KLR COLLEGE OF ENGINEERING & TECHNOLOGY
(Affiliated to JNTUH, Approved by AICTE New Delhi, an ISO 9001:2015 Certified)
BCM Road, Paloncha-507115, Bhadradri Kothagudem (Dist.), Telangana.

Lecture Notes

On

ELECTRICAL CIRCUIT ANALYSIS

II BTECH I SEM (EEE)

Estd.2008

Regulation - R18

Academic Year: 2021-2022

UNIT-I

NETWORK THEOREMS

SUPERPOSITION THEOREM:-

Superposition theorem states that in any linear, active, bilateral network having more than one source, the response across any element is the sum of the responses obtained from each source considered separately and all other sources are replaced by their internal resistance. The superposition theorem is used to solve the network where two or more sources are present and connected. In other words, it can be stated as if a number of voltage or current sources are acting in a linear network, the resulting current in any branch is the algebraic sum of all the currents that would be produced in it when each source acts alone while all the other independent sources are replaced by their internal resistances. It is only applicable to the circuit which is valid for the ohm's law (i.e., for the linear circuit).

Procedure

Step 1 - Consider a single source acting alone. Deactivate all other sources (or simply remove) i.e. replace voltage source by short circuit and current source by open circuit, if internal impedance is zero. If internal impedance is known, replace them by their internal impedance.

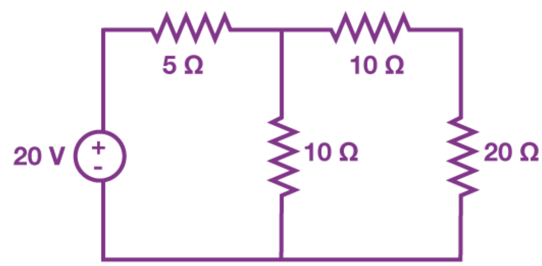
NOTE: Dependent sources must be kept as they are.

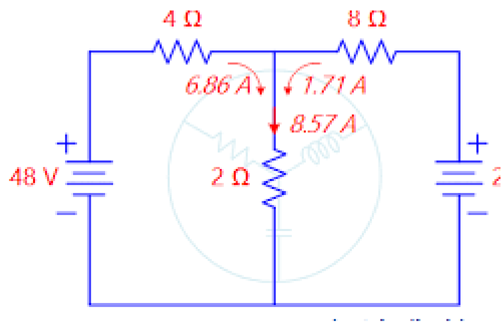
Step 2 - Find the current through or the voltage across the required elements due to source under consideration.

Step 3- Repeat Step 1 and step 2 for each source.

Step 4 -Add up all the responses produced by each source to get cumulative response. Consider the direction for current and polarity for voltages while adding them.

Examples:-





THEVENINS THEOREM:-

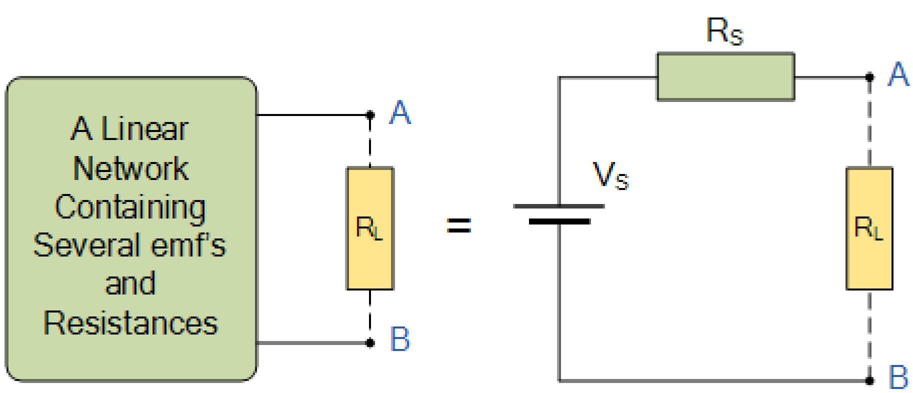
Thevenin's theorem states that it is possible to simplify any linear circuit, irrespective of how complex it is, to an equivalent circuit with a single voltage source and a series resistance.

STATEMENT:-

Any linear circuit containing several voltages and resistances can be replaced by just one single voltage in series with a single resistance connected across the load". In other words, it is possible to simplify any electrical circuit, no matter how complex, to an equivalent two-terminal circuit with just a single constant voltage source in series with a resistance (or impedance) connected to a load as shown below.

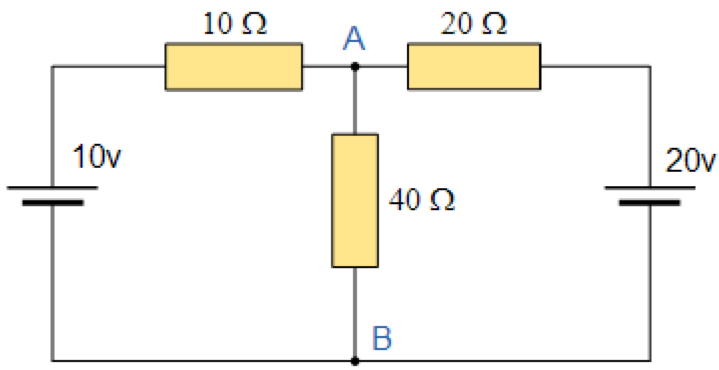
Thevenins Theorem is especially useful in the circuit analysis of power or battery systems and other interconnected resistive circuits where it will have an effect on the adjoining part of the circuit.

Thevenins Theorem Equivalent Circuit



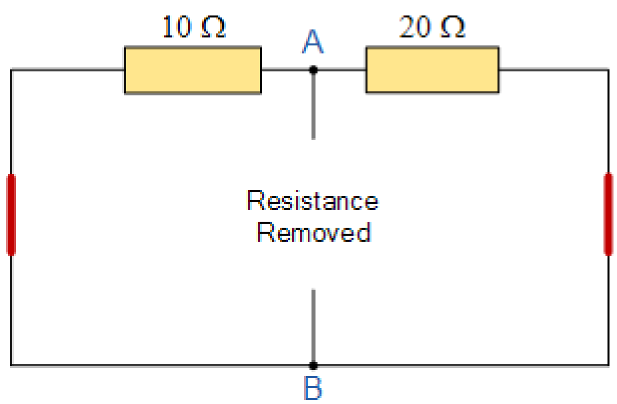
As far as the load resistor RL is concerned, any complex "one-port" network consisting of multiple resistive circuit elements and energy sources can be replaced by one single equivalent resistance Rs and one single equivalent voltage Vs. Rs is the source resistance value looking back into the circuit and Vs is the open circuit voltage at the terminals.

For example, consider the circuit from the previous tutorials.



Firstly, to analyse the circuit we have to remove the centre 40Ω load resistor connected across the terminals A-B, and remove any internal resistance associated with the voltage source(s). This is done by shorting out all the voltage sources connected to the circuit, that is v = 0, or open circuit any connected current sources making i = 0. The reason for this is that we want to have an ideal voltage source or an ideal current source for the circuit analysis.

