic that is slightly angled down by an amount equal to VOUT/RP where RP is the internal source resistance.

Current Source Example No2

A practical current source consists of a 3A ideal current source which has an internal resistance of 500 Ohms. With no-load attached, calculate the current sources open-circuit terminal voltage and the no-load power absorbed by the internal resistor.

1. No-load values:

$I_O = 0$

I_S I_R

$3A$ 500Ω V_{AB}

A

B

$I_S = I_R = 3A, \quad R_P = 500\Omega$

$V_{AB} = V_{RP} = I_S \times R_P$

$\therefore V_{RP} = 3 \times 500 = 1500V \text{ or } 1.5kV$

$P_R = I_S^2 \times R_P$

$P_R = 3^2 \times 500 = 4500W \text{ or } 4.5kW$
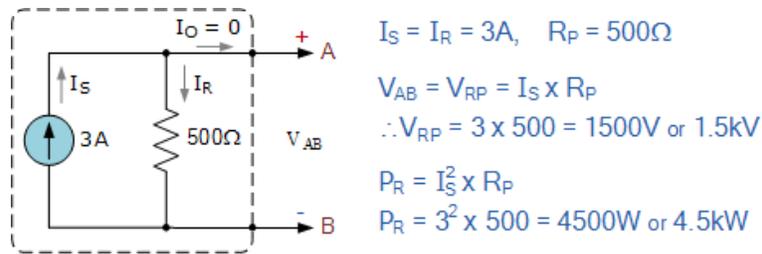
Figure circuit

Then the open circuit voltage across the internal source resistance and terminals A and B (VAB) is calculated at 1500 volts.

Part 2: If a 250 Ohm load resistor is connected to the terminals of the same practical current source, calculate the current through each resistance, the power absorbed by each resistance and the voltage drop across the load resistor. Draw the circuit.

2. Data given with load connected: $IS = 3A$, $RP = 500\Omega$ and $RL = 250\Omega$

I_S I_{RP} I_{RL}

$3A$ $R_P = 500\Omega$ $R_L = 250\Omega$

+   A

−   B
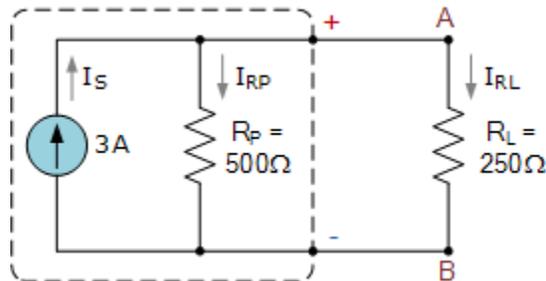
Figure circuit

2a. To find the currents in each resistive branch, we can use the current-division rule.

$$I_{RP} = \frac{R_L}{R_P + R_L} \times I_S = \frac{250}{500 + 250} \times 3 = 1A$$

$$I_{RL} = \frac{R_P}{R_L + R_P} \times I_S = \frac{500}{250 + 500} \times 3 = 2A$$

$$\therefore I_{RP} + I_{RL} = 1 + 2 = 3A = I_S$$

2b. The power absorbed by each resistor is given as:

$$P_{RP} = I_{RP}^2 \times R_P = 1^2 \times 500 = 500W$$

$$P_{RL} = I_{RL}^2 \times R_L = 2^2 \times 250 = 1000W$$

2c. Then the voltage drop across the load resistor, RL is given as:

$$V_{AB} = I_S \times R_T$$

$$R_T = \frac{R_P \times R_L}{R_P + R_L} = \frac{500 \times 250}{500 + 250} = 166.7\Omega$$

$$\therefore V_{AB} = 3 \times 166.7 = 500V$$

We can see that the terminal voltage of an open-circuited practical current source can be very high it will produce whatever voltage is needed, 1500 volts in this example, to supply the specified current. In theory, this terminal voltage can be infinite as the source attempts to deliver the rated current.

Connecting a load across its terminals will reduce the voltage, 500 volts in this example, as now the current has somewhere to go and for a constant current source, the terminal voltage is directly proportional to the load resistance.

In the case of non-ideal current sources that each have an internal resistance, the total internal resistance (or impedance) will be the result of combining them together in parallel, exactly the same as for resistors in parallel.

Dependent Current Source

We now know that an ideal current source provides a specified amount of current completely independent of the voltage across it and as such will produce whatever voltage is necessary to maintain the required current. This then makes it completely independent of the circuit to which it is connected to resulting in it being called an *ideal independent current source*.

A controlled or dependent current source on the other hand changes its available current depending upon the voltage across, or the current through, some other element connected to the circuit. In other words, the output of a dependent current source is controlled by another voltage or current.

Dependent current sources behave similar to the current sources we have looked at so far, both ideal (independent) and practical. The difference this time is that a dependent current source can be controlled by an input voltage or current. A current source that depends on a voltage input is generally referred to as a Voltage Controlled Current Source or VCCS. A current source that depends on a current input is generally referred too as a Current Controlled Current Source or CCCS.

Generally, an ideal current dependent source, either voltage or current controlled is designated by a diamond-shaped symbol where an arrow indicates the direction of the current, i as shown.

Dependent Current Source Symbols
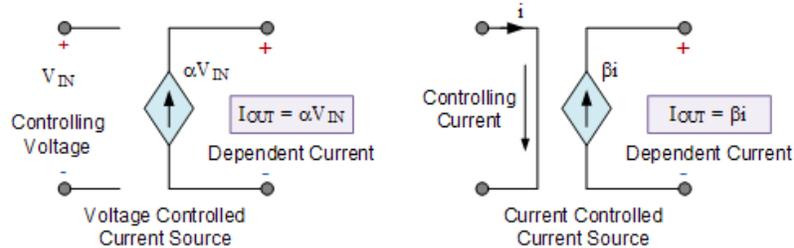
Figure circuit

An ideal dependent voltage-controlled current source, VCCS, maintains an output current, IOUT that is proportional to the controlling input voltage, VIN. In other words, the output current "depends" on the value of input voltage making it a dependent current source.

Then the VCCS output current is defined by the following equation: IOUT = αVIN. This multiplying constant α (alpha) has the SI units of mhos, ℧ (an inverted Ohms sign) because α = IOUT/VIN, and its units will therefore be amperes/volt.

An ideal dependent current-controlled current source, CCCS, maintains an output current that is proportional to a controlling input current. Then the output current "depends" on the value of the input current, again making it a dependent current source.

As a controlling current, IIN determines the magnitude of the output current, IOUT times the magnification constant β (beta), the output current for a CCCS element is determined by the following equation: IOUT = βIIN. Note that the multiplying constant β is a dimensionless scaling factor as β = IOUT/IIN, so therefore its units would be amperes/amperes.

Kirchhoffs voltage law voltage division rule
Kirchhoff's Voltage Law

Kirchhoff's Voltage Law (KVL) is Kirchhoff's second law that deals with the conservation of energy around a closed circuit path.

Gustav Kirchhoff's Voltage Law is the second of his fundamental laws we can use for circuit analysis. His voltage law states that for a closed loop series path **the algebraic sum of all the voltages around any closed loop in a circuit is equal to zero**. This is because a circuit loop is a closed conducting path so no energy is lost.

In other words the algebraic sum of ALL the potential differences around the loop must be equal to zero as: $\Sigma V = 0$. Note here that the term "algebraic sum" means to take into account the polarities and signs of the sources and voltage drops around the loop.

This idea by Kirchhoff is commonly known as the **Conservation of Energy**, as moving around a closed loop, or circuit, you will end up back to where you started in the circuit and therefore back to the same initial potential with no loss of voltage around the loop. Hence any voltage drops around the loop must be equal to any voltage sources met along the way.

So when applying Kirchhoff's voltage law to a specific circuit element, it is important that we pay special attention to the algebraic signs, (+ and -) of the voltage drops across elements and the emf's of sources otherwise our calculations may be wrong.

But before we look more closely at Kirchhoff's voltage law (KVL) lets first understand the voltage drop across a single element such as a resistor.

A Single Circuit Element

For this simple example we will assume that the current, I is in the same direction as the flow of positive charge, that is conventional current flow.

Here the flow of current through the resistor is from point A to point B, that is from positive terminal to a negative terminal. Thus as we are travelling in the same direction as current flow, there will be a *fall* in potential across the resistive element giving rise to a -IR voltage drop across it.

If the flow of current was in the opposite direction from point B to point A, then there would be a *rise* in potential across the resistive element as we are moving from a -potential to a + potential giving us a +I*R voltage drop.

Thus to apply Kirchhoff's voltage law correctly to a circuit, we must first understand the direction of the polarity and as we can see, the sign of the voltage drop across the resistive element will depend on the direction of the current flowing through it. As a general rule, you will loose potential in the same direction of current across an element and gain potential as you move in the direction of an emf source.

The direction of current flow around a closed circuit can be assumed to be either clockwise or anticlockwise and either one can be chosen. If the direction chosen is different from the actual direction of current flow, the result will still be correct and valid but will result in the algebraic answer having a minus sign.

To understand this idea a little more, lets look at a single circuit loop to see if Kirchhoff's Voltage Law holds true.

A Single Circuit Loop

Kirchhoff's voltage law states that the algebraic sum of the potential differences in any loop must be equal to zero as: $\Sigma V = 0$. Since the two resistors, R_1 and R_2 are wired together in a series connection, they are both part of the same loop so the same current must flow through each resistor.

Thus the voltage drop across resistor, $R_1 = I*R_1$ and the voltage drop across resistor, $R_2 = I*R_2$ giving by KVL:

$$V_S + (-IR_1) + (-IR_2) = 0$$

$$\therefore V_S = IR_1 + IR_2$$

$$V_S = I(R_1 + R_2)$$

$$V_S = IR_T$$

$$\text{Where: } R_T = R_1 + R_2$$

We can see that applying Kirchhoff's Voltage Law to this single closed loop produces the formula for the equivalent or total resistance in the series circuit and we can expand on this to find the values of the voltage drops around the loop.

$$R_T = R_1 + R_2$$

$$I = \frac{V_S}{R_T} = \frac{V_S}{R_1 + R_2}$$

$$V_{R1} = IR_1 = V_S\left(\frac{R_1}{R_1 + R_2}\right)$$

$$V_{R2} = IR_2 = V_S\left(\frac{R_2}{R_1 + R_2}\right)$$

Kirchhoff's Voltage Law Example No1

Three resistor of values: 10 ohms, 20 ohms and 30 ohms, respectively are connected in series across a 12 volt battery supply. Calculate: a) the total resistance, b) the circuit current, c) the current through each resistor, d) the voltage drop across each resistor, e) verify that Kirchhoff's voltage law, KVL holds true.

a) Total Resistance (R$_T$)

$R_T = R_1 + R_2 + R_3 = 10\Omega + 20\Omega + 30\Omega = 60\Omega$

Then the total circuit resistance R$_T$ is equal to 60Ω

b) Circuit Current (I)

$$I = \frac{V_S}{R_T} = \frac{12}{60} = 0.2A$$

Thus the total circuit current I is equal to 0.2 amperes or 200mA

c) Current Through Each Resistor

The resistors are wired together in series, they are all part of the same loop and therefore each experience the same amount of current. Thus:

$I_{R1} = I_{R2} = I_{R3} = I_{SERIES} = 0.2$ amperes

d) Voltage Drop Across Each Resistor

$V_{R1} = I \times R_1 = 0.2 \times 10 = 2$ volts

$V_{R2} = I \times R_2 = 0.2 \times 20 = 4$ volts

$V_{R3} = I \times R_3 = 0.2 \times 30 = 6$ volts

e) Verify Kirchhoff's Voltage Law

$$V_S + (-IR_1) + (-IR_2) + (-IR_3) = 0$$

$$12 + (-0.2 \times 10) + (-0.2 \times 20) + (-0.2 \times 30) = 0$$

$$12 + (-2) + (-4) + (-6) = 0$$

$$\therefore 12 - 2 - 4 - 6 = 0$$

Thus Kirchhoff's voltage law holds true as the individual voltage drops around the closed loop add up to the total.

Kirchhoff's Circuit Loop
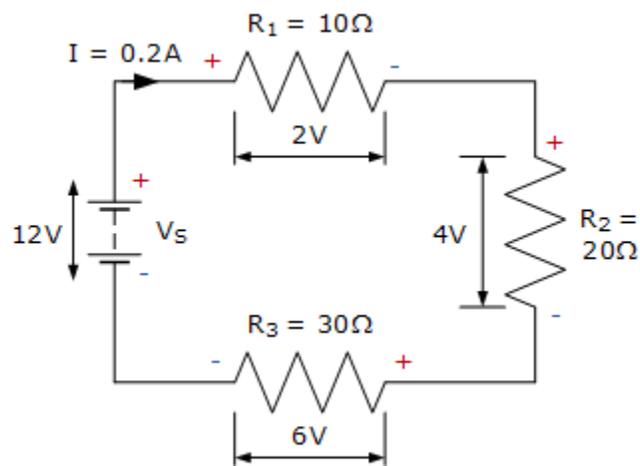
Figure circuit

We have seen here that Kirchhoff's voltage law, KVL is Kirchhoff's second law and states that the algebraic sum of all the voltage drops, as you go around a closed circuit from some fixed point and return back to the same point, and taking polarity into account, is always zero. That is $\Sigma V = 0$

The theory behind Kirchhoff's second law is also known as the law of conservation of voltage, and this is particularly useful for us when dealing with series circuits, as series circuits also act as voltage dividers and the voltage divider circuit is an important application of many series circuits.

Kirchhoff's Current Law

Kirchhoff's Current Law (KCL) is Kirchhoff's first law that deals with the conservation of charge entering and leaving a junction.
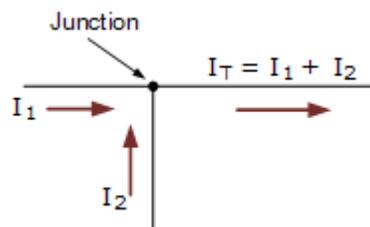
Figure KVL

To determine the amount or magnitude of the electrical current flowing around an electrical or electronic circuit, we need to use certain laws or rules that allows us to write down these currents in the form of an equation. The network equations used are those according to Kirchhoff's laws, and as we are dealing with circuit currents, we will be looking at Kirchhoff's current law, (KCL). Gustav Kirchhoff's Current Law is one of the fundamental laws used for circuit analysis. His current law states that for a parallel path the total current entering a circuits junction is exactly equal to the total current leaving the same junction. This is because it has no other place to go as no charge is lost. In other words the algebraic sum of ALL the currents entering and leaving a junction must be equal to zero as: $\Sigma I_{IN} = \Sigma I_{OUT}$. This idea by Kirchhoff is commonly known as the Conservation of Charge, as the current is conserved around the junction with no loss of current. Lets look at a simple example of Kirchhoff's current law (KCL) when applied to a single junction.

A Single Junction

Here in this simple single junction example, the current I_T leaving the junction is the algebraic sum of the two currents, I_1 and I_2 entering the same junction. That is $I_T = I_1 + I_2$.

Note that we could also write this correctly as the algebraic sum of: $I_T - (I_1 + I_2) = 0$.

So if I_1 equals 3 amperes and I_2 is equal to 2 amperes, then the total current, I_T leaving the junction will be $3 + 2 = 5$ amperes, and we can use this basic law for any number of junctions or nodes as the sum of the currents both entering and leaving will be the same. Also, if we reversed the directions of the currents, the resulting equations would still hold true for I_1 or I_2. As $I_1 = I_T - I_2 = 5 - 2 = 3$ amps, and $I_2 = I_T - I_1 = 5 - 3 = 2$ amps. Thus we can think of the currents entering the junction as being positive (+), while the ones

leaving the junction as being negative (-). Then we can see that the mathematical sum of the currents either entering or leaving the junction and in whatever direction will always be equal to zero, and this forms the basis of Kirchhoff's Junction Rule, more commonly known as Kirchhoff's Current Law, or (KCL).

Resistors in Parallel

Let's look how we could apply Kirchhoff's current law to resistors in parallel, whether the resistances in those branches are equal or unequal. Consider the following circuit diagram:
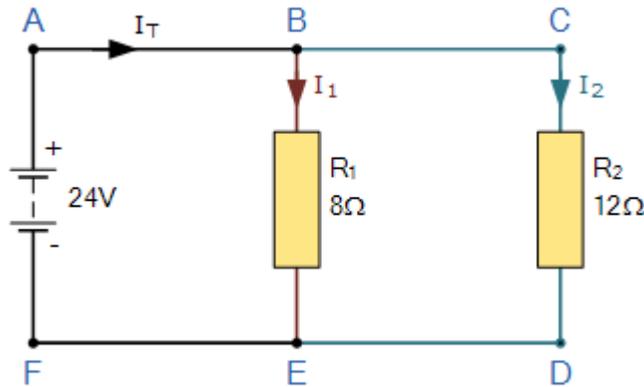
Figure circuit

In this simple parallel resistor example there are two distinct junctions for current. Junction one occurs at node B, and junction two occurs at node E. Thus we can use Kirchhoff's Junction Rule for the electrical currents at both of these two distinct junctions, for those currents entering the junction and for those currents flowing leaving the junction. To start, all the current, I_T leaves the 24 volt supply and arrives at point A and from there it enters node B. Node B is a junction as the current can now split into two distinct directions, with some of the current flowing downwards and through resistor R_1 with the remainder continuing on through resistor R_2 via node C. Note that the currents flowing into and out of a node point are commonly called branch currents. We can use Ohm's Law to determine the individual branch currents through each resistor as: $I = V/R$, thus:

For current branch B to E through resistor R_1

$$I_{B-E} = I_1 = \frac{V}{R_1} = \frac{24}{8} = 3A$$

For current branch C to D through resistor R_2

$$I_{C-D} = I_2 = \frac{V}{R_2} = \frac{24}{12} = 2A$$

From above we know that Kirchhoff's current law states that the sum of the currents entering a junction must equal the sum of the currents leaving the junction, and in our simple example above, there is one current, I_T going into the junction at node B and two currents leaving the junction, I_1 and I_2. Since we now know from calculation that the currents leaving the junction at node B is I_1equals 3 amps and I_2 equals 2 amps, the sum of the currents entering the junction at node B must equal $3 + 2 = 5$ amps. Thus $\Sigma_{IN} = I_T = 5$ amperes. In our example, we have two distinct junctions at node B and node E, thus we can confirm this value for I_T as the two currents recombine again at node E. So, for Kirchhoff's junction rule to hold true, the sum of the currents into point F must equal the sum of the currents flowing out of the junction at node E. As the two currents entering junction E are 3 amps and 2 amps respectively, the sum of the currents entering point F is therefore: $3 + 2 = 5$ amperes. Thus $\Sigma_{IN} = I_T = 5$ amperes and therefore Kirchhoff's current law holds true as this is the same value as the current leaving point A.

Applying KCL to more complex circuits.

We can use Kirchhoff's current law to find the currents flowing around more complex circuits. We hopefully know by now that the algebraic sum of all the currents at a node (junction point) is equal to zero and with this idea in mind, it is a simple case of determining the currents entering a node and those leaving the node. Consider the circuit below.

Kirchhoff's Current Law Example No1
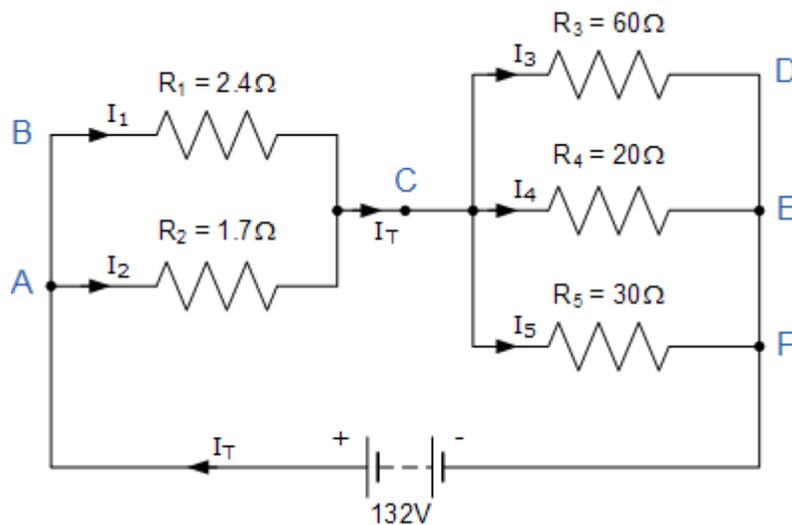
Figure circuit

In this example there are four distinct junctions for current to either separate or merge together at nodes A, C, E and node F. The supply current I_T separates at node A flowing through resistors R_1 and R_2, recombining at node C before separating again through resistors R_3, R_4 and R_5 and finally recombining once again at node F.

But before we can calculate the individual currents flowing through each resistor branch, we must first calculate the circuits total current, I_T. Ohms law tells us that $I = V/R$ and as we know the value of V, 132 volts, we need to calculate the circuit resistances as follows.

33

Circuit Resistance R$_{AC}$

$$\frac{1}{R_{(AC)}} = \frac{1}{R_1} + \frac{1}{R_2} = \frac{1}{2.4} + \frac{1}{1.7}$$

$$\frac{1}{R_{(AC)}} = \frac{1}{1} \qquad \therefore R_{(AC)} = 1\Omega$$

Thus the equivalent circuit resistance between nodes A and C is calculated as 1 Ohm.

Circuit Resistance R$_{CF}$

$$\frac{1}{R_{(CF)}} = \frac{1}{R_3} + \frac{1}{R_4} + \frac{1}{R_5} = \frac{1}{60} + \frac{1}{20} + \frac{1}{30}$$

$$\frac{1}{R_{(CF)}} = \frac{1}{0.1} \qquad \therefore R_{(CF)} = 10\Omega$$

Thus the equivalent circuit resistance between nodes C and F is calculated as 10 Ohms. Then the total circuit current, I$_T$ is given as:

$$R_T = R_{(AC)} + R_{(CF)} = 1 + 10 = 11\Omega$$

$$I_T = \frac{V}{R_T} = \frac{132}{11} = 12 \, \text{Amps}$$

Giving us an equivalent circuit of:

Kirchhoff's Current Law Equivalent Circuit
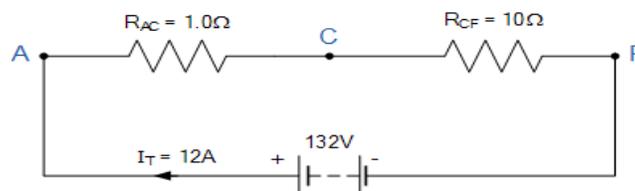
Figure circuit

Therefore, V = 132V, R$_{AC}$ = 1Ω, R$_{CF}$ = 10Ω's and I$_T$ = 12A.

Having established the equivalent parallel resistances and supply current, we can now calculate the individual branch currents and confirm using Kirchhoff's junction rule as follows.

$$V_{AC} = I_T \times R_{AC} = 12 \times 1 = 12 \, \text{Volts}$$

$$V_{CF} = I_T \times R_{CF} = 12 \times 10 = 120 \, \text{Volts}$$

$$I_1 = \frac{V_{AC}}{R_1} = \frac{12}{2.4} = 5 \, \text{Amps}$$

$$I_2 = \frac{V_{AC}}{R_2} = \frac{12}{1.7} = 7 \, \text{Amps}$$

$$I_3 = \frac{V_{CF}}{R_3} = \frac{120}{60} = 2 \, \text{Amps}$$

$$I_4 = \frac{V_{CF}}{R_4} = \frac{120}{20} = 6 \, \text{Amps}$$

$$I_5 = \frac{V_{CF}}{R_5} = \frac{120}{30} = 4 \, \text{Amps}$$

Thus, I_1 = 5A, I_2 = 7A, I_3 = 2A, I_4 = 6A, and I_5 = 4A.

We can confirm that Kirchoff's current law holds true around the circuit by using node C as our reference point to calculate the currents entering and leaving the junction as:

$$\text{At node C} \quad \sum I_{IN} = \sum I_{OUT}$$

$$I_T = I_1 + I_2 = I_3 + I_4 + I_5$$

$$\therefore 12 = (5 + 7) = (2 + 6 + 4)$$

We can also double check to see if Kirchhoffs Current Law holds true as the currents entering the junction are positive, while the ones leaving the junction are negative, thus the algebraic sum is: $I_1 + I_2 - I_3 - I_4 - I_5 = 0$ which equals $5 + 7 - 2 - 6 - 4 = 0$.

So we can confirm by analysis that Kirchhoff's current law (KCL) which states that the algebraic sum of the currents at a junction point in a circuit network is always zero is true and correct in this example.

Kirchhoff's Current Law Example No2

Find the currents flowing around the following circuit using Kirchhoff's Current Law only.
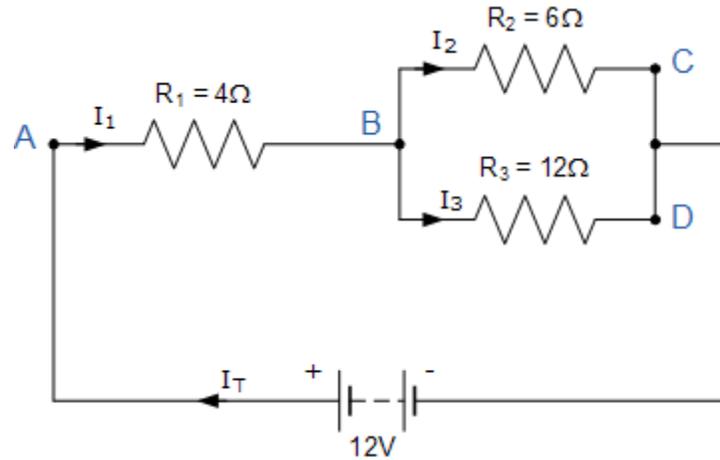
Figure circuit

I_T is the total current flowing around the circuit driven by the 12V supply voltage. At point A, I_1 is equal to I_T, thus there will be an I_1*R voltage drop across resistor R_1.

The circuit has 2 branches, 3 nodes (B, C and D) and 2 independent loops, thus the $I*R$ voltage drops around the two loops will be:

- Loop ABC $\Rightarrow$ $12 = 4I_1 + 6I_2$
- Loop ABD $\Rightarrow$ $12 = 4I_1 + 12I_3$

Since Kirchhoff's current law states that at node B, $I_1 = I_2 + I_3$, we can therefore substitute current I_1 for $(I_2 + I_3)$ in both of the following loop equations and then simplify.

Kirchhoff's Loop Equations

$$\text{Loop}(ABC) \qquad\qquad \text{Loop}(ABD)$$

$$12 = 4I_1 + 6I_2 \qquad\qquad 12 = 4I_1 + 12I_3$$

$$12 = 4(I_2 + I_3) + 6I_2 \qquad 12 = 4(I_2 + I_3) + 12I_3$$

$$12 = 4I_2 + 4I_3 + 6I_2 \qquad 12 = 4I_2 + 4I_3 + 12I_3$$

$$12 = 10I_2 + 4I_3 \qquad\qquad 12 = 4I_2 + 16I_3$$

We now have two simultaneous equations that relate to the currents flowing around the circuit.

Eq. No 1 : $12 = 10I_2 + 4I_3$

Eq. No 2 : $12 = 4I_2 + 16I_3$

By multiplying the first equation (Loop ABC) by 4 and subtracting Loop ABD from Loop ABC, we can be reduced both equations to give us the values of I_2 and I_3

Eq. No 1 : $12 = 10I_2 + 4I_3$ (x4) $\Rightarrow$ $48 = 40I_2 + 16I_3$

Eq. No 2 : $12 = 4I_2 + 16I_3$ (x1) $\Rightarrow$ $12 = 4I_2 + 16I_3$

Eq. No 1 – Eq. No 2 $\Rightarrow$ $36 = 36I_2 + 0$

Substitution of I_2 in terms of I_3 gives us the value of I_2 as 1.0 Amps

Now we can do the same procedure to find the value of I_3 by multiplying the first equation (Loop ABC) by 4 and the second equation (Loop ABD) by 10. Again by subtracting Loop ABC from Loop ABD, we can be reduced both equations to give us the values of I_2 and I_3

Eq. No 1 : $12 = 10I_2 + 4I_3$ (x4) $\Rightarrow$ $48 = 40I_2 + 16I_3$

Eq. No 2 : $12 = 4I_2 + 16I_3$ (x10) $\Rightarrow$ $120 = 40I_2 + 160I_3$

Eq. No 2 – Eq. No 1 $\Rightarrow$ $72 = 0 + 144I_3$

Thus substitution of I_3 in terms of I_2 gives us the value of I_3 as 0.5 Amps

As Kirchhoff's junction rule states that : $I_1 = I_2 + I_3$

The supply current flowing through resistor R_1 is given as : $1.0 + 0.5 = 1.5$ Amps

Thus $I_1 = I_T = 1.5$ Amps, $I_2 = 1.0$ Amps and $I_3 = 0.5$ Amps and from that information we could calculate the I*R voltage drops across the devices and at the various points (nodes) around the circuit.

We could have solved the circuit of example two simply and easily just using Ohm's Law, but we have used Kirchhoff's Current Law here to show how it is possible to solve more complex circuits when we cannot just simply apply Ohm's Law.

Derivation of Current Divider formula

The derivation of CDR formula is very simple. Let's reconsider the general circuit:

Current divider formula: $I_x = \dfrac{I_t}{R_x} R_t$
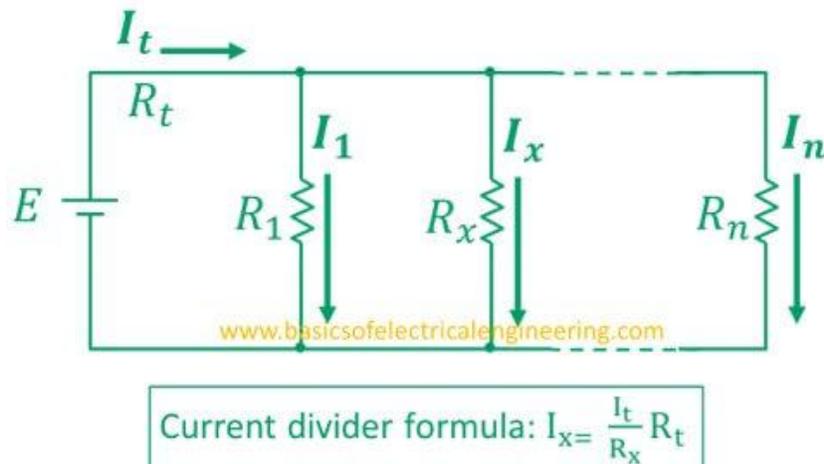
Apply the Ohm's law on R_x.

$I_x = E/R_x$
where $E = I_tR_t$.
$I_x = I_tR_t/R_x$.
or
$I_x = [R_t/R_x] * I_t$

Problem 2

Simplify the circuit and then use current division to find i_x in the circuit below:
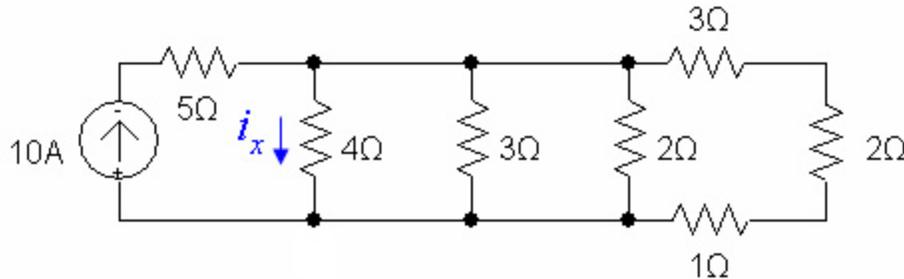
Figure circuit

$i_x = 2A$

Series, parallel connection of resistors, inductors and capacitors and their equivalents:

Resistors in Series and Parallel

Resistor circuits that combine series and parallel resistors networks together are generally known as **Resistor Combination** or mixed resistor circuits. The method of calculating the circuits equivalent resistance is the same as that for any individual series or parallel circuit and hopefully we now know that resistors in series carry exactly the same current and that resistors in parallel have exactly the same voltage across them.

For example, in the following circuit calculate the total current (I_T) taken from the 12v supply.
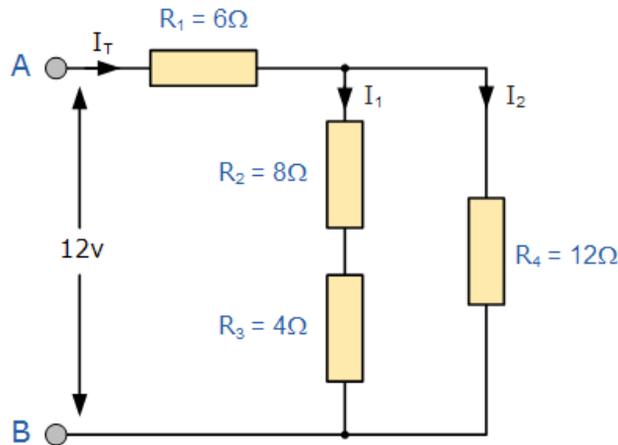
At first glance this may seem a difficult task, but if we look a little closer we can see that the two resistors, R_2 and R_3 are actually both connected together in a "SERIES" combination so we can add them together to produce an equivalent resistance the same as we did in the series resistor tutorial. The resultant resistance for this combination would therefore be:

$$R_2 + R_3 = 8\Omega + 4\Omega = 12\Omega$$

So we can replace both resistor R_2 and R_3 above with a single resistor of resistance value 12Ω
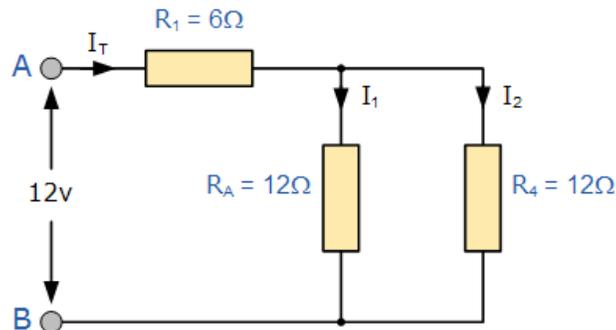
So our circuit now has a single resistor R_A in "PARALLEL" with the resistor R_4. Using our resistors in parallel equation we can reduce this parallel combination to a single equivalent resistor value of $R_{(combination)}$ using the formula for two parallel connected resistors as follows.

$$R_{(combination)} = \frac{1}{R_A} + \frac{1}{R_4} = \frac{1}{12} + \frac{1}{12} = \frac{1}{R_{(com)}} = 6\Omega$$

The resultant resistive circuit now looks something like this:
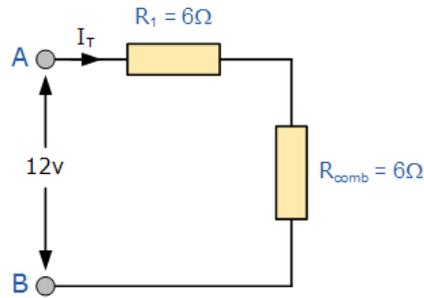
We can see that the two remaining resistances, R_1 and $R_{(comb)}$ are connected together in a "SERIES" combination and again they can be added together (resistors in series) so that the total circuit resistance between points A and B is therefore given as:

$$R_{(AB)} = R_{comb} + R_1 = 6\Omega + 6\Omega = 12\Omega.$$
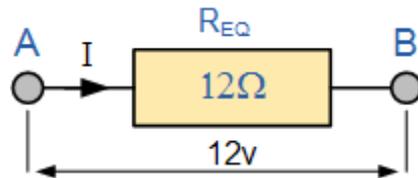
and a single resistance of just 12Ω can be used to replace the original four resistors connected together in the original circuit.

Now by using Ohm´s Law, the value of the circuit current (I) is simply calculated as:

$$\text{Circuit Current (I)} = \frac{V}{R} = \frac{12}{12} = 1 \, \text{Ampere}$$

So any complicated resistive circuit consisting of several resistors can be reduced to a simple single circuit with only one equivalent resistor by replacing all the resistors connected together in series or in parallel using the steps above.

It is sometimes easier with complex resistor combinations and resistive networks to sketch or redraw the new circuit after these changes have been made, as this helps as a visual aid to the maths. Then continue to replace any series or parallel combinations until one equivalent resistance, R_{EQ} is found. Lets try another more complex resistor combination circuit.

Resistors in Series and Parallel Example No2

Find the equivalent resistance, R_{EQ} for the following resistor combination circuit.

Again, at first glance this resistor ladder network may seem a complicated task, but as before it is just a combination of series and parallel resistors connected together. Starting from the right hand side and using the simplified equation for two parallel resistors, we can find the equivalent resistance of the R_8 to R_{10} combination and call it R_A.

$$R_A = \frac{R_9 \times (R_8 + R_{10})}{R_9 + R_8 + R_{10}} = \frac{6 \times (10 + 2)}{6 + 10 + 2} = 4\Omega$$

R_A is in series with R_7 therefore the total resistance will be $R_A + R_7 = 4 + 8 = 12\Omega$ as shown.
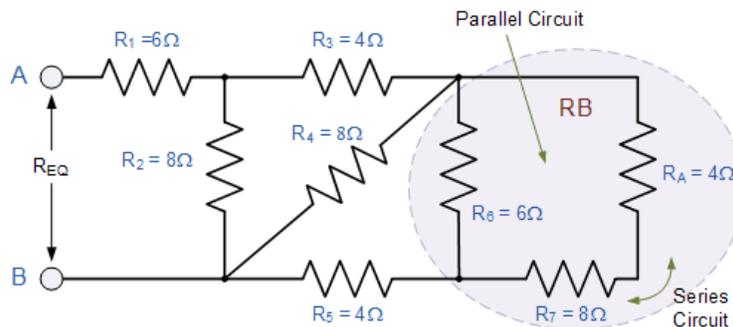
This resistive value of 12Ω is now in parallel with R_6 and can be calculated as R_B.

$$R_B = \frac{R_6 \times (R_A + R_7)}{R_6 + R_A + R_7} = \frac{6 \times (4 + 8)}{6 + 4 + 8} = 4\Omega$$

R_B is in series with R_5 therefore the total resistance will be $R_B + R_5 = 4 + 4 = 8\Omega$ as shown.
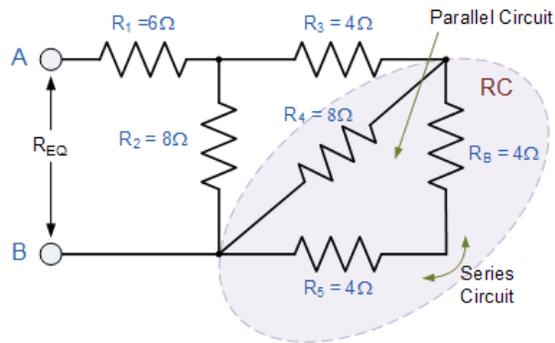
This resistive value of 8Ω is now in parallel with R_4 and can be calculated as R_C as shown.

$$R_C = \frac{R_4 \times (R_B + R_5)}{R_4 + R_B + R_5} = \frac{8 \times (4 + 4)}{8 + 4 + 4} = 4\Omega$$

R_C is in series with R_3 therefore the total resistance will be $R_C + R_3 = 8\Omega$ as shown.
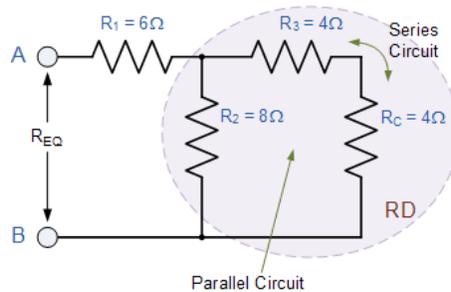
This resistive value of 8Ω is now in parallel with R_2 from which we can calculated R_D as:

$$R_D = \frac{R_2 \times (R_C + R_3)}{R_2 + R_C + R_3} = \frac{8 \times (4 + 4)}{8 + 4 + 4} = 4\Omega$$

R_D is in series with R_1 therefore the total resistance will be $R_D + R_1 = 4 + 6 = 10\Omega$ as shown.
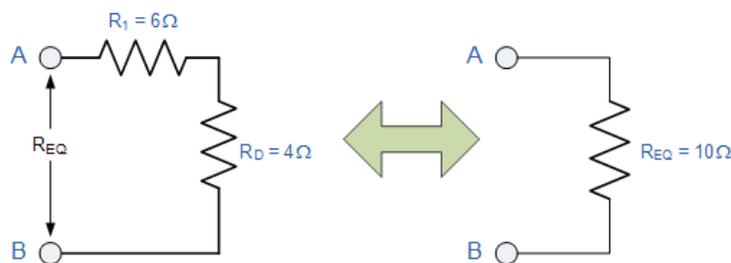
Inductors in Series and parallel:

Inductors can be connected together in a series connection when the are daisy chained together sharing a common electrical current

Inductor in Series Circuit
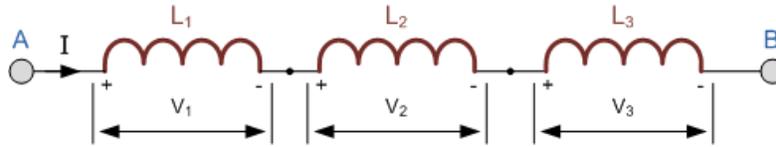
The current, (I) that flows through the first inductor, L_1 has no other way to go but pass through the second inductor and the third and so on. Then, series inductors have a **Common Current** flowing through them, for example:

$$I_{L1} = I_{L2} = I_{L3} = I_{AB} \dots \text{etc.}$$

In the example above, the inductors L_1, L_2 and L_3 are all connected together in series between points A and B. The sum of the individual voltage drops across each inductor can be found using Kirchoff's Voltage Law (KVL) where, $V_T = V_1 + V_2 + V_3$ and we know from the previous tutorials on inductance that the self-induced emf across an inductor is given as: $V = L \, di/dt$.