The value of the equivalent resistance, Rs is found by calculating the total resistance looking back from the terminals A and B with all the voltage sources shorted. We then get the following circuit.



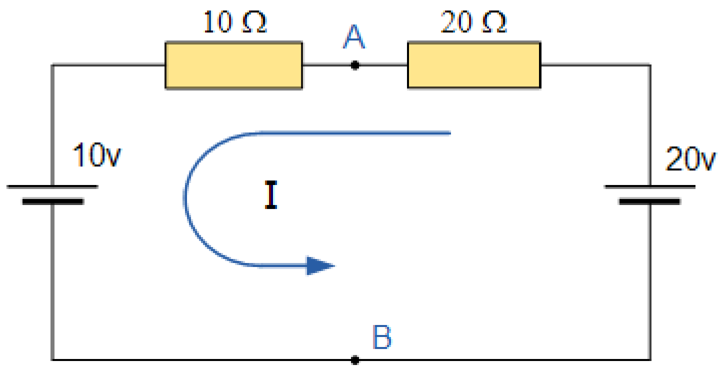
Find the Equivalent Resistance (Rs)

10Ω Resistor in Parallel with the 20Ω Resistor

$$R_T = \frac{R_1 \times R_2}{R_1 + R_2} = \frac{20 \times 10}{20 + 10} = 6.67\Omega$$

The voltage Vs is defined as the total voltage across the terminals A and B when there is an open circuit between them. That is without the load resistor RL connected.

Find the Equivalent Voltage (Vs)



We now need to reconnect the two voltages back into the circuit, and as VS = VAB the current flowing around the loop is calculated as:

$$I = \frac{V}{R} = \frac{20v - 10v}{20\Omega + 10\Omega} = 0.33\ amps$$

This current of 0.33 amperes (330mA) is common to both resistors so the voltage drop across the 20Ω resistor or the 10Ω resistor can be calculated as:

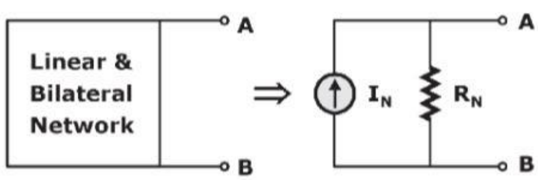
NORTON'S THEOREM:-

Norton's theorem states that any linear circuit can be simplified to an equivalent circuit consisting of a single current source and parallel resistance that is connected to a load.

STATEMENT:-

Norton's theorem states that any 2-terminal linear and bilateral network or circuit having multiple independent and dependent sources can be represented in a simplified equivalent circuit known as Norton's equivalent circuit.

Norton's Theorem Circuit Diagram



Norton's equivalent circuit consists of Norton's current source, IN in parallel with Norton's resistance, RN. The parallel the combination of current source and resistor is a practical current source. Hence, we can say that Norton's equivalent circuit is nothing but a practical current source.

Procedure of Norton's Theorem

It will take more time than the normal methods for finding the response of an element if the network/ circuit is having multiple sources and resistances. That time, we can use Norton's theorem to find the response easily. Now, let's see the steps for finding the response of an element when multiple sources and resistances are present in the network/ circuit by using Norton's theorem.

Step 1: Remove the element, where we are supposed to find the response from the given circuit. After the removal of the element, the terminals will be open.

Step 2: Find the current flowing through the terminals of the circuit obtained in Step 1 after shorting them. This current is known as short circuit current or Norton's equivalent current or Norton's current, IN in short.

Step 3: Replace all the independent sources with their internal resistances in the circuit obtained in Step 1.

Step 4: Find the equivalent resistance across the open-circuited terminals of the circuit obtained in Step 3 indirect methods if there are no dependent sources. This equivalent resistance is known as Norton's equivalent resistance or Norton's resistance, RN in short.

Step 5: If dependent sources are present, then we can find the equivalent resistance across the open-circuited terminals of the circuit obtained in Step 3 by using the Test source method. In the test source method, we will connect a 1V source (or 1A source) across the open terminals and will calculate another parameter current (or voltage). We will get the value of Norton's resistance, RN by taking the ratio of voltage and current across the 2 terminals.

Step 6: Draw Norton's equivalent circuit by connecting Norton's current, in parallel with Norton's resistance, RN.

MAXIMUM POWER TRANSFER THEOREM

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

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

Proof of Maximum Power Transfer Theorem

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

This concept is illustrated in following figures.

Maximum Power Transfer

The amount of power dissipated across the load resistor is

$P_L=I^2R_L$

Substitute $I=\frac{V_{Th}}{R_{Th}+R_L}$

in the above equation.

$P_L=\left(\frac{V_{Th}}{(R_{Th}+R_L)}\right)^2 R_L$

$\Rightarrow P_L=V_{Th}^2\left\{\frac{R_L}{(R_{Th}+R_L)^2}\right\}$

Equation 1

Condition for Maximum Power Transfer

For maximum or minimum, first derivative will be zero. So, differentiate Equation 1 with respect to R_L and make it equal to zero.

$\frac{dP_L}{dR_L}=V_{Th}^2\left\{\frac{(R_{Th}+R_L)^2\times 1-R_L\times 2(R_{Th}+R_L)}{(R_{Th}+R_L)^4}\right\}=0$

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

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

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

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

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

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

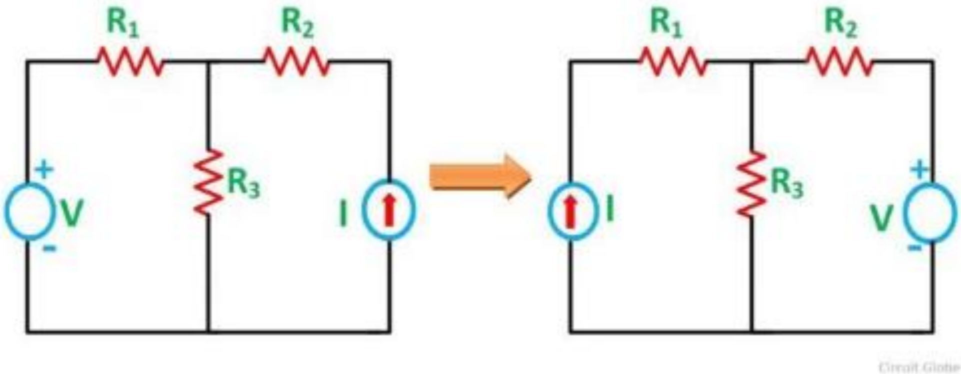
RECIPROCITY THEOREM

The reciprocity theorem states that the current at one point in a circuit due to a voltage at a second point is the same as the current at the second point due to the same voltage at the first. The reciprocity theorem is valid for almost all passive networks. The reciprocity theorem is a feature of a more general principle of reciprocity in electromagnetism.

Explanation of Reciprocity Theorem

The location of the voltage source and the current source may be interchanged without a change in current. However, the polarity of the voltage source should be identical with the direction of the branch current in each position.

The Reciprocity Theorem is explained with the help of the circuit diagram shown below



The various resistances R1, R2, R3 is connected in the circuit diagram above with a voltage source (V) and a current source (I). It is clear from the figure above that the voltage source and current sources are interchanged for solving the network with the help of Reciprocity Theorem.

Steps for Solving a Network Utilizing Reciprocity Theorem

Step 1 – Firstly, select the branches between which reciprocity has to be established.

Step 2 – The current in the branch is obtained using any conventional network analysis method.

Step 3 – The voltage source is interchanged between the branch which is selected.

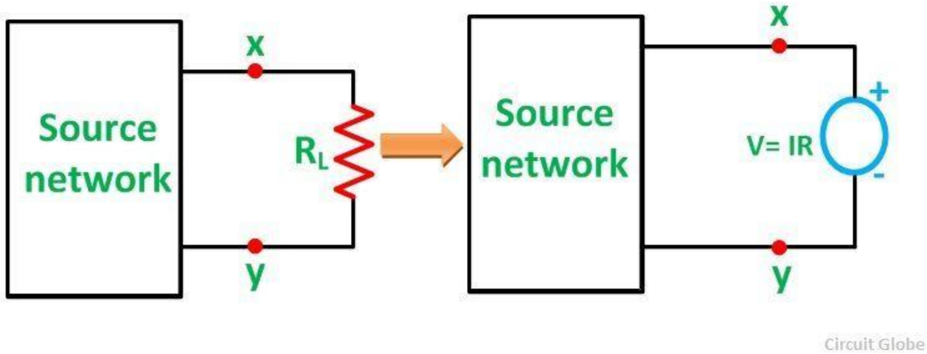
Step 4 – The current in the branch where the voltage source was existing earlier is calculated.

Step 5 – Now, it is seen that the current obtained in the previous connection, i.e., in step 2 and the current which is calculated when the source is interchanged, i.e., in step 4 are identical to each other.

COMPENSATION THEOREM

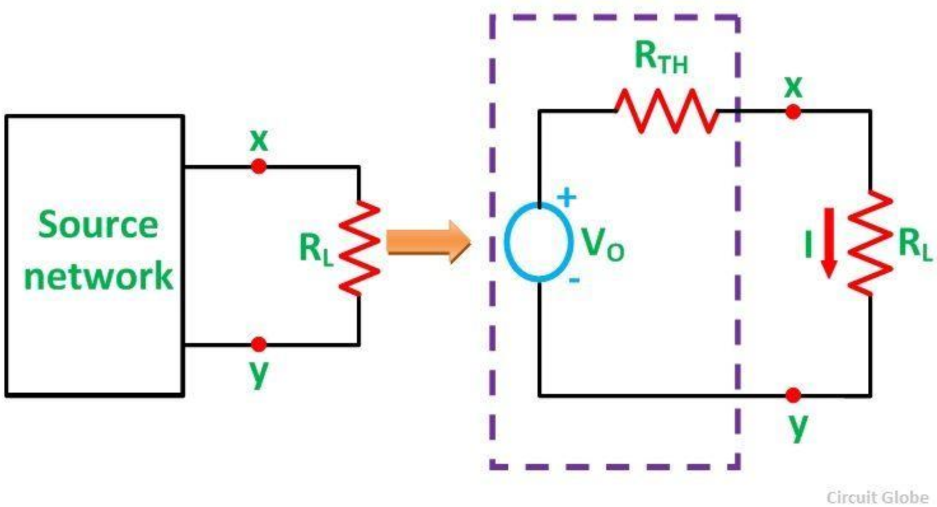
Compensation Theorem states that in a linear time-invariant network when the resistance (R) of an uncoupled branch, carrying a current (I), is changed by (ΔR), then the currents in all the branches would change and can be obtained by assuming that an ideal voltage source of (VC) has been connected such that VC = I (ΔR) in series with (R + ΔR) when all other sources in the network are replaced by their internal resistances.

In Compensation Theorem, the source voltage (VC) opposes the original current. In simple words, compensation theorem can be stated as – the resistance of any network can be replaced by a voltage source, having the same voltage as the voltage drop across the resistance which is replaced.



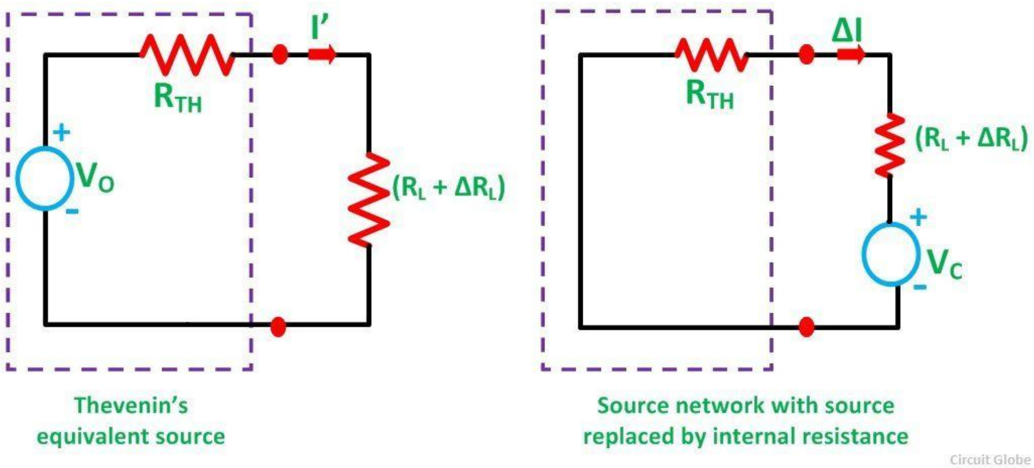
Explanation

Let us assume a load RL be connected to a DC source network whose Thevenin's equivalent gives V0 as the Thevenin's voltage and RTH as the Thevenin's resistance as shown in the figure below:



$$I = \frac{V_0}{R_{TH} + R_L} \dots\dots\dots\dots(1)$$

Let the load resistance RL be changed to (RL + ΔRL). Since the rest of the circuit remains unchanged, the Thevenin's equivalent network remains the same as shown in the circuit diagram below:



Here,

$$I' = \frac{V_0}{R_{TH} + (R_L + \Delta R_L)} \dots \dots \dots \dots (2)$$

The change of current being termed as ΔI

Therefore,

$$\Delta I = I' - I \dots \dots \dots \dots \dots (3)$$

Putting the value of I' and I from the equation (1) and (2) in the equation (3)

we will get the following equation:

$$\Delta I = \frac{V_0}{R_{TH} + (R_L + \Delta R_L)} - \frac{V_0}{R_{TH} + R_L}$$

$$\Delta I = \frac{V_0 \{R_{TH} + R_L - (R_{TH} + R_L + \Delta R_L)\}}{(R_{TH} + R_L + \Delta R_L)(R_{TH} + R_L)}$$

$$\Delta I = -\left[\frac{V_0}{R_{TH} + R_L}\right] \frac{\Delta R_L}{R_{TH} + R_L + \Delta R_L} \dots \dots \dots \dots . (4)$$

Now, putting the value of I from the equation (1) in equation (4),

we will get the following equation:

$$\Delta I = -\frac{I\,\Delta R_L}{R_{TH} + R_L + \Delta R_L} \;\dots\dots\dots\dots (5)$$

As we know, VC = I ΔRL and is known as compensating voltage.