So by taking the values of the individual voltage drops across each inductor in our example above, the total inductance for the series combination is given as:

$$L_T \frac{di}{dt} = L_1 \frac{di}{dt} + L_2 \frac{di}{dt} + L_3 \frac{di}{dt}$$

By dividing through the above equation by di/dt we can reduce it to give a final expression for calculating the total inductance of a circuit when connecting inductors together in series and this is given as:

Inductors in Series Equation

$$L_{total} = L_1 + L_2 + L_3 + \dots + L_n \text{ etc.}$$

Then the total inductance of the series chain can be found by simply adding together the individual inductances of the inductors in series just like adding together resistors in series. However, the above equation only holds true when there is "NO" mutual inductance or magnetic coupling between two or more of the inductors, (they are magnetically isolated from each other).

One important point to remember about inductors in series circuits, the total inductance (L_T) of any two or more inductors connected together in series will always be **GREATER**than the value of the largest inductor in the series chain.

Inductors in Series Example No1

Three inductors of 10mH, 40mH and 50mH are connected together in a series combination with no mutual inductance between them. Calculate the total inductance of the series combination.

$$L_T = L_1 + L_2 + L_3 = 10mH + 40mH + 50mH = 100mH$$

Inductors in Parallel Circuit
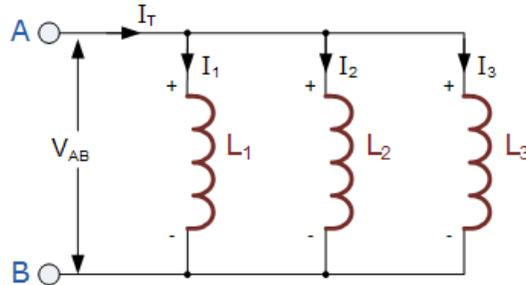
In the previous series inductors tutorial, we saw that the total inductance, L_T of the circuit was equal to the sum of all the individual inductors added together. For inductors in parallel the equivalent circuit inductance L_T is calculated differently.

The sum of the individual currents flowing through each inductor can be found using Kirchoff's Current Law (KCL) where, $I_T = I_1 + I_2 + I_3$ and we know from the previous tutorials on inductance that the self-induced emf across an inductor is given as: $V = L\, di/dt$

Then by taking the values of the individual currents flowing through each inductor in our circuit above, and substituting the current i for $i_1 + i_2 + i_3$ the voltage across the parallel combination is given as:

$$V_{AB} = L_T \frac{d}{dt}\left(i_1 + i_2 + i_3\right) = L_T\left(\frac{di_1}{dt} + \frac{di_2}{dt} + \frac{di_3}{dt}\right)$$

By substituting di/dt in the above equation with v/L gives:

$$V_{AB} = L_T\left(\frac{v}{L_1} + \frac{v}{L_2} + \frac{v}{L_3}\right)$$

We can reduce it to give a final expression for calculating the total inductance of a circuit when connecting inductors in parallel and this is given as:

Parallel Inductor Equation

$$\frac{1}{L_T} = \frac{1}{L_1} + \frac{1}{L_2} + \frac{1}{L_3} \ldots\ldots + \frac{1}{L_N}$$

Here, like the calculations for parallel resistors, the reciprocal (1/Ln) value of the individual inductances are all added together instead of the inductances themselves. But again as with series connected inductances, the above equation only holds true when there is "NO" mutual inductance or magnetic coupling between two or more of the inductors, (they are magnetically isolated from each

other). Where there is coupling between coils, the total inductance is also affected by the amount of coupling.

This method of calculation can be used for calculating any number of individual inductances connected together within a single parallel network. If however, there are only two individual inductors in parallel then a much simpler and quicker formula can be used to find the total inductance value, and this is:

$$L_T = \frac{L_1 \times L_2}{L_1 + L_2}$$

One important point to remember about inductors in parallel circuits, the total inductance (L_T) of any two or more inductors connected together in parallel will always be **LESS** than the value of the smallest inductance in the parallel chain.

Inductors in Parallel Example No1

Three inductors of 60mH, 120mH and 75mH respectively, are connected together in a parallel combination with no mutual inductance between them. Calculate the total inductance of the parallel combination in millihenries.

$$\frac{1}{L_T} = \frac{1}{L_1} + \frac{1}{L_2} + \frac{1}{L_3}$$

$$\therefore L_T = \frac{1}{\frac{1}{L_1} + \frac{1}{L_2} + \frac{1}{L_3}} = \frac{1}{\frac{1}{60\,\mathrm{mH}} + \frac{1}{120\,\mathrm{mH}} + \frac{1}{75\,\mathrm{mH}}}$$

$$L_T = \frac{1}{38.333} = 26\,\mathrm{mH}$$

Capacitors in a Series Connection
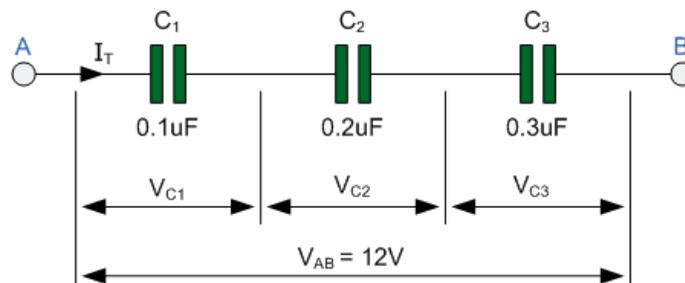
In the previous parallel circuit we saw that the total capacitance, C_T of the circuit was equal to the sum of all the individual capacitors added together. In a series connected circuit however, the total or equivalent capacitance C_T is calculated differently.

In the series circuit above the right hand plate of the first capacitor, C_1 is connected to the left hand plate of the second capacitor, C_2 whose right hand plate is connected to the left hand plate of the

third capacitor, C_3. Then this series connection means that in a DC connected circuit, capacitor C_2 is effectively isolated from the circuit.

The result of this is that the effective plate area has decreased to the smallest individual capacitance connected in the series chain. Therefore the voltage drop across each capacitor will be different depending upon the values of the individual capacitance's.

Then by applying Kirchoff's Voltage Law, (KVL) to the above circuit, we get:

$$V_{AB} = V_{C1} + V_{C2} + V_{C3} = 12V$$

$$V_{C1} = \frac{Q_T}{C_1}, \quad V_{C2} = \frac{Q_T}{C_2}, \quad V_{C3} = \frac{Q_T}{C_3}$$

Since Q = CV and rearranging for V = Q/C, substituting Q/C for each capacitor voltage V_C in the above KVL equation will give us:

$$V_{AB} = \frac{Q_T}{C_T} = \frac{Q_T}{C_1} + \frac{Q_T}{C_2} + \frac{Q_T}{C_3}$$

dividing each term through by Q gives

Series Capacitors Equation

$$\frac{1}{C_T} = \frac{1}{C_1} + \frac{1}{C_2} + \frac{1}{C_3} + \ldots etc$$

When adding together **Capacitors in Series**, the reciprocal (1/C) of the individual capacitors are all added together (just like resistors in parallel) instead of the capacitance's themselves. Then the total value for capacitors in series equals the reciprocal of the sum of the reciprocals of the individual capacitances.

Capacitors in Series Example No1

Taking the three capacitor values from the above example, we can calculate the total capacitance, C_T for the three capacitors in series as:

$$\frac{1}{C_T} = \frac{1}{0.1\mu F} + \frac{1}{0.2\mu F} + \frac{1}{0.3\mu F} = \frac{1}{18.33 \times 10^6}$$

$$\therefore C_T = 0.055\mu F \text{ or } 55nF$$

One important point to remember about capacitors that are connected together in a series configuration, is that the total circuit capacitance (C_T) of any number of capacitors connected together in

series will always be **LESS** than the value of the smallest capacitor in the series and in our example above $C_T = 0.055uF$ with the value of the smallest capacitor in the series chain is only 0.1uF.

This reciprocal method of calculation can be used for calculating any number of individual capacitors connected together in a single series network. If however, there are only two capacitors in series, then a much simpler and quicker formula can be used and is given as:

$$C_T = \frac{C_1 \times C_2}{C_1 + C_2}$$

If the two series connected capacitors are equal and of the same value, that is: $C_1 = C_2$, we can simplify the above equation further as follows to find the total capacitance of the series combination.

$$C_T = \frac{C^2}{2C} = \frac{C}{2} = \frac{1}{2}C$$

Then we can see that if and only if the two series connected capacitors are the same and equal, then the total capacitance, C_T will be exactly equal to one half of the capacitance value, that is: C/2.

With series connected resistors, the sum of all the voltage drops across the series circuit will be equal to the applied voltage V_S (Kirchoff's Voltage Law) and this is also true about capacitors in series.

With series connected capacitors, the capacitive reactance of the capacitor acts as an impedance due to the frequency of the supply. This capacitive reactance produces a voltage drop across each capacitor, therefore the series connected capacitors act as a capacitive voltage divider network.

The result is that the voltage divider formula applied to resistors can also be used to find the individual voltages for two capacitors in series. Then:

$$V_{CX} = V_S \frac{C_T}{C_X}$$

Where: C_X is the capacitance of the capacitor in question, V_S is the supply voltage across the series chain and V_{CX} is the voltage drop across the target capacitor.

Capacitors in Parallel

When capacitors are connected together in parallel the total or equivalent capacitance, C_T in the circuit is equal to the sum of all the individual capacitors added together. This is because the top plate of capacitor, C_1 is connected to the top plate of C_2 which is connected to the top plate of C_3 and so on.

The same is also true of the capacitors bottom plates. Then it is the same as if the three sets of plates were touching each other and equal to one large single plate thereby increasing the effective plate area in m^2.

Since capacitance, C is related to plate area ($C = \varepsilon A/d$) the capacitance value of the combination will also increase. Then the total capacitance value of the capacitors connected together in parallel is

actually calculated by adding the plate area together. In other words, the total capacitance is equal to the sum of all the individual capacitance's in parallel. You may have noticed that the total capacitance of parallel capacitors is found in the same way as the total resistance of series resistors.

The currents flowing through each capacitor and as we saw in the previous tutorial are related to the voltage. Then by applying Kirchoff's Current Law, (KCL) to the above circuit, we have

$$i_1 = C_1 \frac{dv}{dt}, \quad i_2 = C_2 \frac{dv}{dt}, \quad i_3 = C_3 \frac{dv}{dt}$$

$$i_T = i_1 + i_2 + i_3$$

$$\therefore i_T = C_1 \frac{dv}{dt} + C_2 \frac{dv}{dt} + C_3 \frac{dv}{dt}$$

and this can be re-written as:

$$i_T = \left(C_1 + C_2 + C_3 \right) \frac{dv}{dt}$$

$$\text{or} \quad i_T = C_T \frac{dv}{dt}$$

Then we can define the total or equivalent circuit capacitance, C_T as being the sum of all the individual capacitance's add together giving us the generalized equation of:

Parallel Capacitors Equation

$$C_T = C_1 + C_2 + C_3 + \ldots etc$$

When adding together capacitors in parallel, they must all be converted to the same capacitance units, whether it is uF, nF or pF. Also, we can see that the current flowing through the total capacitance value, C_T is the same as the total circuit current, i_T

We can also define the total capacitance of the parallel circuit from the total stored coulomb charge using the $Q = CV$ equation for charge on a capacitors plates. The total charge Q_T stored on all the plates equals the sum of the individual stored charges on each capacitor therefore,

$$Q_T = Q_1 + Q_2 + Q_3 \quad \text{but, } Q = CV$$

$$\therefore Q_T = CV_T = CV_1 + CV_2 + CV_3$$

$$\text{or} \quad C_T = C_1 + C_2 + C_3$$

As the voltage, (V) is common for parallel connected capacitors, we can divide both sides of the above equation through by the voltage leaving just the capacitance and by simply adding together the

48

value of the individual capacitances gives the total capacitance, C_T. Also, this equation is not dependent upon the number of **Capacitors in Parallel** in the branch, and can therefore be generalized for any number of N parallel capacitors connected together.

Capacitors in Parallel Example No1

So by taking the values of the three capacitors from the above example, we can calculate the total equivalent circuit capacitance C_T as being:

$$C_T = C_1 + C_2 + C_3 = 0.1uF + 0.2uF + 0.3uF = 0.6uF$$
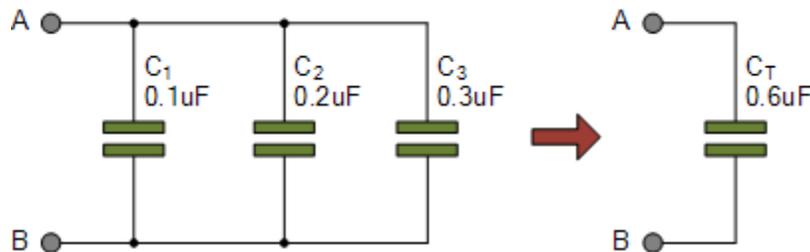
One important point to remember about parallel connected capacitor circuits, the total capacitance (C_T) of any two or more capacitors connected together in parallel will always be **GREATER** than the value of the largest capacitor in the group as we are adding together values. So in our example above $C_T = 0.6uF$ whereas the largest value capacitor is only 0.3uF.

When 4, 5, 6 or even more capacitors are connected together the total capacitance of the circuit C_T would still be the sum of all the individual capacitors added together and as we know now, the total capacitance of a parallel circuit is always greater than the highest value capacitor.

This is because we have effectively increased the total surface area of the plates. If we do this with two identical capacitors, we have doubled the surface area of the plates which in turn doubles the capacitance of the combination and so on.

Resistors in Series and Parallel Example No2

Find the equivalent resistance, REQ for the following resistor combination circuit.

Again, at first glance this resistor ladder network may seem a complicated task, but as before it is just a combination of series and parallel resistors connected together. Starting from the right hand side and using the simplified equation for two parallel resistors, we can find the equivalent resistance of the R8 to R10 combination and call it RA.

$$R_A = \frac{R_9 \times (R_8 + R_{10})}{R_9 + R_8 + R_{10}} = \frac{6 \times (10 + 2)}{6 + 10 + 2} = 4\Omega$$

RA is in series with R7 therefore the total resistance will be RA + R7 = 4 + 8 = 12Ω as shown.
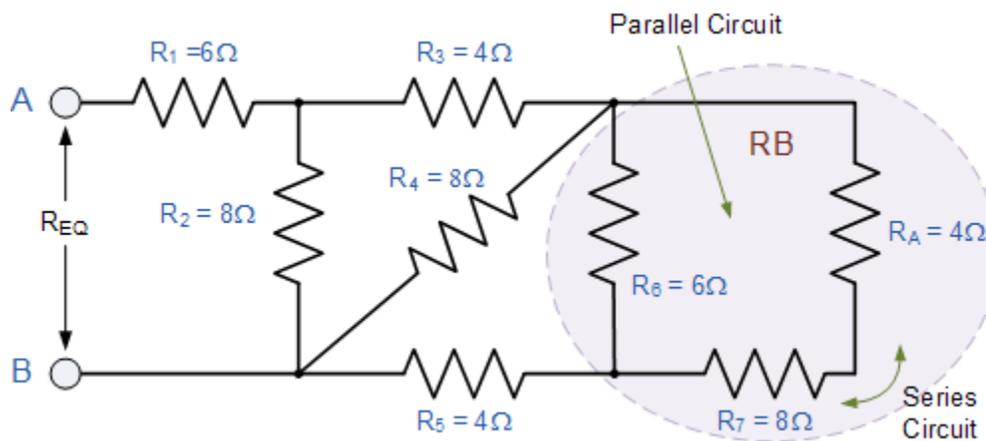
This resistive value of 12Ω is now in parallel with R6 and can be calculated as RB.

$$R_B = \frac{R_6 \times (R_A + R_7)}{R_6 + R_A + R_7} = \frac{6 \times (4 + 8)}{6 + 4 + 8} = 4\Omega$$

RB is in series with R5 therefore the total resistance will be RB + R5 = 4 + 4 = 8Ω as shown.
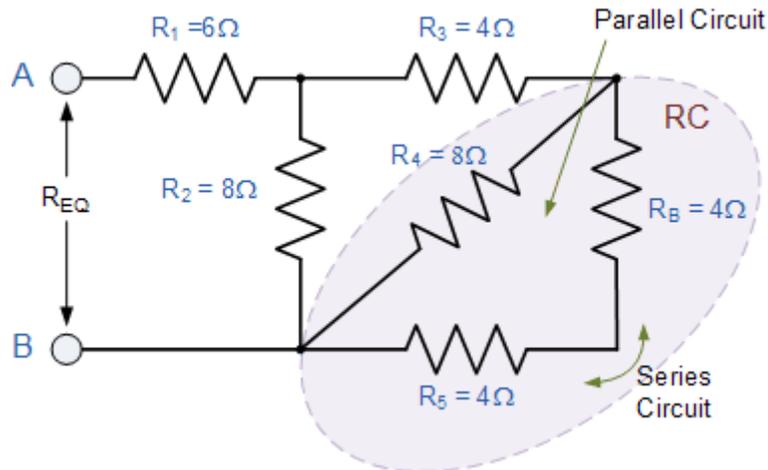
This resistive value of 8Ω is now in parallel with R4 and can be calculated as RC as shown.

$$R_C = \frac{R_4 \times (R_B + R_5)}{R_4 + R_B + R_5} = \frac{8 \times (4+4)}{8+4+4} = 4\Omega$$

RC is in series with R3 therefore the total resistance will be RC + R3 = 8Ω as shown.
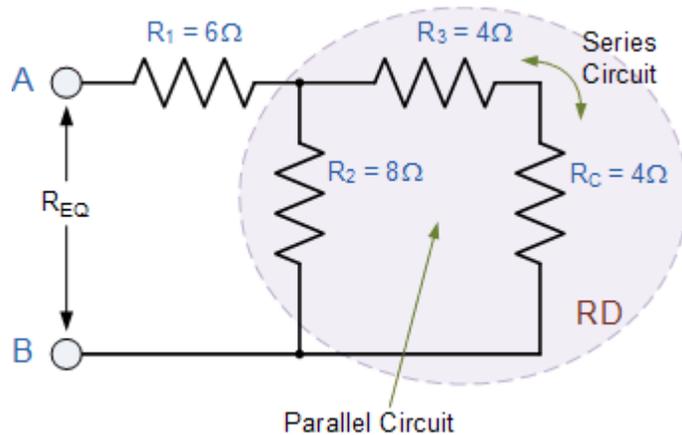
This resistive value of 8Ω is now in parallel with R2 from which we can calculated RD as:

$$R_D = \frac{R_2 \times (R_C + R_3)}{R_2 + R_C + R_3} = \frac{8 \times (4+4)}{8+4+4} = 4\Omega$$

RD is in series with R1 therefore the total resistance will be RD + R1 = 4 + 6 = 10Ω as shown.

Then the complex combinational resistive network above comprising of ten individual resistors connected together in series and parallel combinations can be replaced with just one single equivalent resistance (REQ) of value 10Ω.

When solving any combinational resistor circuit that is made up of resistors in series and parallel branches, the first step we need to take is to identify the simple series and parallel resistor branches and replace them with equivalent resistors.

MODULE – II
ANALYSIS OF ELECTRICAL CIRCUITS

Source transformation technique

Background Study

An electrical network can consist of sources and passive elements. Sources are circuit components which possess their own energy and are capable of transferring this energy to other circuit elements.

There are two basic types of sources: voltage sources and current sources. These can be further classified as independent or dependent. In the case of independent sources, the voltage or current is fixed. If the source is dependent, the value of the voltage or current depends on the amount of current or voltage elsewhere in the circuit.

Passive components do not have their own energy. As a result of this, they are regarded as sinks. However, they influence the amount of current or voltage in a given portion of the circuit. Resistors, capacitors, and inductors are passive components.

Analyzing Electrical Networks

The complexity of electrical networks ranges from very simple—e.g., a voltage divider—to very complicated—e.g., the internal structure of an integrated circuit (IC).

A good electrical designer is expected to have a sound knowledge of the entire system irrespective of its complexity. This is absolutely essential when the issue arises of upgrading or troubleshooting the system.

At this point, it is essential to note that analyzing an electrical circuit is sometimes easy and straightforward, taking only a couple of minutes in its entirety. Sometimes, though, it can involve a lot of hard work (or rather, smart work), and it may even force the analyzer to resort to help from software. Nevertheless, the mode of analysis is based on certain basic rules and theorems.

Here's a list of important theorems along with a brief explanation:

1. Superposition Theorem: Aids in finding the current and voltage in a circuit which has multiple sources; the effects produced by each of the sources individually can be summed.
2. Thevenin's Theorem: Aids in circuit simplification; multiple sources and resistances can be represented by an equivalent circuit with just a single voltage source and a single resistor.
3. Norton's Theorem: Aids in circuit simplification; multiple sources and resistances can be represented by an equivalent circuit with just a single current source and a single resistor.
4. Millman's Theorem: A simplification technique involving circuits with parallel branches.

At this point, it should be noted that all these theorems are based on the basic rules governing the field of electronics, namely, Ohm's Law and Kirchoff's Laws.

In addition, we may sometimes find a circuit which has the resistors connected either in a delta/pi or a star/Y/T configuration. In such cases, we can employ star-to-delta or delta-to-star transformation when analyzing the circuit.

Source Transformation for Independent Sources

Consider the circuit shown in Figure 1; the goal is to find the current (denoted by i) through the central 5 Ω resistor. Here, mesh analysis (Kirchoff's Voltage Law, KVL) cannot be applied readily because the circuit has a branch which has a current source. Thus, we need to devise a method by which we can eliminate this current source from our circuit. However, while doing so, we need to take care that the current and the voltages in the circuit remain unaltered.
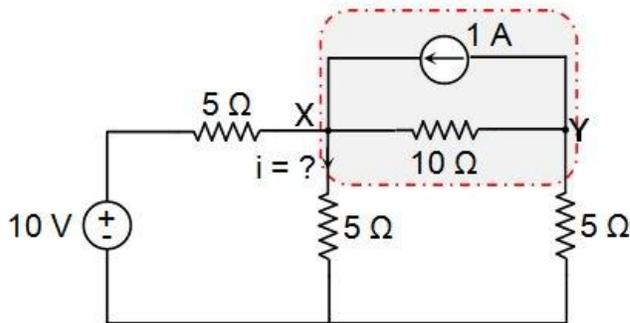
Figure 1

Recall Ohm's Law, which states that

$$I=VRI=VR$$

Step 1: Current-to-Voltage Source Transformation

Looking back at the circuit (Figure 1) again, we can see that the 1 A current source has a 10 Ω resistor in parallel with it. Let us now replace this combination with a voltage source, V = 1 A × 10 Ω = 10 V, and a 10 Ω series resistor. You can see what this looks like in Figure 2. Note that the positive terminal of the voltage source is placed to the left, because the current-source arrow was pointing to the left. These two circuits (Figure 1 and Figure 2) are considered to be equivalent: the 1 A current entering node X from node Y has not changed.
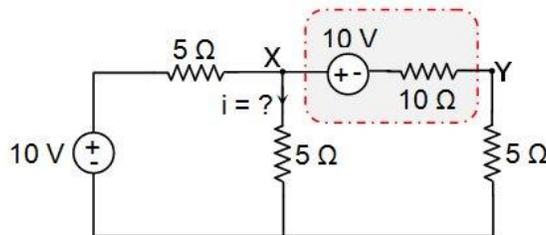
Figure 2

The process carried out here is called source transformation. We have transformed an existing current source with a parallel resistor into an equivalent voltage source with a series resistor.

Step 2: Voltage-to-Current Source Transformation

The circuit in Figure 2 can be further simplified as it has a 10 Ω resistor in series with a 5 Ω resistor. These can be replaced by an equivalent 15 Ω (= 10 Ω + 5 Ω) resistor. The simplified circuit is shown in Figure 3(a).

Now we can easily apply mesh analysis to solve the problem before us. However, there is also a much easier graphical way of achieving this: apply source transformation again!

Previously we transformed a current source with a parallel resistor, but we can also apply source transformation to a voltage source with a series resistor. We have two such arrangements, as indicated in Figure 3(b). This circuit is equivalent to the one shown in Figure 3(a).
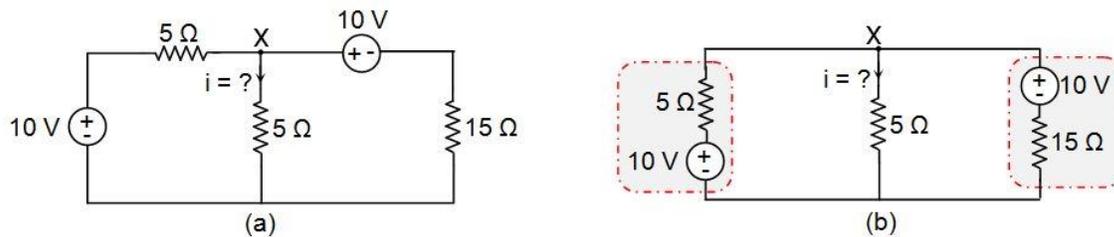
Figure 3

So here we will apply voltage-to-current source transformation, which is very similar to current-to-voltage source transformation. The process involves replacing the voltage source V in series with a resistor R with an equivalent network which has a current source

$$I = \frac{V}{R}$$

in parallel with a resistor R. The current source is oriented such that the arrow points towards the positive terminal of the voltage source being replaced (see Figure 4).

Thus, for the leftmost branch, we have a current source

$$I = \frac{10}{5} = 2 \text{ A}$$

in parallel with a 5 Ω resistor. Similarly, for the rightmost branch, we get

$$I = \frac{10}{15} = \frac{2}{3} \text{ A}$$

in parallel with a 15 Ω resistor. The resulting circuit is shown in Figure 4.

The circuit in Figure 4 has two current sources pointing in the same direction, and hence these can be replaced by a single current source whose value is equal to their sum, i.e.,

$$\frac{8}{3} \text{ A}$$
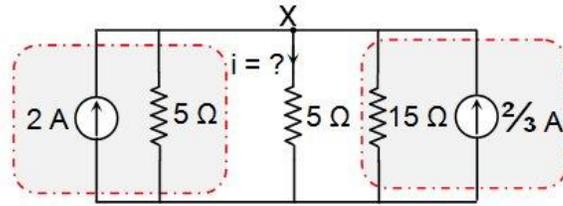
Figure 4

There are three resistors: two 5 Ω resistors and one 15 Ω resistor, all in parallel. We could replace all three of them with an equivalent resistance (R_{EQ}), but our goal is to find the current through the 5 Ω resistor, so we will combine only the other two.

$$R_{EQ}=5×155+15=5×1520=154 \ Ω$$

$$R_{EQ}=5×155+15=5×1520=154 \ Ω$$

Having made these changes, we get the circuit shown in Figure 5.
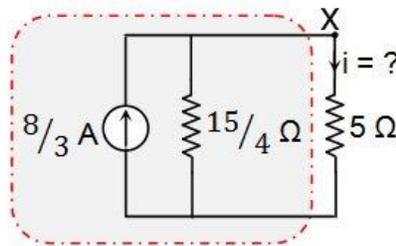
Figure 5

Step 3: Current-to-Voltage Source Transformation (again)

Now, let us apply current-to-voltage source transformation once again for the combination indicated in Figure 5.

Here, the voltage source will be of the value, with the positive terminal towards node X, in series with a resistor.

$$V=83×154=10 \ VV=83×154=10 \ V$$

$$154 \ Ω154 \ Ω$$

The resulting circuit is shown in Figure 6.
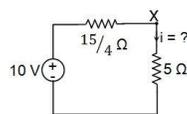
Figure 6

For Figure 6, we can readily apply KVL to obtain the current through the 5 Ω resistor:

$$10-154i-5i=010-154i-5i=0$$

$$10-354i=010-354i=0$$

$$10=354i10=354i$$

$$i=10\times435=87\ Ai=10\times435=87\ A$$

Source Transformation for Dependent Sources

Source transformation is applicable even for the circuits which have dependent sources. Let's consider the circuit shown in Figure 7(a).

Here, one needs to resort to source shifting for the 3 A current source before applying source transformation. This yields the circuit shown in Figure 7(b).
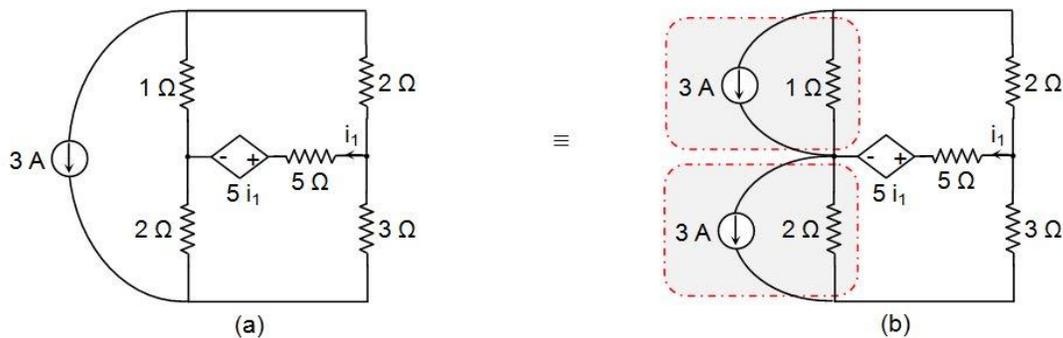
Figure 7

Step 1: Current-to-Voltage Source Transformation

Now let us apply current-to-voltage source transformation for the circuit elements indicated in Figure 7(b).

For the upper portion, we have V = 3 × 1 = 3 V, with the positive terminal oriented downwards, in series with a 1 Ω resistor. Similarly, for the lower portion we get V = 3 × 2 = 6 V, oriented downwards, in series with a 2 Ω resistor. This leads to the circuit shown in Figure 8(a).
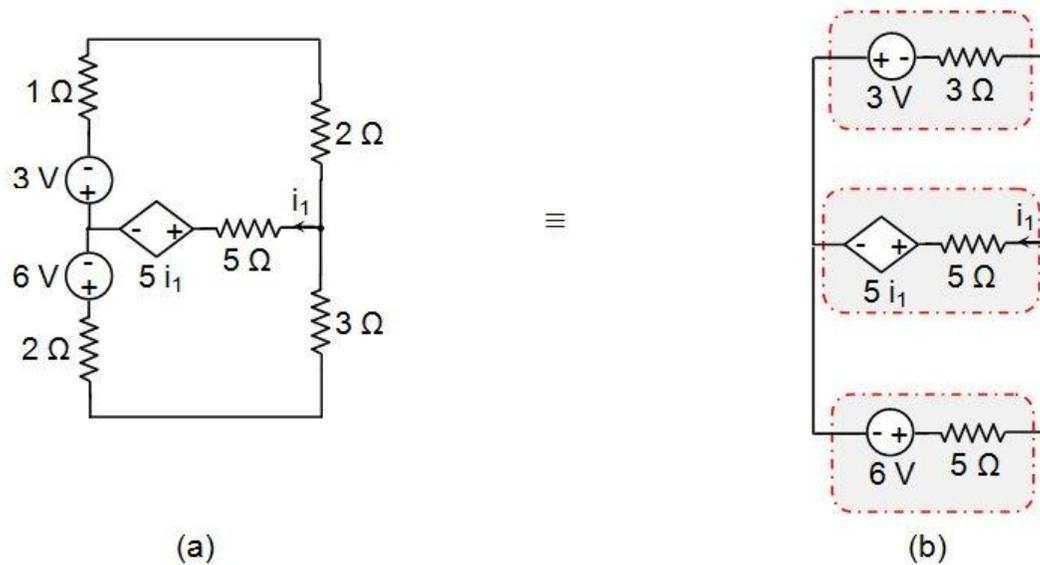
Figure 8

The circuit can be further reduced: the upper mesh has a 1 Ω resistor in series with a 2 Ω resistor, forming an equivalent resistance of 3 Ω, and the lower mesh has a 2 Ω resistor in series with a 3 Ω resistor, which can be replaced by a single 5 Ω resistor. This results in the circuit shown in Figure 8(b).

Step 2: Voltage-to-Current Source Transformation

By looking at Figure 8(b), you can see that we need to apply voltage-to-current source transformation three times (once for each source-plus-resistor combination).

Case 1: For the 3 V source in series with 3 Ω

$$I = \frac{3}{3} = 1 \text{ A}$$

in parallel with a 3 Ω resistor, directed towards the left.

Case 2: For the $5i_1$ dependent voltage source in series with 5 Ω

$$I = \frac{5i_1}{5} = 1i_1 \text{ A}$$

in parallel with a 5 Ω resistor, directed towards the right.

Case 3: For the 6 V source in series with 5 Ω

$$I = \frac{6}{5} \text{ A}$$

in parallel with a 5 Ω resistor directed towards the right.

58

These steps lead to the circuit shown in Figure 9.
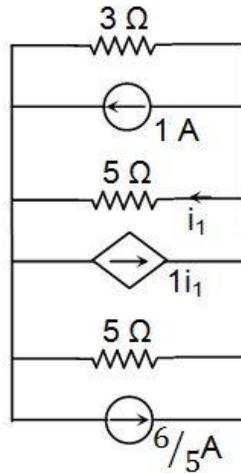
Figure 9

Here, the two independent current sources are oriented in opposite directions and thus can be replaced by a single current source whose value is given by

$$I = \frac{6}{5} - 1 = \frac{1}{5} \text{ A}$$

, directed towards the right.

Next, we can express the value of the dependent current source as

$$I = \frac{1}{5} + 1i_1 = \frac{1 + 5i_1}{5} \text{ A}$$

, towards the right.

We need to find the current flowing through the 5 Ω resistor (the one in the middle). Therefore, we will leave it as it is and replace the other two (i.e., the 3 Ω and 5 Ω resistors on top and bottom) with an equivalent resistance: R = 5 ∥ 3 =

$$\frac{5 \times 3}{5 + 3} = \frac{15}{8} \text{ Ω}$$

.

The circuit can now be depicted as follows:
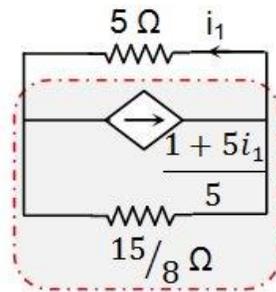
Figure 10

Step 3: Current-to-Voltage Source Transformation (again)

Finally, we can transform the dependent current source into a dependent voltage source with the value

$$V' = \frac{1+5i_1}{5} \times \frac{15}{8} = \frac{3+15i_1}{8} \ V$$

This will be in series with a

$$\frac{15}{8} \ \Omega$$

resistor and will have its positive terminal towards the right.
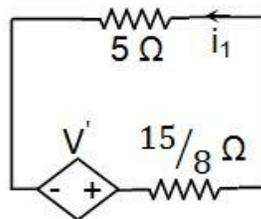
Figure 11

Now, KVL can be used to compute i_1:

$$\frac{3+15i_1}{8} - \frac{15}{8} \times i_1 - 5i_1 = 0$$

$$\frac{3}{8} + \frac{15i_1}{8} - \frac{15}{8} \times i_1 - 5i_1 = 0$$

$$\frac{3}{8} - 5i_1 = 0$$

$$\frac{3}{8} = 5i_1$$

$$i_1 = \frac{3}{40} \ A$$

Example:

Consider the below voltage source circuit with a voltage of 20 V and a internal resistance of 5 ohms. This circuit is transformed into the current source by placing a resistor of the same value with a current source. This current source value can be determined by,

$$\text{Is} = \text{Vs/Rs}$$

$$= 20/\ 5$$

$$= 4 \text{ amps}$$

The equivalent current source with a current of 4A and parallel resistor of 5 ohms is shown below.
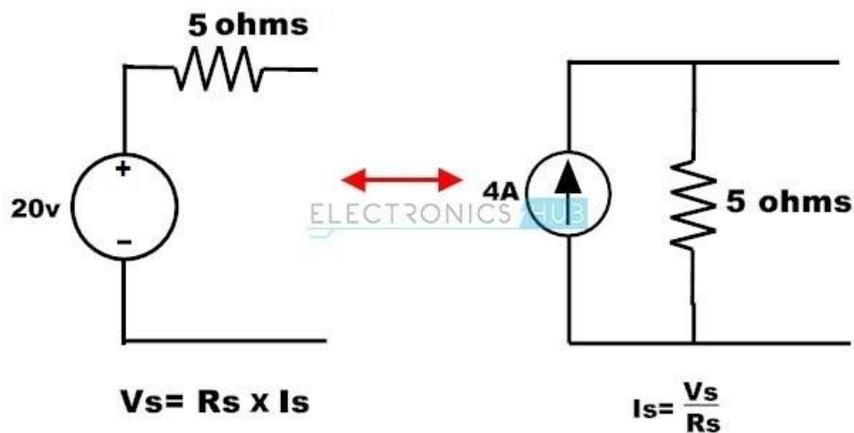
Conversion of Current Source to Voltage Source

The current source transformed into a voltage source by interchanging parallel resistor in series. Let us see how it could work.