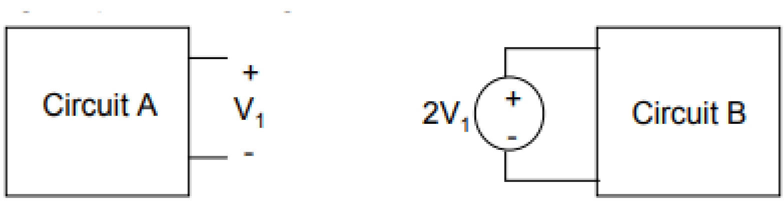
Therefore, equation (5) becomes,

$$\Delta I = \frac{-V_C}{R_{TH} + R_L + \Delta R_L}$$

Hence, Compensation theorem tells that with the change of branch resistance, branch currents changes and the change is equivalent to an ideal compensating voltage source in series with the branch opposing the original current, where all other sources in the network being replaced by their internal resistances.

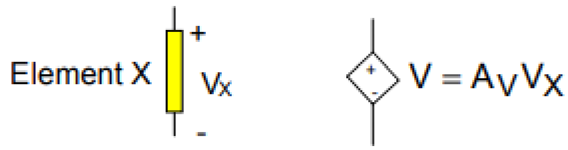
ANALYSIS WITH DEPENDENT CURRENT AND VOLTAGE SOURCES:-

A linear dependent source is a voltage or current source that depends linearly on some other circuit current or voltage. Example: You watch a certain voltmeter V1 and manually adjust a voltage source Vs to be 2 times this value. This constitutes a voltage-dependent voltage source.



This is just a manual example, but we can create such dependent sources electronically. We will create a new symbol for dependent sources.

We can have voltage or current sources depending on voltages or currents elsewhere in the circuit. A diamond-shaped symbol is used for dependent sources, just as a reminder that it’s a dependent source. Circuit analysis is performed just as with independent sources.



Here, the voltage V provided by the dependent source (right) is proportional to the voltage drop over Element X. The dependent source does not need to be attached to the Element X in any way.

NODE AND MESH ANALYSIS

MESH ANALYSIS:-

Mesh analysis is defined as

The method in which the current flowing through a planar circuit is calculated.

A planar circuit is defined as the circuits that are drawn on the plane surface in which there are no wires crossing each other. Therefore, a mesh analysis can also be known as loop analysis or mesh-current method.

Procedure of Mesh Analysis

The following steps are to be followed while solving the given electrical network using mesh analysis:

Step 1:

To identify the meshes and label these mesh currents in either clockwise or counter clockwise direction.

Step 2:

To observe the amount of current that flows through each element in terms of mesh current.

Step 3:

Writing the mesh equations to all meshes using Kirchhoff's voltage law and then Ohm's law.

Step 4:

The mesh currents are obtained by following Step 3 in which the mesh equations are solved.

Hence, for a given electrical circuit the current flowing through any element and the voltage across any element can be determined using the node voltages.

NODE ANALYSIS:-

Nodal analysis is used for solving any electrical network, and it is defined as

The mathematical method for calculating the voltage distribution between the circuit nodes.

This method is also known as the node-voltage method since the node voltages are with respect to the ground. The following are the three laws that define the equation related to the voltage that is measured between each circuit node:

- Ohm's law
- Kirchhoff's voltage law
- Kirchhoff's current law

Features of Nodal Analysis

- Nodal analysis is an application of Kirchhoff's current law.
- When there are 'n' nodes in a given electrical circuit, there will be 'n-1' simultaneous equations to be solved.
- To obtain all the node voltages, 'n-1' should be solved.
- The number of non-reference nodes and the number of nodal equations obtained are equal.

Procedure of Nodal Analysis

The following steps are to be followed while solving any electrical circuit using nodal analysis:

Step 1:

To identify the principal nodes and select one of them as a reference node. This reference node will be treated as the ground.

Step 2:

All the node voltages with respect to the ground from all the principal nodes should be labelled except the reference node.

Step 3:

The nodal equations at all the principal nodes except the reference node should have a nodal equation. The nodal equation is obtained from Kirchhoff's current law and then from Ohm's law.

Step 4:

To obtain the node voltages, the nodal equations can be determined by following Step 3.

Hence, for a given electrical circuit, the current flowing through any element and the voltage across any element can be determined using the node voltages.

CONCEPT OF DUALITY AND DUAL NETWORKS

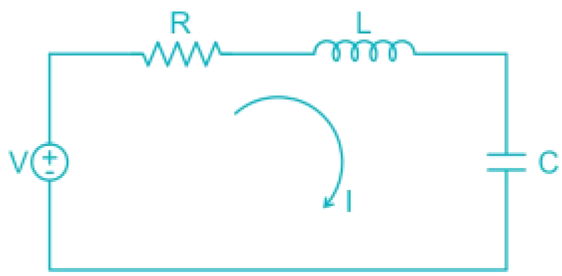
CONCEPT OF DUALITY:-

Two electrical networks are said to be dual networks if the mesh equations of one network are equal to the node equation of others.

Identical behaviour patterns observed between voltages and currents in two circuits illustrate the principle of duality.

Duality of Networks Identical behaviour patterns observed between voltages and currents in two circuits, illustrate the principal of duality.

Series RLC Circuit

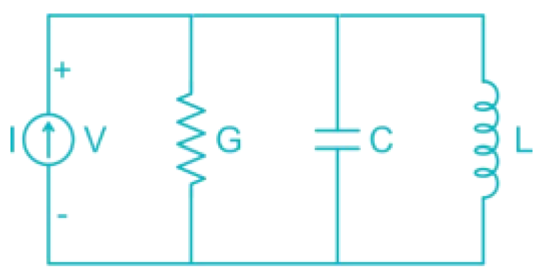


Mesh equation –

$-V + IR + L\ dI\ dt + 1\ C \int I\ dt = 0$

$V = IR + L\ dI\ dt + 1\ C \int I\ dt \rightarrow (1)$

Parallel GCL Circuit



Node equation –

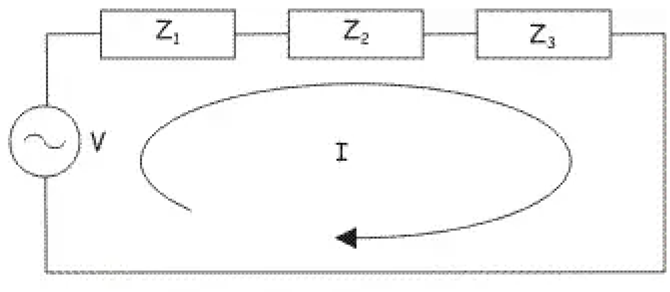
I + VG + C dV dt + 1 L ∫ Vdt = 0

I = VG + C dV dt + 1 L ∫ Vdt → (2)

CONCEPT OF DUAL NETWORKS:-

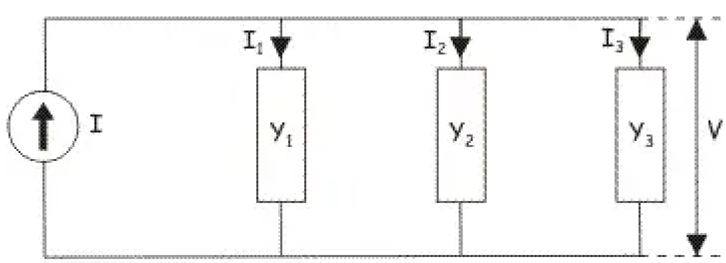
Two electrical networks are said to be dual networks if the mesh equations of one network is equal to the node equation of the other.