Steps:

- Find the parallel resistance of the constant current source and place in series with a voltage source.
- Determine the open circuit voltage value of the voltage source by applying ohms law.
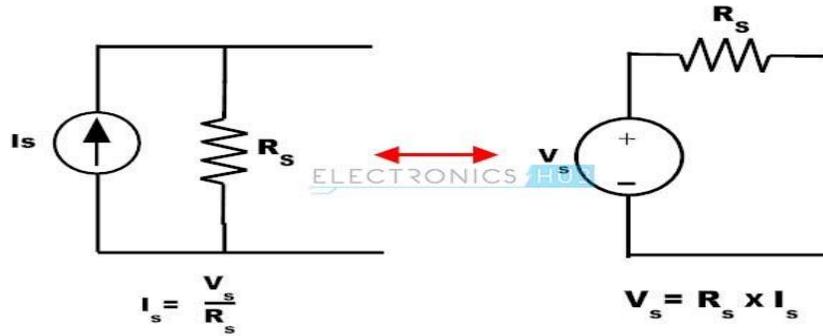
In the above figure, a current source is converted into a voltage source by placing resistance Rs in series with a voltage source and the value of the voltage source is calculated as,

$$Vs = Is * Rs$$

Example:

Consider the below example for current source transformation, where current source is of 10A with a parallel resistance of 3 ohms. To calculate the value of voltage in voltage source apply the simple ohms law, then,

$$Vs = Is * Rs$$

$$Vs = 10 * 3$$

$$= 30 \text{ Volts.}$$

Therefore the equivalent voltage source of this transformation consists a voltage source 30 V with a series resistance 3 ohms.
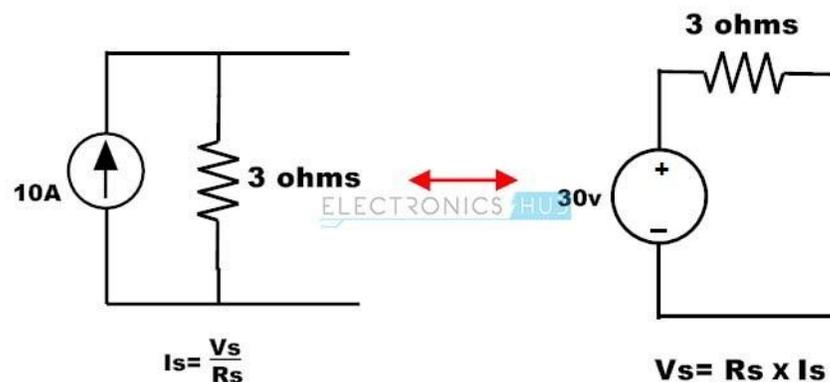
Example of Source Transformations

- Consider the below example where we have to find the voltage Vo by applying the source transformation. This circuit consists of both current and voltage sources. Let us see how we can apply source transformation to simplify the circuit below.
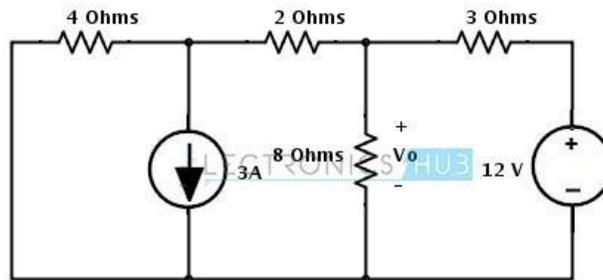
- In the circuit there are two areas where we can apply the source transformation since current source has a parallel resistor and voltage source has a series resistor as shown in figure. So these configurations are necessary requirements to apply the source transformation.
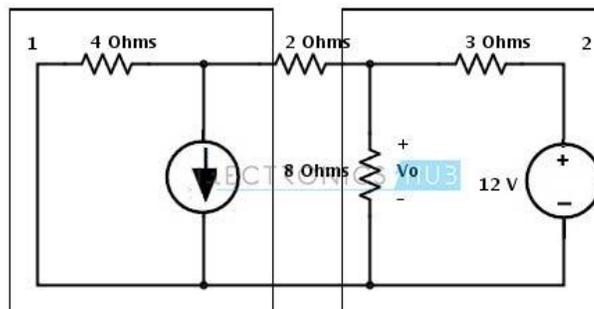
- First, consider the current source with a parallel resistance of 4 ohms. This current source can be transformed into a voltage source by 4 ohms series with a voltage source and voltage source value is determined as

$$Vs = Is * R$$

$$= 3 * 4$$

$$= 12 \text{ Volts}$$

Consider the direction of the current as it downwards so the voltage terminals in voltage source are also changes as shown in figure.
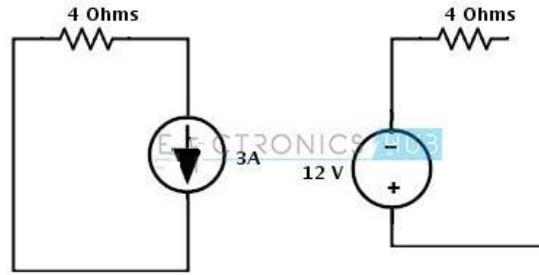
- Place the above voltage source with a series resistance in the circuit, then we get below figure.
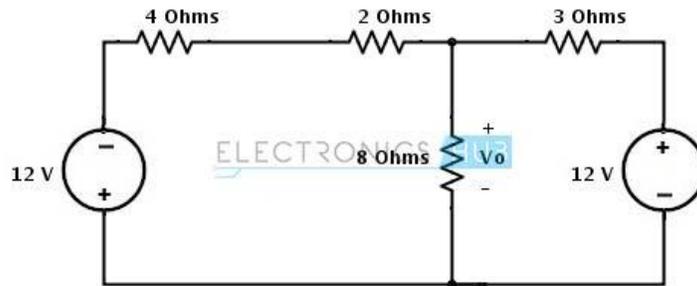
- Resistors 4 ohms and 2 ohms are in series , hence the total series resistance will be 6 ohms as shown below.
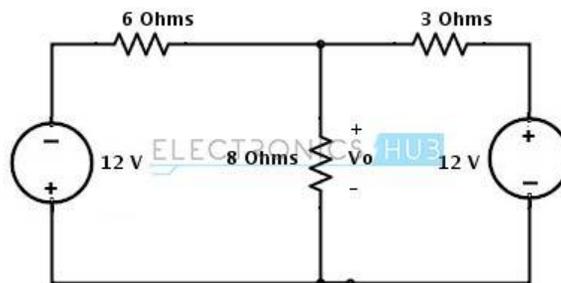
- Again the voltage source of 12V with series resistor 6 ohms can be transformed into a current source. Therefore consider to transform it.
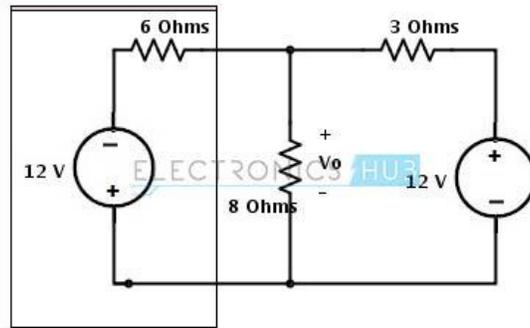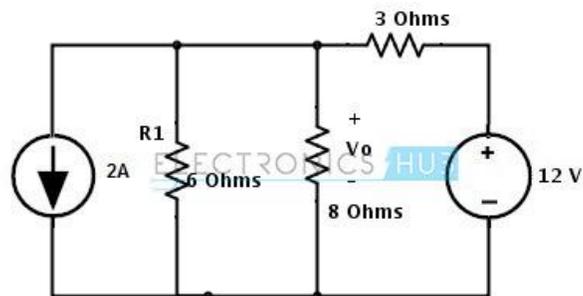
- This 12V voltage source with 6 ohm resistor combination can be converted into the current source by placing 6 ohms resistor in parallel with a current source. And the value of current in current source can be determined as

$$Is = Vs/R$$

$$= 12/6$$

$$= 2 Amps$$

The direction of current flow is represented in below figure.
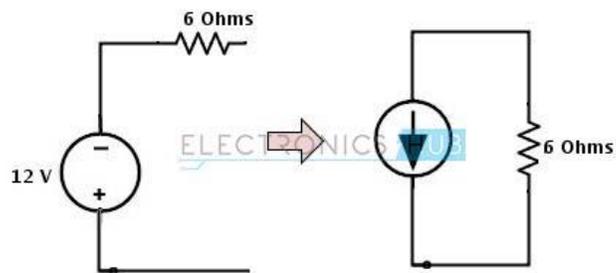
- Insert the above current source in the main circuit, then we get

- On the right hand side, there is a voltage source with a 3 ohms resistor so this can be transformed into a current source by placing a 3 ohm resistor in parallel with a current source and this current source value is calculated as

$$Is = Vs/Rs$$

$$= 12/3$$

$$= 4\ Amps$$

The direction of current in current source is shown in figure.
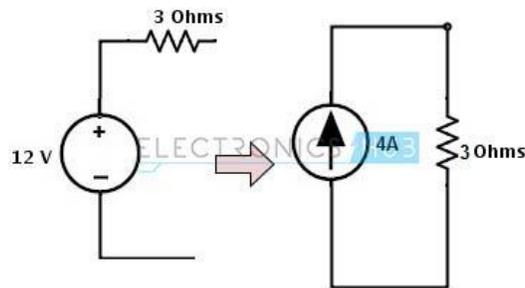
- Insert the above current source in simplified circuit, then we get a final circuit as
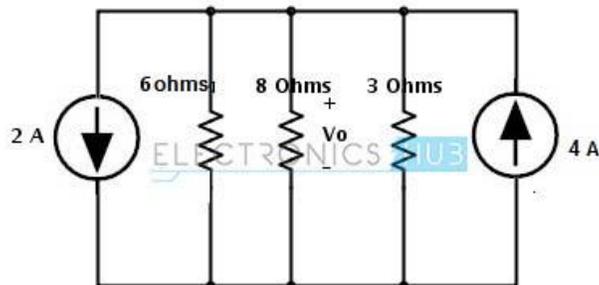
From the above simplified circuit the current sources are appeared to be opposite to each other. The node current through the circuit will be

$$Is = I1 - I2$$

$$= 4-2$$

$$= 2\ amps$$

By applying the divider rule, the current through the resistor 8 ohms is

$$Io = Is * (1/Ro/ ((1/Ro) + (1/R1) + (1/R2))$$

$$= 2 * (1/8/((1/8) + (1/6) + (1/3))$$

$$= 0.4 \text{ Amps}$$

Therefore, the voltage across the resistor 8 ohms is

$$Vo = Io * Ro$$

$$= 0.4 * 8$$

$$= 3.2 \text{ Volts}$$

STAR DELTA TRANSFORMATION

Star-Delta Transformations and Delta-Star Transformations allow us to convert impedances connected together in a 3-phase configuration from one type of connection to another

We can now solve simple series, parallel or bridge type resistive networks using Kirchhoff's laws mesh current analysis or nodal voltage analysis techniques but in a balanced 3-phase circuit we can use different mathematical techniques to simplify the analysis of the circuit and thereby reduce the amount of math's involved which in itself is a good thing.

Standard 3-phase circuits or networks take on two major forms with names that represent the way in which the resistances are connected, a **Star** connected network which has the symbol of the letter, Y (wye) and a **Delta** connected network which has the symbol of a triangle, Δ (delta).

If a 3-phase, 3-wire supply or even a 3-phase load is connected in one type of configuration, it can be easily transformed or changed it into an equivalent configuration of the other type by using either the **Star Delta Transformation** or **Delta Star Transformation** process.

A resistive network consisting of three impedances can be connected together to form a Tor "Tee" configuration but the network can also be redrawn to form a **Star** or Y type network as shown below.

T-connected and Equivalent Star Network

T-Network   Star-Network
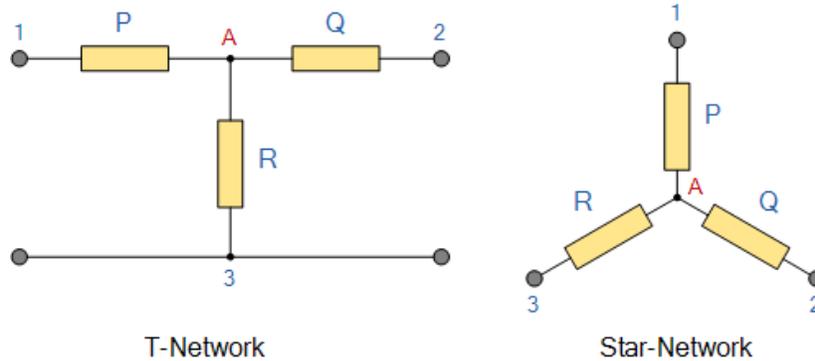
As we have already seen, we can redraw the T resistor network above to produce an electrically equivalent **Star** or Y type network. But we can also convert a Pi or π type resistor network into an electrically equivalent **Delta** or Δ type network as shown below.

Pi-connected and Equivalent Delta Network

Pi-Network   Delta-Network
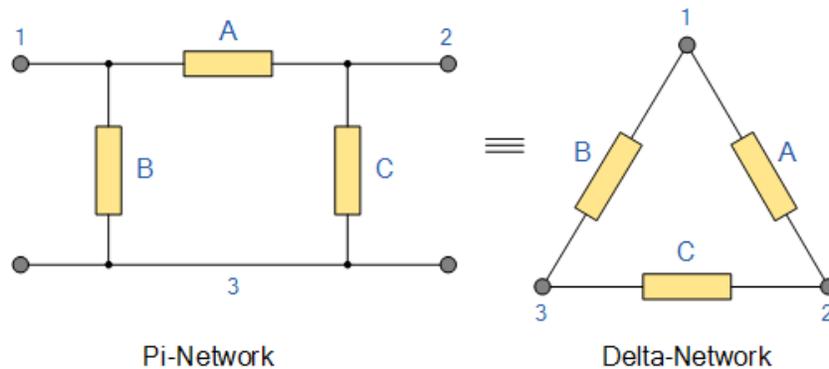
Having now defined exactly what is a **Star** and **Delta** connected network it is possible to transform the Y into an equivalent Δ circuit and also to convert a Δ into an equivalent Ycircuit using a the transformation process.

This process allows us to produce a mathematical relationship between the various resistors giving us a **Star Delta Transformation** as well as a **Delta Star Transformation**.

These circuit transformations allow us to change the three connected resistances (or impedances) by their equivalents measured between the terminals 1-2, 1-3 or 2-3 for either a star or delta connected circuit. However, the resulting networks are only equivalent for voltages and currents external to the star or delta networks, as internally the voltages and currents are different but each network will consume the same amount of power and have the same power factor to each other.

Delta Star Transformation

To convert a delta network to an equivalent star network we need to derive a transformation formula for equating the various resistors to each other between the various terminals. Consider the circuit below.

Delta to Star Network
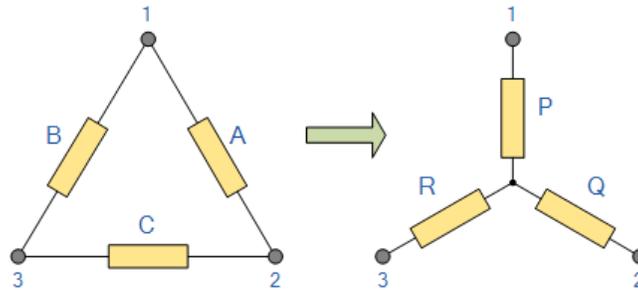
Compare the resistances between terminals 1 and 2.

$$P + Q = A \text{ in parallel with } (B + C)$$

$$P + Q = \frac{A(B + C)}{A + B + C} \quad \text{...EQ1}$$

Resistance between the terminals 2 and 3.

$$Q + R = C \text{ in parallel with } (A + B)$$

$$Q + R = \frac{C(A + B)}{A + B + C} \quad \text{...EQ2}$$

Resistance between the terminals 1 and 3.

$$P + R = B \text{ in parallel with } (A + C)$$

$$P + R = \frac{B(A + C)}{A + B + C} \quad \text{...EQ3}$$

This now gives us three equations and taking equation 3 from equation 2 gives:

$$EQ3 - EQ2 = (P + R) - (Q + R)$$

$$P + R = \frac{B(A + C)}{A + B + C} \ - \ Q + R = \frac{C(A + B)}{A + B + C}$$

$$\therefore P - Q = \frac{BA + CB}{A + B + C} \ - \ \frac{CA + CB}{A + B + C}$$

$$\therefore \ P - Q = \frac{BA - CA}{A + B + C}$$

Then, re-writing Equation 1 will give us:

$$P + Q = \frac{AB + AC}{A + B + C}$$

Adding together equation 1 and the result above of equation 3 minus equation 2 gives:

$$(P - Q) + (P + Q)$$

$$= \frac{BA - CA}{A + B + C} + \frac{AB + AC}{A + B + C}$$

$$= 2P = \frac{2AB}{A + B + C}$$

From which gives us the final equation for resistor P as:

$$P = \frac{AB}{A + B + C}$$

Then to summarize a little about the above maths, we can now say that resistor P in a Star network can be found as Equation 1 plus (Equation 3 minus Equation 2) or Eq1 + (Eq3 – Eq2).

Similarly, to find resistor Q in a star network, is equation 2 plus the result of equation 1 minus equation 3 or Eq2 + (Eq1 – Eq3) and this gives us the transformation of Q as:

$$Q = \frac{AC}{A + B + C}$$

and again, to find resistor R in a Star network, is equation 3 plus the result of equation 2 minus equation 1 or Eq3 + (Eq2 – Eq1) and this gives us the transformation of R as:

$$R = \frac{BC}{A + B + C}$$

When converting a delta network into a star network the denominators of all of the transformation formulas are the same: A + B + C, and which is the sum of ALL the delta resistances. Then to convert any delta connected network to an equivalent star network we can summarized the above transformation equations as:

Delta to Star Transformations Equations

$$P = \frac{AB}{A+B+C} \quad Q = \frac{AC}{A+B+C} \quad R = \frac{BC}{A+B+C}$$

If the three resistors in the delta network are all equal in value then the resultant resistors in the equivalent star network will be equal to one third the value of the delta resistors. This gives each resistive branch in the star network a value of: $R_{STAR} = 1/3*R_{DELTA}$ which is the same as saying: $(R_{DELTA})/3$

Delta – Star Example No1

Convert the following Delta Resistive Network into an equivalent Star Network.
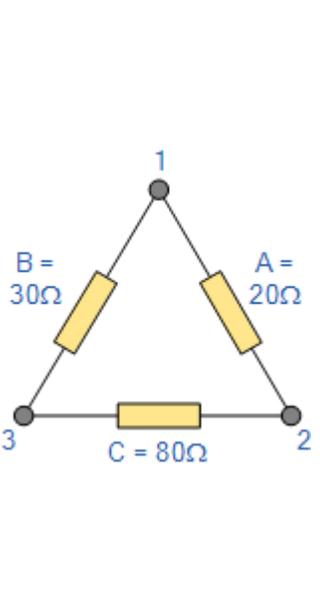
$$Q = \frac{AC}{A+B+C} = \frac{20 \times 80}{130} = 12.31\Omega$$

$$P = \frac{AB}{A+B+C} = \frac{20 \times 30}{130} = 4.61\Omega$$

$$R = \frac{BC}{A+B+C} = \frac{30 \times 80}{130} = 18.46\Omega$$

Star Delta Transformation

Star Delta transformation is simply the reverse of above. We have seen that when converting from a delta network to an equivalent star network that the resistor connected to one terminal is the product of the two delta resistances connected to the same terminal, for example resistor P is the product of resistors A and B connected to terminal 1.

By rewriting the previous formulas a little we can also find the transformation formulas for converting a resistive star network to an equivalent delta network giving us a way of producing a star delta transformation as shown below.

Star to Delta Transformation
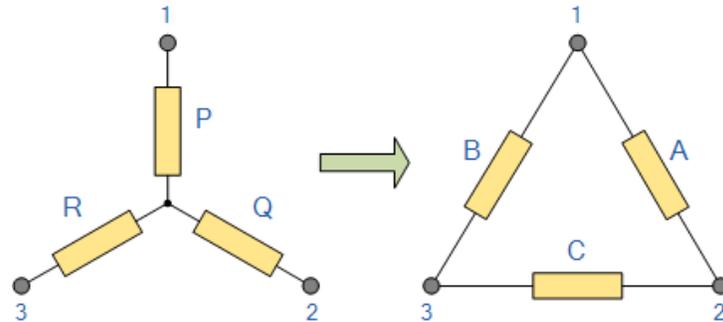
The value of the resistor on any one side of the delta, Δ network is the sum of all the two-product combinations of resistors in the star network divide by the star resistor located "directly opposite" the delta resistor being found. For example, resistor A is given as:

$$A = \frac{PQ + QR + RP}{R}$$

with respect to terminal 3 and resistor B is given as:

$$B = \frac{PQ + QR + RP}{Q}$$

with respect to terminal 2 with resistor C given as:

$$C = \frac{PQ + QR + RP}{P}$$

with respect to terminal 1.

By dividing out each equation by the value of the denominator we end up with three separate transformation formulas that can be used to convert any Delta resistive network into an equivalent star network as given below.

Star Delta Transformation Equations

$$A = \frac{PQ}{R} + Q + P \quad B = \frac{RP}{Q} + P + R \quad C = \frac{QR}{P} + Q + R$$

72

One final point about converting a star resistive network to an equivalent delta network. If all the resistors in the star network are all equal in value then the resultant resistors in the equivalent delta network will be three times the value of the star resistors and equal, giving: $R_{DELTA} = 3*R_{STAR}$
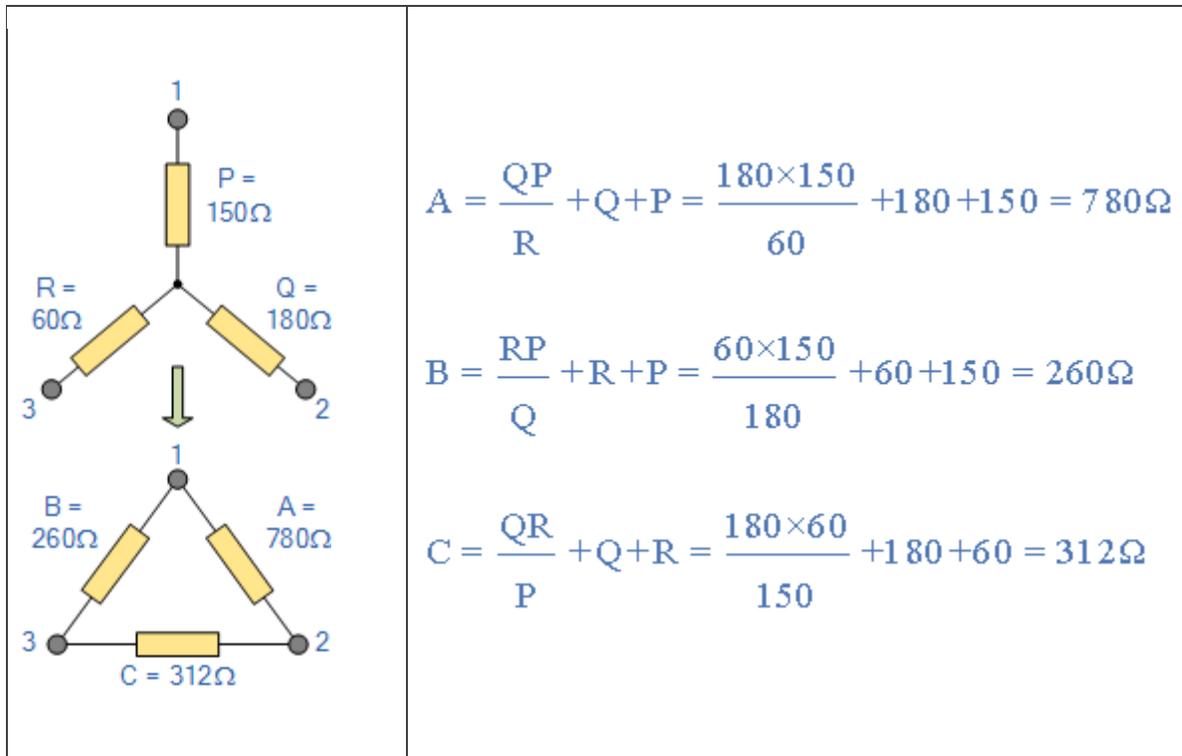
Star – Delta Example No2

Convert the following Star Resistive Network into an equivalent Delta Network.

$$A = \frac{QP}{R} + Q + P = \frac{180 \times 150}{60} + 180 + 150 = 780\,\Omega$$

$$B = \frac{RP}{Q} + R + P = \frac{60 \times 150}{180} + 60 + 150 = 260\,\Omega$$

$$C = \frac{QR}{P} + Q + R = \frac{180 \times 60}{150} + 180 + 60 = 312\,\Omega$$

Both **Star Delta Transformation** and **Delta Star Transformation** allows us to convert one type of circuit connection into another type in order for us to easily analyse the circuit. These transformation techniques can be used to good effect for either star or delta circuits containing resistances or impedances.

Star – Delta Example

Convert the following Star Resistive Network into an equivalent Delta Network.

$$A = \frac{QP}{R} + Q + P = \frac{180 \times 150}{60} + 180 + 150 = 780\,\Omega$$

$$B = \frac{RP}{Q} + R + P = \frac{60 \times 150}{180} + 60 + 150 = 260\,\Omega$$

$$C = \frac{QR}{P} + Q + R = \frac{180 \times 60}{150} + 180 + 60 = 312\,\Omega$$

Both **Star Delta Transformation** and **Delta Star Transformation** allows us to convert one type of circuit connection into another type in order for us to easily analyse the circuit. These transformation techniques can be used to good effect for either star or delta circuits containing resistances or impedances.

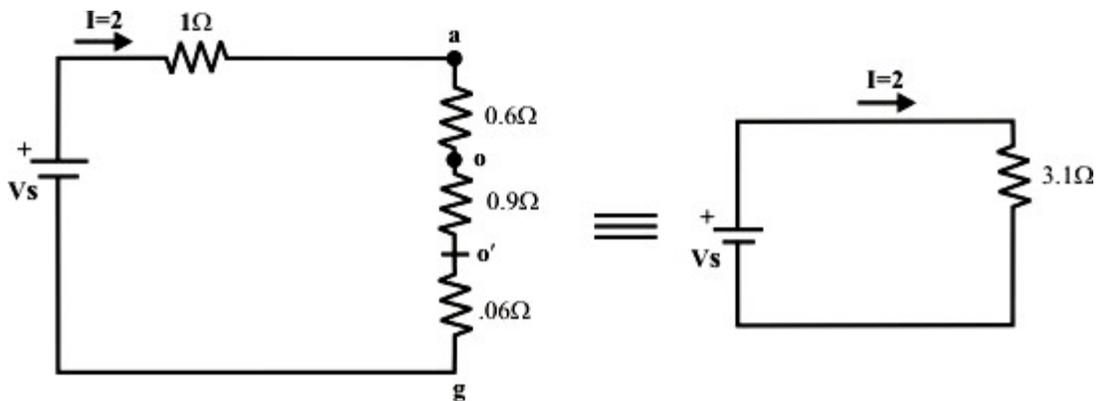Find the value of the voltage source ( ) that delivers 2 Amps current through the circuit as shown in fig.6.3.
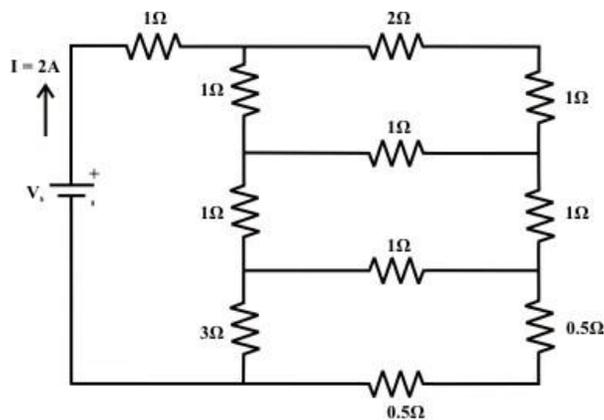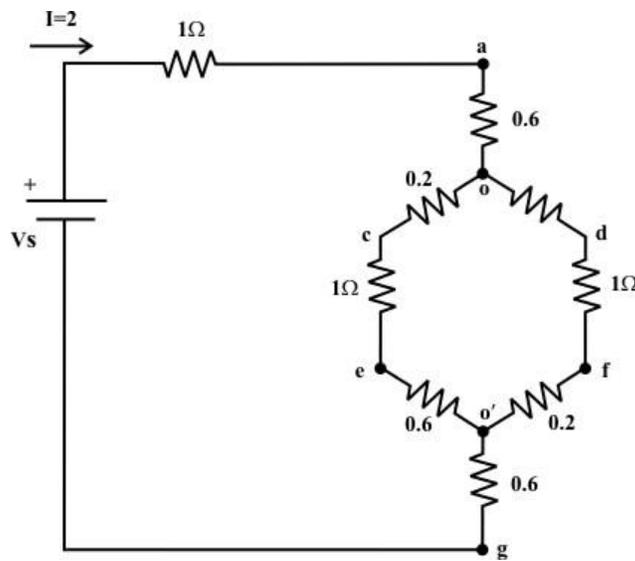

Fig. 6.3

Solution:
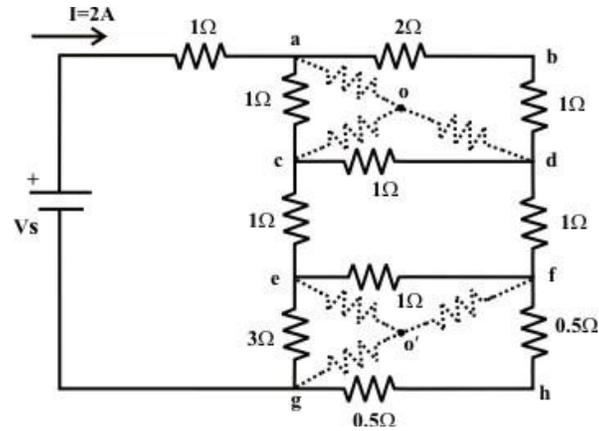
Convert the three terminals Δ-network (a-c-d & e-f-g) into an equivalent Y-connected network.

Consider the Δ-connected network 'a-c-d' and the corresponding equivalent Y-connected

resistor values are given as

The source V_s that delivers $2A$ current through the circuit can be obtained as
$V_s = I \times 3.2 = 2 \times 3.1 = 6.2 Volts$.

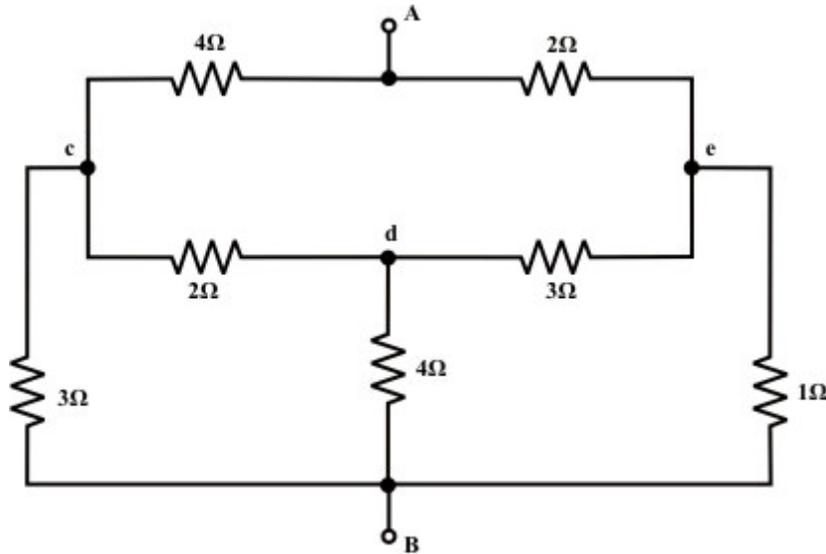**Example:** Determine the equivalent resistance between the terminals *AandB* of network shown in fig.6.4 (a).

Fig. 6.4 (a)

A 'Δ' is substituted for the 'Y' between points c, d, and e as   shown in fig.6.4(b); then unknown resistances value for Y *to* Δ transformation are computed below.

$$R_{cB} = 2+4+\frac{2\times 4}{3} = 8.66\Omega\,;\ R_{eB} = 3+4+\frac{4\times 3}{2} = 13\Omega\,;\ R_{ce} = 2+3+\frac{2\times 3}{4} = 6.5\Omega$$
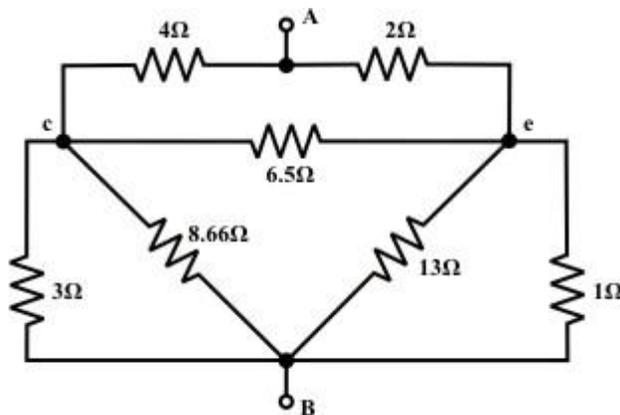

Fig. 6.4 (b)

Next we transform 'Δ'connected 3-terminal resistor to an equivalent 'Y' connected network between points 'A'; 'c' and 'e' (see fig.6.4(b)) and the corresponding Y connected resistances value are obtained using the following expression. Simplified circuit after conversion is shown in fig
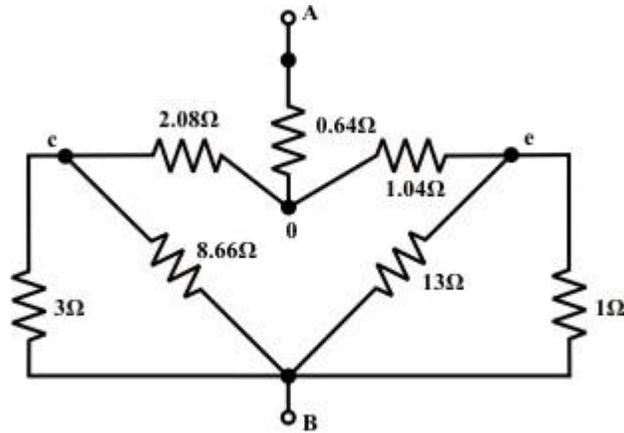
Fig. 6.4 (c)

$$R_{Ao} = \frac{4 \times 2}{4 + 2 + 6.5} = 0.64\Omega; \quad R_{co} = \frac{4 \times 6.5}{4 + 2 + 6.5} = 2.08\Omega; \quad R_{eo} = \frac{6.5 \times 2}{4 + 2 + 6.5} = 1.04\Omega;$$

The circuit shown in fig.6.5(c) can further be reduced by considering two pairs of parallel branches $3 \| 8.66$ *and* $13 \| 1$ and the corresponding simplified circuit is shown in
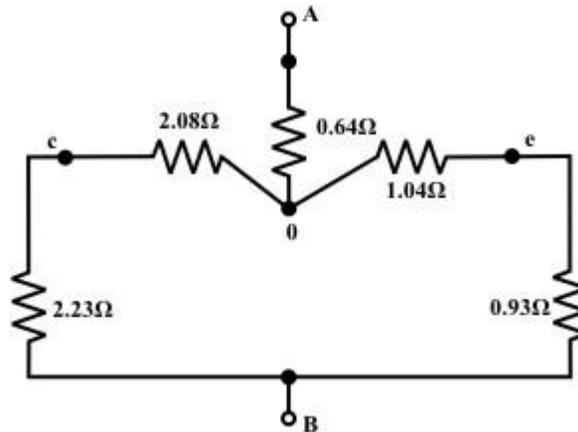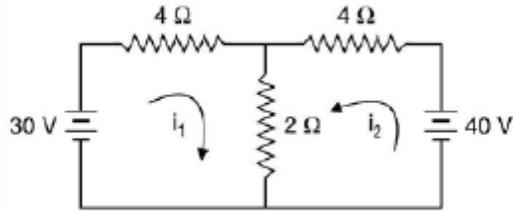
Fig

Fig. 6.4 (d)

MESH ANALYSIS:

Mesh analysis is an technique used to solve the complex networks consisting of more number of meshes. Mesh analysis is nothing but applying KVL to each and every loop in circuit and solving for mesh currents. By finding the mesh currents we can solve any require data of the network.

Let V1 = 30v, V2 = 40v, R1 =4 ohms, R2 = 2 ohms and R3 = 4 ohms

   i1, i2 are the mesh currents, here positive direction of currents are assumed , but in general we can assume current directions in any fashion and can solve.

 Applying KVL to first loop we get,   V1 – i1R1 – (i1+i2)R2.
$$V1 = (R1+R2)i1 + i2.R2. \quad ----1$$

Applying KVL to second loop we get,   V2 – i2R3 – (i1+i2)R2.
$$V2 = (R1+R2)i1 + i2.R3. \quad ----2$$

Hence by solving eq. 1 and 2 we can get mesh currents i1 and i2.

Mesh analysis by inspection method:

Inspection method is direct method using mesh currents can be find directly without applying KVL. Let us take same network as above , representing eq 1 and 2 in matrix form.

V1	=	R1	R1+R2	i1
V2		R1+R2	R3	i2

 Generally we can write in matrix form as,

V1	=	R11	R12	i1
V2		R21	R22	i2

Where, V1 – sum of all the voltage sources in 1st mesh according current flow.

V2 – sum of all the voltage sources in 2nd mesh according current flow.

R11- self resistance of first loop, adding total resistance in 1st loop

R21= R12- mutual resistance between 1st and 2nd loop ,

+ve if mesh currents are in same direction

-ve if mesh currents are in opposite direction

R22- self resistance of second loop, adding total resistance in 2nd loop

NODAL ANALYSIS :

Nodal analysis is an technique used to solve the complex networks consisting of more number of nodes. Node analysis is nothing but applying KCL to each and every node in circuit and solving for node voltages. By finding the node voltages we can solve any require data of the network.
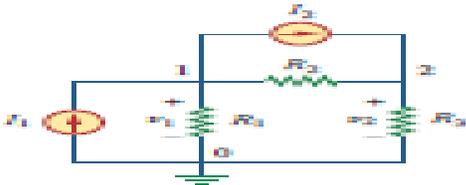
For the above circuit current division takes place at two nodes 1 and 2.

Let , I3,I4,I5 are the currents flowing through R1,R2 and R3.

Applying KCL at node 1,   I1 = I3+I4

$$= Va / R1 + (Va-Vb) / R2$$

$$= Va(1/R1 + 1/R2) - Vb / R2 \text{ ----1}$$

Applying KCL at node 2,   I2+I4 = I5

$$I2+(Va-Vb) / R2 = Vb / R3$$

$$I2 = - Va / R2 + Vb (1/R3 + 1/R2) \text{ ----2}$$

By solving above eq.1 and 2 we can get node voltages V1 and V2.

Nodal analysis by inspection method:

Inspection method is direct method using node voltages can be find directly without applying KCL. Let us take same network as above , representing eq 1 and 2 in matrix form.

I1	=	(1/R1 +1/R2)	-1/ R2	V1
I2		-1/ R2	(1/R3 +1/R2)	V2

Genereally we can write in matrix form as,

I1	=	G11	G12	V1
I2		G21	G22	V2

Where, I1 – sum of all the current sources in 1st node according current flow.

 I2 – sum of all the voltage sources in 2nd node according current flow.

 G11- self conductance of first node, adding total conductance in 1st node

 G21= G12- mutual conductance between 1st and 2nd loop ,

 Always   -ve

 G22- self conductance of second node, adding total conductance in 2nd node

CONVENTIONS:

- power delivered = Vs.Is
- power lost   = I^2 . R
- volatge is positive if flow is from negative to positive called as poetntial rise.(Vs)
- volatge is negative if flow is from positive to negative called as poetntial drop.(I.R)

Mesh analysis exercise problems

Given the Circuit below, find the currents   ,   .
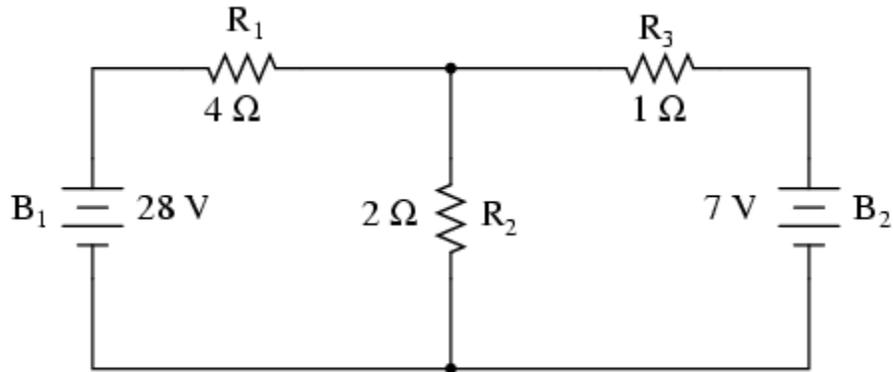
Solution:-

The first step in the Mesh Current method is to identify "loops" within the circuit encompassing all components. In our example circuit, the loop formed by B_1, R_1, and R_2 will be the first while the loop formed by B_2, R_2, and R_3 will be the second. The strangest part of the Mesh Current method is envisioning circulating currents in each of the loops. In fact, this method gets its name from the idea of these currents meshing together between loops like sets of spinning gears:
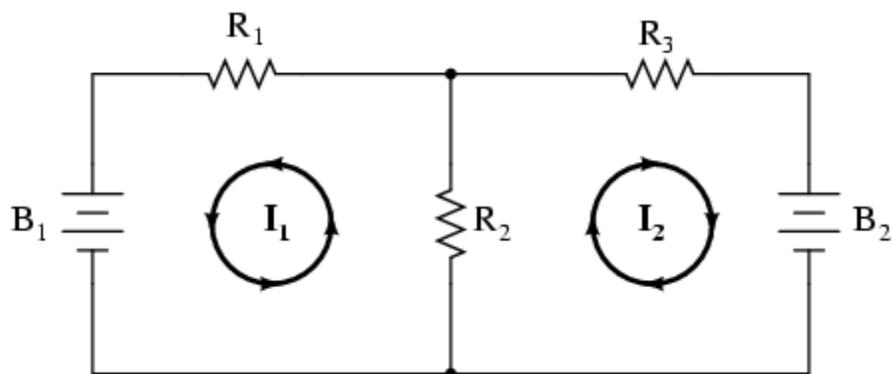
The choice of each current's direction is entirely arbitrary, just as in the Branch Current method, but the resulting equations are easier to solve if the currents are going the same direction through intersecting components (note how currents I_1 and I_2 are both going "up" through resistor R_2, where they "mesh," or intersect). If the assumed direction of a mesh current is wrong, the answer for that current will have a negative value.

The next step is to label all voltage drop polarities across resistors according to the assumed directions of the mesh currents. Remember that the "upstream" end of a resistor will always be negative, and the "downstream" end of a resistor positive with respect to each other, since electrons are negatively charged. The battery polarities, of course, are dictated by their symbol orientations in the diagram, and may or may not "agree" with the resistor polarities (assumed current directions):
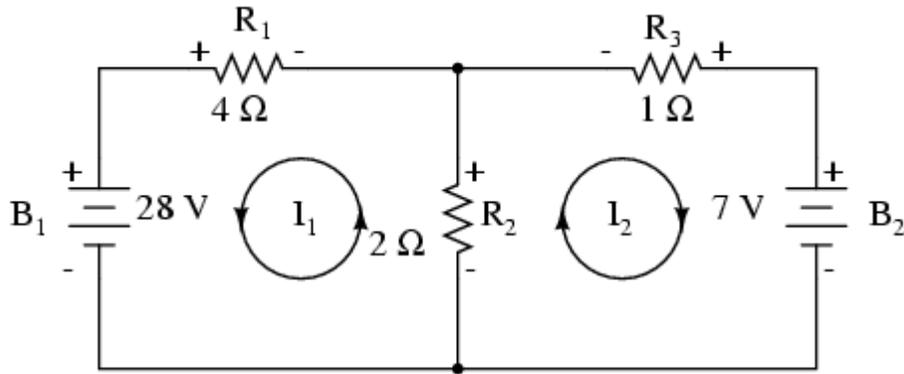
Using Kirchhoff's Voltage Law, we can now step around each of these loops, generating equations representative of the component voltage drops and polarities. As with the Branch Current method, we will denote a resistor's voltage drop as the product of the resistance (in ohms) and its respective mesh current (that quantity being unknown at this point). Where two currents mesh together, we will write that term in the equation with resistor current being the *sum* of the two meshing currents.

Tracing the left loop of the circuit, starting from the upper-left corner and moving counter-clockwise (the choice of starting points and directions is ultimately irrelevant), counting polarity as if we had a voltmeter in hand, red lead on the point ahead and black lead on the point behind, we get this equation:

$$-28 + 2(I_1 + I_2) + 4I_1 = 0$$

Notice that the middle term of the equation uses the sum of mesh currents I_1 and I_2 as the current through resistor R_2. This is because mesh currents I_1 and I_2 are going the same direction through R_2, and thus complement each other. Distributing the coefficient of 2 to the I_1 and I_2 terms, and then combining I_1 terms in the equation, we can simplify as such:

$-28 + 2(I_1 + I_2) + 4I_1 = 0$ *Original form of equation*

. . . distributing to terms within parentheses . . .

$-28 + 2I_1 + 2I_2 + 4I_1 = 0$

. . . combining like terms . . .