The dual network is based on Kirchhoff Current Law and Kirchhoff Voltage Law.



Applying Kirchhoff Voltage Law in the network A, above we get,

$$V = IZ_1 + IZ_2 + IZ_3$$
$$\Rightarrow V = I(Z_1 + Z_2 + Z_3) \quad \cdots\cdots \quad (i)$$



Applying Kirchhoff Current Law in the network B, above we get,

$$
\begin{aligned}
& I = I_1 + I_2 + I_3 \\
\Rightarrow\ & I = VY_1 + VY_2 + VY_3 \\
\Rightarrow\ & I = V(Y_1 + Y_2 + Y_3) \quad \cdots\cdots \quad (ii)
\end{aligned}
$$

Here we have found that equations (i) and (ii) are similar in their mathematical form. Equation (i) is in mesh form and equation (ii) is in nodal form.

Here, the left side variable of equation (i) is voltage, and the left side variable of equation (ii) is current.

Similarly, the right side of equation (i) is a product of the current and total impedance of the circuit.

Similarly, the right side of equation (ii) is the product of voltage and admittance of the circuit.

So, it is needless to say these two networks are dual networks. From, the examples it is also clear that dual networks may not be equivalent networks.
The circuit equation of the two dual networks are similar in form but the variable is interchanged.

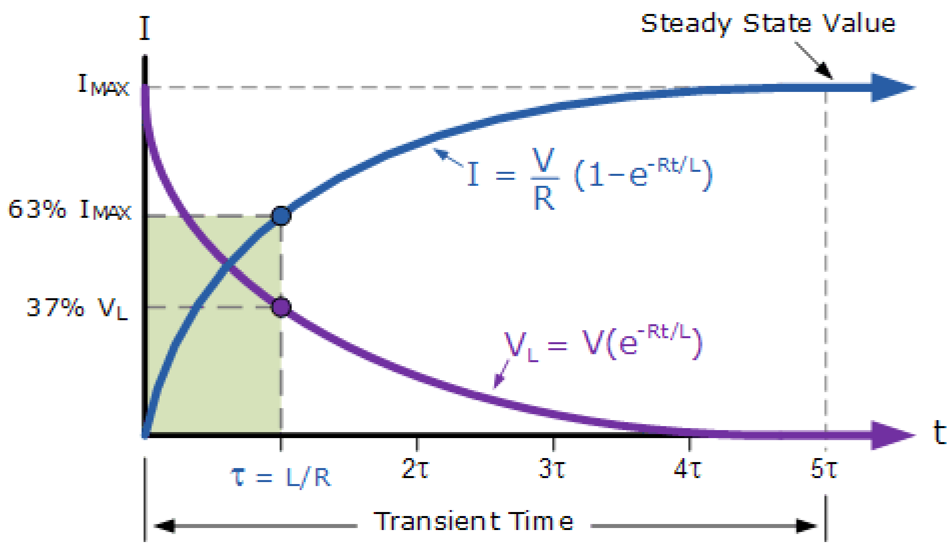
UNIT-II

SOLUTION OF FIRST ORDER AND SECOND ORDER NETWORKS

SOLUTION OF FIRST ORDER AND SECOND ORDER DIFFERENTIAL EQUATIONS FOR SERIES RL, RC, AND RLC CIRCUITS:-

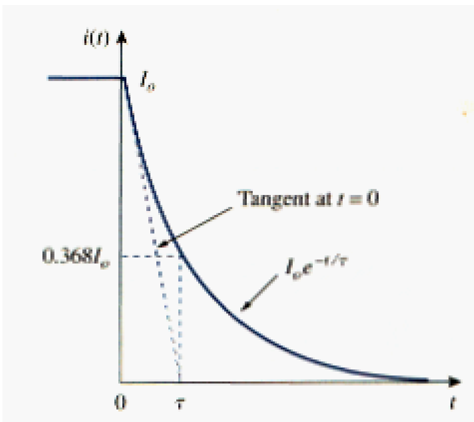
RL CIRCUIT:-

Differential equation of a LR circuit for growing current



$$\epsilon - Ridi = Ldt$$

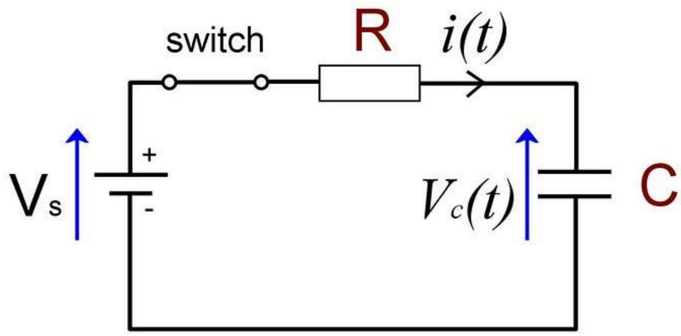
Differential equation of a LR circuit for decaying current



$$idi = -LRdt$$

RC CIRCUIT:-

In the circuit below, the switch is initially open, so before time t = 0, there is no voltage feeding the circuit. Once the switch closes, the supply voltage Vs is applied indefinitely. This is known as a *step input.* The response of the RC circuit is called a transient response, or step response for a step input.



$$V_s - i(t)R - V_c(t) = 0$$

Kirchoff’s voltage law around an RC circuit.

When a step voltage is first applied to an RC circuit, the output voltage of the circuit doesn’t change instantly. There is a time delay due to the fact that current needs to charge the capacitance. The time taken for the output voltage (the voltage on the capacitor) to reach 63% of its final value is known as the time constant, often represented by the Greek letter tau (τ). The time constant = RC where R is the resistance in ohms and C is the capacitance in farads.

RLC CIRCUIT:-

Now that we have all of the basic relationships for the elements in the circuit, we can find a model for the circuit.

The next step involves using Kirchoff’s Voltage Law in order to find the circuit.

By substituting the circuit elements into Kirchoff’s law, we get

: vr + vl + vc = v(t) (1)

Where vr is the voltage drop across the resistor, vl is the voltage drop across the inductor, vc is the voltage drop across the capacitor, and vt is a time varying voltage.

It is now necessary to find the voltage drops across each circuit element.
To find the voltage drop across the resistor, we take the resistance equation and modify it for a time varying voltage giving us:

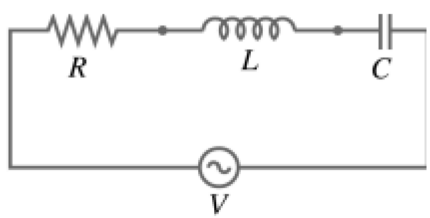
$Ri(t) = V(t)$

Where R is the resistance, i(t) is a time varying current, and v(t) is the time varying voltage.