$-28 + 6I_1 + 2I_2 = 0$ *Simplified form of equation*

At this time we have one equation with two unknowns. To be able to solve for two unknown mesh currents, we must have two equations. If we trace the other loop of the circuit, we can obtain another

KVL equation and have enough data to solve for the two currents. Creature of habit that I am, I'll start at the upper-left hand corner of the right loop and trace counter-clockwise:

$$- 2(I_1 + I_2) + 7 - 1I_2 = 0$$

Simplifying the equation as before, we end up with:

$$- 2I_1 - 3I_2 + 7 = 0$$

Now, with two equations, we can use one of several methods to mathematically solve for the unknown currents I_1 and I_2:

$$- 28 + 6I_1 + 2I_2 = 0$$

$$- 2I_1 - 3I_2 + 7 = 0$$

. . . rearranging equations for easier solution . . .
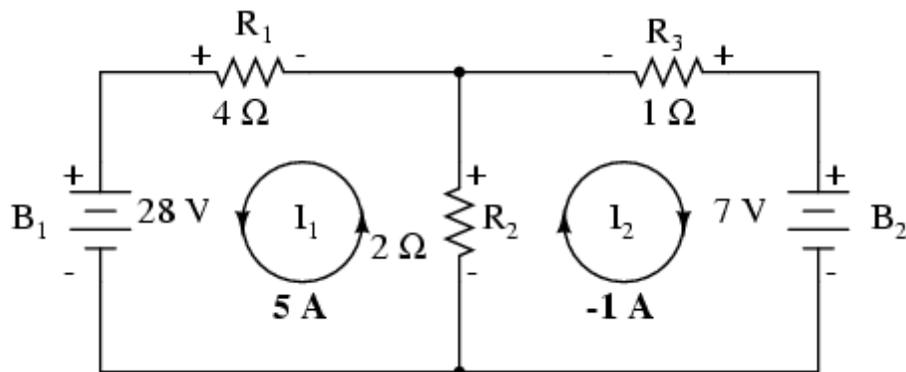
$$6I_1 + 2I_2 = 28$$

$$-2I_1 - 3I_2 = -7$$

Solutions:

$$I_1 = 5 \text{ A}$$

$$I_2 = -1 \text{ A}$$

Knowing that these solutions are values for *mesh* currents, not *branch* currents, we must go back to our diagram to see how they fit together to give currents through all components:

The solution of -1 amp for I_2 means that our initially assumed direction of current was incorrect. In actuality, I_2 is flowing in a counter-clockwise direction at a value of (positive) 1 amp:
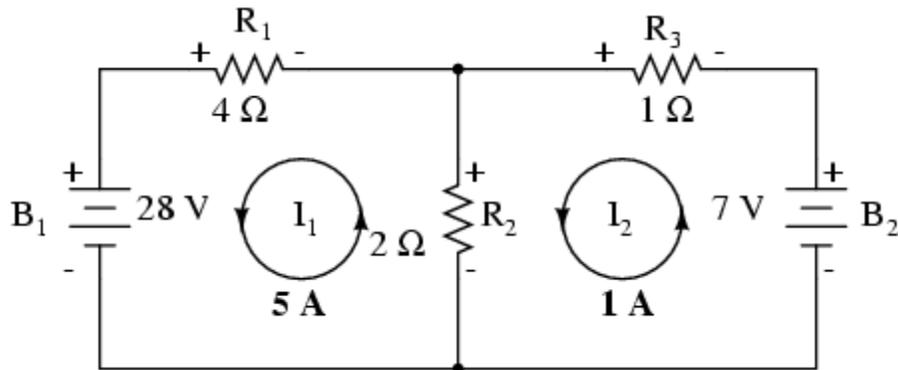

This change of current direction from what was first assumed will alter the polarity of the voltage drops across R_2 and R_3 due to current I_2. From here, we can say that the current through R_1 is 5 amps, with the voltage drop across R_1 being the product of current and resistance (E=IR), 20 volts (positive on the left and negative on the right). Also, we can safely say that the current through R_3 is 1 amp, with a voltage drop of 1 volt (E=IR), positive on the left and negative on the right. But what is happening at R_2?

Mesh current I_1 is going "up" through R_2, while mesh current I_2 is going "down" through R_2. To determine the actual current through R_2, we must see how mesh currents I_1 and I_2 interact (in this case they're in opposition), and algebraically add them to arrive at a final value. Since I_1 is going "up" at 5 amps, and I_2 is going "down" at 1 amp, the *real* current through R_2 must be a value of 4 amps, going "up:"
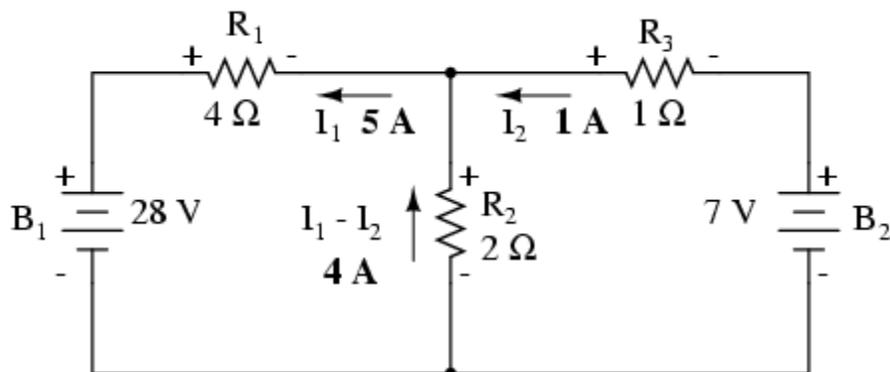

A current of 4 amps through R_2's resistance of 2 Ω gives us a voltage drop of 8 volts (E=IR), positive on the top and negative on the bottom.

The primary advantage of Mesh Current analysis is that it generally allows for the solution of a large network with fewer unknown values and fewer simultaneous equations. Our example problem took three equations to solve the Branch Current method and only two equations using the Mesh Current method. This advantage is much greater as networks increase in complexity:
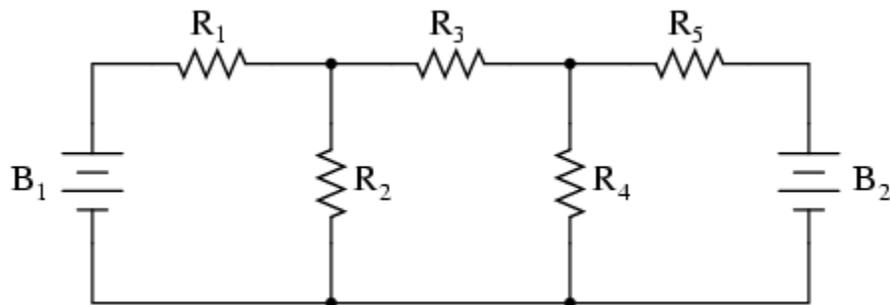
To solve this network using Branch Currents, we'd have to establish five variables to account for each and every unique current in the circuit (I_1 through I_5). This would require five equations for solution, in the form of two KCL equations and three KVL equations (two equations for KCL at the nodes, and three equations for KVL in each loop):
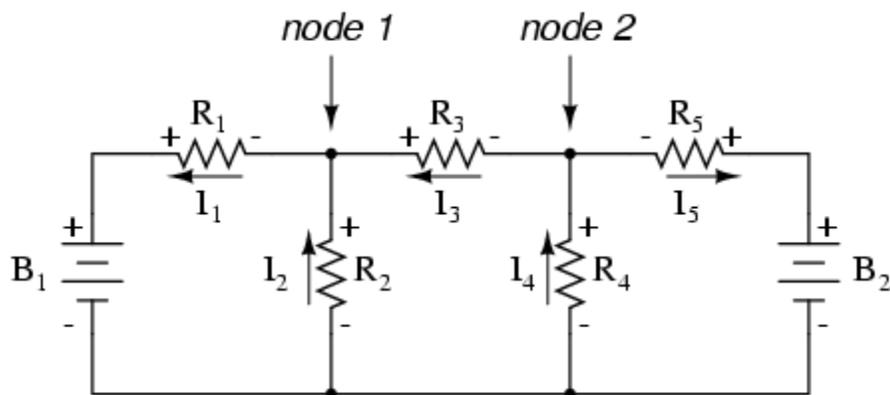
$- I_1 + I_2 + I_3 = 0$ *Kirchhoff's Current Law at node 1*
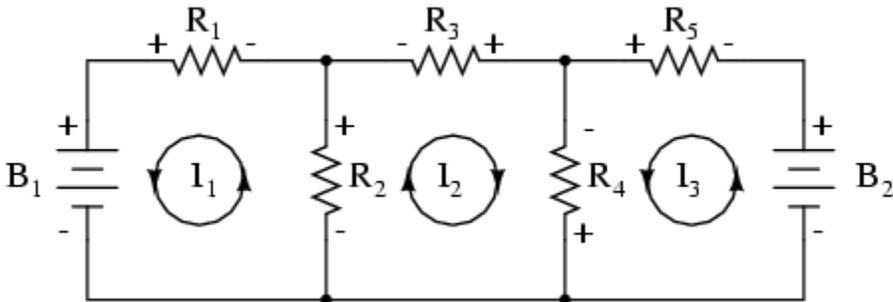
$- I_3 + I_4 - I_5 = 0$ *Kirchhoff's Current Law at node 2*

$- E_{B1} + I_2 R_2 + I_1 R_1 = 0$ *Kirchhoff's Voltage Law in left loop*

$- I_2 R_2 + I_4 R_4 + I_3 R_3 = 0$ *Kirchhoff's Voltage Law in middle loop*

$- I_4 R_4 + E_{B2} - I_5 R_5 = 0$ *Kirchhoff's Voltage Law in right loop*

I suppose if you have nothing better to do with your time than to solve for five unknown variables with five equations, you might not mind using the Branch Current method of analysis for this circuit. For those of us who *have* better things to do with our time, the Mesh Current method is a whole lot easier, requiring only three unknowns and three equations to solve:

$$- E_{B1} + R_2(I_1 + I_2) + I_1 R_1 = 0 \qquad \text{Kirchhoff's Voltage Law in left loop}$$

$$- R_2(I_2 + I_1) - R_4(I_2 + I_3) - I_2 R_3 = 0 \qquad \text{Kirchhoff's Voltage Law in middle loop}$$

$$R_4(I_3 + I_2) + E_{B2} + I_3 R_5 = 0 \qquad \text{Kirchhoff's Voltage Law in right loop}$$

Less equations to work with is a decided advantage, especially when performing simultaneous equation solution by hand (without a calculator).

Another type of circuit that lends itself well to Mesh Current is the unbalanced Wheatstone Bridge. Take this circuit, for example:
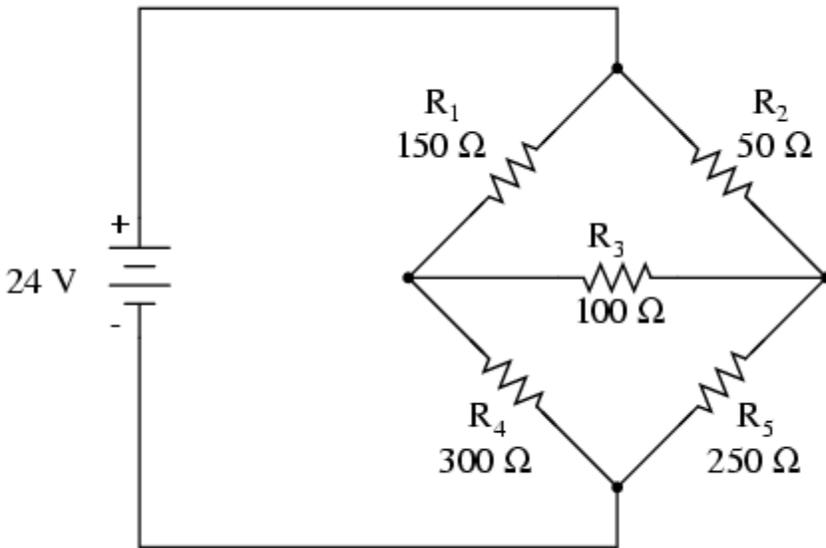
Since the ratios of R_1/R_4 and R_2/R_5 are unequal, we know that there will be voltage across resistor R_3, and some amount of current through it. As discussed at the beginning of this chapter, this type of circuit is irreducible by normal series-parallel analysis, and may only be analyzed by some other method.

We could apply the Branch Current method to this circuit, but it would require *six* currents (I_1 through I_6), leading to a very large set of simultaneous equations to solve. Using the Mesh Current method, though, we may solve for all currents and voltages with much fewer variables.

The first step in the Mesh Current method is to draw just enough mesh currents to account for all components in the circuit. Looking at our bridge circuit, it should be obvious where to place two of these currents:
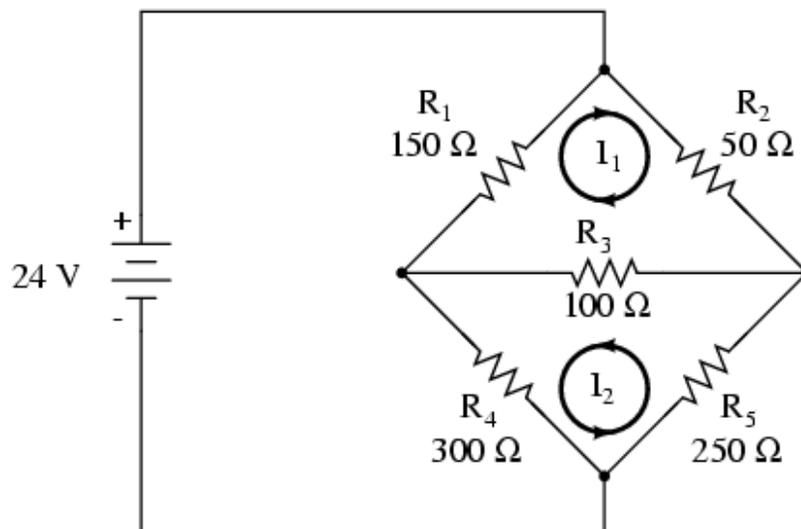
The directions of these mesh currents, of course, is arbitrary. However, two mesh currents is not enough in this circuit, because neither I_1 nor I_2 goes through the battery. So, we must add a third mesh current, I_3:
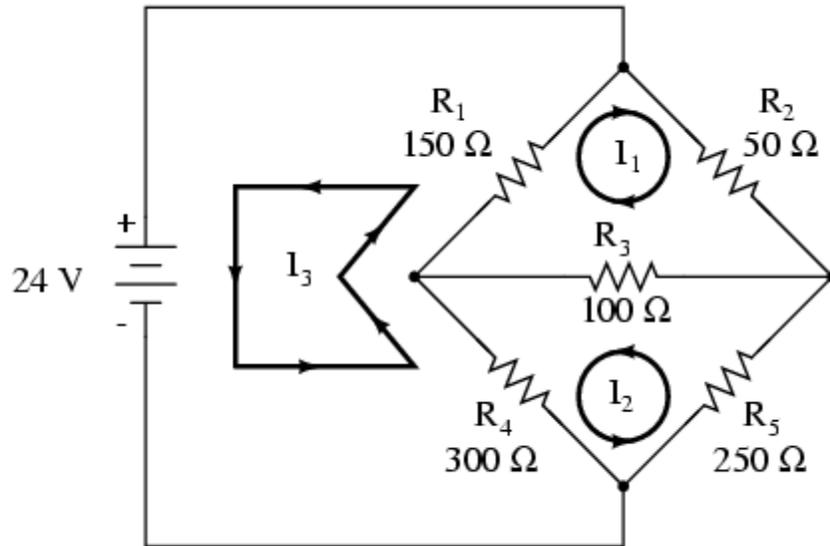
Here, I have chosen I_3 to loop from the bottom side of the battery, through R_4, through R_1, and back to the top side of the battery. This is not the only path I could have chosen for I_3, but it seems the simplest.

Now, we must label the resistor voltage drop polarities, following each of the assumed currents' directions:
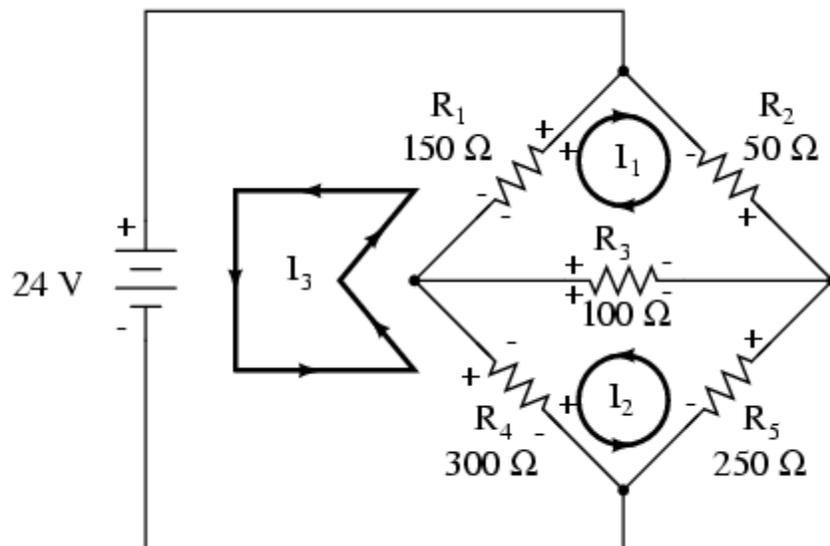
Notice something very important here: at resistor R_4, the polarities for the respective mesh currents do not agree. This is because those mesh currents (I_2 and I_3) are going through R_4 in different directions. This does not preclude the use of the Mesh Current method of analysis, but it does complicate it a bit. Though later, we will show how to avoid the R_4 current clash. (See Example below)

Generating a KVL equation for the top loop of the bridge, starting from the top node and tracing in a clockwise direction:

$$50I_1 + 100(I_1 + I_2) + 150(I_1 + I_3) = 0 \qquad \textit{Original form of equation}$$

$$\textit{. . . distributing to terms within parentheses . . .}$$

$$50I_1 + 100I_1 + 100I_2 + 150I_1 + 150I_3 = 0$$

$$\textit{. . . combining like terms . . .}$$

$$\mathbf{300I_1 + 100I_2 + 150I_3 = 0} \qquad \textit{Simplified form of equation}$$

In this equation, we represent the common directions of currents by their *sums* through common resistors. For example, resistor R_3, with a value of 100 Ω, has its voltage drop represented in the above KVL equation by the expression $100(I_1 + I_2)$, since both currents I_1 and I_2 go through R_3 from right to left. The same may be said for resistor R_1, with its voltage drop expression shown as $150(I_1 + I_3)$, since both I_1 and I_3 go from bottom to top through that resistor, and thus work *together* to generate its voltage drop.

Generating a KVL equation for the bottom loop of the bridge will not be so easy, since we have two currents going against each other through resistor R_4. Here is how I do it (starting at the right-hand node, and tracing counter-clockwise):

$$100(I_1 + I_2) + 300(I_2 - I_3) + 250I_2 = 0 \qquad \textit{Original form of equation}$$

$$\textit{. . . distributing to terms within parentheses . . .}$$

$$100I_1 + 100I_2 + 300I_2 - 300I_3 + 250I_2 = 0$$

$$\textit{. . . combining like terms . . .}$$

$$\mathbf{100I_1 + 650I_2 - 300I_3 = 0} \qquad \textit{Simplified form of equation}$$

Note how the second term in the equation's original form has resistor R_4's value of 300 Ω multiplied by the *difference* between I_2 and I_3 (I_2 - I_3). This is how we represent the combined effect of two mesh currents going in opposite directions through the same component. Choosing the appropriate mathematical signs is very important here: $300(I_2$ - $I_3)$ does not mean the same thing as $300(I_3$ - $I_2)$. I chose to write $300(I_2$ - $I_3)$ because I was thinking first of I_2's effect (creating a positive voltage drop, measuring with an imaginary voltmeter across R_4, red lead on the bottom and black lead on the top), and secondarily of I_3's effect (creating a negative voltage drop, red lead on the bottom and black lead on the top). If I had thought in terms of I_3's effect first and I_2's effect secondarily, holding my imaginary voltmeter leads in the same positions (red on bottom and black on top), the expression would have been $-300(I_3$ - $I_2)$. Note that this expression *is* mathematically equivalent to the first one: $+300(I_2$ - $I_3)$.

Well, that takes care of two equations, but I still need a third equation to complete my simultaneous equation set of three variables, three equations. This third equation must also include the battery's voltage, which up to this point does not appear in either two of the previous KVL equations. To generate this equation, I will trace a loop again with my imaginary voltmeter starting from the battery's bottom (negative) terminal, stepping clockwise (again, the direction in which I step is arbitrary, and does not need to be the same as the direction of the mesh current in that loop):

$$24 - 150(I_3 + I_1) - 300(I_3 - I_2) = 0 \qquad \textit{Original form of equation}$$

$$\textit{. . . distributing to terms within parentheses . . .}$$

$$24 - 150I_3 - 150I_1 - 300I_3 + 300I_2 = 0$$

$$\textit{. . . combining like terms . . .}$$

$$\mathbf{-150I_1 + 300I_2 - 450I_3 = -24} \qquad \textit{Simplified form of equation}$$

Solving for I_1, I_2, and I_3 using whatever simultaneous equation method we prefer:

$$300I_1 + 100I_2 + 150I_3 = 0$$
$$100I_1 + 650I_2 - 300I_3 = 0$$
$$-150I_1 + 300I_2 - 450I_3 = -24$$

$$\textit{Solutions:}$$

$$I_1 = -93.793 \text{ mA}$$
$$I_2 = 77.241 \text{ mA}$$
$$I_3 = 136.092 \text{ mA}$$

SUPERMESH Circuit Analysis | Step by Step with Solved Example

Supermesh or Supermesh Analysis is a better technique instead of using **Mesh analysis** to analysis such a complex electric circuit or network, where two meshes have a current source as a common element. This is the same where we use **Supernode circuit analysis** instead of **Node or Nodal circuit analysis** to simplify such a network where the assign supernode, fully enclosing the voltage source inside the supernode and reducing the number of none reference nodes by one (1) for each voltage source.

In supermesh circuit analysis technique, the current source is in the inner area of the supermesh. Therefore, we are able to reduce the number of meshes by one (1) for each current source which is present in the circuit.

The single mesh can be ignored, if current source (in that mesh) lies on the perimeter of the circuit. Alternatively, KVL (Kirchhoff's Voltage Law) is applied only to those meshes or supermeshes in the renewed circuit.

By the way, it is difficult to understand by Preamble, so we will first solve a simple circuit by supermesh circuit analyses, and then, we will summarize the whole supermesh analysis (step by step).

Solved Example of Supermesh Analysis

Determin V₃ by Supermesh in the circuit of fig below

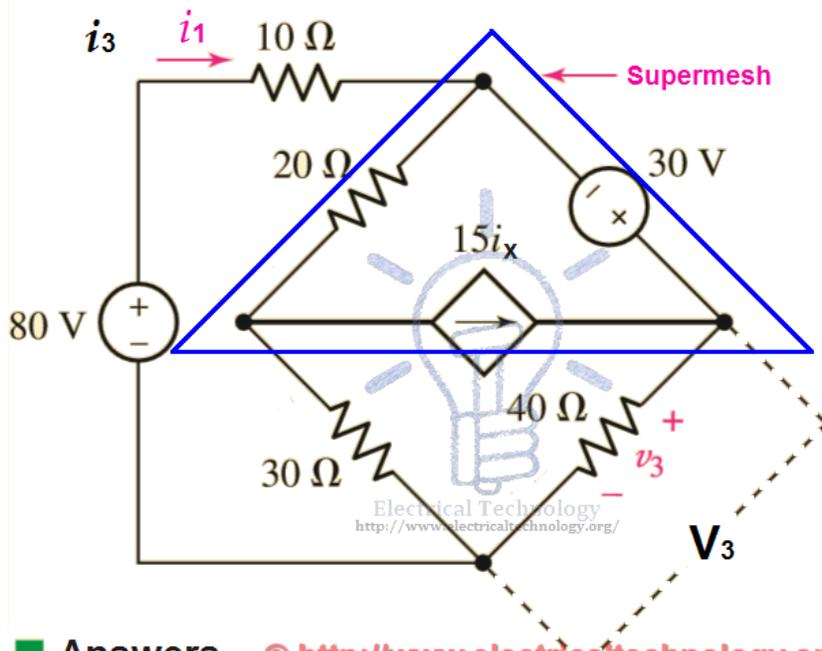■ **Answers** © http://www.electricaltechnology.org
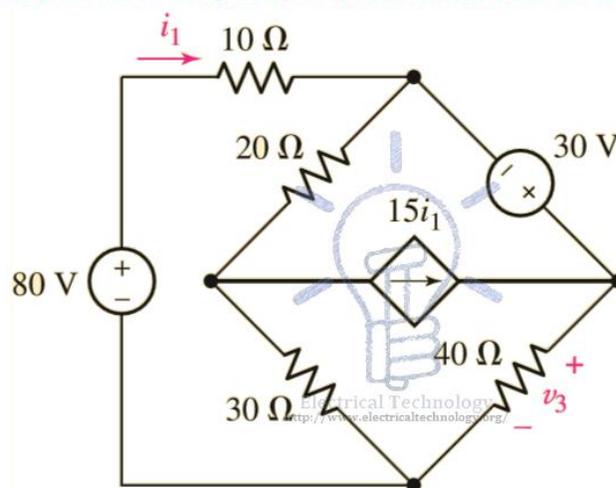
$$V3 = 104.2\ V,\ \dot{i}_1 = 0.583\ A,\ \dot{i}_2 = -6.15\ A,\ \dot{i}_3 = 2.6\ A$$

Example:
Use Mesh analysis to find V_3 and Current i_1, i_2 and i_3 in the following fig?

Solution:

Solving by Supermesh Analysis

Answers © http://www.electricaltechnology.org

V3 = 104.2 V, i_1 = 0.583 A, i_2 = -6.15 A, i_3 = 2.6 A

Supermesh Circuit Analysis. Step by step with solved example

UsingKVA on Mesh 1.

$80 = 10i_1 + 20(i_1-i_2) + 30 (i_1-i_3)$

Simplifying

$80 = 10i_1 + 20i_1 -20i_2 + 30i_1-30i_3$

$80 = 60i_1 - 20i_2 - 30i_3$ ..... $\rightarrow$ Eq 1.

Now apply **KVL** on **Supermesh** (which is integration of *mesh 2* and *mesh 3*, but we have reduced it by single mesh which is known as *supermesh*)

$30 = 40i_3 + 30(i_3-i_1) +20(i_2-i_1)$

$30 = 40i_3 + 30i_3 - 30i_1 +20i_2-20i_1$

$30 = 70i_3 - 50i_1 +20i_2$ ..... $\rightarrow$ Eq 2.

But here, we have three (3) variables i.e. i1, 12 and i3. And there are two equations. So we must need three equations as well.

The independent current source (in the **supermesh**) is related to the assumed mesh currents, i.e.

15ix $= i_3-i_2$

I3=15*ix*+i_2 ..... $\rightarrow$ Eq 3.

Solving equations 1, 2 and 3 by **Cramer's rule or Cramer'r rule calculator**, **Elimination**, **Gauss Elimination** or **computer aided program** such as **MATLAB**, we find

i_1 =0.583A
i_2 =-6.15A
i_3 = 2.6 A

Also, we can find the value of $\mathbf{v_3}$,

$$\mathbf{V_3 = i_3 \ x \ R_3}$$

Putting the values,

V_3 =2.6Ax40Ω
V_3 = 104 V.
Summary of Supermesh Analysis (Step by Step)
1. **Evaluate if the circuit is a planer circuit**. if yes, apply Supermesh. If no, perform nodal analysis instead.
2. **Redraw the circuit** if necessary and count the number of meshes in the circuit.
3. **Label each of mesh currents in the circuit**. As a rule of thumb, defining all the mesh currents to flow clockwise result in a simpler circuit analysis.
4. **Form a supermesh if the circuit contains current sources by two meshes**. So that, the supermesh would enclose both meshes.
5. **Write a KVL (Kirchoff's Voltage Law) around each mesh and supermesh in the circuit**. Begin with an easy and will fitted one node. Now proceed in the direction of the mesh current. Take the "-" sign in the account while writing KVL equations and solving the circuit. No KVL equation is needed if a current source lies on the periphery of a mesh. So, the mesh current is determined and evaluated by inspection.
6. **One KCL (Kirchhoff's Current Law) is needed for each supermesh defined and can be accomplished by simple application of KCL.** in simple words, relate the current flowing from each current source to mesh currents.
7. **An additional case can be occurred if the circuit contains on further dependent sources.** In this case, express any additional unknown values and qantitis like currents ir voltages other than the mesh currents in terms of suitable mesh currents.
8. **Arrange and organize the system of equations.**
9. **At last, solve the system of equations for the Nodal voltages** such as V1, V2, and V3 etc. there will be Mesh of them. if you find difficulties to solve the system of equations, refer to the above example.

Nodal Voltage Analysis

Nodal Voltage Analysis finds the unknown voltage drops around a circuit between different nodes that provide a common connection for two or more circuit components

Nodal Voltage Analysis complements the previous mesh analysis in that it is equally powerful and based on the same concepts of matrix analysis. As its name implies, **Nodal Voltage Analysis** uses the "Nodal" equations of Kirchhoff's first law to find the voltage potentials around the circuit.

So by adding together all these nodal voltages the net result will be equal to zero. Then, if there are "n" nodes in the circuit there will be "n-1" independent nodal equations and these alone are sufficient to describe and hence solve the circuit.

At each node point write down Kirchhoff's first law equation, that is: "*the currents entering a node are exactly equal in value to the currents leaving the node*" then express each current in terms of the voltage across the branch. For "n" nodes, one node will be used as the reference node and all the other voltages will be referenced or measured with respect to this common node.

For example, consider the circuit from the previous section.

Nodal Voltage Analysis Circuit
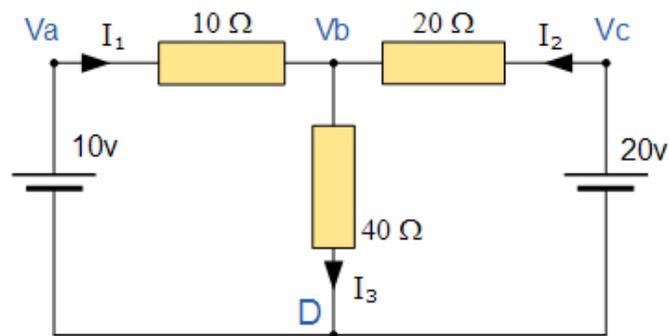
In the above circuit, node D is chosen as the reference node and the other three nodes are assumed to have voltages, Va, Vb and Vc with respect to node D. For example;

$$\frac{(V_a - V_b)}{10} + \frac{(V_c - V_b)}{20} = \frac{V_b}{40}$$

As Va = 10v and Vc = 20v , Vb can be easily found by:

$$\left(1 - \frac{Vb}{10}\right) + \left(1 - \frac{Vb}{20}\right) = \frac{Vb}{40}$$

$$2 = Vb\left(\frac{1}{40} + \frac{1}{20} + \frac{1}{10}\right)$$

$$Vb = \frac{80}{7} V$$

$$\therefore I_3 = \frac{2}{7} \text{ or } 0.286 Amps$$

again is the same value of 0.286 amps, we found using Kirchhoff's Circuit Law in the previous tutorial.

From both Mesh and Nodal Analysis methods we have looked at so far, this is the simplest method of solving this particular circuit. Generally, nodal voltage analysis is more appropriate when there are a larger number of current sources around. The network is then defined as: [I] = [Y] [V] where [I] are the driving current sources, [V] are the nodal voltages to be found and [Y] is the admittance matrix of the network which operates on [V] to give [I].

Nodal Voltage Analysis Summary.

The basic procedure for solving **Nodal** Analysis equations is as follows:

1. Write down the current vectors, assuming currents into a node are positive. ie, a (N x 1) matrices for "N" independent nodes.
2. Write the admittance matrix [Y] of the network where:
    i. Y_{11} = the total admittance of the first node.
    ii. Y_{22} = the total admittance of the second node.
    iii. R_{JK} = the total admittance joining node J to node K.
3. For a network with "N" independent nodes, [Y] will be an (N x N) matrix and that Ynnwill be positive and Yjk will be negative or zero value.
4. The voltage vector will be (N x L) and will list the "N" voltages to be found.

We have now seen that a number of theorems exist that simplify the analysis of linear circuits. In the next tutorial we will look at Thevenins Theorem which allows a network consisting of linear resistors and sources to be represented by an equivalent circuit with a single voltage source and a series resistance.

Nodal analysis exercise problems super nodal analysis:

The node voltage method of analysis solves for unknown voltages at circuit nodes in terms of a system of KCL equations. This analysis looks strange because it involves replacing voltage sources with equivalent current sources. Also, resistor values in ohms are replaced by equivalent conductances in siemens, $G = 1/R$. The siemens (S) is the unit of conductance, having replaced the mho unit. In any event $S = \Omega^{-1}$. And S = mho (obsolete).

We start with a circuit having conventional voltage sources. A common node E_0 is chosen as a reference point. The node voltages E_1 and E_2 are calculated with respect to this point.
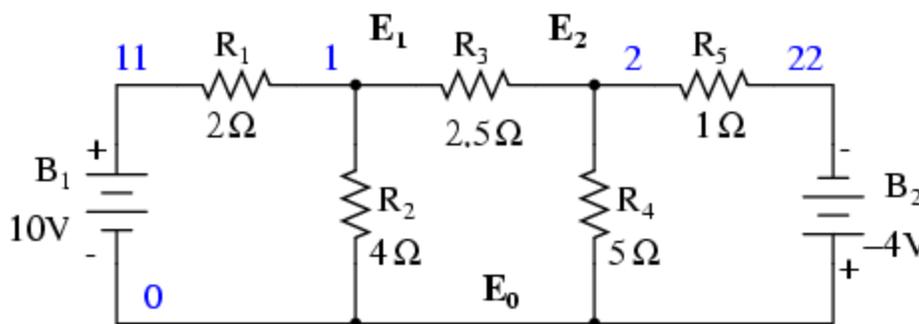
A voltage source in series with a resistance must be replaced by an equivalent current source in parallel with the resistance. We will write KCL equations for each node. The right hand side of the equation is the value of the current source feeding the node.

$$1_1 = B_1/R_1 = 10/2 = 5\ A$$

(a)   (b)
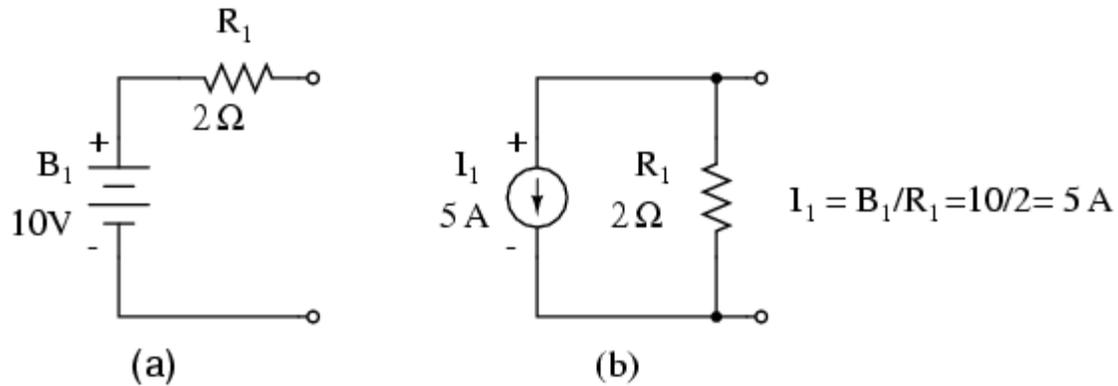
Replacing voltage sources and associated series resistors with equivalent current sources and parallel resistors yields the modified circuit. Substitute resistor conductances in siemens for resistance in ohms.
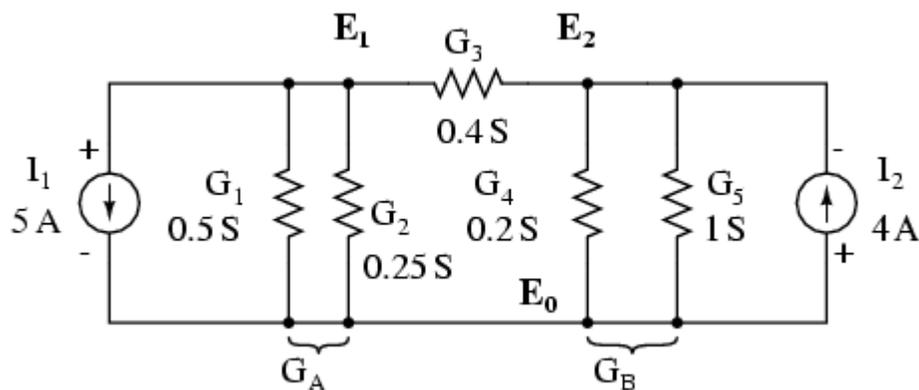
$I1 = E1/R1 = 10/2 = 5\ A$

$I2 = E2/R5 = 4/1\ = 4\ A$

$G1 = 1/R1 = 1/2\ \Omega\ = 0.5\ S$

$G2 = 1/R2 = 1/4\ \Omega\ = 0.25\ S$

$G3 = 1/R3 = 1/2.5\ \Omega = 0.4\ S$

$G4 = 1/R4 = 1/5\ \Omega\ = 0.2\ S$

$G5 = 1/R5 = 1/1\ \Omega\ = 1.0\ S$

The parallel conductances (resistors) may be combined by addition of the conductances. Though, we will not redraw the circuit. The circuit is ready for application of the node voltage method.

$$GA = G1 + G2 = 0.5\ S + 0.25\ S = 0.75\ S$$

$$GB = G4 + G5 = 0.2\ S + 1\ S = 1.2\ S$$

Deriving a general node voltage method, we write a pair of KCL equations in terms of unknown node voltages V_1 and V_2 this one time. We do this to illustrate a pattern for writing equations by inspection.

$$GAE1 + G3(E1 - E2) = I1\ (1)$$

$$GBE2 - G3(E1 - E2) = I2\ (2)$$

$$(GA + G3\ )E1 - G3E2 = I1\ (1)$$

$$-G3E1 + (GB + G3)E2 = I2\ (2)$$

The coefficients of the last pair of equations above have been rearranged to show a pattern. The sum of conductances connected to the first node is the positive coefficient of the first voltage in equation (1). The sum of conductances connected to the second node is the positive coefficient of the second voltage in equation (2). The other coefficients are negative, representing conductances between nodes. For both equations, the right hand side is equal to the respective current source connected to the node. This pattern allows us to quickly write the equations by inspection. This leads to a set of rules for the node voltage method of analysis.

Node voltage rules:

1. Convert voltage sources in series with a resistor to an equivalent current source with the resistor in parallel.
2. Change resistor values to conductances.
3. Select a reference node(E_0)
4. Assign unknown voltages $(E_1)(E_2)$ ... $(E_N)$to remaining nodes.
5. Write a KCL equation for each node 1,2, ... N. The positive coefficient of the first voltage in the first equation is the sum of conductances connected to the node. The coefficient for the second voltage in the second equation is the sum of conductances connected to that node. Repeat for coefficient of third voltage, third equation, and other equations. These coefficients fall on a diagonal.
6. All other coefficients for all equations are negative, representing conductances between nodes. The first equation, second coefficient is the conductance from node 1 to node 2, the third coefficient is the conductance from node 1 to node 3. Fill in negative coefficients for other equations.
7. The right hand side of the equations is the current source connected to the respective nodes.
8. Solve system of equations for unknown node voltages.

SUPERNODE Circuit Analysis | Step by Step with Solved Example

Consider both circuits in the following fig 1. did you notice something different?

Fig 1
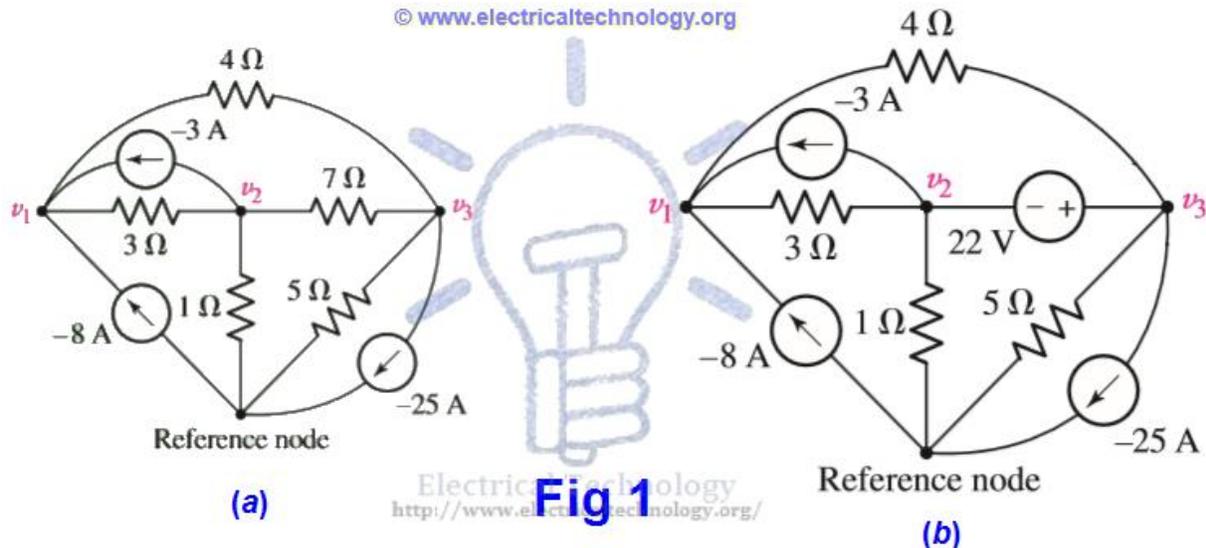
(a) Best to solve by Node or Nodal Analysis

(b) Best to solve with Supernode Analysis

Difference between Node / Nodal & Supernode Analysis

The difference in both circuits is that there is an additional voltage source of 22V instead of 7Ω resistor between node 2 and node 3. And this is the main point.

In Node or Nodal analysis, we apply the KCL (Kirchhoff's Current Law) at each non-reference nodes i.e. we apply the simple KCL at once on three nodes in fig 1(a).

If we do the same i.e. apply the Nodal analysis instead on Supernode circuit analysis on the circuit in fig 1 (b), we face some difficulty at Node1 and Node2, because we don't know that what is the current in the branch with the voltage source? In addition, there is no such a way by which we adjust the situation i.e. we can't express the current as a function of the voltage, where the definition of the voltage source is that the voltage is independent of the current. Due to these difficulties and troubles, we use supernode circuit analysis instead of Nodal analysis in the above fig 1 (b).

There are two methods to simplify the circuit in the above fig 1 (b).

The 1st one, which is more complex, is that to assign an unknown current value to the branch contains the voltage source. Then apply KCL three times on the 3 Nodes (one KCL equation for each node). At last, apply KVL (Kirchhoff's Voltage Law) which is v3–v2 = 22V between Node2 and Node3. In this case, we get four (4) equations for unknown values in the above example, which is little bit complex to simplify.

The 2nd method is easier than the above method which is called Supernode analysis. In this method, we treat Node2, Node3 and the voltage source of 22V together as a sort of Supernode and apply KCL to both nodes (Nod2 and Node3) at once. The supernode is indicated by the region enclosed by the dotted line. This is possible because, if the total current leaving Node2 is zero (0) and the total current leaving Node3

98

is zero (0), the the total current leaving the combination is zero. This concept is shown in the following fig 2 (b) with the supernode (the area enclosed by the broken line).

© www.electricaltechnology.org
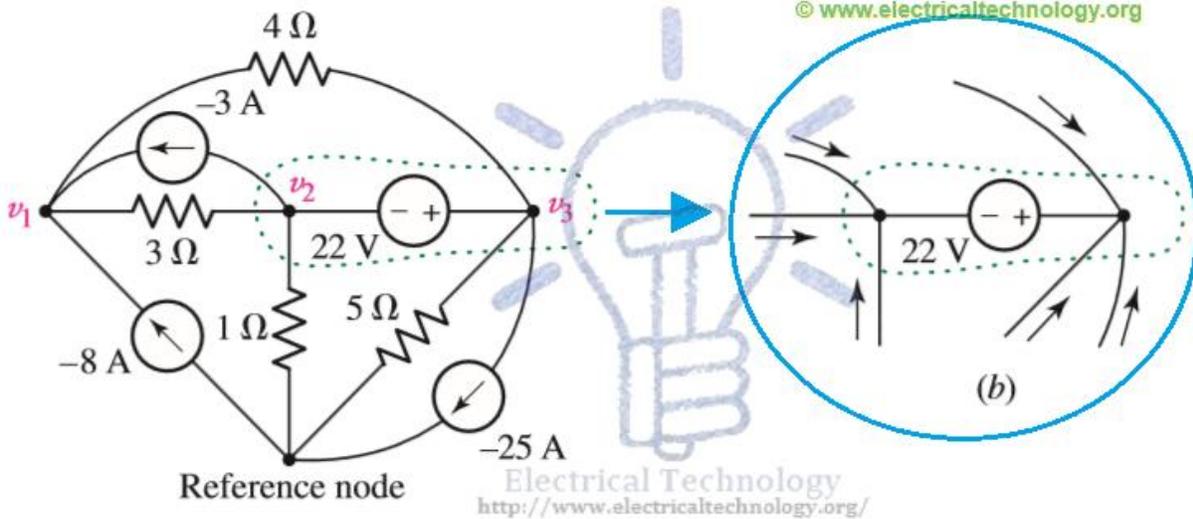
Expanded View of the Region defined as a Supernode

Fig 2 KCL requires that all currents flowing into the region sum to Zero

Expanded view of the region defined as Supernode

Now, we will solve the circuit below by step by step supernode circuit analysis and then, we will summarize the whole supernode analysis (step by step).

Solved Example of Supernode Analysis

Example:
Use Supernode analysis to find voltage across each current source i.e. v1 & v2 in the following fig 3 (a)?

A four (4) Nodes Circuit **Fig 3** Redrawn Circuit with Reference Node Chosen and Voltage labeled

SUPERNODE Circuit Analysis | Step by Step with Solved Example

SUPERNODE Circuit Analysis Step by Step with Solved Example

Solution:
First, we redraw the circuit as shown in fig 3(b)

We begin by writing a KCL equation for Node1.

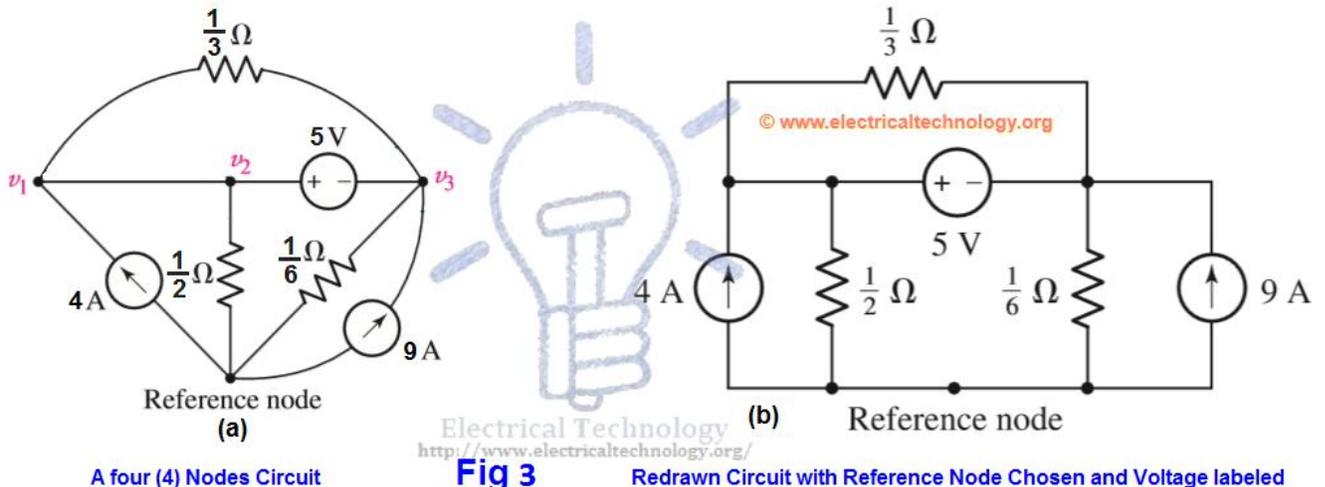$$4 = \frac{3v_1}{0} + \frac{3v_3}{1/3}$$

$4 = 0 + 3v1 + 3v3$ …. → Eq 1.

Now, consider the supernode (Combination of Node1 and Node2). Moreover, one current source and three resistors are connected. Thus,

Apply KCL at Supernode (Node1 & Node2)

$$9 = \frac{v_2}{1/2} - \frac{v_3}{1/6} + \frac{v_3 - v_1}{1/3} + \frac{v_2 - v_1}{0}$$

$9 = 2v2 + 6v3 + 3v3 - 3v1 + 0.$
$9 = -3v1 + 2v2 + 9v3$ …. → Eq 2.

Since we have three unknown values, therefore, we need one additional equation. Obliviously, we will go for the 5V voltage source between Nodes 2 and 3, which is;

$v2 - v3 = 5$ …. → Eq 3.

Solving equations 1, 2 and 3 by Cramer's rule or Cramer'r rule calculator, Elimination, Gauss Elimination or computer aided program such as MATLAB, we find,

$v3 = 0.575$ V or 375mV.
$v2 = 5.375$ V.
$v1 = 1.708$ V.

Summary of Supernode Analysis (Step by Step)

1. Redraw the circuit if possible.
2. Count the Number of Nodes in the circuit.
3. Design a Reference Node. This may be the node with the greatest number of branches. So that, we may minimize the number of of equations.
4. Label the Nodal Voltages. Which are (N-1), where N= number of Nodes.
5. Form a Supernode if the circuit or network contains voltage sources. This job is done by enclosing the source terminal, and other circuit element connected between the two terminals with the dotted line enclosure. This is shown in the above fig 2 (b).

6. Write a KCL (Kirchhoff's Current law) equation for each non-reference node as well as for each supernode which does not contain the reference node. On the first side, add the currents flowing into a supernode or node from the current sources. On the other hand, add the currents leaving the supernode or node through resistors. Take the "-" sign in the account while writing KCL equations and solving the circuit.
7. One KCL (Kirchhoff's Current Law) is needed for each defined Supernode which can be accomplished by simple application of KCL. In simple words, relate the voltage across each voltage source to nodal voltages.
8. If dependent sources appear in the circuit, In this case, express any additional unknown values and quantities like currents or voltages other than the Nodal voltages in terms of suitable nodal voltages.
9. Arrange and organize the system of equations.
10. At last, solve the system of equations for the Nodal voltages such as V1, V2, and V3 etc. there will be (N-1, where "N" = Number of Nodes) of them. If you find difficulties to solve the system of equations, refer to the above solved example.

Mesh analysis and nodal analysis by inspection , exercise problems

Mesh analysis by inspectio

We take a second look at the "mesh current method" with all the currents running counterclockwise (ccw). The motivation is to simplify the writing of mesh equations by ignoring the resistor voltage drop polarity. Though, we must pay attention to the polarity of voltage sources with respect to assumed current direction. The sign of the resistor voltage drops will follow a fixed pattern.

If we write a set of conventional mesh current equations for the circuit below, where we do pay attention to the signs of the voltage drop across the resistors, we may rearrange the coefficients into a fixed pattern:
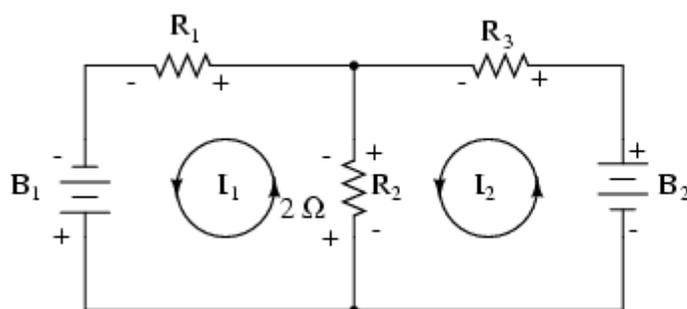
Mesh equations

$$(I_1 - I_2)R_2 + I_1R_1 - B_1 = 0$$
$$I_2R_3 - (I_1 - I_2)R_2 - B_2 = 0$$

Simplified

$$(R_1 + R_2)I_1 \qquad - R_2I_2 = B_1$$
$$-R_2I_1 + (R_2 + R_3)I_2 = B_2$$

Once rearranged, we may write equations by inspection. The signs of the coefficients follow a fixed pattern in the pair above, or the set of three in the rules below.

- **Mesh current rules:**

- This method assumes electron flow (not conventional current flow) voltage sources. Replace any current source in parallel with a resistor with an equivalent voltage source in series with an equivalent resistance.
- Ignoring current direction or voltage polarity on resistors, draw counterclockwise current loops traversing all components. Avoid nested loops.
- Write voltage-law equations in terms of unknown currents currents: I_1, I_2, and I_3. Equation 1 coefficient 1, equation 2, coefficient 2, and equation 3 coefficient 3 are the positive sums of resistors around the respective loops.
- All other coefficients are negative, representative of the resistance common to a pair of loops. Equation 1 coefficient 2 is the resistor common to loops 1 and 2, coefficient 3 the resistor common to loops 1 an 3. Repeat for other equations and coefficients.

> +(sum of R's loop 1)I1 - (common R loop 1-2)I2 - (common R loop 1-3)I3 = E1
> -(common R loop 1-2)I1 + (sum of R's loop 2)I2 - (common R loop 2-3)I3 = E2
> -(common R loop 1-3)I1 - (common R loop 2-3)I2 + (sum of R's loop 3)I3 = E3

- The right hand side of the equations is equal to any electron current flow voltage source. A voltage rise with respect to the counterclockwise assumed current is positive, and 0 for no voltage source.
- Solve equations for mesh currents:I_1, I_2, and I_3 . Solve for currents through individual resistors with KCL. Solve for voltages with Ohms Law and KVL.

While the above rules are specific for a three mesh circuit, the rules may be extended to smaller or larger meshes. The figure below illustrates the application of the rules. The three currents are all drawn in the same direction, counterclockwise. One KVL equation is written for each of the three loops. Note that there is no polarity drawn on the resistors. We do not need it to determine the signs of the coefficients. Though we do need to pay attention to the polarity of the voltage source with respect to current direction. The I_3counterclockwise current traverses the 24V source from (+) to (-). This is a voltage rise for electron current flow. Therefore, the third equation right hand side is +24V.
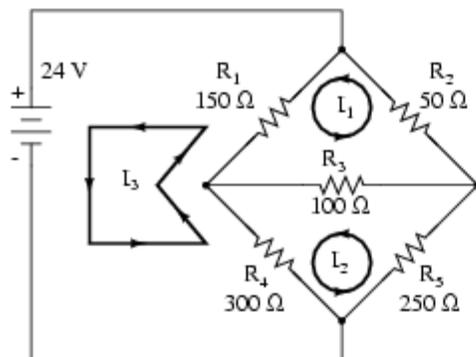
$$+(R_1+R_2+R_3)I_1 \qquad -(R_3)I_2 \qquad -(R_1)I_3 = 0$$
$$-(R_3)I_1 +(R_3+R_4+R_5)I_2 \qquad -(R_4)I_3 = 0$$
$$-(R_1)I_1 \qquad -(R_4)I_2 +(R_1+R_4)I_3 =24$$

$$+(150+50+100)I_1 \qquad - (100)I_2 \qquad - (150)I_3 = 0$$
$$-(100)I_1 +(100+300+250)I_2 \qquad - (300)I_3 = 0$$
$$-(150)I_1 \qquad - (300)I_2 +(150+300)I_3 =24$$

$$+(300)I_1 -(100)I_2 -(150)I_3 = 0$$
$$-(100)I_1 + (650)I_2 -(300)I_3 = 0$$
$$-(150)I_1 -(300)I_2 +(450)I_3 =24$$

In Octave, enter the coefficients into the A matrix with column elements comma separated, and rows

semicolon separated. Enter the voltages into the column vector b. Solve for the unknown currents: I_1, I_2, and I_3 with the command: x=A\b. These currents are contained within the x column vector. The positive values indicate that the three mesh currents all flow in the assumed counterclockwise direction.

```
octave:2> A=[300,-100,-150;-100,650,-300;-150,-300,450]
    A =
      300 -100 -150
      -100 650 -300
      -150 -300 450

    octave:3> b=[0;0;24]
    b =
      0
      0
      24

    octave:4> x=A\b
    x =
      0.093793
      0.077241
      0.136092
```

The mesh currents match the previous solution by a different mesh current method.. The calculation of resistor voltages and currents will be identical to the previous solution. No need to repeat here.

Note that electrical engineering texts are based on conventional current flow. The loop-current, mesh-current method in those text will run the assumed mesh currents **clockwise**.[aef] The conventional current flows out the (+) terminal of the battery through the circuit, returning to the (-) terminal. A conventional current voltage rise corresponds to tracing the assumed current from (-) to (+) through any voltage sources.

One more example of a previous circuit follows. The resistance around loop 1 is 6 Ω, around loop 2: 3 Ω. The resistance common to both loops is 2 Ω. Note the coefficients of I_1 and I_2 in the pair of equations. Tracing the assumed counterclockwise loop 1 current through B_1 from (+) to (-) corresponds to an electron current flow voltage rise. Thus, the sign of the 28 V is positive. The loop 2 counter clockwise assumed current traces (-) to (+) through B_2, a voltage drop. Thus, the sign of B_2 is negative, -7 in the 2nd mesh equation. Once again, there are no polarity markings on the resistors. Nor do they figure into the equations.
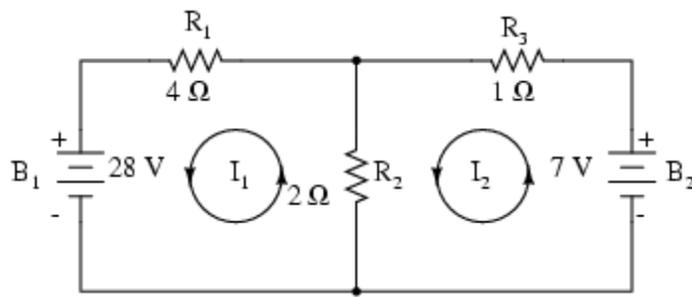
$$6I_1 - 2I_2 = 28$$
$$-2I_1 + 3I_2 = -7$$
Mesh equations

$6I_1 - 2I_2 = 28$	$6I_1 - 2I_2 = 28$
$-6I_1 + 9I_2 = -21$	$6I_1 - 2(1) = 28$
$7I_2 = 7$	$6I_1 = 30$
$I_2 = 1$	$I_1 = 5$

The currents $I_1 = 5$ A, and $I_2 = 1$ A are both positive. They both flow in the direction of the counterclockwise loops. This compares with previous results.
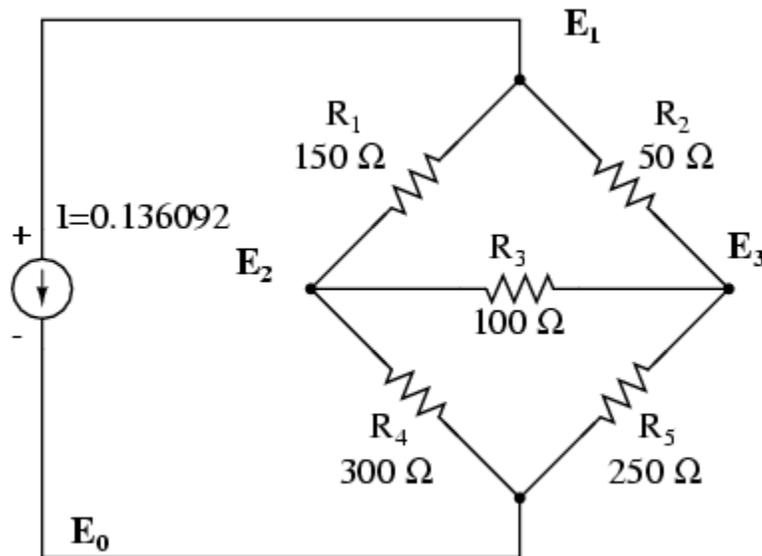
There are three nodes to write equations for by inspection. Note that the coefficients are positive for equation (1) E_1, equation (2) E_2, and equation (3) E_3. These are the sums of all conductances connected to the nodes. All other coefficients are negative, representing a conductance between nodes. The right hand side of the equations is the associated current source, 0.136092 A for the only current source at node 1. The other equations are zero on the right hand side for lack of current sources. We are too lazy to calculate the conductances for the resistors on the diagram. Thus, the subscripted G's are the coefficients.

(G1 + G2)E1 -G1E2 -G2E3 = 0.136092
-G1E1 +(G1 + G3 + G4)E2 -G3E3 = 0
-G2E1 -G3E2 +(G2 + G3 + G5)E3 = 0

104

We are so lazy that we enter reciprocal resistances and sums of reciprocal resistances into the octave "A" matrix, letting octave compute the matrix of conductances after "A=".[octav] The initial entry line was so long that it was split into three rows. This is different than previous examples. The entered "A" matrix is delineated by starting and ending square brackets. Column elements are space separated. Rows are "new line" separated. Commas and semicolons are not need as separators. Though, the current vector at "b" is semicolon separated to yield a column vector of currents.

```
octave:12> A = [1/150+1/50 -1/150 -1/50
    > -1/150 1/150+1/100+1/300 -1/100
    > -1/50 -1/100 1/50+1/100+1/250]
    A =
      0.0266667  -0.0066667  -0.0200000
     -0.0066667   0.0200000  -0.0100000
     -0.0200000  -0.0100000   0.0340000

    octave:13> b = [0.136092;0;0]
    b =
      0.13609
      0.00000
      0.00000

    octave:14> x=A\b
    x =
      24.000
      17.655
      19.310
```

Note that the "A" matrix diagonal coefficients are positive, That all other coefficients are negative.
The solution as a voltage vector is at "x". E_1 = 24.000 V, E_2 = 17.655 V, E_3 = 19.310 V. These three voltages compare to the previous mesh current and SPICE solutions to the unbalanced bridge problem. This is no coincidence, for the 0.13609 A current source was purposely chosen to yield the 24 V used as a voltage source in that problem

5.4 THEVENIN'S THEOREM:

DC: " An complex network consisting of number voltage and current sources cand be replaced by simple series circuit consisting of equivalent voltage source in series with equivalent resistance, where equivalen voltage is called as open circuit voltage and equivalent resistance is called as thevenin's resistance calculated across open circuit terminals while all energy sources are non-operative"
AC: " An complex network consisting of number voltage and current sources cand be replaced by simple series circuit consisting of equivalent voltage source in series with equivalent impedance, where equivalen voltage is called as open circuit voltage and equivalent impedance is called as thevenin's impedance calculated across open circuit terminals while all energy sources are non-operative"

Eg:
Here w   n.

Open c   Fig 1   Fig 2   Fig 3
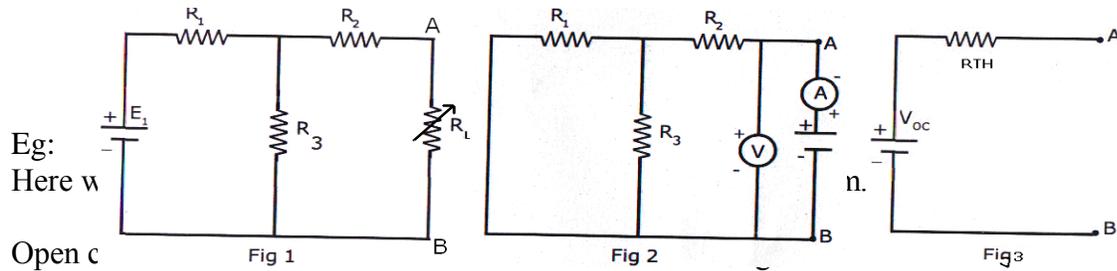
Thevenin's voltage , Vth = E1. R3 / (R1+R3) ----1 from figure .1

Thevenin's resistance, Rth = (R1.R3)/ (R1+R3) + R2 ----2 from figure 2.

Now draw the thevenin's equivalent circuit as shown in figure 3 with calculated values.

Eg: **With AC excitation**

Here we need to find current through ZL using thevenin's theorem.

Open circuit the AB terminals to find the Thevenin's voltage.

Thevenin's voltage , Vth = E1. R3 / (R1+R3) ----1 from figure .1

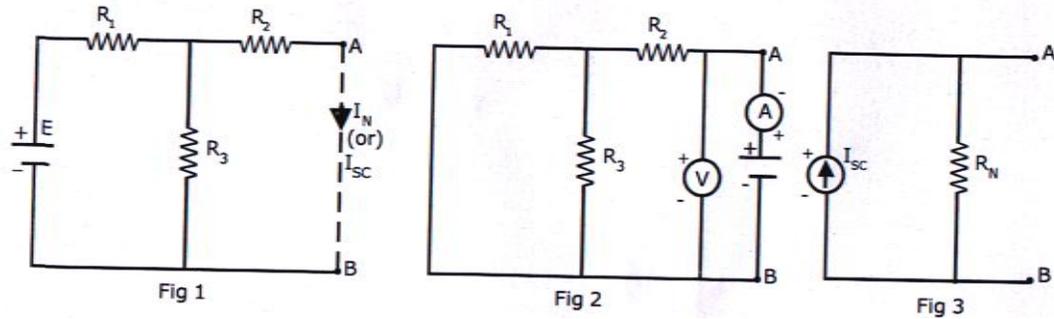Thevenin's impedance, Zth = (Z1. Z3)/ (Z1+ Z3) + Z2 ----2 from figure 2.

Now draw the thevenin's equivalent circuit as shown in figure 3 with calculated values.

5.5 NORTON'S THEOREM:

DC: " An complex network consisting of number voltage and current sources cand be replaced by simple parallel circuit consisting of equivalent current source in parallel with equivalent resistance, where equivalent current source is called as short circuit current and equivalent resistance is called as norton's resistance calculated across open circuit terminals while all energy sources are non-operative"

AC: "An complex network consisting of number voltage and current sources cand be replaced by simple parallel circuit consisting of equivalent current source in parallel with equivalent impedance, where equivalent current source is called as short circuit current and equivalent impedance is called as norton's impedance calculated across open circuit terminals while all energy sources are non-operative"

Fig 1   Fig 2   Fig 3

Here we need to find current through RL using norton's theorem.

Short circuit the AB terminals to find the norton's current.

Total resistance of circuit is, Rt = (R2.R3) / (R2+R3) + R1

Source current, I = E / Rt

Norton's current , IN = I. R3 / (R2+R3) ----1 from figure .1

Norton's resistance, RN = (R1.R3)/ (R1+R3) + R2 ----2 from figure 2.

Now draw the Norton's equivalent circuit as shown in figure 3 with calculated values.

*** These two theorems are useful in determining the load value for which maximum power transfer can be happened.

MODULE – III
INTRODUCTION TO AC CIRCUITS

. **Analysis of AC circuits:**

Direct Current or **D.C.** as it is more commonly called, is a form of electrical current or voltage that flows around an electrical circuit in one direction only, making it a "Uni-directional" supply.

Generally, both DC currents and voltages are produced by power supplies, batteries, dynamos and solar cells to name a few. A DC voltage or current has a fixed magnitude (amplitude) and a definite direction associated with it. For example, +12V represents 12 volts in the positive direction, or -5V represents 5 volts in the negative direction.

We also know that DC power supplies do not change their value with regards to time, they are a constant value flowing in a continuous steady state direction. In other words, DC maintains the same value for all times and a constant uni-directional DC supply never changes or becomes negative unless its connections are physically reversed. An example of a simple DC or direct current circuit is shown below.
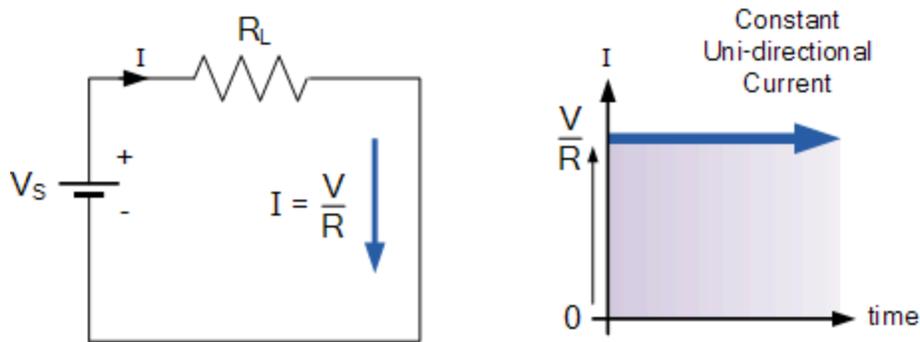
DC Circuit and Waveform

Figure DC circuit

An alternating function or **AC Waveform** on the other hand is defined as one that varies in both magnitude and direction in more or less an even manner with respect to time making it a "Bi-directional" waveform. An AC function can represent either a power source or a signal source with the shape of an *AC waveform* generally following that of a mathematical sinusoid being defined as: $A(t) = A_{max}*\sin(2\pi ft)$.

The term AC or to give it its full description of Alternating Current, generally refers to a time-varying waveform with the most common of all being called a **Sinusoid** better known as a **Sinusoidal Waveform**. Sinusoidal waveforms are more generally called by their short description as **Sine Waves**. Sine waves are by far one of the most important types of AC waveform used in electrical engineering.

The shape obtained by plotting the instantaneous ordinate values of either voltage or current against time is called an **AC Waveform**. An AC waveform is constantly changing its polarity every half cycle alternating between a positive maximum value and a negative maximum value respectively with regards to time with a common example of this being the domestic mains voltage supply we use in our homes.

This means then that the *AC Waveform* is a "time-dependent signal" with the most common type of time-dependant signal being that of the **Periodic Waveform**. The periodic or AC waveform is the resulting product of a rotating electrical generator. Generally, the shape of any periodic waveform can be generated using a fundamental frequency and superimposing it with harmonic signals of varying frequencies and amplitudes but that's for another tutorial.

Alternating voltages and currents cannot be stored in batteries or cells like direct current (DC) can, it is much easier and cheaper to generate these quantities using alternators or waveform generators when they are needed. The type and shape of an AC waveform depends upon the generator or device producing them, but all AC waveforms consist of a zero voltage line that divides the waveform into two symmetrical halves. The main characteristics of an **AC Waveform** are defined as:

AC Waveform Characteristics

- The Period, (T) is the length of time in seconds that the waveform takes to repeat itself from start to finish. This can also be called the *Periodic Time* of the waveform for sine waves, or the *Pulse Width* for square waves.

- The Frequency, (f) is the number of times the waveform repeats itself within a one second time period. Frequency is the reciprocal of the time period, ($f = 1/T$) with the unit of frequency being the *Hertz*, (Hz).

- The Amplitude (A) is the magnitude or intensity of the signal waveform measured in volts or amps.

In our tutorial about waveforms,we looked at different types of waveforms and said that "**Waveforms** are basically a visual representation of the variation of a voltage or current plotted to a base of time". Generally, for AC waveforms this horizontal base line represents a zero condition of either voltage or current. Any part of an AC type waveform which lies above the horizontal zero axis represents a voltage or current flowing in one direction.

Likewise, any part of the waveform which lies below the horizontal zero axis represents a voltage or current flowing in the opposite direction to the first. Generally for sinusoidal AC waveforms the shape of the waveform above the zero axis is the same as the shape below it. However, for most non-power AC signals including audio waveforms this is not always the case.

The most common periodic signal waveforms that are used in Electrical and Electronic Engineering are the *Sinusoidal Waveforms*. However, an alternating AC waveform may not always take the shape of a smooth shape based around the trigonometric sine or cosine function. AC waveforms can also take the shape of either *Complex Waves*, *Square Waves* or *Triangular Waves* and these are shown below.
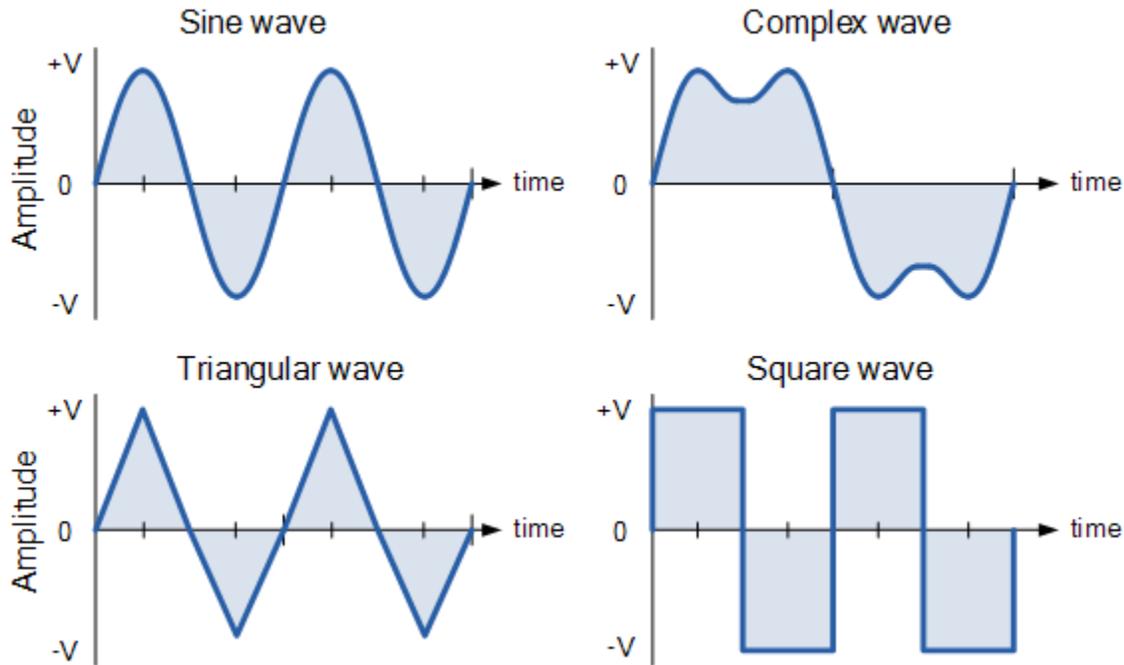
Types of Periodic Waveform

Figure AC waveforms

The time taken for an **AC Waveform** to complete one full pattern from its positive half to its negative half and back to its zero baseline again is called a **Cycle** and one complete cycle contains both a positive half-cycle and a negative half-cycle. The time taken by the waveform to complete one full cycle is called the **Periodic Time** of the waveform, and is given the symbol "T".

The number of complete cycles that are produced within one second (cycles/second) is called the **Frequency**, symbol f of the alternating waveform. Frequency is measured in **Hertz**, (Hz) named after the German physicist Heinrich Hertz.

Then we can see that a relationship exists between cycles (oscillations), periodic time and frequency (cycles per second), so if there are f number of cycles in one second, each individual cycle must take $1/f$ seconds to complete.

Relationship Between Frequency and Periodic Time

$$\text{Frequency}, (f) = \frac{1}{\text{Periodic Time}} = \frac{1}{T} \text{ Hertz}$$

or

$$\text{Periodic Time}, (T) = \frac{1}{\text{Frequency}} = \frac{1}{f} \text{ seconds}$$

AC Waveform Example No1

1. What will be the periodic time of a 50Hz waveform and 2. what is the frequency of an AC waveform that has a periodic time of 10mS.

1).

$$\text{Periodic Time, } (T) = \frac{1}{f} = \frac{1}{50} = 0.02 \text{secs or } 20\text{ms}$$

2).

$$\text{Frequency, } (f) = \frac{1}{T} = \frac{1}{10 \times 10^{-3}} = 100\text{Hz}$$

Frequency used to be expressed in "cycles per second" abbreviated to "cps", but today it is more commonly specified in units called "Hertz". For a domestic mains supply the frequency will be either 50Hz or 60Hz depending upon the country and is fixed by the speed of rotation of the generator. But one hertz is a very small unit so prefixes are used that denote the order of magnitude of the waveform at higher frequencies such as **kHz**, **MHz** and even **GHz**.

Amplitude of an AC Waveform

As well as knowing either the periodic time or the frequency of the alternating quantity, another important parameter of the AC waveform is Amplitude, better known as its Maximum or Peak value represented by the terms, V*max* for voltage or I*max* for current.

The peak value is the greatest value of either voltage or current that the waveform reaches during each half cycle measured from the zero baseline. Unlike a DC voltage or current which has a steady state that can be measured or calculated using Ohm's Law, an alternating quantity is constantly changing its value over time.

For pure sinusoidal waveforms this peak value will always be the same for both half cycles (+Vm = -Vm) but for non-sinusoidal or complex waveforms the maximum peak value can be very different for each half cycle. Sometimes, alternating waveforms are given a *peak-to-peak*, V*p-p* value and this is simply the distance or the sum in voltage between the maximum peak value, +V*max* and the minimum peak value, -V*max* during one complete cycle.

The Average Value of an AC Waveform

The average or mean value of a continuous DC voltage will always be equal to its maximum peak value as a DC voltage is constant. This average value will only change if the duty cycle of the DC voltage changes. In a pure sine wave if the average value is calculated over the full cycle, the average value would be equal to zero as the positive and negative halves will cancel each other out. So the average or mean value of an AC waveform is calculated or measured over a half cycle only and this is shown below.
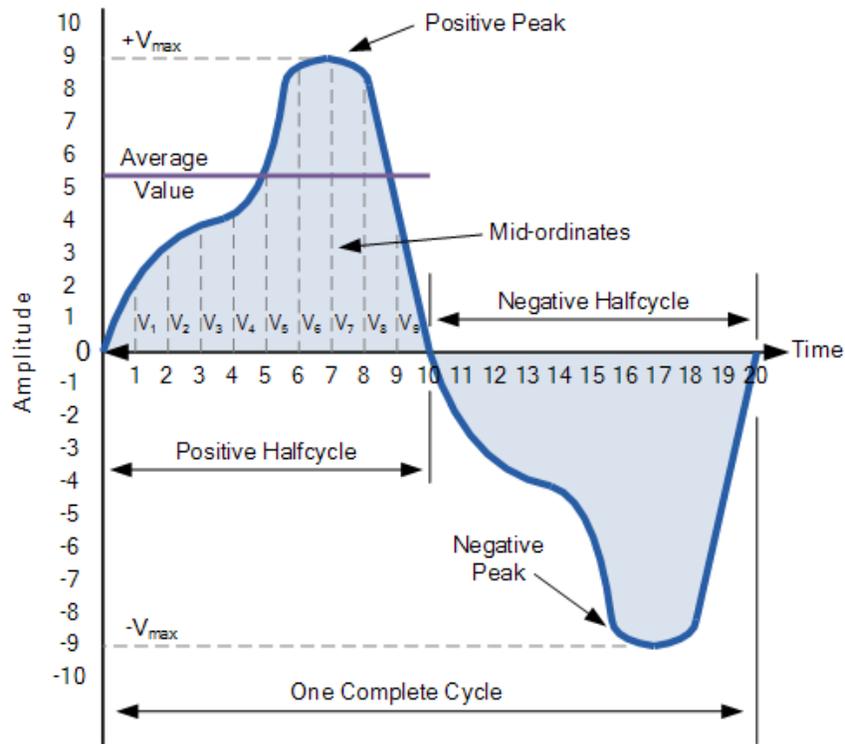
Average Value of a Non-sinusoidal Waveform

Figure Average value

To find the average value of the waveform we need to calculate the area underneath the waveform using the mid-ordinate rule, trapezoidal rule or the Simpson's rule found commonly in mathematics. The approximate area under any irregular waveform can easily be found by simply using the mid-ordinate rule.

The zero axis base line is divided up into any number of equal parts and in our simple example above this value was nine, (V_1 to V_9). The more ordinate lines that are drawn the more accurate will be the final average or mean value. The average value will be the addition of all the instantaneous values added together and then divided by the total number. This is given as.

Average Value of an AC Waveform

$$V_{average} = \frac{V_1 + V_2 + V_3 + V_4 + ... + V_n}{n}$$

Where: n equals the actual number of mid-ordinates used.

For a pure sinusoidal waveform this average or mean value will always be equal to $0.637*V_{max}$ and this relationship also holds true for average values of current.

The RMS Value of an AC Waveform

The average value of an AC waveform that we calculated above as being: $0.637*V_{max}$ is NOT the same value we would use for a DC supply. This is because unlike a DC supply which is constant and and of a fixed value, an AC waveform is constantly changing over time and has no fixed value. Thus the equivalent value for an alternating current system that provides the same amount of electrical power to a load as a DC equivalent circuit is called the "effective value".

The effective value of a sine wave produces the same I^2*R heating effect in a load as we would expect to see if the same load was fed by a constant DC supply. The effective value of a sine wave is more commonly known as the **Root Mean Squared** or simply **RMS** value as it is calculated as the square root of the mean (average) of the square of the voltage or current.

That is V_{rms} or I_{rms} is given as the square root of the average of the sum of all the squared mid-ordinate values of the sine wave. The RMS value for any AC waveform can be found from the following modified average value formula as shown.

RMS Value of an AC Waveform

$$V_{RMS} = \sqrt{\frac{V_1^2 + V_2^2 + V_3^2 + V_4^2 + \ldots + V_n^2}{n}}$$

Where: n equals the number of mid-ordinates.

For a pure sinusoidal waveform this effective or R.M.S. value will always be equal too: $1/\sqrt{2}*V_{max}$ which is equal to $0.707*V_{max}$ and this relationship holds true for RMS values of current. The RMS value for a sinusoidal waveform is always greater than the average value except for a rectangular waveform. In this case the heating effect remains constant so the average and the RMS values will be the same.

One final comment about R.M.S. values. Most multimeters, either digital or analogue unless otherwise stated only measure the R.M.S. values of voltage and current and not the average. Therefore when using a multimeter on a direct current system the reading will be equal to $I = V/R$ and for an alternating current system the reading will be equal to Irms = Vrms/R.

Also, except for average power calculations, when calculating RMS or peak voltages, only use V_{RMS} to find I_{RMS} values, or peak voltage, Vp to find peak current, Ip values. Do not mix them together as Average, RMS or Peak values of a sine wave are completely different and your results will definitely be incorrect.

Form Factor and Crest Factor

Although little used these days, both **Form Factor** and **Crest Factor** can be used to give information about the actual shape of the AC waveform. Form Factor is the ratio between the average value and the RMS value and is given as.

$$\text{Form Factor} = \frac{\text{R.M.S value}}{\text{Average value}} = \frac{0.707 \times \text{Vmax}}{0.637 \times \text{Vmax}}$$

For a pure sinusoidal waveform the Form Factor will always be equal to 1.11. Crest Factor is the ratio between the R.M.S. value and the Peak value of the waveform and is given as.

$$\text{Crest Factor} = \frac{\text{Peak value}}{\text{R.M.S. value}} = \frac{\text{Vmax}}{0.707 \times \text{Vmax}}$$

For a pure sinusoidal waveform the Crest Factor will always be equal to 1.414.

AC Waveform Example No2

A sinusoidal alternating current of 6 amps is flowing through a resistance of 40Ω. Calculate the average voltage and the peak voltage of the supply.

The R.M.S. Voltage value is calculated as:

$$V_{RMS} = I \times R = 6 \times 40 = 240V$$

The Average Voltage value is calculated as:

$$\text{Form Factor} = \frac{V_{RMS}}{V_{average}}$$

$$\therefore V_{average} = \frac{V_{RMS}}{\text{Form Factor}} = \frac{240}{1.11} = 216.2\,\text{volts}$$

The Peak Voltage value is calculated as:

$$\text{Peak Voltage} = \text{R.M.S.} \times 1.414$$

$$\therefore 240 \times 1.414 = 339.4\,\text{volts}$$

The use and calculation of Average, R.M.S, Form factor and Crest Factor can also be use with any type of periodic waveform including Triangular, Square, Sawtoothed or any other irregular or complex voltage/current waveform shape. Conversion between the various sinusoidal values can sometimes be confusing so the following table gives a convenient way of converting one sine wave value to another.

Phasor Diagrams and Phasor Algebra

Phasor Diagrams are a graphical way of representing the magnitude and directional relationship between two or more alternating quantities

Sinusoidal waveforms of the same frequency can have a Phase Difference between themselves which represents the angular difference of the two sinusoidal waveforms. Also the terms "lead" and "lag" as well as "in-phase" and "out-of-phase" are commonly used to indicate the relationship of one waveform to the other with the generalized sinusoidal expression given as: $A_{(t)} = A_m \sin(\omega t \pm \Phi)$ representing the sinusoid in the time-domain form.

But when presented mathematically in this way it is sometimes difficult to visualise this angular or phasor difference between two or more sinusoidal waveforms. One way to overcome this problem is to represent the sinusoids graphically within the spacial or phasor-domain form by using **Phasor Diagrams**, and this is achieved by the rotating vector method.

Basically a rotating vector, simply called a "**Phasor**" is a scaled line whose length represents an AC quantity that has both magnitude ("peak amplitude") and direction ("phase") which is "frozen" at some point in time.

A phasor is a vector that has an arrow head at one end which signifies partly the maximum value of the vector quantity (V or I) and partly the end of the vector that rotates.