$L\ di\ dt = V(t)$

To find the voltage drop across the capacitor, it is necessary to examine the actual component in physical sense. The current through any circuit element is defined by the rate of charge passing through it with respect to time. However, electrons do not pass through a capacitor, but instead electrons stick to the negative plate of the capacitor for each electron that leave the positive plate. The charge on plate is equal and opposite to the charge on the other plate of the capacitor, which means that the charge on each plate is the integral of the time varying current function. The equation is then as follows:

$1\ C\ Z\ t\ 0\ i(\tau)d\tau + V(0) = V(t)$



Where C is the capacitance, t is the time interval. i(t) is the time varying current, and V (0) is an initial condition, of the voltage when the system is turned on. The last thing that needs to be determined is the time varying voltage function.
This function will have a sinusoidal curve given the nature of radio waves, and will also be time dependent.
Thus, the equation is $V0 \sin \omega t$ Substituting into equation (1) we get the following equation:

$L\ di\ dt + Ri(t) + 1\ C\ Z\ t\ 0\ i(\tau)d\tau + V(0) = V0 \sin \omega t$ (2)

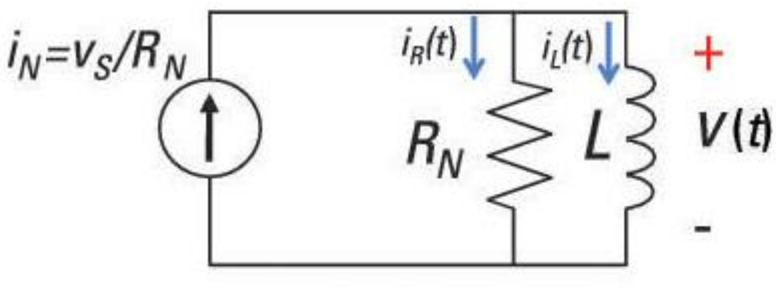
Differentiating this equation with respect to time gives:

$L\ di2\ d\ 2t + R\ di\ dt + 1\ C\ i(t) = Vo\omega \cos \omega t$ (3)

SOLUTION OF FIRST ORDER AND SECOND ORDER DIFFERENTIAL EQUATIONS FOR PARALLEL RL, RC, AND RLC CIRCUITS:-

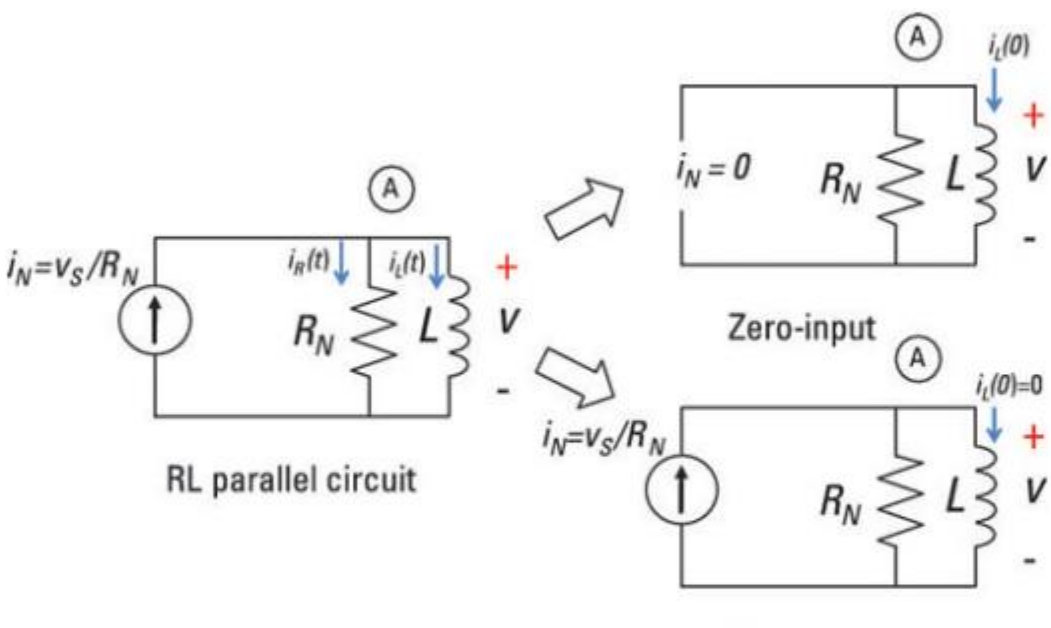
RL CIRCUIT:

A first-order RL parallel circuit has one resistor (or network of resistors) and a single inductor. First-order circuits can be analyzed using first-order differential equations. By analyzing a first-order circuit, you can understand its timing and delays.
Analyzing such a parallel RL circuit, like the one shown here, follows the same process as analyzing an RC series circuit. So if you are familiar with that procedure, this should be a breeze.



RL-parallel circuit

Here is how the RL parallel circuit is split up into two problems: the zero-input response and the zero-state response. Here, you'll start by analyzing the zero-input response.



To simplify matters, you set the input source (or forcing function) equal to 0.

iN(t) = 0 amps.

This means no input current for all time — a big, fat zero. The first-order differential equation reduces to

$$i_N(t) = 0 = \left(\frac{L}{R}\right)\frac{di_{zI}(t)}{dt} + i_{zI}(t) \quad \text{or} \quad i(t) = -\left(\frac{L}{R}\right)\frac{di_{zI}(t)}{dt}$$

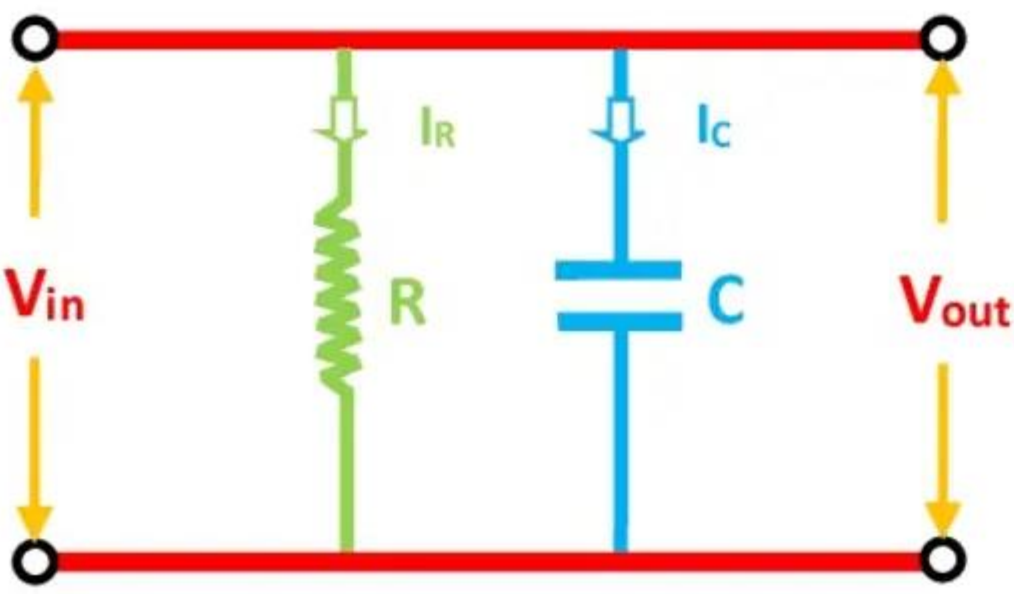
For an input source of no current, the inductor current iZI is called a zero-input response.