Generally, vectors are assumed to pivot at one end around a fixed zero point known as the "point of origin" while the arrowed end representing the quantity, freely rotates in an **anti-clockwise** direction at an angular velocity, (ω) of one full revolution for every cycle. This anti-clockwise rotation of the vector is considered to be a positive rotation. Likewise, a clockwise rotation is considered to be a negative rotation.

Although the both the terms vectors and phasors are used to describe a rotating line that itself has both magnitude and direction, the main difference between the two is that a vectors magnitude is the "peak value" of the sinusoid while a phasors magnitude is the "rms value" of the sinusoid. In both cases the phase angle and direction remains the same.

The phase of an alternating quantity at any instant in time can be represented by a phasor diagram, so phasor diagrams can be thought of as "functions of time". A complete sine wave can be constructed by a single vector rotating at an angular velocity of $\omega = 2\pi f$, where f is the frequency of the waveform. Then a **Phasor** is a quantity that has both "Magnitude" and "Direction".

Generally, when constructing a phasor diagram, angular velocity of a sine wave is always assumed to be: ω in rad/sec. Consider the phasor diagram below.

Phasor Diagram of a Sinusoidal Waveform
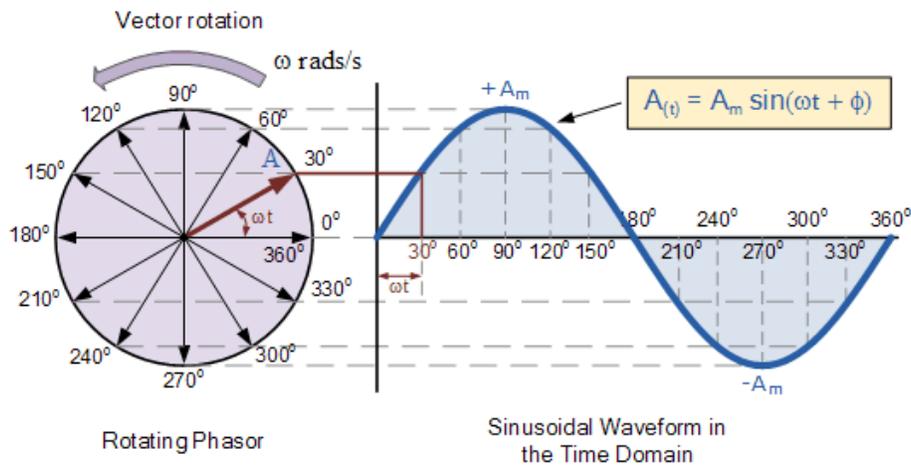

Figure AC waveform

As the single vector rotates in an anti-clockwise direction, its tip at point A will rotate one complete revolution of 360° or 2π representing one complete cycle. If the length of its moving tip is transferred at different angular intervals in time to a graph as shown above, a sinusoidal waveform would be drawn starting at the left with zero time. Each position along the horizontal axis indicates the time that has elapsed since zero time, $t = 0$. When the vector is horizontal the tip of the vector represents the angles at 0°, 180° and at 360°.

Likewise, when the tip of the vector is vertical it represents the positive peak value, ($+Am$) at 90° or $\pi/2$ and the negative peak value, ($-Am$) at 270° or $3\pi/2$. Then the time axis of the waveform represents the angle either in degrees or radians through which the phasor has moved. So we can say that a phasor represent a scaled voltage or current value of a rotating vector which is "frozen" at some point in time, (t) and in our example above, this is at an angle of 30°.

Sometimes when we are analysing alternating waveforms we may need to know the position of the phasor, representing the Alternating Quantity at some particular instant in time especially when we want to compare two different waveforms on the same axis. For example, voltage and current. We have assumed in the waveform above that the waveform starts at time $t = 0$ with a corresponding phase angle in either degrees or radians.

But if a second waveform starts to the left or to the right of this zero point or we want to represent in phasor notation the relationship between the two waveforms then we will need to take into account this phase difference, Φ of the waveform.
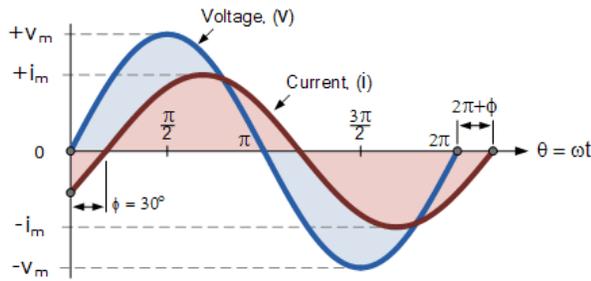
Phase Difference of a Sinusoidal Waveform

Figure phase difference

The generalised mathematical expression to define these two sinusoidal quantities will be written as:

$$v_{(t)} = V_m \sin(\omega t)$$

$$i_{(t)} = I_m \sin(\omega t - \phi)$$

The current, i is lagging the voltage, v by angle Φ and in our example above this is 30°. So the difference between the two phasors representing the two sinusoidal quantities is angle Φ and the resulting phasor diagram will be.

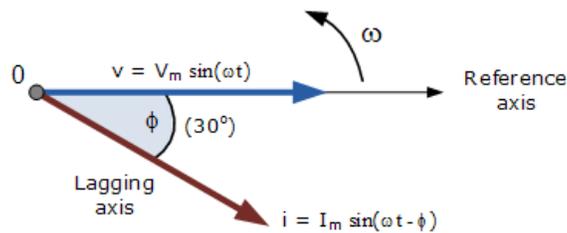## Phasor Diagram of a Sinusoidal Waveform

Figure vector difference

The phasor diagram is drawn corresponding to time zero (t = 0) on the horizontal axis. The lengths of the phasors are proportional to the values of the voltage, (V) and the current, (I) at the instant in time that the phasor diagram is drawn. The current phasor lags the voltage phasor by the angle, Φ, as the two phasors rotate in an *anticlockwise*direction as stated earlier, therefore the angle, Φ is also measured in the same anticlockwise direction.
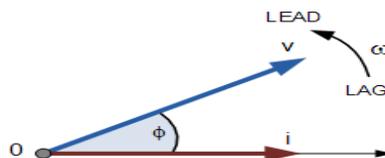

Figure vector difference

117

If however, the waveforms are frozen at time, t = 30°, the corresponding phasor diagram would look like the one shown on the right. Once again the current phasor lags behind the voltage phasor as the two waveforms are of the same frequency.

However, as the current waveform is now crossing the horizontal zero axis line at this instant in time we can use the current phasor as our new reference and correctly say that the voltage phasor is "leading" the current phasor by angle, Φ. Either way, one phasor is designated as the *reference* phasor and all the other phasors will be either leading or lagging with respect to this reference.

Phasor Addition

Sometimes it is necessary when studying sinusoids to add together two alternating waveforms, for example in an AC series circuit, that are not in-phase with each other. If they are in-phase that is, there is no phase shift then they can be added together in the same way as DC values to find the algebraic sum of the two vectors. For example, if two voltages of say 50 volts and 25 volts respectively are together "in-phase", they will add or sum together to form one voltage of 75 volts (50 + 25).

If however, they are not in-phase that is, they do not have identical directions or starting point then the phase angle between them needs to be taken into account so they are added together using phasor diagrams to determine their **Resultant Phasor** or **Vector Sum** by using the *parallelogram law*.

Consider two AC voltages, V_1 having a peak voltage of 20 volts, and V_2 having a peak voltage of 30 volts where V_1 leads V_2 by 60°. The total voltage, V_T of the two voltages can be found by firstly drawing a phasor diagram representing the two vectors and then constructing a parallelogram in which two of the sides are the voltages, V_1 and V_2 as shown below.

Phasor Addition of two Phasors

By drawing out the two phasors to scale onto graph paper, their phasor sum $V_1 + V_2$ can be easily found by measuring the length of the diagonal line, known as the "resultant r-vector", from the zero point to the intersection of the construction lines 0-A. The downside of this graphical method is that it is time consuming when drawing the phasors to scale.

Also, while this graphical method gives an answer which is accurate enough for most purposes, it may produce an error if not drawn accurately or correctly to scale. Then one way to ensure that the correct answer is always obtained is by an analytical method.

Mathematically we can add the two voltages together by firstly finding their "vertical" and "horizontal" directions, and from this we can then calculate both the "vertical" and "horizontal" components for the resultant "r vector", V_T. This analytical method which uses the cosine and sine rule to find this resultant value is commonly called the **Rectangular Form**.

In the rectangular form, the phasor is divided up into a real part, x and an imaginary part, yforming the generalised expression $Z = x \pm jy$. (we will discuss this in more detail in the next tutorial). This then gives us a mathematical expression that represents both the magnitude and the phase of the sinusoidal voltage as:

$$V_m = \cos(\phi) + jV_m(\sin\phi)$$

So the addition of two vectors, A and B using the previous generalised expression is as follows:

$$A = x + jy \qquad B = w + jz$$

$$A + B = (x + w) + j(y + z)$$

Phasor Addition using Rectangular Form

Voltage, V_2 of 30 volts points in the reference direction along the horizontal zero axis, then it has a horizontal component but no vertical component as follows.

Horizontal Component = 30 cos 0° = 30 volts

Vertical Component = 30 sin 0° = 0 volts

This then gives us the rectangular expression for voltage V_2 of: $30 + j0$

Voltage, V_1 of 20 volts leads voltage, V_2 by 60°, then it has both horizontal and vertical components as follows.

Horizontal Component = 20 cos 60° = 20 x 0.5 = 10 volts

Vertical Component = 20 sin 60° = 20 x 0.866 = 17.32 volts

This then gives us the rectangular expression for voltage V_1 of: $10 + j17.32$

The resultant voltage, V_T is found by adding together the horizontal and vertical components as follows.

$V_{Horizontal}$ = sum of real parts of V_1 and V_2 = 30 + 10 = 40 volts

$V_{Vertical}$ = sum of imaginary parts of V_1 and V_2 = 0 + 17.32 = 17.32 volts

Now that both the real and imaginary values have been found the magnitude of voltage, V_T is determined by simply using **Pythagoras's Theorem** for a 90° triangle as follows.

$$V_T = \sqrt{\left(\begin{array}{c}\text{Real or Horizontal}\\\text{Component}\end{array}\right)^2 + \left(\begin{array}{c}\text{Imaginary or Vertical}\\\text{Component}\end{array}\right)^2}$$

$$V_T = \sqrt{40^2 + 17.32^2}$$

$$\therefore V_T = 43.6\,\text{volts}$$

Then the resulting phasor diagram will be:

Resultant Value of V_T

Phasor Subtraction

Phasor subtraction is very similar to the above rectangular method of addition, except this time the vector difference is the other diagonal of the parallelogram between the two voltages of V_1 and V_2 as shown.

Vector Subtraction of two Phasors

This time instead of "adding" together both the horizontal and vertical components we take them away, subtraction.

$$A = x + jy \qquad B = w + jz$$

$$A - B = (x - w) + j(y - z)$$

Phasor Diagram of a Sinusoidal Waveform

As the single vector rotates in an anti-clockwise direction, its tip at point A will rotate one complete revolution of 360o or 2π representing one complete cycle. If the length of its moving tip is transferred at different angular intervals in time to a graph as shown above, a sinusoidal waveform would be drawn starting at the left with zero time. Each position along the horizontal axis indicates the time that has elapsed since zero time, t = 0. When the vector is horizontal the tip of the vector represents the angles at 0o, 180o and at 360o.

Likewise, when the tip of the vector is vertical it represents the positive peak value, (+Am) at 90o or $\pi/2$ and the negative peak value, (-Am) at 270o or $3\pi/2$. Then the time axis of the waveform represents the angle either in degrees or radians through which the phasor has moved. So we can say that a phasor represent a scaled voltage or current value of a rotating vector which is "frozen" at some point in time, (t) and in our example above, this is at an angle of 30o.

Sometimes when we are analysing alternating waveforms we may need to know the position of the phasor, representing the Alternating Quantity at some particular instant in time especially when we want to compare two different waveforms on the same axis. For example, voltage and current. We have assumed in the waveform above that the waveform starts at time t = 0 with a corresponding phase angle in either degrees or radians.

But if a second waveform starts to the left or to the right of this zero point or we want to represent in phasor notation the relationship between the two waveforms then we will need to take into account this phase difference, Φ of the waveform. Consider the diagram below from the previous Phase Difference tutorial.

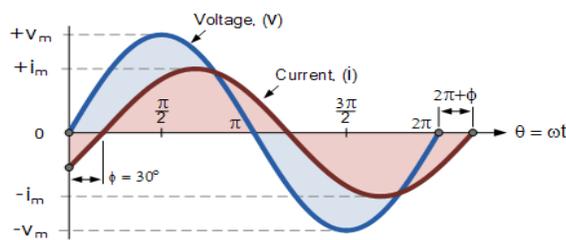**Phase Difference of a Sinusoidal Waveform**

Figure vector difference

The generalised mathematical expression to define these two sinusoidal quantities will be written as:

$$v_{(t)} = V_m \sin(\omega t)$$

$$i_{(t)} = I_m \sin(\omega t - \phi)$$

The current, i is lagging the voltage, v by angle Φ and in our example above this is 30o. So the difference between the two phasors representing the two sinusoidal quantities is angle Φ and the resulting phasor diagram will be.

Phasor Diagram of a Sinusoidal Waveform

The phasor diagram is drawn corresponding to time zero (t = 0) on the horizontal axis. The lengths of the phasors are proportional to the values of the voltage, (V) and the current, (I) at the instant in time that the phasor diagram is drawn. The current phasor lags the voltage phasor by the angle, Φ, as the two phasors rotate in an anticlockwisedirection as stated earlier, therefore the angle, Φ is also measured in the same anticlockwise direction.

If however, the waveforms are frozen at time, t = 30o, the corresponding phasor diagram would look like the one shown on the right. Once again the current phasor lags behind the voltage phasor as the two waveforms are of the same frequency.

However, as the current waveform is now crossing the horizontal zero axis line at this instant in time we can use the current phasor as our new reference and correctly say that the voltage phasor is "leading" the current phasor by angle, Φ. Either way, one phasor is designated as the reference phasor and all the other phasors will be either leading or lagging with respect to this reference.

Phase Difference and Phase Shift

Phase Difference is used to describe the difference in degrees or radians when two or more alternating quantities reach their maximum or zero values

Previously we saw that a Sinusoidal Waveform is an alternating quantity that can be presented graphically in the time domain along an horizontal zero axis. We also saw that as an alternating quantity, sine waves have a positive maximum value at time $\pi/2$, a negative maximum value at time $3\pi/2$, with zero values occurring along the baseline at 0, π and 2π.

However, not all sinusoidal waveforms will pass exactly through the zero axis point at the same time, but may be "shifted" to the right or to the left of 0° by some value when compared to another sine wave.

For example, comparing a voltage waveform to that of a current waveform. This then produces an angular shift or **Phase Difference** between the two sinusoidal waveforms. Any sine wave that does not pass through zero at t = 0 has a phase shift.

The **phase difference** or phase shift as it is also called of a Sinusoidal Waveform is the angle Φ (Greek letter Phi), in degrees or radians that the waveform has shifted from a certain reference point along the horizontal zero axis. In other words phase shift is the lateral difference between two or more waveforms along a common axis and sinusoidal waveforms of the same frequency can have a phase difference.

The phase difference, Φ of an alternating waveform can vary from between 0 to its maximum time period, T of the waveform during one complete cycle and this can be anywhere along the horizontal axis between, Φ = 0 to 2π (radians) or Φ = 0 to 360°depending upon the angular units used.

Phase difference can also be expressed as a *time shift* of τ in seconds representing a fraction of the time period, T for example, +10mS or − 50uS but generally it is more common to express phase difference as an angular measurement.

Then the equation for the instantaneous value of a sinusoidal voltage or current waveform we developed in the previous Sinusoidal Waveform will need to be modified to take account of the phase angle of the waveform and this new general expression becomes.

Phase Difference Equation

$$A_{(t)} = A_{max} \times \sin(\omega t \pm \Phi)$$
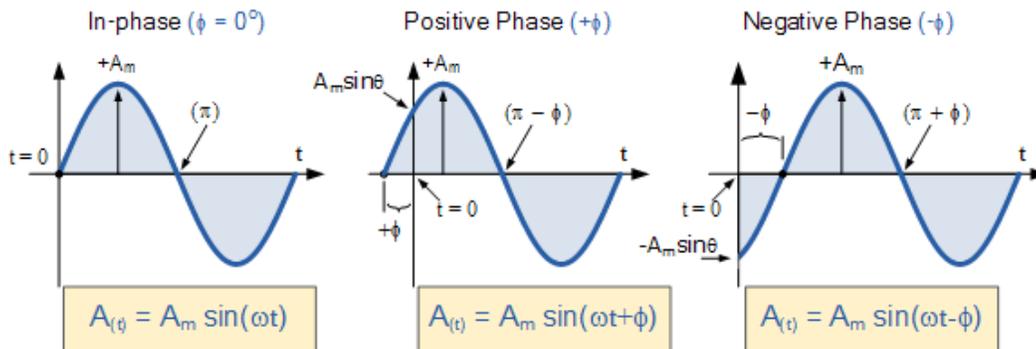
Where:

A_m - is the amplitude of the waveform.

ωt - is the angular frequency of the waveform in radian/sec.

Φ (phi) - is the phase angle in degrees or radians that the waveform has shifted either left or right from the reference point.

If the positive slope of the sinusoidal waveform passes through the horizontal axis "before" t = 0 then the waveform has shifted to the left so Φ >0, and the phase angle will be positive in nature, +Φ giving a leading phase angle. In other words it appears earlier in time than 0° producing an anticlockwise rotation of the vector.

Likewise, if the positive slope of the sinusoidal waveform passes through the horizontal x-axis some time "after" t = 0 then the waveform has shifted to the right so Φ <0, and the phase angle will be negative in nature -Φ producing a lagging phase angle as it appears later in time than 0° producing a clockwise rotation of the vector. Both cases are shown below.

Phase Relationship of a Sinusoidal Waveform

In-phase ($\phi = 0°$) · Positive Phase ($+\phi$) · Negative Phase ($-\phi$)

$$A_{(t)} = A_m \sin(\omega t)$$

$$A_{(t)} = A_m \sin(\omega t + \phi)$$

$$A_{(t)} = A_m \sin(\omega t - \phi)$$

Firstly, lets consider that two alternating quantities such as a voltage, v and a current, ihave the same frequency f in Hertz. As the frequency of the two quantities is the same the angular velocity, ω must also be the same. So at any instant in time we can say that the phase of voltage, v will be the same as the phase of the current, i.

Then the angle of rotation within a particular time period will always be the same and the phase difference between the two quantities of v and i will therefore be zero and $\Phi = 0$. As the frequency of the voltage, v and the current, i are the same they must both reach their maximum positive, negative and zero values during one complete cycle at the same time (although their amplitudes may be different). Then the two alternating quantities, v and iare said to be "in-phase".

Two Sinusoidal Waveforms – "in-phase"
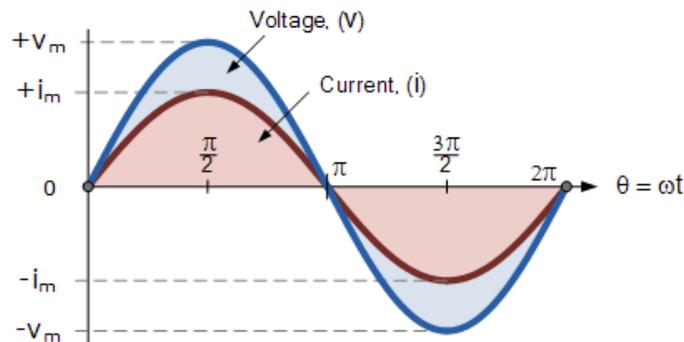
Now lets consider that the voltage, v and the current, i have a phase difference between themselves of 30°, so ($\Phi = 30°$ or $\pi/6$ radians). As both alternating quantities rotate at the same speed, i.e. they have the same frequency, this phase difference will remain constant for all instants in time, then the phase difference of 30° between the two quantities is represented by phi, Φ as shown below.

124

Phase Difference of a Sinusoidal Waveform
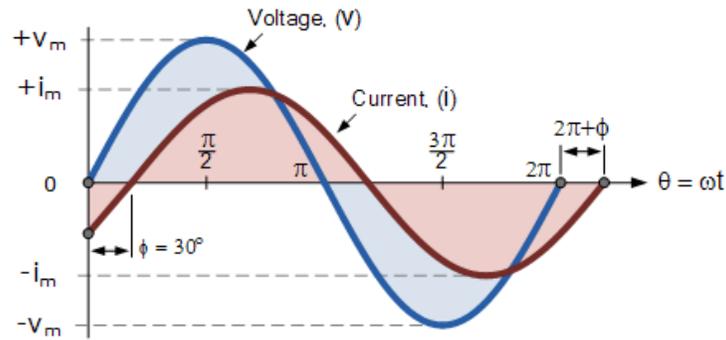
The voltage waveform above starts at zero along the horizontal reference axis, but at that same instant of time the current waveform is still negative in value and does not cross this reference axis until 30° later. Then there exists a **Phase difference** between the two waveforms as the current cross the horizontal reference axis reaching its maximum peak and zero values after the voltage waveform.

As the two waveforms are no longer "in-phase", they must therefore be "out-of-phase" by an amount determined by phi, Φ and in our example this is 30°. So we can say that the two waveforms are now 30° out-of phase. The current waveform can also be said to be "lagging" behind the voltage waveform by the phase angle, Φ. Then in our example above the two waveforms have a **Lagging Phase Difference** so the expression for both the voltage and current above will be given as.

$$\text{Voltage,} (v_t) = V_m \sin \omega t$$

$$\text{Current,} (i_t) = I_m \sin(\omega t - \theta)$$

where, i lags v by angle Φ

Likewise, if the current, i has a positive value and crosses the reference axis reaching its maximum peak and zero values at some time before the voltage, v then the current waveform will be "leading" the voltage by some phase angle. Then the two waveforms are said to have a **Leading Phase Difference** and the expression for both the voltage and the current will be.

$$\text{Voltage,} (v_t) = V_m \sin \omega t$$

$$\text{Current,} (i_t) = I_m \sin(\omega t + \theta)$$

where, i leads v by angle Φ

The phase angle of a sine wave can be used to describe the relationship of one sine wave to another by using the terms "Leading" and "Lagging" to indicate the relationship between two sinusoidal waveforms of the same frequency, plotted onto the same reference axis. In our example above the two waveforms

are out-of-phase by 30°. So we can correctly say that i lags v or we can say that v leads i by 30° depending upon which one we choose as our reference.

The relationship between the two waveforms and the resulting phase angle can be measured anywhere along the horizontal zero axis through which each waveform passes with the "same slope" direction either positive or negative.

In AC power circuits this ability to describe the relationship between a voltage and a current sine wave within the same circuit is very important and forms the bases of AC circuit analysis.

AC Inductance and Inductive Reactance:

The opposition to current flow through an AC Inductor is called Inductive Reactance and which depends lineally on the supply frequency

Inductors and chokes are basically coils or loops of wire that are either wound around a hollow tube former (air cored) or wound around some ferromagnetic material (iron cored) to increase their inductive value called **inductance**.

Inductors store their energy in the form of a magnetic field that is created when a voltage is applied across the terminals of an inductor. The growth of the current flowing through the inductor is not instant but is determined by the inductors own self-induced or back emf value. Then for an inductor coil, this back emf voltage V_L is proportional to the *rate of change of the current* flowing through it.

This current will continue to rise until it reaches its maximum steady state condition which is around five time constants when this self-induced back emf has decayed to zero. At this point a steady state current is flowing through the coil, no more back emf is induced to oppose the current flow and therefore, the coil acts more like a short circuit allowing maximum current to flow through it.

However, in an alternating current circuit which contains an **AC Inductance**, the flow of current through an inductor behaves very differently to that of a steady state DC voltage. Now in an AC circuit, the opposition to the current flowing through the coils windings not only depends upon the inductance of the coil but also the frequency of the applied voltage waveform as it varies from its positive to negative values.

The actual opposition to the current flowing through a coil in an AC circuit is determined by the AC Resistance of the coil with this AC resistance being represented by a complex number. But to distinguish a DC resistance value from an AC resistance value, which is also known as Impedance, the term **Reactance** is used.

Like resistance, reactance is measured in Ohm's but is given the symbol "X" to distinguish it from a purely resistive "R" value and as the component in question is an inductor, the reactance of an inductor is called **Inductive Reactance**, (X_L) and is measured in Ohms. Its value can be found from the formula.

## Inductive Reactance

$$X_L = 2\pi f L$$

Where: X_L is the Inductive Reactance in Ohms, f is the frequency in Hertz and L is the inductance of the coil in Henries.

We can also define inductive reactance in radians, where Omega, ω equals $2\pi f$.

$$X_L = \omega L$$

So whenever a sinusoidal voltage is applied to an inductive coil, the back emf opposes the rise and fall of the current flowing through the coil and in a purely inductive coil which has zero resistance or losses, this impedance (which can be a complex number) is equal to its inductive reactance. Also reactance is represented by a vector as it has both a magnitude and a direction (angle). Consider the circuit below.

## AC Inductance with a Sinusoidal Supply
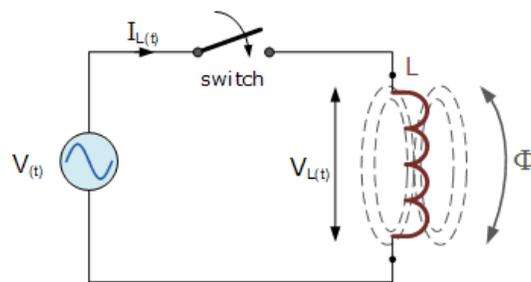
This simple circuit above consists of a pure inductance of L Henries (H), connected across a sinusoidal voltage given by the expression: $V(t) = V_{max} \sin \omega t$. When the switch is closed this sinusoidal voltage will cause a current to flow and rise from zero to its maximum value. This rise or change in the current will induce a magnetic field within the coil which in turn will oppose or restrict this change in the current.

But before the current has had time to reach its maximum value as it would in a DC circuit, the voltage changes polarity causing the current to change direction. This change in the other direction once again being delayed by the self-induced back emf in the coil, and in a circuit containing a pure inductance only, the current is delayed by 90°.

The applied voltage reaches its maximum positive value a quarter ($1/4f$) of a cycle earlier than the current reaches its maximum positive value, in other words, a voltage applied to a purely inductive circuit "LEADS" the current by a quarter of a cycle or 90° as shown below.

Sinusoidal Waveforms for AC Inductance
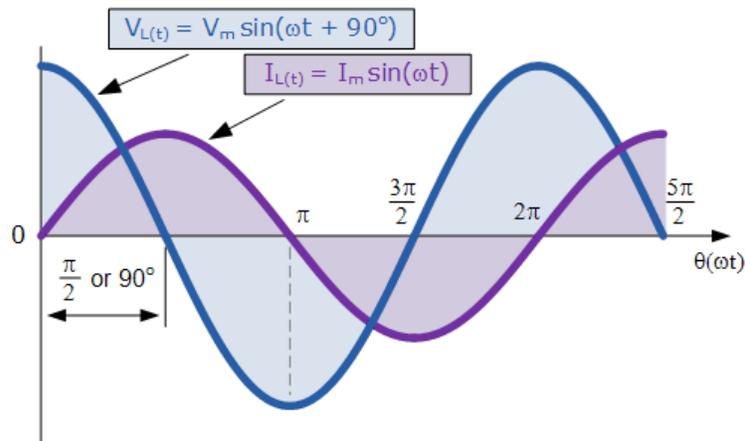
This effect can also be represented by a phasor diagram were in a purely inductive circuit the voltage "LEADS" the current by 90°. But by using the voltage as our reference, we can also say that the current "LAGS" the voltage by one quarter of a cycle or 90° as shown in the vector diagram below.

Phasor Diagram for AC Inductance
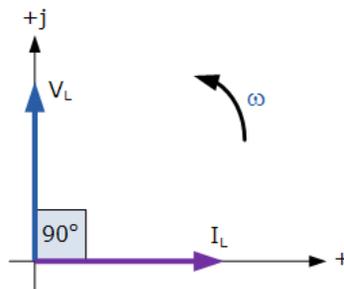
So for a pure loss less inductor, V_L "leads" I_L by 90°, or we can say that I_L "lags" V_L by 90°.

There are many different ways to remember the phase relationship between the voltage and current flowing through a pure inductor circuit, but one very simple and easy to remember way is to use the mnemonic expression "ELI" (pronounced *Ellie* as in the girls name). ELI stands for Electromotive force first in an AC inductance, L before the current I. In other words, voltage before the current in an inductor, E, L, I equals "ELI", and whichever phase angle the voltage starts at, this expression always holds true for a pure inductor circuit.

## The Effect of Frequency on Inductive Reactance

When a 50Hz supply is connected across a suitable AC Inductance, the current will be delayed by 90° as described previously and will obtain a peak value of I amps before the voltage reverses polarity at the end of each half cycle, i.e. the current rises up to its maximum value in "T secs".

If we now apply a 100Hz supply of the same peak voltage to the coil, the current will still be delayed by 90° but its maximum value will be lower than the 50Hz value because the time it requires to reach its maximum value has been reduced due to the increase in frequency because now it only has "1/2 T secs" to reach its peak value. Also, the rate of change of the flux within the coil has also increased due to the increase in frequency.

Then from the above equation for inductive reactance, it can be seen that if either the **Frequency** OR the **Inductance** is increased the overall inductive reactance value of the coil would also increase. As the frequency increases and approaches infinity, the inductors reactance and therefore its impedance would also increase towards infinity acting like an open circuit.

Likewise, as the frequency approaches zero or DC, the inductors reactance would also decrease to zero, acting like a short circuit. This means then that inductive reactance is "directly proportional to frequency" and has a small value at low frequencies and a high value at higher frequencies as shown.

Inductive Reactance against Frequency
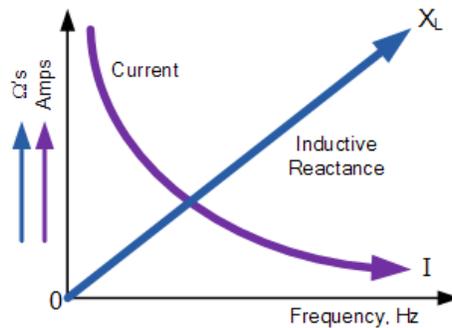
The inductive reactance of an inductor increases as the frequency across it increases therefore inductive reactance is proportional to frequency ($X_L \alpha f$) as the back emf generated in the inductor is equal to its inductance multiplied by the rate of change of current in the inductor.

Also as the frequency increases the current flowing through the inductor also reduces in value.

We can present the effect of very low and very high frequencies on a the reactance of a pure AC Inductance as follows:
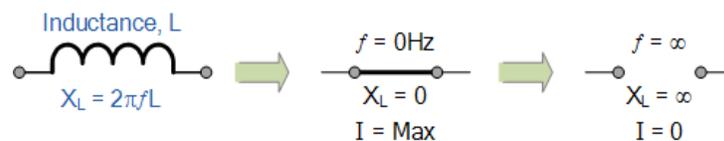
In an AC circuit containing pure inductance the following formula applies:

$$\text{Current}, I = \frac{\text{Voltage}}{\text{Opposition to current flow}} = \frac{V}{X_L}$$

So how did we arrive at this equation. Well the self induced emf in the inductor is determined by Faraday's Law that produces the effect of self-induction in the inductor due to the rate of change of the current and the maximum value of the induced emf will correspond to the maximum rate of change. Then the voltage in the inductor coil is given as:

$$V_{L(t)} = L\frac{di_{L(t)}}{dt}$$

$$\text{If,} \quad i_{L(t)} = I_{max} \sin(\omega t) \quad \text{then:}$$

$$V_{L(t)} = L\frac{d}{dt}I_{max}\sin(\omega t + \theta)$$

$$= \omega L I_{max}\cos(\omega t + \theta)$$

then the voltage across an AC inductance will be defined as:

$$V_L = \omega L I_{max}\sin(\omega t + 90^o)$$

Where: $V_L = I\omega L$ which is the voltage amplitude and $\theta = +90^o$ which is the phase difference or phase angle between the voltage and current.
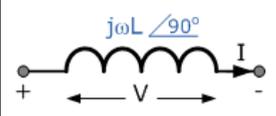
In the Phasor Domain

In the phasor domain the voltage across the coil is given as:

$$V_L = j\omega L I$$

$$\text{where:} \quad j\omega L = jX_L = 2\pi f L = \text{IMPEDANCE, } Z$$

and in Polar Form this would be written as: $X_L \angle 90^o$ where:

$$X_L \angle \theta = \frac{V_L \angle +90^o}{I_L \angle 0^o}$$

$$X_L \angle \theta = j\omega L = 0 + jX_L = \omega L\angle +90^o = Z\angle +90^o$$

AC Capacitance and Capacitive Reactance

The opposition to current flow through an AC Capacitor is called Capacitive Reactance and which itself is inversely proportional to the supply frequency

Capacitors store energy on their conductive plates in the form of an electrical charge. When a capacitor is connected across a DC supply voltage it charges up to the value of the applied voltage at a rate determined by its time constant.

A capacitor will maintain or hold this charge indefinitely as long as the supply voltage is present. During this charging process, a charging current, i flows into the capacitor opposed by any changes to the voltage at a rate which is equal to the rate of change of the electrical charge on the plates. A capacitor therefore has an opposition to current flowing onto its plates.

The relationship between this charging current and the rate at which the capacitors supply voltage changes can be defined mathematically as: $i = C(dv/dt)$, where C is the capacitance value of the capacitor in farads and dv/dt is the rate of change of the supply voltage with respect to time. Once it is "fully-charged" the capacitor blocks the flow of any more electrons onto its plates as they have become saturated and the capacitor now acts like a temporary storage device.

A pure capacitor will maintain this charge indefinitely on its plates even if the DC supply voltage is removed. However, in a sinusoidal voltage circuit which contains "AC Capacitance", the capacitor will alternately charge and discharge at a rate determined by the frequency of the supply. Then capacitors in AC circuits are constantly charging and discharging respectively.

When an alternating sinusoidal voltage is applied to the plates of an AC capacitor, the capacitor is charged firstly in one direction and then in the opposite direction changing polarity at the same rate as the AC supply voltage. This instantaneous change in voltage across the capacitor is opposed by the fact that it takes a certain amount of time to deposit (or release) this charge onto the plates and is given by $V = Q/C$. Consider the circuit below.

AC Capacitance with a Sinusoidal Supply
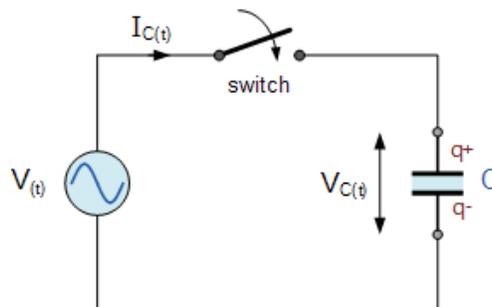
When the switch is closed in the circuit above, a high current will start to flow into the capacitor as there is no charge on the plates at $t = 0$. The sinusoidal supply voltage, V is increasing in a positive direction at its maximum rate as it crosses the zero reference axis at an instant in time given as 0°. Since the rate of change of the potential difference across the plates is now at its maximum value, the flow of current into the capacitor will also be at its maximum rate as the maximum amount of electrons are moving from one plate to the other.

As the sinusoidal supply voltage reaches its 90° point on the waveform it begins to slow down and for a very brief instant in time the potential difference across the plates is neither increasing nor decreasing therefore the current decreases to zero as there is no rate of voltage change. At this 90° point the potential difference across the capacitor is at its maximum (V_{max}), no current flows into the capacitor as the capacitor is now fully charged and its plates saturated with electrons.

At the end of this instant in time the supply voltage begins to decrease in a negative direction down towards the zero reference line at 180°. Although the supply voltage is still positive in nature the capacitor starts to discharge some of its excess electrons on its plates in an effort to maintain a constant voltage. This results in the capacitor current flowing in the opposite or negative direction.

When the supply voltage waveform crosses the zero reference axis point at instant 180° the rate of change or slope of the sinusoidal supply voltage is at its maximum but in a negative direction, consequently the current flowing into the capacitor is also at its maximum rate at that instant. Also at this 180° point the potential difference across the plates is zero as the amount of charge is equally distributed between the two plates.

Then during this first half cycle 0° to 180° the applied voltage reaches its maximum positive value a quarter $(1/4f)$ of a cycle after the current reaches its maximum positive value, in other words, a voltage applied to a purely capacitive circuit "LAGS" the current by a quarter of a cycle or 90° as shown below.

Sinusoidal Waveforms for AC Capacitance
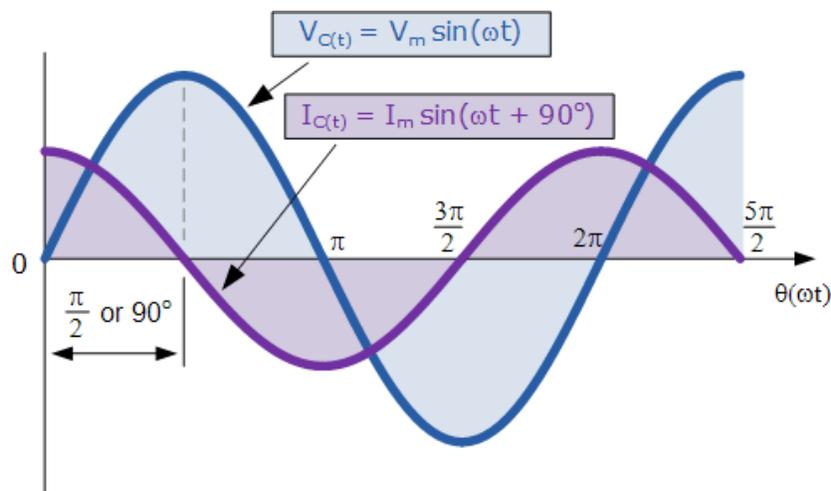
During the second half cycle 180° to 360°, the supply voltage reverses direction and heads towards its negative peak value at 270°. At this point the potential difference across the plates is neither

decreasing nor increasing and the current decreases to zero. The potential difference across the capacitor is at its maximum negative value, no current flows into the capacitor and it becomes fully charged the same as at its 90° point but in the opposite direction.

As the negative supply voltage begins to increase in a positive direction towards the 360° point on the zero reference line, the fully charged capacitor must now loose some of its excess electrons to maintain a constant voltage as before and starts to discharge itself until the supply voltage reaches zero at 360° at which the process of charging and discharging starts over again.

From the voltage and current waveforms and description above, we can see that the current is always leading the voltage by 1/4 of a cycle or $\pi/2 = 90^\circ$ "out-of-phase" with the potential difference across the capacitor because of this charging and discharging process.

This effect can also be represented by a phasor diagram where in a purely capacitive circuit the voltage "LAGS" the current by 90°. But by using the voltage as our reference, we can also say that the current "LEADS" the voltage by one quarter of a cycle or 90° as shown in the vector diagram below.

Phasor Diagram for AC Capacitance
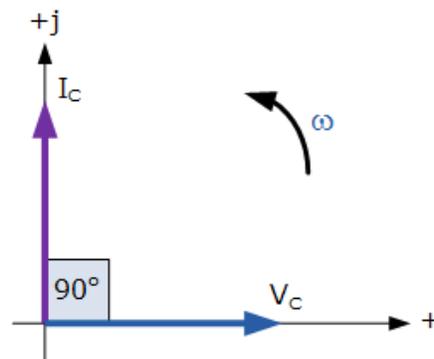
So for a pure capacitor, V_C "lags" I_C by 90°, or we can say that I_C "leads" V_C by 90°.

There are many different ways to remember the phase relationship between the voltage and current flowing in a pure AC capacitance circuit, but one very simple and easy to remember way is to use the mnemonic expression called "ICE". ICE stands for current I first in an AC capacitance, C before Electromotive force. In other words, current before the voltage in a capacitor, I, C, E equals "ICE", and whichever phase angle the voltage starts at, this expression always holds true for a pure AC capacitance circuit.

Capacitive Reactance

So we now know that capacitors oppose changes in voltage with the flow of electrons onto the plates of the capacitor being directly proportional to the rate of voltage change across its plates as the capacitor charges and discharges. Unlike a resistor where the opposition to current flow is its actual resistance, the opposition to current flow in a capacitor is called **Reactance**.

133

Like resistance, reactance is measured in Ohm's but is given the symbol X to distinguish it from a purely resistive R value and as the component in question is a capacitor, the reactance of a capacitor is called **Capacitive Reactance**, (X_C) which is measured in Ohms.

Since capacitors charge and discharge in proportion to the rate of voltage change across them, the faster the voltage changes the more current will flow. Likewise, the slower the voltage changes the less current will flow. This means then that the reactance of an AC capacitor is "inversely proportional" to the frequency of the supply as shown.

Capacitive Reactance

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

Where: X_C is the Capacitive Reactance in Ohms, f is the frequency in Hertz and C is the AC capacitance in Farads, symbol F.

When dealing with AC capacitance, we can also define capacitive reactance in terms of radians, where Omega, ω equals $2\pi f$.

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

From the above formula we can see that the value of capacitive reactance and therefore its overall impedance (in Ohms) decreases towards zero as the frequency increases acting like a short circuit. Likewise, as the frequency approaches zero or DC, the capacitors reactance increases to infinity, acting like an open circuit which is why capacitors block DC.

The relationship between capacitive reactance and frequency is the exact opposite to that of inductive reactance, (X_L) we saw in the previous tutorial. This means then that capacitive reactance is "inversely proportional to frequency" and has a high value at low frequencies and a low value at higher frequencies as shown.

Capacitive Reactance against Frequency
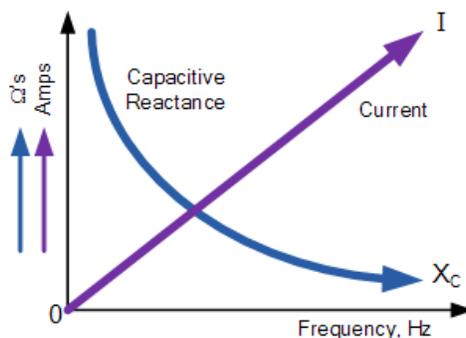
Capacitive reactance of a capacitor decreases as the frequency across its plates increases. Therefore, capacitive reactance is inversely proportional to frequency. Capacitive reactance opposes current flow but the electrostatic charge on the plates (its AC capacitance value) remains constant.

This means it becomes easier for the capacitor to fully absorb the change in charge on its plates during each half cycle. Also as the frequency increases the current flowing into the capacitor increases in value because the rate of voltage change across its plates increases.

We can present the effect of very low and very high frequencies on the reactance of a pure AC Capacitance as follows:
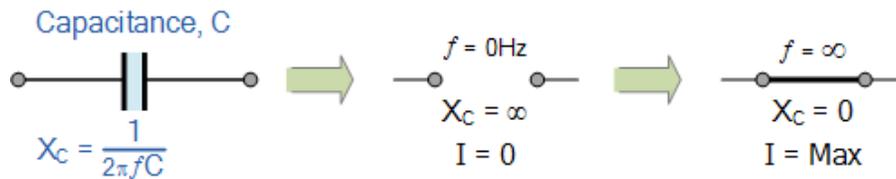
In an AC circuit containing pure capacitance the current (electron flow) flowing into the capacitor is given as:

$$I_{C(t)} = \frac{dq}{dt} \quad \text{where: } q = CV_C = CV_{max}\sin(\omega t)$$

$$\therefore I_{C(t)} = \frac{d}{dt}CV_{max}\sin(\omega t) = \omega CV_{max}\cos(\omega t)$$

$$\text{If: } I_{max} = \frac{V_{max}}{X_C} \quad \text{where: } X_C = \frac{1}{2\pi f C} = \frac{1}{\omega C}$$

$$\text{then: } I_{max} = \omega CV_{max}$$

and therefore, the rms current flowing into an AC capacitance will be defined as:

$$I_{C(t)} = I_{max}\sin\left(\omega t + 90^\circ\right)$$

Where: $I_C = V/(1/\omega C)$ (or $I_C = V/X_C$) is the current magnitude and $\theta = +90^\circ$ which is the phase difference or phase angle between the voltage and current. For a purely capacitive circuit, Ic leads Vc by 90°, or Vc lags Ic by 90°.
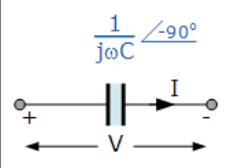
Phasor Domain

In the phasor domain the voltage across the plates of an AC capacitance will be:

135

$$V_C = \frac{1}{j\omega C} \times I_C$$

$$\text{where: } \frac{1}{j\omega C} = jX_C = \frac{1}{2\pi f C} = \text{IMPEDANCE, Z}$$

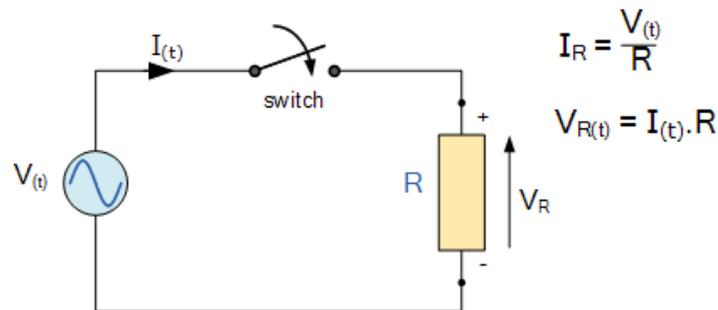and in Polar Form this would be written as: $X_C \angle -90^o$ where:

$$X_C \angle \theta = \frac{V_C \angle 0^o}{I_C \angle +90^o}$$

$$X_C \angle \theta = \frac{1}{j\omega C} = 0 - jX_C = \frac{1}{\omega C} \angle -90^o = Z \angle -90^o$$

AC Resistance and Impedance

Impedance, measured in Ohms, is the effective resistance to current flow around an AC circuit containing resistances and reactances

AC Resistance with a Sinusoidal Supply

$$I_R = \frac{V_{(t)}}{R}$$

$$V_{R(t)} = I_{(t)}.R$$

When the switch is closed, an AC voltage, V will be applied to resistor, R. This voltage will cause a current to flow which in turn will rise and fall as the applied voltage rises and falls sinusoidally. As the load is a resistance, the current and voltage will both reach their maximum or peak values and fall through zero at exactly the same time, i.e. they rise and fall simultaneously and are therefore said to be "*in-phase*".

Then the electrical current that flows through an AC resistance varies sinusoidally with time and is represented by the expression, $I(t) = Im \times \sin(\omega t + \theta)$, where Im is the maximum amplitude of the

current and θ is its phase angle. In addition we can also say that for any given current, i flowing through the resistor the maximum or peak voltage across the terminals of R will be given by Ohm's Law as:

$$V_{(t)} = R.I_{(t)} = R.I_m \sin\left(\omega t + \theta\right)$$

and the instantaneous value of the current, i will be:

$$i_{R(t)} = I_{R(max)} \sin \omega t$$

So for a purely resistive circuit the alternating current flowing through the resistor varies in proportion to the applied voltage across it following the same sinusoidal pattern. As the supply frequency is common to both the voltage and current, their phasors will also be common resulting in the current being "in-phase" with the voltage, (θ = 0).

In other words, there is no phase difference between the current and the voltage when using an AC resistance as the current will achieve its maximum, minimum and zero values whenever the voltage reaches its maximum, minimum and zero values as shown below.

Sinusoidal Waveforms for AC Resistance
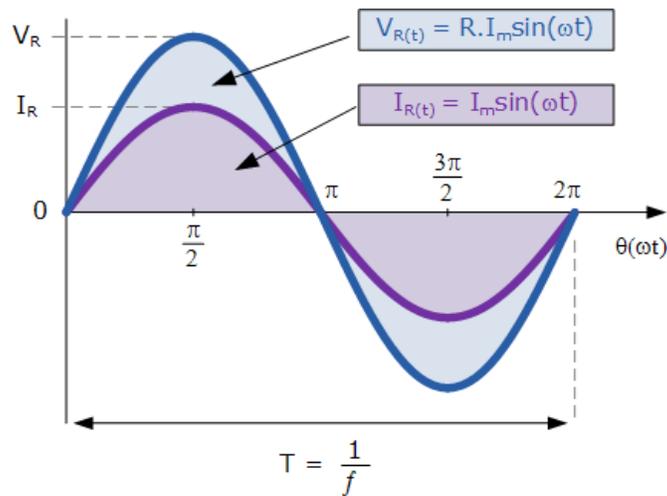
This "in-phase" effect can also be represented by a phasor diagram. In the complex domain, resistance is a real number only meaning that there is no "j" or imaginary component. Therefore, as the voltage and current are both in-phase with each other, there will be no phase difference (θ = 0) between them, so the vectors of each quantity are drawn super-imposed upon one another along the same reference axis. The transformation from the sinusoidal time-domain into the phasor-domain is given as.

Phasor Diagram for AC Resistance
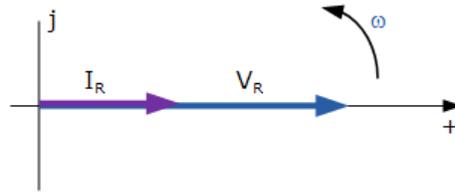
As a phasor represents the RMS values of the voltage and current quantities unlike a vector which represents the peak or maximum values, dividing the peak value of the time-domain expressions above by $\sqrt{2}$ the corresponding voltage-current phasor relationship is given as.

RMS Relationship

$$I = \frac{I_m}{\sqrt{2}}\angle\theta \ A \quad \text{and} \quad V = \frac{R.I_m}{\sqrt{2}}\angle\theta \ V$$

$$\therefore R = \frac{V}{I} = \frac{(R.I_m/\sqrt{2})\angle\theta}{(I_m/\sqrt{2})\angle\theta}$$

Phase Relationship

$$V = R.I_{RMS}\angle\theta \quad \text{and} \quad I = I_{RMS}\angle\theta$$

$$V\angle\theta_v = I\angle\theta_i$$

$$\therefore \theta_v = \theta_i \ (in-phase)$$

This shows that a pure resistance within an AC circuit produces a relationship between its voltage and current phasors in exactly the same way as it would relate the same resistors voltage and current relationship within a DC circuit. However, in a DC circuit this relationship is commonly called **Resistance**, as defined by Ohm's Law but in a sinusoidal AC circuit this voltage-current relationship is now called **Impedance**. In other words, in an AC circuit electrical resistance is called "Impedance".

In both cases this voltage-current (V-I) relationship is always linear in a pure resistance. So when using resistors in AC circuits the term **Impedance**, symbol **Z** is the generally used to mean its resistance. Therefore, we can correctly say that for a resistor, DC resistance = AC impedance , or R = Z.

The impedance vector is represented by the letter, (Z) for an AC resistance value with the units of Ohm's (Ω) the same as for DC. Then Impedance (or AC resistance) can be defined as:

AC Impedance

$$Z = \frac{V}{I} \ \Omega's$$

Impedance can also be represented by a complex number as it depends upon the frequency of the circuit, ω when reactive components are present. But in the case of a purely resistive circuit this reactive component will always be zero and the general expression for impedance in a purely resistive circuit given as a complex number will be:

$$Z = R + j0 = R \ \Omega's$$

Since the phase angle between the voltage and current in a purely resistive AC circuit is zero, the power factor must also be zero and is given as: $\cos 0^o = 1.0$, Then the instantaneous power consumed in the resistor is given by:

$$P = V.I = V_m \sin \omega t \times I_m \sin \omega t = V_m I_m \sin^2 \omega t$$

$$\therefore P_{max} = \sin^2(\omega t), \qquad where \ P_{max} = V_{max} I_{max}$$

However, as the average power in a resistive or reactive circuit depends upon the phase angle and in a purely resistive circuit this is equal to $\theta = 0$, the power factor is equal to one so the average power consumed by an AC resistance can be defined simply by using Ohm's Law as:

$$P = V.I = I^2 R = \frac{V^2}{R} \ watts$$

which are the same Ohm's Law equations as for DC circuits. Then the effective power consumed by an AC resistance is equal to the power consumed by the same resistor in a DC circuit.

Many AC circuits such as heating elements and lamps consist of a pure ohmic resistance only and have negligible values of inductance or capacitance containing on impedance.

In such circuits we can use both Ohm's Law ,Kirchoff's Law as well as simple circuit rules for calculating and finding the voltage, current, impedance and power as in DC circuit analysis. When working with such rules it is usual to use RMS values only.

AC Resistance Example No1

An electrical heating element which has an AC resistance of 60 Ohms is connected across a 240V AC single phase supply. Calculate the current drawn from the supply and the power consumed by the heating element. Also draw the corresponding phasor diagram showing the phase relationship between the current and voltage.

1. The supply current:

$$I = \frac{V}{R} = \frac{240}{60} = 4.0\ A$$

2. The Active power consumed by the AC resistance is calculated as:

$$P = I^2 R = 4^2 . 60 = 960\ W$$

3. As there is no phase difference in a resistive component, ($\theta = 0$), the corresponding phasor diagram is given as:

AC Resistance Example No2

A sinusoidal voltage supply defined as: $V(t) = 100 \times \cos(\omega t + 30°)$ is connected to a pure resistance of 50 Ohms. Determine its impedance and the peak value of the current flowing through the circuit. Draw the corresponding phasor diagram.
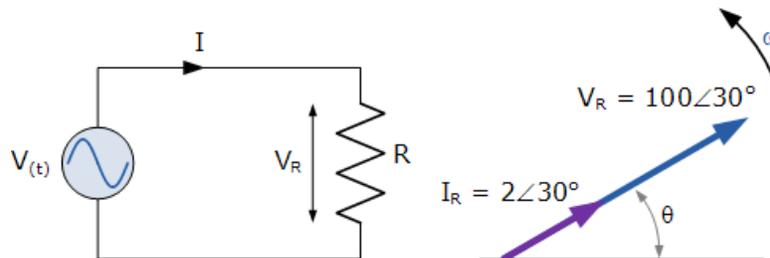
The sinusoidal voltage across the resistance will be the same as for the supply in a purely resistive circuit. Converting this voltage from the time-domain expression into the phasor-domain expression gives us:

$$V_{R(t)} = 100\cos(\omega t + 30°) \quad \Rightarrow \quad V_R = 100\angle 30° \text{ volts}$$

Applying Ohms Law gives us:

$$I_R = \frac{V_R}{R} = \frac{100\angle 30°}{50\Omega} = 2\angle 30° \text{ Amps}$$

The corresponding phasor diagram will therefore be:

j-notation:

The mathematics used in Electrical Engineering to add together resistances, currents or DC voltages use what are called "real numbers" used as either integers or as fractions. But real numbers are

not the only kind of numbers we need to use especially when dealing with frequency dependent sinusoidal sources and vectors. As well as using normal or real numbers, **Complex Numbers** were introduced to allow complex equations to be solved with numbers that are the square roots of negative numbers, $\sqrt{-1}$.

In electrical engineering this type of number is called an "imaginary number" and to distinguish an imaginary number from a real number the letter " j " known commonly in electrical engineering as the **j-operator**, is used. Thus the letter "j" is placed in front of a real number to signify its imaginary number operation.

Examples of imaginary numbers are: j3, j12, j100 etc. Then a **complex number** consists of two distinct but very much related parts, a " Real Number " plus an " Imaginary Number ".

Complex Numbers represent points in a two dimensional complex or s-plane that are referenced to two distinct axes. The horizontal axis is called the "real axis" while the vertical axis is called the "imaginary axis". The real and imaginary parts of a complex number are abbreviated as Re(z) and Im(z), respectively.

Complex numbers that are made up of real (the active component) and imaginary (the reactive component) numbers can be added, subtracted and used in exactly the same way as elementary algebra is used to analyse DC circuits.

The rules and laws used in mathematics for the addition or subtraction of imaginary numbers are the same as for real numbers, j2 + j4 = j6 etc. The only difference is in multiplication because two imaginary numbers multiplied together becomes a negative real number. Real numbers can also be thought of as a complex number but with a zero imaginary part labelled j0.

The **j-operator** has a value exactly equal to $\sqrt{-1}$, so successive multiplication of " j ", (j x j) will result in j having the following values of, -1, -j and +1. As the j-operator is commonly used to indicate the anticlockwise rotation of a vector, each successive multiplication or power of " j", j^2, j^3 etc, will force the vector to rotate through an angle of 90° anticlockwise as shown below. Likewise, if the multiplication of the vector results in a -j operator then the phase shift will be -90°, i.e. a clockwise rotation.
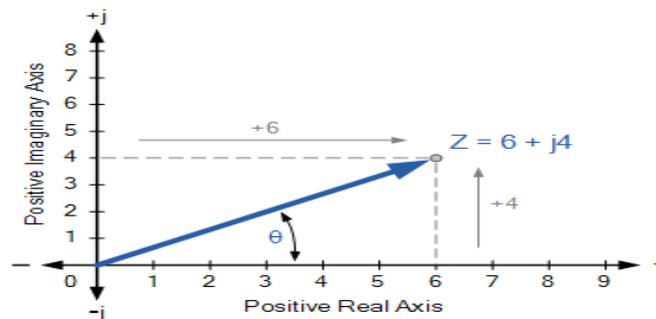
Vector Rotation of the j-operator

$$90° \text{ rotation}: \quad j^1 = \sqrt{-1} = +j$$

$$180° \text{ rotation}: \quad j^2 = \left(\sqrt{-1}\right)^2 = -1$$

$$270° \text{ rotation}: \quad j^3 = \left(\sqrt{-1}\right)^3 = -j$$

$$360° \text{ rotation}: \quad j^4 = \left(\sqrt{-1}\right)^4 = +1$$
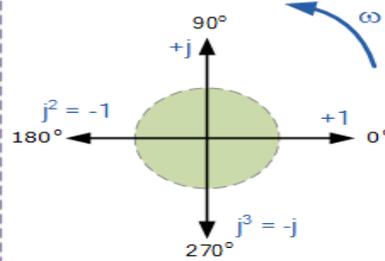
So by multiplying an imaginary number by j2 will rotate the vector by 180o anticlockwise, multiplying by j3 rotates it 270o and by j4 rotates it 360o or back to its original position. Multiplication by j10 or by j30 will cause the vector to rotate anticlockwise by the appropriate amount. In each successive rotation, the magnitude of the vector always remains the same.

In Electrical Engineering there are different ways to represent a complex number either graphically or mathematically. One such way that uses the cosine and sine rule is called the Cartesian or Rectangular Form.

Complex Numbers using the Rectangular Form

In the last tutorial about phasors, we saw that a complex number is represented by a real part and an imaginary part that takes the generalised form of:

$$Z = x + jy$$

Where:

Z - is the Complex Number representing the Vector

x - is the Real part or the Active component

y - is the Imaginary part or the Reactive component

j - is defined by $\sqrt{-1}$

In the rectangular form, a complex number can be represented as a point on a two dimensional plane called the complex or s-plane. So for example, $Z = 6 + j4$ represents a single point whose coordinates represent 6 on the horizontal real axis and 4 on the vertical imaginary axis as shown.

Complex Numbers using the Complex or s-plane
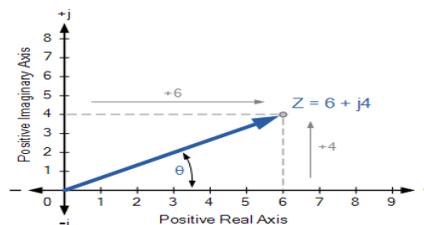
But as both the real and imaginary parts of a complex number in the rectangular form can be either a positive number or a negative number, then both the real and imaginary axis must also extend in both the positive and negative directions. This then produces a complex plane with four quadrants called an Argand Diagram as shown above.

Four Quadrant Argand Diagram
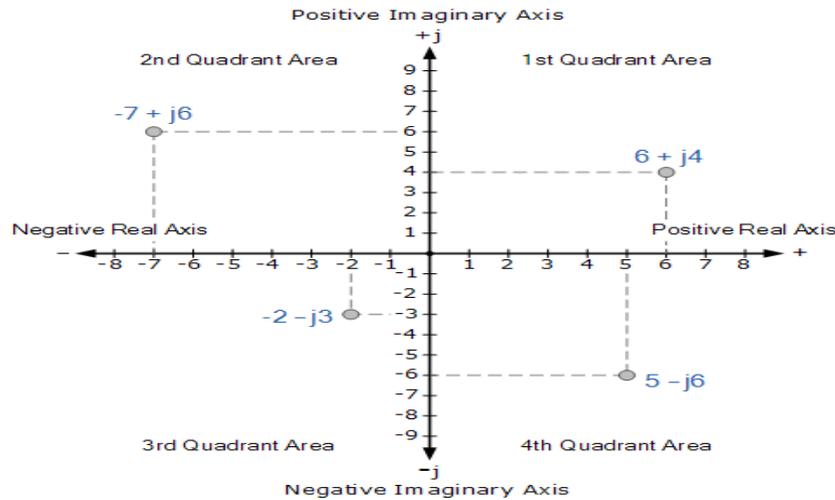
On the Argand diagram, the horizontal axis represents all positive real numbers to the right of the vertical imaginary axis and all negative real numbers to the left of the vertical imaginary axis. All positive imaginary numbers are represented above the horizontal axis while all the negative imaginary numbers are below the horizontal real axis. This then produces a two dimensional complex plane with four distinct quadrants labelled, QI, QII, QIII, and QIV.

The Argand diagram above can also be used to represent a rotating phasor as a point in the complex plane whose radius is given by the magnitude of the phasor will draw a full circle around it for every $2\pi/\omega$ seconds.

Then we can extend this idea further to show the definition of a complex number in both the polar and rectangular form for rotations of 90o.

$$0° = ±360° = +1 = 1\angle 0° = 1 + j0$$

$$+90° = +\sqrt{-1} = +j = 1\angle +90° = 0 + j1$$

$$-90° = -\sqrt{-1} = -j = 1\angle -90° = 0 - j1$$

$$±180° = \left(\sqrt{-1}\right)^2 = -1 = 1\angle ±180° = -1 + j0$$

Complex Numbers can also have "zero" real or imaginary parts such as: Z = 6 + j0 or Z = 0 + j4. In this case the points are plotted directly onto the real or imaginary axis. Also, the angle of a complex number can be calculated using simple trigonometry to calculate the angles of right-angled triangles, or measured anti-clockwise around the Argand diagram starting from the positive real axis.

Then angles between 0 and 90o will be in the first quadrant (I), angles (θ) between 90 and 180o in the second quadrant (II). The third quadrant (III) includes angles between 180 and 270o while

the fourth and final quadrant (IV) which completes the full circle, includes the angles between 270 and 360o and so on. In all the four quadrants the relevant angles can be found from:

tan-1(imaginary component ÷ real component)

Addition and Subtraction of Complex Numbers:

The addition or subtraction of complex numbers can be done either mathematically or graphically in rectangular form. For addition, the real parts are firstly added together to form the real part of the sum, and then the imaginary parts to form the imaginary part of the sum and this process is as follows using two complex numbers A and B as examples.

Complex Addition and Subtraction

$$A = x + jy \qquad B = w + jz$$

$$A + B = (x + w) + j(y + z)$$

$$A - B = (x - w) + j(y - z)$$

Complex Numbers Example No1

Two vectors are defined as, $A = 4 + j1$ and $B = 2 + j3$ respectively. Determine the sum and difference of the two vectors in both rectangular ($a + jb$) form and graphically as an Argand Diagram.

Mathematical Addition and Subtraction

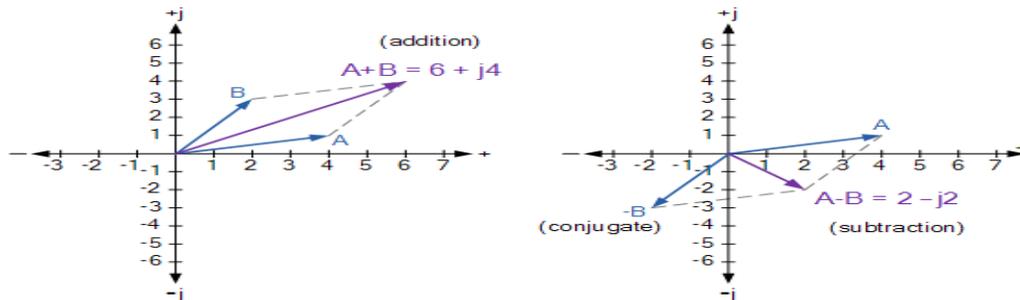Addition

$$A + B = (4 + j1) + (2 + j3)$$

$$A + B = (4 + 2) + j(1 + 3) = 6 + j4$$

Subtraction

$$A - B = (4 + j1) - (2 + j3)$$

$$A + B = (4 - 2) + j(1 - 3) = 2 - j2$$

Graphical Addition and Subtraction

Multiplication and Division of Complex Numbers

The multiplication of complex numbers in the rectangular form follows more or less the same rules as for normal algebra along with some additional rules for the successive multiplication of the j-operator where: $j2 = -1$. So for example, multiplying together our two vectors from above of $A = 4 + j1$ and $B = 2 + j3$ will give us the following result.

$$A \times B = (4 + j1)(2 + j3)$$

$$= 8 + j12 + j2 + j^2 3$$

$$\text{but } j^2 = -1,$$

$$= 8 + j14 - 3$$

$$A \times B = 5 + j14$$

Mathematically, the division of complex numbers in rectangular form is a little more difficult to perform as it requires the use of the denominators conjugate function to convert the denominator of the equation into a real number. This is called "rationalising". Then the division of complex numbers is best carried out using "Polar Form", which we will look at later. However, as an example in rectangular form lets find the value of vector A divided by vector B.

$$\frac{A}{B} = \frac{4 + j1}{2 + j3}$$

Multiply top & bottom by Conjugate of $2 + j3$

$$\frac{4 + j1}{2 + j3} \times \frac{2 - j3}{2 - j3} = \frac{8 - j12 + j2 - j^2 3}{4 - j6 + j6 - j^2 9}$$

$$= \frac{8 - j10 + 3}{4 + 9} = \frac{11 - j10}{13}$$

$$= \frac{11}{13} + \frac{-j10}{13} = 0.85 - j0.77$$

The Complex Conjugate

The **Complex Conjugate**, or simply **Conjugate** of a complex number is found by reversing the algebraic sign of the complex numbers imaginary number only while keeping the algebraic sign of the real number the same and to identify the complex conjugate of z the symbol z is used. For example, the conjugate of $z = 6 + j4$ is $z = 6 - j4$, likewise the conjugate of $z = 6 - j4$ is $z = 6 + j4$.

The points on the Argand diagram for a complex conjugate have the same horizontal position on the real axis as the original complex number, but opposite vertical positions. Thus, complex conjugates can be thought of as a reflection of a complex number. The following example shows a complex number, $6 + j4$ and its conjugate in the complex plane.

Conjugate Complex Numbers
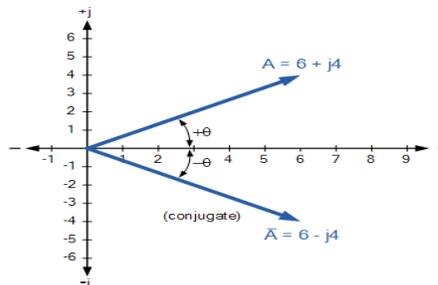
The sum of a complex number and its complex conjugate will always be a real number as we have seen above. Then the addition of a complex number and its conjugate gives the result as a real number or active component only, while their subtraction gives an imaginary number or reactive

145

component only. The conjugate of a complex number is an important element used in Electrical Engineering to determine the apparent power of an AC circuit using rectangular form.

Complex Numbers using Polar Form

Unlike rectangular form which plots points in the complex plane, the **Polar Form** of a complex number is written in terms of its magnitude and angle. Thus, a polar form vector is presented as: $Z = A \angle \pm \theta$, where: Z is the complex number in polar form, A is the magnitude or modulo of the vector and θ is its angle or argument of A which can be either positive or negative. The magnitude and angle of the point still remains the same as for the rectangular form above, this time in polar form the location of the point is represented in a "triangular form" as shown below.

Polar Form Representation of a Complex Number
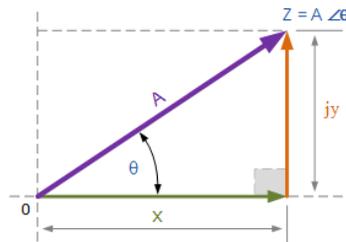
As the polar representation of a point is based around the triangular form, we can use simple geometry of the triangle and especially trigonometry and Pythagoras's Theorem on triangles to find both the magnitude and the angle of the complex number. As we remember from school, trigonometry deals with the relationship between the sides and the angles of triangles so we can describe the relationships between the sides as:

$$A^2 = x^2 + y^2$$

$$A = \sqrt{x^2 + y^2}$$

$$\text{Also,} \quad x = A.\cos\theta, \quad y = A.\sin\theta$$

Using trigonometry again, the angle θ of A is given as follows.

$$\theta = \tan^{-1}\frac{y}{x}$$

Then in Polar form the length of A and its angle represents the complex number instead of a point. Also in polar form, the conjugate of the complex number has the same magnitude or modulus it is the sign of the angle that changes, so for example the conjugate of $6 \angle 30^o$ would be $6 \angle -30^o$.

Converting between Rectangular Form and Polar Form In the rectangular form we can express a vector in terms of its rectangular coordinates, with the horizontal axis being its real axis and the vertical axis being its imaginary axis or j-component. In polar form these real and imaginary axes are simply represented by "A $\angle \theta$". Then using our example above, the relationship between rectangular form and polar form can be defined as.

Converting Polar Form into Rectangular Form, (P→R)

$$6\angle30^o = x + jy$$

However,

$$x = A.\cos\theta \quad y = A.\sin\theta$$

Therefore,

$$6\angle30^0 = (6\cos\theta) + j(6\sin\theta)$$
$$= (6\cos30^o) + j(6\sin30^o)$$
$$= (6 \times 0.866) + j(6 \times 0.5)$$
$$= 5.2 + j3$$

We can also convert back from rectangular form to polar form as follows.

Converting Rectangular Form into Polar Form, (R→P)

$$(5.2 + j3) = A\angle\theta$$

where: $A = \sqrt{5.2^2 + 3^2} = 6$

and $\theta = \tan^{-1}\dfrac{3}{5.2} = 30^o$

Hence, $(5.2 + j3) = 6\angle30^0$

Polar Form Multiplication and Division

Rectangular form is best for adding and subtracting complex numbers as we saw above, but polar form is often better for multiplying and dividing. To multiply together two vectors in polar form, we must first multiply together the two modulus or magnitudes and then add together their angles.

Multiplication in Polar Form

$$Z_1 \times Z_2 = A_1 \times A_2 \angle\theta_1 + \theta_2$$

Multiplying together 6 ∠30o and 8 ∠– 45o in polar form gives us.

$$Z_1 \times Z_2 = 6 \times 8 \angle30^o + (-45^o) = 48 \angle-15^o$$

Division in Polar Form

Likewise, to divide together two vectors in polar form, we must divide the two modulus and then subtract their angles as shown.

$$\frac{Z_1}{Z_2} = \left(\frac{A_1}{A_2}\right)\angle\theta_1 - \theta_2$$

$$\frac{Z_1}{Z_2} = \left(\frac{6}{8}\right)\angle30^o - (-45^o) = 0.75\angle75^o$$

Fortunately today's modern scientific calculators have built in mathematical functions (check your book) that allows for the easy conversion of rectangular to polar form, (R → P) and back from polar to rectangular form, (R → P).

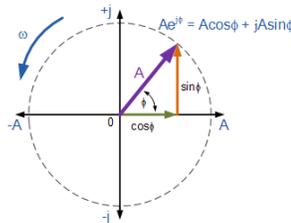**Complex Numbers using Exponential Form**

So far we have considered complex numbers in the **Rectangular Form**, (a + jb) and the **Polar Form**, (A ∠±θ). But there is also a third method for representing a complex number which is similar to the polar form that corresponds to the length (magnitude) and phase angle of the sinusoid but uses the base of the natural logarithm, e = 2.718 281.. to find the value of the complex number. This third method is called the **Exponential Form**.

The **Exponential Form** uses the trigonometric functions of both the sine (sin) and the cosine (cos) values of a right angled triangle to define the complex exponential as a rotating point in the complex plane. The exponential form for finding the position of the point is based around Euler's Identity, named after Swiss mathematician, Leonhard Euler and is given as:

$$Z = Ae^{j\phi}$$

$$Z = A\left(\cos\phi + j\sin\phi\right)$$

Then Euler's identity can be represented by the following rotating phasor diagram in the complex plane.

We can see that Euler's identity is very similar to the polar form above and that it shows us that a number such as Ae jθ which has a magnitude of 1 is also a complex number. Not only can we convert complex numbers that are in exponential form easily into polar form such as: 2e j30 = 2∠30, 10e j120 = 10∠120 or -6e j90 = -6∠90, but Euler's identity also gives us a way of converting a complex number from its exponential form into its rectangular form. Then the relationship between, Exponential, Polar and Rectangular form in defining a complex number is given as.

Complex Number Forms

$$Z = x + jy = A\angle\theta = A\left(\cos\phi + j\sin\phi\right)$$

Phasor Notation

So far we have look at different ways to represent either a rotating vector or a stationary vector using complex numbers to define a point on the complex plane. Phasor notation is the process of constructing a single complex number that has the amplitude and the phase angle of the given sinusoidal waveform.

Then phasor notation or phasor transform as it is sometimes called, transfers the real part of the sinusoidal function: A(t) = Am cos(ωt ± Φ) from the time domain into the complex number domain which is also called the frequency domain. For example:

$$V_{(t)} = V_m \cos\left(\omega t + \theta\right) \quad \Leftrightarrow \quad \text{Euler's identity: } e^{\pm j\theta} = \cos\theta \pm j\sin\theta$$

$$V_{(t)} = 20\cos\left(\omega t + 30^\circ\right) \text{Volts} \Rightarrow \left(\frac{20}{\sqrt{2}}\right)\angle 30^\circ \Rightarrow V_{RMS} = 14.14\angle 30^\circ$$

or

$$V_{(t)} = 35\cos\left(\omega t + 45^\circ\right) \text{Volts} \Rightarrow \left(\frac{35}{\sqrt{2}}\right)\angle 45^\circ \Rightarrow V_{RMS} = 24.75\angle 45^\circ$$
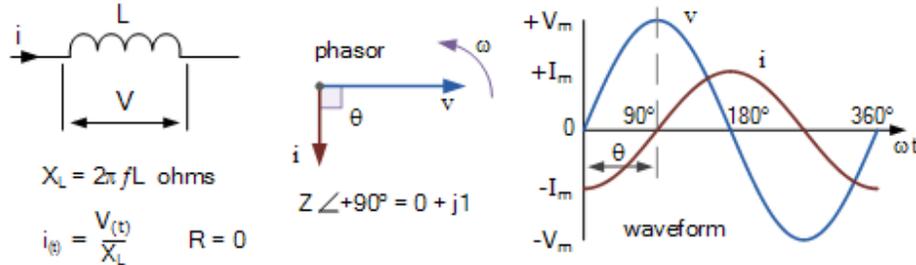
or

$$V_{(t)} = 10\cos\left(\omega t - 30^\circ\right) \text{Volts} \Rightarrow \left(\frac{10}{\sqrt{2}}\right)\angle\text{-}30^\circ \Rightarrow V_{RMS} = 7.07\angle\text{-}30^\circ$$

or

$$V_{(t)} = 100\cos\left(\omega t\right) \text{Volts} \Rightarrow \left(\frac{100}{\sqrt{2}}\right)\angle 0^\circ \Rightarrow V_{RMS} = 70.72\angle 0^\circ$$

Please note that the √2 converts the maximum amplitude into an effective or RMS value with the phase angle given in radians, (ω).

Purely Inductive Circuit

The waveforms above show us the instantaneous voltage and instantaneous current across a purely inductive coil as a function of time. Maximum current, I_m occurs at one full quarter of a cycle (90°) after the maximum (peak) value of the voltage. Here the current is shown with its negative maximum value at the start of the voltage cycle and passes through zero increasing to its positive maximum value when the voltage waveform is at its maximum value at 90°.

Thus as the voltage and current waveforms are no longer rising and falling together, but instead a phase shift of 90° ($\pi/2$) is introduced in the coil, then the voltage and current waveforms are "out-of-phase" with each other as the voltage leads the current by 90°. Since the phase difference between the voltage waveform and the current waveform is 90o, then the phase angle resulting in cos 90° = 0.

Therefore the electrical power consumed by a pure inductor, Q_L is given by:

Real Power in a Pure Inductor

$$P = V \times I \cos\theta$$

$$\cos\left(90^o\right) = 0$$

$$\therefore P = V \times I \times 0 = 0 \ \ (watts)$$

Clearly then, a pure inductor does not consume or dissipate any real or true power, but as we have both voltage and current the use of cosθ in the expression: P = VI cosθ for a pure inductor is no longer valid. The product of the current and the voltage in this case is imaginary power, commonly called "Reactive Power", (Q) measured in volt-amperes reactive, (VAr), Kilo-volt-amperes reactive (KVAr), etc.

Volt-amperes reactive, VAr should not be confused with watts, W which is used for real power. VAr represents the product of the volts and amperes that are 90° out-of-phase with each other. To identify the reactive average power mathematically, the sine function is used. Then the equation for the average reactive power in an inductor becomes:
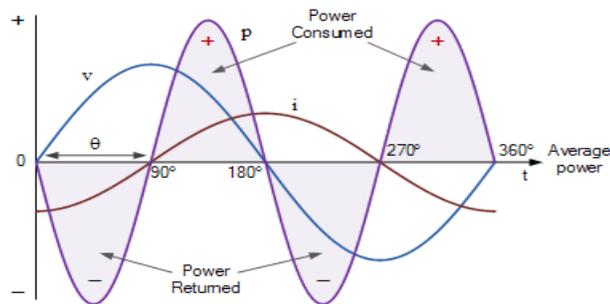
Reactive Power in a Pure Inductor

$$Q_L = V \times I \sin\theta$$

$$\sin\left(+90^\circ\right) = +1$$

$$\therefore Q_L = V \times I \times +1 = V \times I \quad (VAr)$$

Like real power, P, reactive power, Q also depends on voltage and current, but also the phase angle between them. It is therefore the product of the applied voltage and the component part of the current which is 90o out-of-phase with the voltage as shown.

AC Power Waveforms for a Pure Inductor

In the positive half of the voltage waveform between the angle of 0o and 90o, the inductor current is negative while the supply voltage is positive. Therefore, the volts and ampere product gives a negative power as negative times a positive equals a negative. Between 90o and 180o, both current and voltage waveforms are positive in value resulting in positive power. This positive power indicates that the coil is consuming electrical energy from the supply.

In the negative half of the voltage waveform between 180° and 270°, there is a negative voltage and positive current indicating a negative power. This negative power indicates that the coil is returning the stored electrical energy back to the supply. Between 270° and 360°, both the inductors current and the supply voltage are both negative resulting in a period of positive power.

Then during one full-cycle of the voltage waveform we have two identical positive and negative pulses of power whose average value is zero so no real power is used up since the power alternately flows to and from the source. This means then that the total power taken by a pure inductor over one full-cycle is zero, so an inductors reactive power does not perform any real work.

Electrical Power Example

Example-1

A solenoid coil with a resistance of 30 ohms and an inductance of 200mH is connected to a 230VAC, 50Hz supply. Calculate: (a) the solenoids impedance, (b) the current consumed by the solenoid, (c) the phase angle between the current and the applied voltage, and (d) the average power consumed by the solenoid.

230V 50Hz
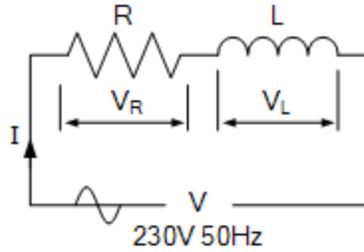
Solution:

Data given: $R = 30\Omega$, $L = 200mH$, $V = 230V$ and $f = 50Hz$.

    (a) Impedance (Z) of the solenoid coil:

$$R = 30\Omega$$

$$X_L = 2\pi fL = 2\pi \times 50 \times 200 \times 10^{-3} = 62.8\Omega$$

$$Z = \sqrt{R^2 + X_L^2} = \sqrt{30^2 + 62.8^2} = 69.6\Omega$$

    (b) Current (I) consumed by the solenoid coil:

$$V = I \times Z$$

$$\therefore I = \frac{V}{Z} = \frac{230}{69.6} = 3.3\,A_{(rms)}$$

    (c) The phase angle, θ:

$$\cos\theta = \frac{R}{Z}, \quad or \quad \sin\theta = \frac{X_L}{Z}, \quad or \quad \tan\theta = \frac{X_L}{R}$$

$$\therefore \cos\theta = \frac{R}{Z} = \frac{30}{69.6} = 0.431$$

$$\cos^{-1}(0.431) = 64.5^{\circ}\ lagging$$

    (d) Average AC power consumed by the solenoid coil:

$$P = V \times I \times \cos\theta$$

$$P = 230 \times 3.3 \times \cos\left(64.5^{\circ}\right)$$

$$\therefore P = 327\ watts$$

Example-2

Three coils of resistances 20, 30 and 40 W and inductance 0.5, 0.3 and 0.2*H*, respectively are connected in series across a 230 V, 50 c/s supply. Calculate the total current, power factor and the power consumed in the circuit.

Solution:

Total resistance $R = 20 + 30 + 40 = 90$ W

Total resistance $L = 0.5 + 0.3 + 0.2 = 1.0$ W

$X_L = 2\pi fL = 2\pi *50* 1.0 = 314\ \Omega$

Impedance $Z = R^2 + X_L^2$

$$= 90^2 + 314^2 = 327\ \Omega$$

Current $I = \dfrac{E}{Z} = \dfrac{230}{327} = 0.704$ A

$$power\ factor\ \cos\varphi = \frac{R}{Z} = \frac{90}{327} = 0.275\ lagging$$

Power consumed $= EI \cos\varphi.$

$$= 230*0.704*0.275 = 44.5\ watts.$$

Example 3:-
In an $R - L$ series circuit $R = 10$ W and $X_L = 8.66$ W if current in the circuit is $(5 - j\ 10)$ A, find (i) the applied voltage (ii) power factor and (iii) active power and reactive power.
Solution:

$Z = R + jX_L = (10 + j\ 8.66)$ W $= 13.23 \angle 40.9°$ W
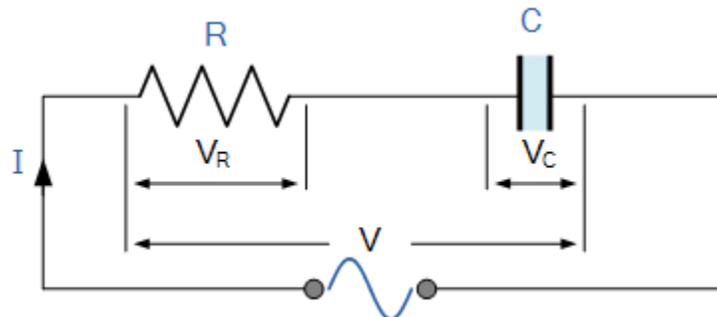$I = (5 - j\ 10)A = 11.18 \angle -63.43°$ A

(i) $V = IZ = 11.18 \angle -63.43° ´ 13.23 \angle 40.9° = 148 \angle -22.53°$ V
$V = 148$ Volts.

(ii) $\Phi = 63.43° - 22.53° = 40.9°$

p.f. $= \cos\Phi = \cos 40.9° = 0.756$ lag.

(iii) S = phasor voltage * conjugate of phasor current.

or $P + jQ = 148 \angle -22.53° * 11.18 \angle 63.43° = 1654.64 \angle 40.9°$ VA.

$= (1250.66 + j\ 1083.36)$ VA

Active power, P $= 1250.66$ W

Reactive power Q $= 1083.36$ VAr.

Series Resistance-Capacitance Circuit

In the RC series circuit above, we can see that the current flowing into the circuit is common to both the resistance and capacitance, while the voltage is made up of the two component voltages, V_R and V_C. The resulting voltage of these two components can be found mathematically but since vectors V_R and V_C are $90°$ out-of-phase, they can be added vectorially by constructing a vector diagram.