No external forces are acting on the circuit except for its initial state (or inductor current, in this case).

The output is due to some initial inductor current I0 at time t = 0.

RC Circuit

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



PARALLEL R-C CIRCUIT

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

$$V = V_R = V_C$$

$$I = I_R + I_C$$

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

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

The voltage-current relationship for the capacitor is:

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

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

$$I_R + I_C = 0$$

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

RLC CIRCUIT:-

Consider a parallel RLC

- Switch at t=0 applies a current source
- For parallel will use KCL
- Proceeding just as for series but now in voltage

(1) Using KCL to write the equations:

0 0 1 vdt I R L v dt di C t + + = ∫ vdt+Io

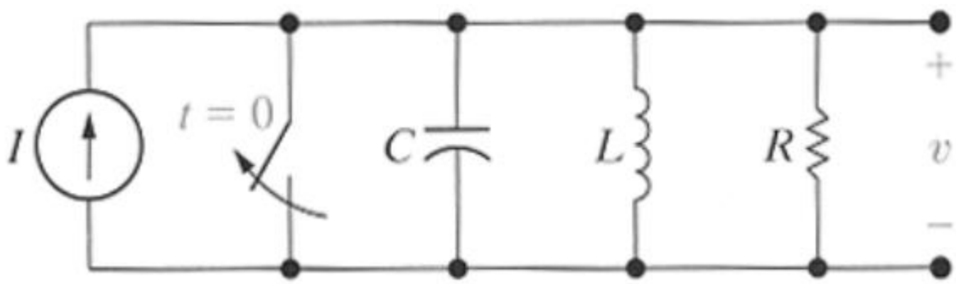
(2) Want full differential equation

•Differentiating with respect to time

0 1 1 2 2 + + v = dt L dv dt R d v C

(3) This is the differential equation of second order

- Second order equations involve 2nd order derivatives.



Solving the Second Order Systems Parallel RLC
•Continuing with the simple parallel RLC circuit as with the series
(4) Make the assumption that solutions are of the exponential form:
i(t) = A exp(st)
•Where A and s are constants of integration.
•Then substituting into the differential equation
0 1 1 2 2 + + v = dt L dv dt R d v C ( ) exp( ) exp( ) 0 1 exp 2 + + st = L A sA st R Cs A st
•Dividing out the exponential for the characteristic equation 0 2 1 1 + + = LC s RC s
•Giving the Homogeneous equation
•Get the 3 same types of solutions but now in voltage
• Just parameters are going to be different.

INITIAL AND FINAL CONDITIONS IN NETWORK ELEMENTS:-

- Most of the transmission lines, electrical circuits and communication networks are made up of network
- Elements like resistor R, inductor L, and Capacitor C.
- These networks are connected by voltage and current
- Sources. It is most useful to understand the behaviour of the network when we switched on the network by
- Supplying voltage source.
- It is most important to determine the transient response of R-L, R-C, R-L-C
- Series circuits for d.c and a.c excitations.

- Assuming that at reference time t= 0, the switch in the circuit is closed and also assuming that switch
- Act in zero time.
- To differentiate between the time immediately before and immediately after the operation
- Of a switch, is represented as t= 0−and t= 0+signs are used.
- The condition existing just before the switch
- Is operated will be designated as i(0−), v(0−), q(0−) and the conditions existing after closing of a switch is
- Designated as as i(0+),v(0+), q(0+).
- Also initial conditions of a network depend on the past history

- Network prior the closing of the network at t= 0−and the network structure at t= 0+, after switching.
- The evaluation of voltages and currents and their derivatives at t= 0+, are known as initial conditions
- And evaluation of condition at t=∞are known as final conditions.
- The initial conditions give knowledge of the behavior of the circuit elements at the instant of switching.
- The final conditions give knowledge of the behavior of the circuit elements after the settling of circuit att=∞

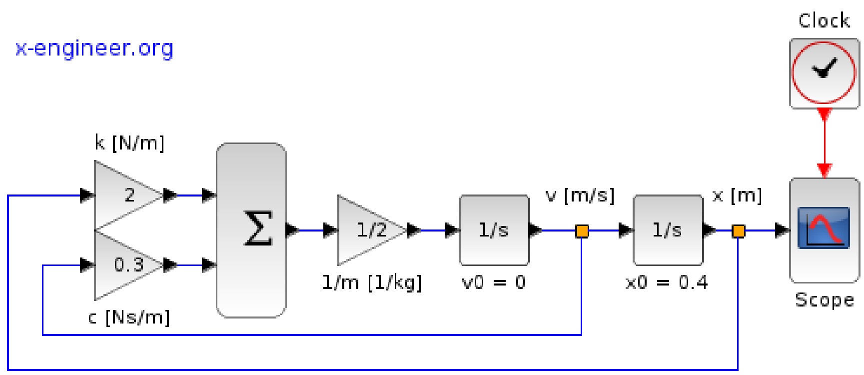
FORCED AND FREE RESPONSE

FREE RESPONSE:-

The free response of a system is the solution of the describing differential equation of the system, when the input is zero.
In our case the input into the system is the force F [N]. Therefore, in order to verify the free response of the system we have to solve the differential equation:

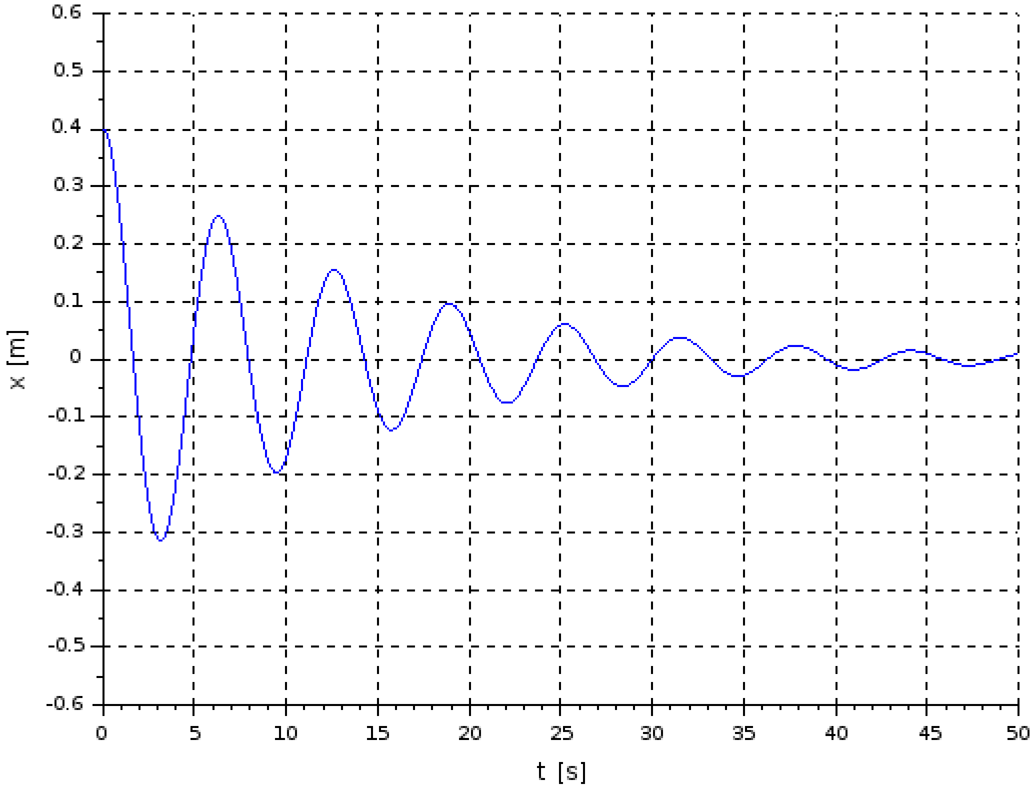
$$m \cdot d2xdt2+c \cdot dxdt+k \cdot x=0(2)$$