To be able to produce a vector diagram for an AC capacitance a reference or common component must be found. In a series AC circuit the current is common and can therefore be used as the reference source because the same current flows through the resistance and into the capacitance. The individual vector diagrams for a pure resistance and a pure capacitance are given as:

Vector Diagrams for the Two Pure Components
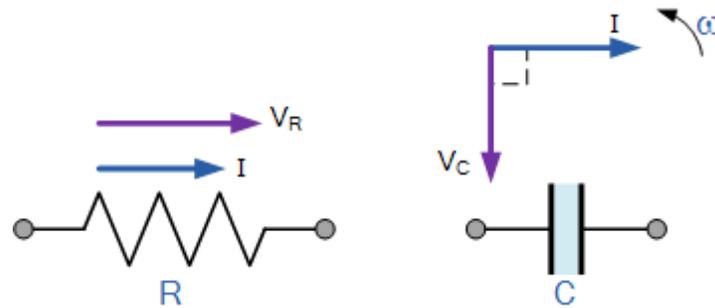
Both the voltage and current vectors for an AC Resistance are in phase with each other and therefore the voltage vector V_R is drawn superimposed to scale onto the current vector. Also we know that the current leads the voltage (ICE) in a pure AC capacitance circuit, therefore the voltage vector V_C is drawn $90°$ behind (lagging) the current vector and to the same scale as V_R as shown.

Vector Diagram of the Resultant Voltage
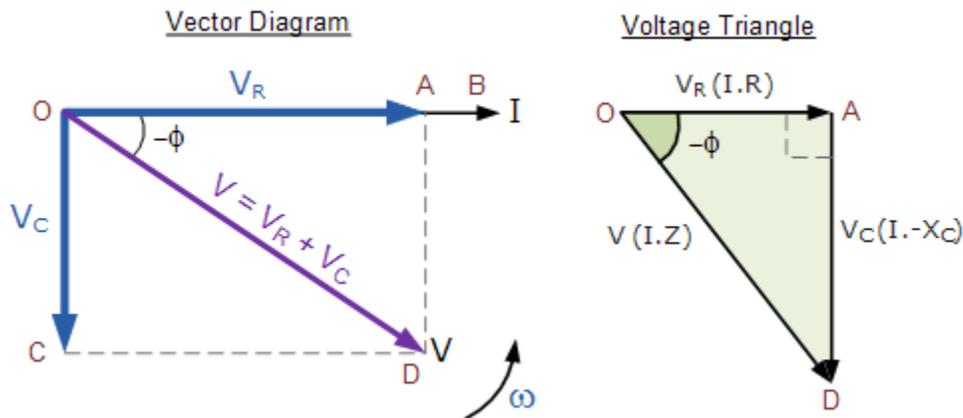
In the vector diagram above, line OB represents the horizontal current reference and line OA is the voltage across the resistive component which is in-phase with the current. Line OC shows the capacitive voltage which is $90°$ behind the current therefore it can still be seen that the current leads the purely capacitive voltage by $90°$. Line OD gives us the resulting supply voltage.

As the current leads the voltage in a pure capacitance by 90° the resultant phasor diagram drawn from the individual voltage drops V_R and V_C represents a right angled voltage triangle shown above as OAD. Then we can also use Pythagoras theorem to mathematically find the value of this resultant voltage across the resistor/capacitor (RC) circuit.

As V_R = I.R and V_C = I.X_C the applied voltage will be the vector sum of the two as follows.

$$V^2 = V_R^2 + V_C^2$$

$$V_R = I.R \quad \text{and} \quad V_C = I.X_C$$

$$V^2 = I^2.R^2 + I^2.X_C^2$$

$$V = \sqrt{(I.R)^2 + (I.X_C)^2}$$

$$\therefore I = \frac{V}{\sqrt{R^2 + X_C^2}} = \frac{V}{Z}$$

The quantity $\sqrt{R^2 + X_C^2}$ represents the **impedance**, Z of the circuit.

The Impedance of an AC Capacitance

Impedance, **Z** which has the units of Ohms, Ω's is the "TOTAL" opposition to current flowing in an AC circuit that contains both Resistance, (the real part) and Reactance (the imaginary part). A purely resistive impedance will have a phase angle of 0° while a purely capacitive impedance will have a phase angle of -90°.

However when resistors and capacitors are connected together in the same circuit, the total impedance will have a phase angle somewhere between 0° and 90° depending upon the value of the components used. Then the impedance of our simple RC circuit shown above can be found by using the impedance triangle.

The RC Impedance Triangle

$$\text{Impedance, } Z = \frac{V}{I}$$
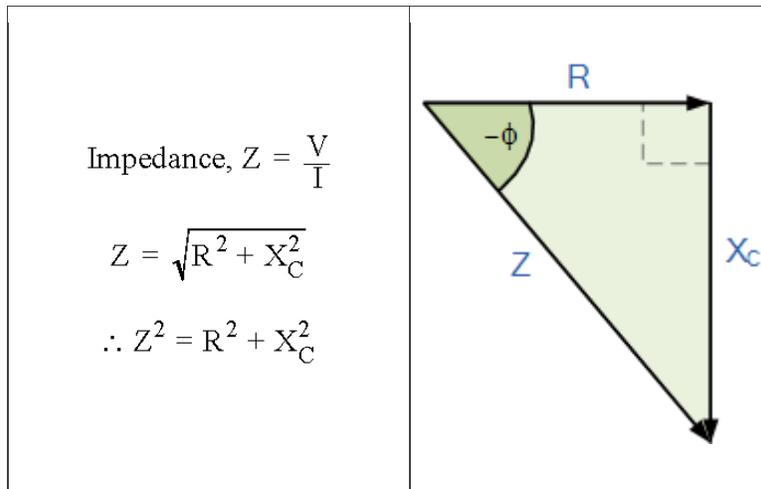
$$Z = \sqrt{R^2 + X_C^2}$$

$$\therefore Z^2 = R^2 + X_C^2$$

Then: (Impedance)^2 = (Resistance)^2 + (j Reactance)^2 where j represents the 90°phase shift.

This means then by using Pythagoras theorem the negative phase angle, θ between the voltage and current is calculated as.
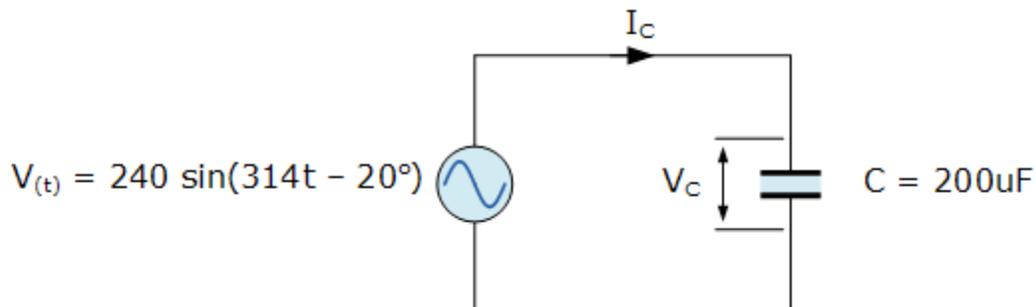
Phase Angle

$$Z^2 = R^2 + X_C^2$$

$$\cos\phi = \frac{R}{Z}, \quad \sin\phi = \frac{X_C}{Z}, \quad \tan\phi = \frac{X_C}{R}$$

AC Capacitance Example No1

A single-phase sinusoidal AC supply voltage defined as: $V_{(t)}$ = 240 sin(314t - 20°) is connected to a pure AC capacitance of 200uF. Determine the value of the current flowing into the capacitor and draw the resulting phasor diagram.


The voltage across the capacitor will be the same as the supply voltage. Converting this time domain value into polar form gives us: V_C = 240 ∠-20° (v). The capacitive reactance will be: X_C = 1/(ω.200uF). Then the current flowing into the capacitor can be found using Ohms law as:

$$X_C = \frac{1}{j\omega C} = \frac{1}{314 \times 200\mu F} = 16\angle\text{-}90°$$

$$I_C = \frac{V_C}{jX_C} = \frac{240\angle\text{-}20°}{16\angle\text{-}90°} = 15\angle 70° \ (A)$$

With the current leading the voltage by 90° in an AC capacitance circuit the phasor diagram will be.
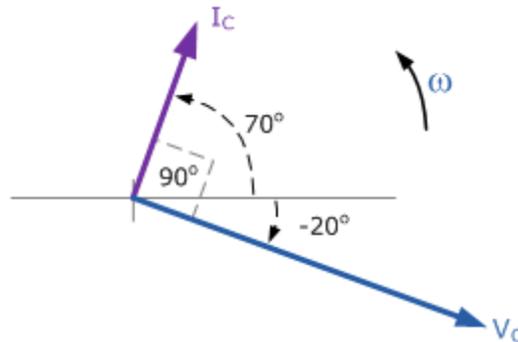
AC Capacitance

Example No2

A capacitor which has an internal resistance of 10Ω's and a capacitance value of 100uF is connected to a supply voltage given as $V_{(t)} = 100 \sin (314t)$. Calculate the current flowing into the capacitor. Also construct a voltage triangle showing the individual voltage drops.
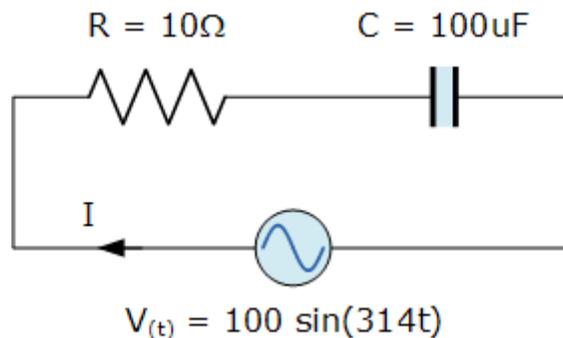
The capacitive reactance and circuit impedance is calculated as:

$$X_C = \frac{1}{\omega C} = \frac{1}{314 \times 100uF} = 31.85\Omega$$

$$Z = \sqrt{R^2 + X_C^2} = \sqrt{10^2 + 31.85^2} = 33.4\Omega$$

Then the current flowing into the capacitor and the circuit is given as:

$$I = \frac{V_C}{Z} = \frac{100}{33.4} = 3\text{Amps}$$

The phase angle between the current and voltage is calculated from the impedance triangle above as:

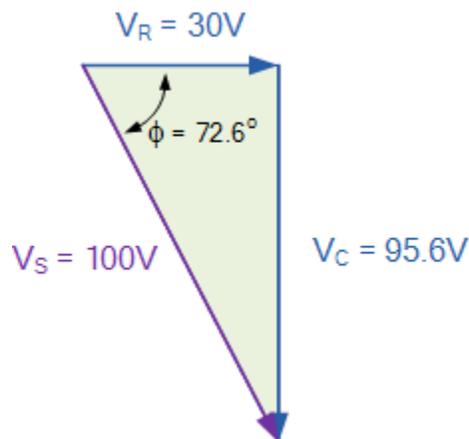$$\phi = \tan^{-1}\left(\frac{X_C}{R}\right) = \frac{31.85}{10} = 72.6° \text{ leading}$$

Then the individual voltage drops around the circuit are calculated as:

$$V_R = I \times R = 3 \times 10 = 30V$$

$$V_C = I \times X_C = 3 \times 31.85 = 95.6V$$

$$V_S = \sqrt{V_R^2 + V_C^2} = \sqrt{30^2 + 95.6^2} = 100V$$

Then the resultant voltage triangle will be.

AC Capacitance Summary

In a pure **AC Capacitance** circuit, the voltage and current are both "out-of-phase" with the current leading the voltage by 90° and we can remember this by using the mnemonic expression "ICE". The AC resistive value of a capacitor called impedance, (Z) is related to frequency with the reactive value of a capacitor called "capacitive reactance", X_C. In an AC Capacitance circuit, this capacitive reactance value is equal to $1/(2\pi f C)$ or $1/(j\omega C)$. Thus far we have seen that the relationship between voltage and current is not the same and changes in all three pure passive components. In the *Resistance* the phase angle is 0°, in the *Inductance* it is +90° while in the *Capacitance* it is -90°.

157
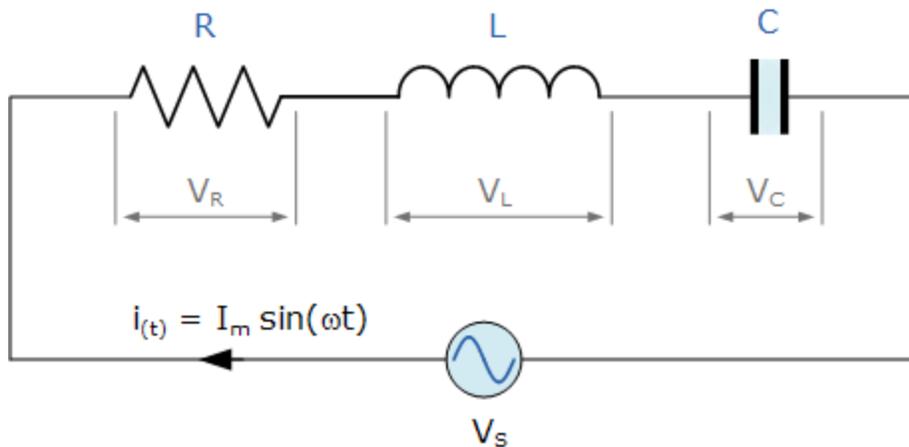
Series RLC Circuit

The series RLC circuit above has a single loop with the instantaneous current flowing through the loop being the same for each circuit element. Since the inductive and capacitive reactance's X_L and X_C are a function of the supply frequency, the sinusoidal response of a series RLC circuit will therefore vary with frequency, f. Then the individual voltage drops across each circuit element of R, L and C element will be "out-of-phase" with each other as defined by:

$$i(t) = I_{max} \sin(\omega t)$$

The instantaneous voltage across a pure resistor, V_R is "in-phase" with current

The instantaneous voltage across a pure inductor, V_L "leads" the current by 90°

The instantaneous voltage across a pure capacitor, V_C "lags" the current by 90°

Therefore, V_L and V_C are 180° "out-of-phase" and in opposition to each other.

For the series RLC circuit above, this can be shown as:
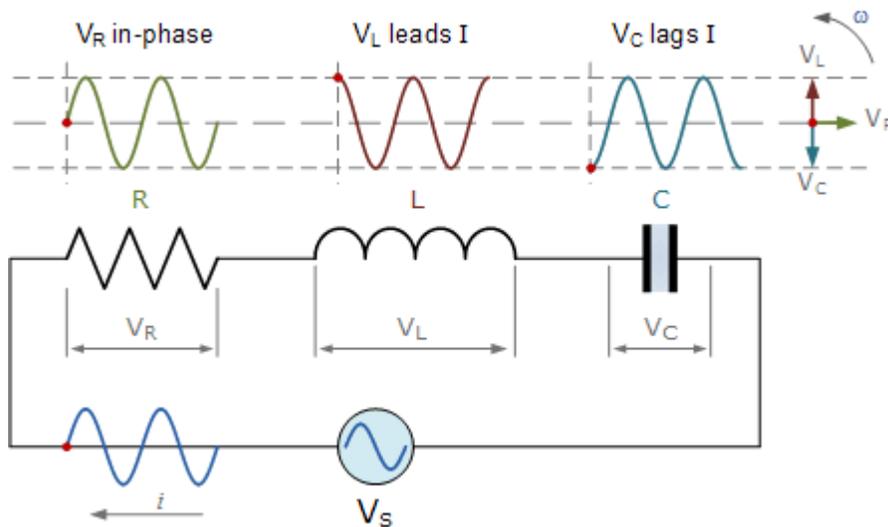
The amplitude of the source voltage across all three components in a series RLC circuit is made up of the three individual component voltages, V_R, V_L and V_C with the current common to all three components. The vector diagrams will therefore have the current vector as their reference with the three voltage vectors being plotted with respect to this reference as shown below.

Individual Voltage Vectors
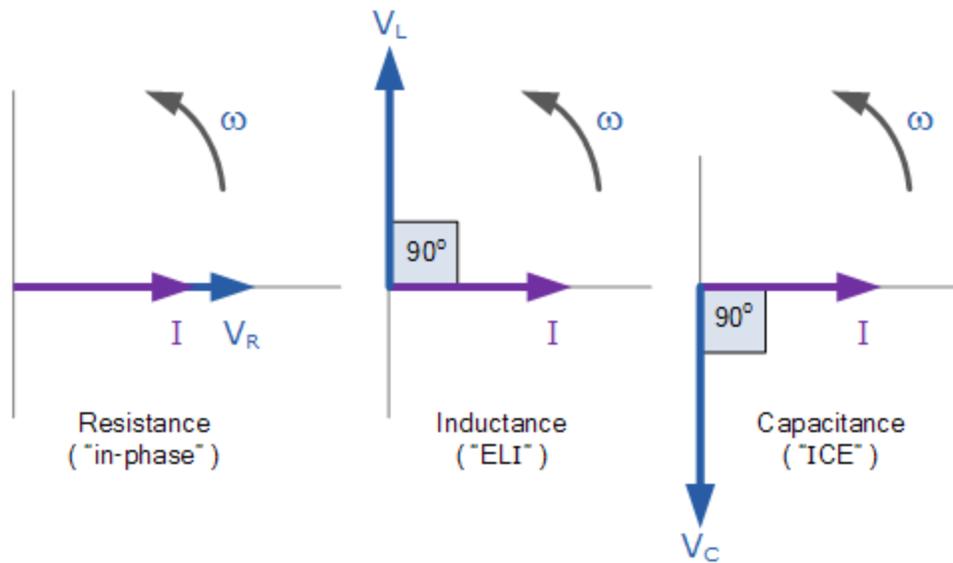
This means then that we can not simply add together V_R, V_L and V_C to find the supply voltage, V_S across all three components as all three voltage vectors point in different directions with regards to the current vector. Therefore we will have to find the supply voltage, V_S as the **Phasor Sum** of the three component voltages combined together vectorially.

Kirchoff's voltage law (KVL) for both loop and nodal circuits states that around any closed loop the sum of voltage drops around the loop equals the sum of the EMF's. Then applying this law to the these three voltages will give us the amplitude of the source voltage, V_S as.

Instantaneous Voltages for a Series RLC Circuit

$$\text{KVL:} \quad V_S - V_R - V_L - V_C = 0$$

$$V_S - IR - L\frac{di}{dt} - \frac{Q}{C} = 0$$

$$\therefore V_S = IR + L\frac{di}{dt} + \frac{Q}{C}$$

The phasor diagram for a series RLC circuit is produced by combining together the three individual phasors above and adding these voltages vectorially. Since the current flowing through the circuit is common to all three circuit elements we can use this as the reference vector with the three voltage vectors drawn relative to this at their corresponding angles.

The resulting vector V_S is obtained by adding together two of the vectors, V_L and V_C and then adding this sum to the remaining vector V_R. The resulting angle obtained between V_S and i will be the circuits phase angle as shown below.

We can see from the phasor diagram on the right hand side above that the voltage vectors produce a rectangular triangle, comprising of hypotenuse V_S, horizontal axis V_R and vertical axis $V_L - V_C$ Hopefully you will notice then, that this forms our old favourite the **Voltage Triangle** and we can therefore use Pythagoras's theorem on this voltage triangle to mathematically obtain the value of V_S as shown.

Voltage Triangle for a Series RLC Circuit

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

$$V_S = \sqrt{V_R^2 + (V_L - V_C)^2}$$

Please note that when using the above equation, the final reactive voltage must always be positive in value, that is the smallest voltage must always be taken away from the largest voltage we can not have a negative voltage added to V_R so it is correct to have $V_L - V_C$ or $V_C - V_L$. The smallest value from the largest otherwise the calculation of V_S will be incorrect.

We know from above that the current has the same amplitude and phase in all the components of a series RLC circuit. Then the voltage across each component can also be described mathematically according to the current flowing through, and the voltage across each element as.

$$V_R = iR \sin(\omega t + 0°) = i.R$$

$$V_L = iX_L \sin(\omega t + 90°) = i.j\omega L$$

$$V_C = iX_C \sin(\omega t - 90°) = i.\frac{1}{j\omega C}$$

By substituting these values into Pythagoras's equation above for the voltage triangle will give us:

$$V_R = I.R \qquad V_L = I.X_L \qquad V_C = I.X_C$$

$$V_S = \sqrt{(I.R)^2 + (I.X_L - I.X_C)^2}$$

$$V_S = I.\sqrt{R^2 + (X_L - X_C)^2}$$

$$\therefore V_S = I \times Z \qquad \text{where: } Z = \sqrt{R^2 + (X_L - X_C)^2}$$

So we can see that the amplitude of the source voltage is proportional to the amplitude of the current flowing through the circuit. This proportionality constant is called the **Impedance** of the circuit which ultimately depends upon the resistance and the inductive and capacitive reactance's.

Then in the series RLC circuit above, it can be seen that the opposition to current flow is made up of three components, X_L, X_C and R with the reactance, X_T of any series RLC circuit being defined as: $X_T = X_L - X_C$ or $X_T = X_C - X_L$ with the total impedance of the circuit being thought of as the voltage source required to drive a current through it.

The Impedance of a Series RLC Circuit

As the three vector voltages are out-of-phase with each other, X_L, X_C and R must also be "out-of-phase" with each other with the relationship between R, X_L and X_C being the vector sum of these three components thereby giving us the circuits overall impedance, Z. These circuit impedance's can be drawn and represented by an **Impedance Triangle** as shown below.

The Impedance Triangle for a Series RLC Circuit
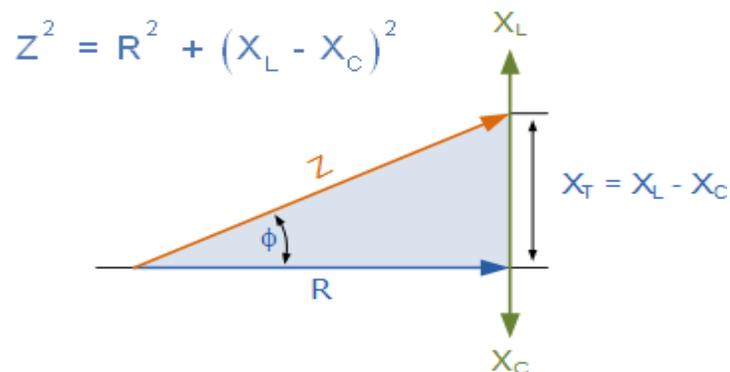
The impedance Z of a series RLC circuit depends upon the angular frequency, ω as do X_L and X_C If the capacitive reactance is greater than the inductive reactance, $X_C > X_L$ then the overall circuit reactance is capacitive giving a leading phase angle.

Likewise, if the inductive reactance is greater than the capacitive reactance, $X_L > X_C$ then the overall circuit reactance is inductive giving the series circuit a lagging phase angle. If the two reactance's

are the same and $X_L = X_C$ then the angular frequency at which this occurs is called the resonant frequency and produces the effect of **resonance** which we will look at in more detail in another tutorial.

Then the magnitude of the current depends upon the frequency applied to the series RLC circuit. When impedance, Z is at its maximum, the current is a minimum and likewise, when Z is at its minimum, the current is at maximum. So the above equation for impedance can be re-written as:

$$\text{Impedance, } Z = \sqrt{R^2 + \left(\omega L - \frac{1}{\omega C}\right)^2}$$

The phase angle, θ between the source voltage, V_S and the current, i is the same as for the angle between Z and R in the impedance triangle. This phase angle may be positive or negative in value depending on whether the source voltage leads or lags the circuit current and can be calculated mathematically from the ohmic values of the impedance triangle as:

$$\cos\phi = \frac{R}{Z} \qquad \sin\phi = \frac{X_L - X_C}{Z} \qquad \tan\phi = \frac{X_L - X_C}{R}$$

Real Power in AC Circuits

Real power (P), also known as true or active power, performs the "real work" within an electrical circuit. Real power, measured in watts, defines the power consumed by the resistive part of a circuit. Then real power, (P) in an AC circuit is the same as power, P in a DC circuit. So just like DC circuits, it is always calculated as I^2*R, where R is the total resistive component of the circuit.
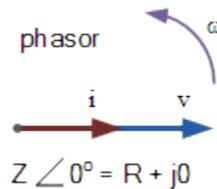

Figure voltage current vectors

As resistances do not produce any phasor difference (phase shift) between voltage and current waveforms, all the useful power is delivered directly to the resistance and converted to heat, light and work. Then the power consumed by a resistance is real power which is fundamentally the circuits average power. To find the corresponding value of the real power the rms voltage and current values are multiplied by the cosine of the phase angle, Φ as shown.

$$\text{Real Power} \quad P = I^2R = V*I*\cos(\Phi) \quad \text{Watts, (W)}$$

But as their is no phase difference between the voltage and the current in a resistive circuit, the phase shift between the two waveforms will be zero (0). Then:

Real Power in an AC Circuit

$$P = V_{rms} \times I_{rms} \times \cos\phi$$

$$\cos\left(0^{o}\right) = 1$$

$$P = V_{rms} \times I_{rms} \times 1$$

$$\therefore P = V_{rms} \times I_{rms} \quad (\text{Watts})$$

Where real power (P) is in watts, voltage (V) is in rms volts and current (I) is in rms amperes. Then real power is the I^2*R resistive element measured in watts, which is what you read on your utility energy

163

meter and has units in Watts (W), Kilowatts (kW), and Megawatts (MW). Note that real power, P is always positive.

Reactive Power in AC Circuits

Reactive power (Q), (sometimes called wattless power) is the power consumed in an AC circuit that does not perform any useful work but has a big effect on the phase shift between the voltage and current waveforms. Reactive power is linked to the reactance produced by inductors and capacitors and counteracts the effects of real power. Reactive power does not exist in DC circuits.
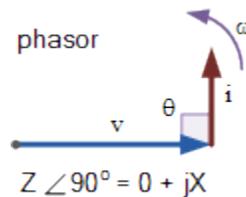

Figure voltage current vectors

Unlike real power (P) which does all the work, reactive power (Q) takes power away from a circuit due to the creation and reduction of both inductive magnetic fields and capacitive electrostatic fields, thereby making it harder for the true power to supply power directly to a circuit or load. The power stored by an inductor in its magnetic field tries to control the current, while the power stored by a capacitors electrostatic field tries to control the voltage. The result is that capacitors "generate" reactive power and inductors "consume" reactive power. This means that they both consume and return power to the source so none of the real power is consumed. To find reactive power, the rms voltage and current values are multiplied by the sine of the phase angle, Φ as shown.

$$\text{Reactive Power} \quad Q = I^2X = V*I*\sin(\Phi) \quad \text{volt-amperes reactive, (VAr's)}$$

As there is a 90° phase difference between the voltage and the current waveforms in a pure reactance (either inductive or capacitive), multiplying V*I by sin(Φ) gives a vertical component that is 90° out-of-phase with each other, so:

164

Reactive Power in an AC Circuit

$$Q = V_{rms} \times I_{rms} \times \sin\phi$$

$$\sin\left(90^o\right) = 1$$

$$Q = V_{rms} \times I_{rms} \times 1$$

$$\therefore Q = V_{rms} \times I_{rms} \quad (VAr)$$

Where reactive power (Q) is in volt-amperes reactive, voltage (V) is in rms volts and current (I) is in rms amperes. Then reactive power represents the product of volts and amperes that are 90° out-of-phase with each other, but in general, there can be any phase angle, Φ between the voltage and the current. Thus reactive power is the I^2X reactive element that has units in volt-amperes reactive (VAr), Kilovolt-amperes reactive (kVAr), and Megavolt-amperes reactive (MVAr).

Apparent Power in AC Circuits

We have seen above that real power is dissipated by resistance and that reactive power is supplied to a reactance. As a result of this the current and voltage waveforms are not in-phase due to the difference between a circuits resistive and reactive components. Then there is a mathematical relationship between the real power (P), and the reactive power (Q), called the complex power. The product of the rms voltage, V applied to an AC circuit and the rms current, I flowing into that circuit is called the "volt-ampere product" (VA) given the symbol S and whose magnitude is known generally as apparent power. This complex Power is not equal to the algebraic sum of the real and reactive powers added together, but is instead the vector sum of P and Q given in volt-amps (VA).

It is complex power that is represented by the power triangle. The rms value of the volt-ampere product is known more commonly as the apparent power as, "apparently" this is the total power consumed by a circuit even though the real power that does the work is a lot less. As apparent power is made up of two parts, the resistive power which is the in-phase power or real power in watts and the reactive power which is the out-of-phase power in volt-amperes, we can show the vector addition of these two power components in the form of a **power triangle**. A power triangle has four parts: P, Q, S and θ.

The three elements which make up power in an AC circuit can be represented graphically by the three sides of a right-angled triangle, in much the same way as the previous impedance triangle. The horizontal (adjacent) side represents the circuits real power (P), the vertical (opposite) side represents the circuits

reactive power (Q) and the hypotenuse represents the resulting apparent power (S), of the power triangle as shown.
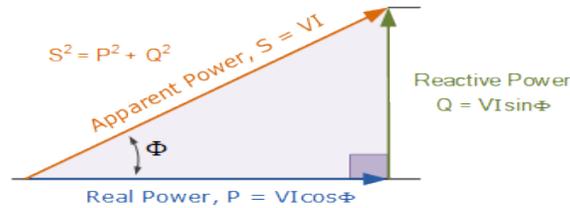
Power Triangle of an AC Circuit

Figure power triangle

- Where:

- **P** is the $I^2 *R$ or Real power that performs work measured in watts, W

- **Q** is the $I^2 *X$ or Reactive power measured in volt-amperes reactive, VAr

- **S** is the $I^2 *Z$ or Apparent power measured in volt-amperes, VA

- **Φ** is the phase angle in degrees. The larger the phase angle, the greater the reactive power

- Cos(Φ) = P/S = W/VA = power factor, p.f.

- Sin(Φ) = Q/S = VAr/VA

- Tan(Φ) = Q/P = VAr/W

The power factor is calculated as the ratio of the real power to the apparent power because this ratio equals cos(Φ).

Power Factor in AC Circuits

Power factor, cos(Φ), is an important part of an AC circuit that can also be expressed in terms of circuit impedance or circuit power. Power factor is defined as the ratio of real power (P) to apparent power (S), and is generally expressed as either a decimal value, for example 0.95, or as a percentage: 95%.

Power factor defines the phase angle between the current and voltage waveforms, were I and V are the magnitudes of rms values of the current and voltage. Note that it does not matter whether the phase angle is the difference of the current with respect to the voltage, or the voltage with respect to the current. The mathematical relationship is given as:

Power Factor of an AC Circuit

$$\text{Power Factor} = \frac{\text{watts}}{\text{volt-amperes}}$$

$$= \frac{P}{S} = \frac{VI\cos\phi}{VI} = \cos\phi$$

We said previously that in a pure resistive circuit, the current and voltage waveforms are in-phase with each other so the real power consumed is the same as the apparent power as the phase difference is zero degrees (0°). So the power factor will be:

$$\text{Power Factor, pf} = \cos 0^o = 1.0$$

That is the number of watts consumed is the same as the number of volt-amperes consumed producing a power factor of 1.0, or 100%. In this case it is referred to a unity power factor. We also said above that in a purely reactive circuit, the current and voltage waveforms are out-of-phase with each other by 90°. As the phase difference is ninety degrees (90°), the power factor will be:

$$\text{Power Factor, pf} = \cos 90^o = 0$$

That is the number of watts consumed is zero but there is still a voltage and current supplying the reactive load. Clearly then reducing the reactive VAr component of the power triangle will cause θ to reduce improving the power factor towards one, unity. It is also desirable to have a high power factor as this makes the most efficient use of the circuit delivering current to a load. Then we can write the relationship between the real power, the apparent power and the circuits power factor as:

$$\text{Real Power}(P) = \text{Apparant Power}(S) \times \text{Power Factor}(pf)$$

$$\text{Power Factor}(pf) = \frac{\text{Real Power}(P) \text{ in Watts}}{\text{Apparant Power}(S) \text{ in volt-amps}}$$

An inductive circuit where the current "lags" the voltage (ELI) is said to have a lagging power factor, and a capacitive circuit where the current "leads" the voltage (ICE) is said to have a leading power factor.

Power Triangle Example No1
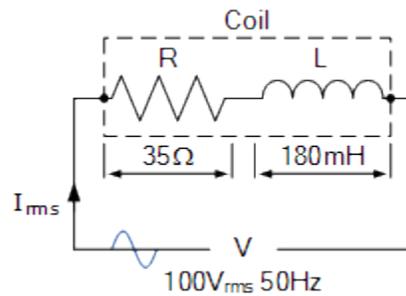
Figure circuit

A wound coil that has an inductance of 180mH and a resistance of 35Ω is connected to a 100V 50Hz supply. Calculate: a) the impedance of the coil, b) the current, c) the power factor, and d) the apparent power consumed. Also draw the resulting power triangle for the above coil.

Data given: R = 35Ω, L = 180mH, V = 100V and f = 50Hz.

(a) Impedance (Z) of the coil:

$$R = 35\Omega$$

$$X_L = 2\pi f L = 2\pi \times 50 \times 0.18 = 56.6\Omega$$

$$Z = \sqrt{R^2 + X_L^2} = \sqrt{35^2 + 56.6^2} = 66.5\Omega$$

(b) Current (I) consumed by the coil:

$$V = I \times Z$$

$$\therefore I = \frac{V}{Z} = \frac{100}{66.5} = 1.5\,A_{(rms)}$$

(c) The power factor and phase angle, Φ:

$$\cos\phi = \frac{R}{Z}, \quad \text{or} \quad \sin\phi = \frac{X_L}{Z}, \quad \text{or} \quad \tan\phi = \frac{X_L}{R}$$

$$\therefore \cos\phi = \frac{R}{Z} = \frac{35}{66.5} = 0.5263$$

$$\cos^{-1}(0.5263) = 58.2° \text{ (lagging)}$$

(d) Apparent power (S) consumed by the coil:

$$P = V \times I \cos\phi = 100 \times 1.5 \times \cos(58.2°) = 79W$$

$$Q = V \times I \sin\phi = 100 \times 1.5 \times \sin(58.2°) = 127.5VAr$$

$$S = V \times I = 100 \times 1.5 = 150VA$$

$$\text{or} \quad S^2 = P^2 + Q^2$$

$$\therefore S = \sqrt{P^2 + Q^2} = \sqrt{79^2 + 127.5^2} = 150VA$$

(e) Power triangle for the coil:

S = VI = 150VA

$Q_L = VI.\sin\phi$ = 127.5 VAr (lag)

Φ = 58.2°

P = VI.cosθ = 79.1W
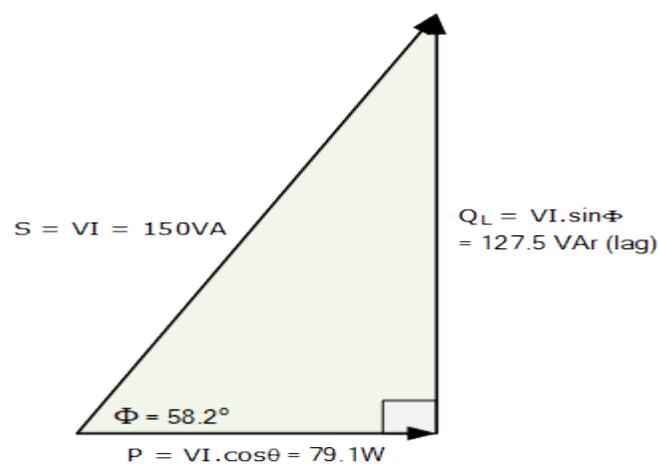
Figure power triangle

As the power triangle relationships of this simple example demonstrates, at 0.5263 or 52.63% power factor, the coil requires 150 VA of power to produce 79 Watts of useful work. In other words, at 52.63% power factor, the coil takes about 89% more current to do the same work, which is a lot of wasted current. Adding a power factor correction capacitor (for this example a 32.3uF) across the coil, in order to increase the power factor to over 0.95, or 95%, would greatly reduce the reactive power consumed by the coil as these capacitors act as reactive current generators, thus reducing the total amount of current consumed.

MODULE – V
NETWORK TOPOLOGY

Network topology is the one of the technique to solve electrical networks consisting of number of meshes or number of nodes, where it is difficult to apply mesh and nodal analysis. Graph theory is the technique where all the elements of the network are Represented by straight lines irrespective of their behavior. Here matrix methods are used to solve complex networks. Before seeing the actual matrices, the knowledge of some of the definitions is very important.

5.1 DEFINITONS:

Node: A node is junction where two or more than two elements are connected.

Degree of the node: number of elements connected to the node is defined as degree of the node.

Branch: A branch is a element(s) connected between pair of nodes.

Path: It is traversal of signal between pair of nodes.

Loop: It is the path started from an node and ends at the same node.

Graph: A graph is formed when all the elements of the network are replaced by straight line irrespective of their behaviour.

Oriented and non-oriented graph: If the graph of the network is represented with directions in each and every branch then it is oriented, if atleast one branch of graph has no direction then iti is non-oriented graph.

Planar and non-planar graph: if a graph can be on plane surface without cross over then system is planar and vice-versa is non-linear.

Tree: Tree is the sub-graph of the graph, which consists of same number of nodes as that of original graph without any closed path. Branches of the tree are called as twigs. Numbers of possible twigs in a tree are (n - 1). n – Number of nodes

Co-tree: The set of braches which are removed to form tree are called as co-tree.

Links: Branches removed to form tree are called as links.

5.2 INCIDENCE MATRIX (A):

Incidence matrix is formed between number of nodes and number of branches. This matrix is useful easy understanding of network and any complex network can be easily feed into system for coding. Order of incidence matrix is (n*b)

Procedure to incidence matrix:

aij = 1, if jth branch is incidence to ith node and direction is away from node.

aij = -1, if jth branch is incidence to ith node and direction is towards from node.

aij = 0, if jth branch is not incidence to ith node .

Properties of incidence matrix:

1. Number of non zero entries of row indicates degree of the node.
2. The non zero entries of the coloumn represents branch connections.

If two coloumns has same entries then they are in parallel.

Reduced incidence matrix:

Reduced incidence matrix is formed by eliminating one of the row from incidence matrix generally ground node row is eliminating, which is representing by A!. Number of possible trees of for any graph are det(A! A!T). Once we form the incidence matrix , we can write KCL equations any complex without appling KCL as follows,

$$A!.Ib = 0$$

Where, A! - reduced incidence matrix
Ib - branch current matrix(an coloumn matrix)

5.3 TIE-SET MATRIX:

Tie-set matrix is formed between link currents and branch currents. The order of tie-set matrix is (Il*Ib). it is represented by B. Tie-set is defined as a loop which consists of one link any number of branches. Hence total number of tie-set possible with above definition are number of links.

Procedure to form tie-set matrix:
Bij = 1, if current direction of jth branch coincides with direction of ith link.
Bij = -1, if current direction of jth branch opposes with direction of ith link.

Bij = 0, if jth branch has no relation with ith link.

Once we form tie-set matrix KVL equations of any complex network can be written as B. V_b = 0
Where, B = tie-set matrix

$$V_b = \text{branch voltage matrix (coloumn matrix)}$$

From the knowledge of tie-set matrix we can calculate all the branch currents in terms of link
currents, which is given as

$$Ib = BT . Il$$

Where, Ib - branch current matrix(an coloumn matrix) Il
- link current matrix(an coloumn matrix) BT-
transpose of tie-set matrix

5.4 CUT-SET MATRIX:

Cut-set matrix is formed between twig voltages and branch voltages. The order of cut-set matrix is ((n-1)*b). it is represented by C. Cut-set is defined as minimum set of branches by removing which graph is divided into two sub-graphs, where one of the part is an isolated node. Hence total number of cut-set possible with above definition is number of twigs. Generally cut-set direction is assumed in direction of branch, as cut-set consists of one branch any number of links.
Procedure to form tie-set matrix:

cij = 1, if direction of jth branch coincides with direction of ith cut-set.
cij = -1, if direction of jth branch opposes with direction of ith cut-set.
cij = 0, if direction of jth branch is not in the ith cut-set.

Once we form cut-set matrix KCL equations of any complex network can be written as C.Ib = 0

Where, C= cut-set matrix

$$Ib = \text{branch current matrix(coloumn matrix)}$$

From the knowledge of cut-set matrix we can calculate all the branch voltages in terms of twig
voltages, which is given as

$$Vt = CT . Vb$$

Where, Vb - branch voltage matrix(an coloumn matrix)
Vt – twig voltage matrix(an coloumn matrix)
CT- transpose of cut-set matrix

Eg: Determine all studied above for given circuit with its graph.
Nodes,n = 3
Branches = 4
Twigs = 3-1 = 2

Links   = 4-3+1 = 2

5.5 Incidence matrix:

A	=		1	2	3	4
		a	1	1	0	0
		b	0	-1	1	1
		c	-1	0	-1	-1

Reduced Incidence matrix:

A!	=		1	2	3	4
		a	1	1	0	0
		b	0	-1	1	1

Tie-set matrix:

As tree can be formed with two branches remaining two are links, let 2,4 arethe links.

B	=		i1	i2	i3	i4
		i2	-1	1	1	0
		i4	0	0	-1	1

We can write all branch currents as, Ib = BT . Il

i1		-1	0	i2
i2	=	1	0	
i3		1	-1	i4
i4		0	1	

Hence , i1 = -i2

      i2 = i2

      i3 =

      i2-i4

      i4 =

      i4

5.6 Cut-set matrix:

Let the two cut-set are (1,2) and (2,3,4) as possible cut-sets are (n-1) = (3-1) =2. Here 1 and 3 are branches or twigs hence cutsets direction is same as these branch directions.

C	=		1	2	3	4
		C(1,2)	1	-1	0	0
		C(2,3,4)	0	-1	1	1

We can write all branch voltages as, Vb =

V1	=	CT . Vt	0	V1
V2		-1	-1	
V3		0	1	V3
V4		0	1	