The easiest way to to this is to integrate the differential equation in Xcos. For a complete explanation how handle it, read the article How to solve (integrate) a differential equation in Xcos.
The equation (2) is for the system in equilibrium. This means that, if we solve it in Xcos, the output of the system (displacement x [m] of the body) will be zero. To be able to see the free response, we need an **i**nitial condition. For our example we are going to set the initial conditions:



$$x(0)v(0)=0.4=0$$

which translates into: at time t = 0 (when simulation starts), the position of the mass is 0.4 m to the right and the speed is 0 m/s. The free response of the mass spring-damper system will be the variation in time of the displacement x [m].



At time t = 0, the position x [m] of the mass is 0.4 m. After being released it begins to oscillate around the equilibrium value, 0 m, with smaller and smaller amplitudes. Due to the damping coefficient, after a while, it will stabilise at 0 m (the equilibrium position).
The general equation of a free response system has the differential equation in the form:

∑i=0naidixdti=0(4)

The solution x(t) of the equation (4) depends only on the n initial conditions.

FORCED RESPONSE;-

The forced response of a system is the solution of the differential equation describing the system, taking into account the impact of the input. In our case the input is force F [N].
Therefore, in order to verify the forced response of the system, we have to solve the differential equation:

m·d2xdt2+c·dxdt+k·x=F(t)(3)

To visualise the forced system response of a system, all the initial conditions must be zero:

x(0)v(0)=0=0

which translates into: at time t = 0 (when simulation starts), the position of the mass is 0 m and the speed is 0 m/s. At time t = 10 s, the input force will become 0.5 N and it will pull the mass to the right. The forced response of the mass spring-damper system will be the variation in time of the displacement x [m].
To integrate equation (3), we have to write it in the following form:

d2xdt2=1m·(F–c·dxdt–k·x)(4)

We can reuse the Xcos model from the free system response, the only differences being that there is an input step force added and the initial condition of the position is 0 m.

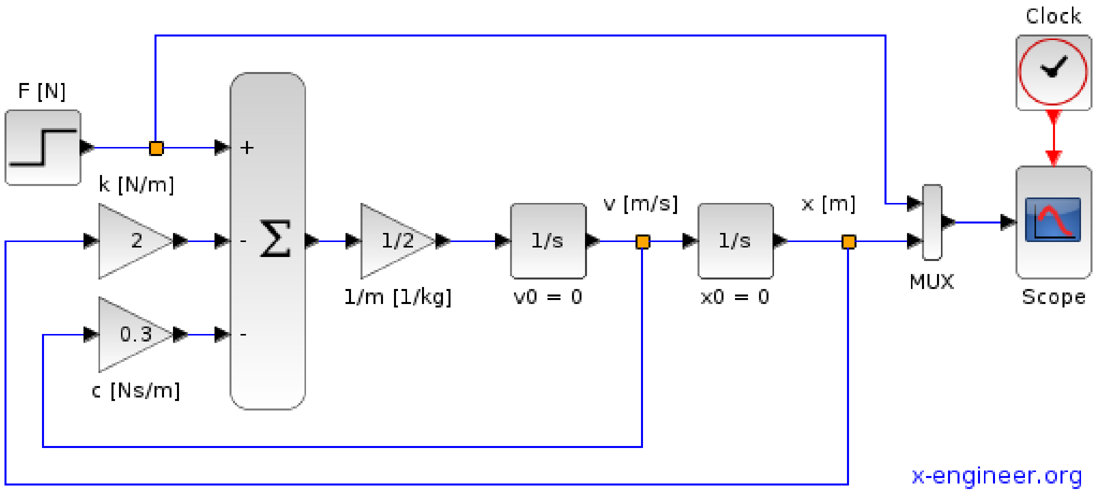


Image: Forced response control system – Xcos block diagram

The input step force will have value 0.5 N when the simulation time is 10 s.
Running the Xcos block diagram will output the following plot:

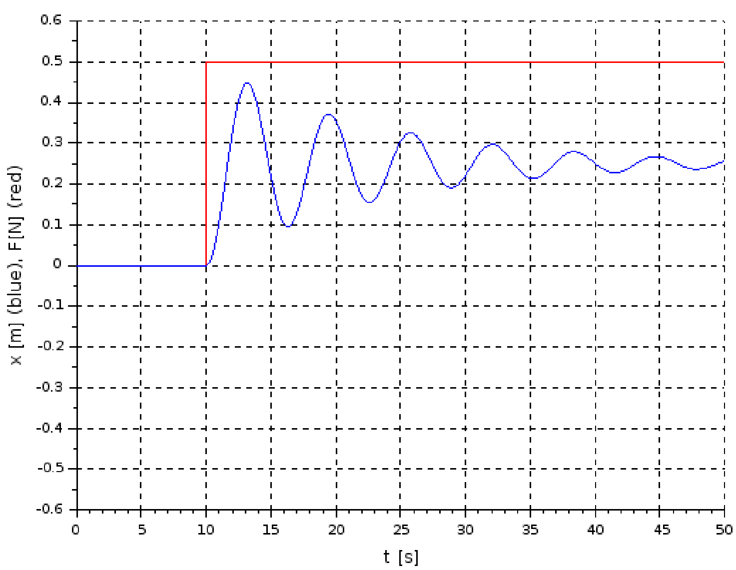


Image: Forced response control system – plot

In this case, with an input force of 0.5 N, the equilibrium position of the system is around 0.25 m. We can notice how the position oscillates around the equilibrium point, settling down in time due to the damping coefficient.

TIME CONSTANTS

The meaning of the time constant of an RC circuit is the time required to charge the capacitor to 63.2% of the value through an applied DC voltage.

τ =RC

Where R is circuit resistance and C is circuit capacitance

Conversely, the period required to discharge a capacitor to about 36.8% of its value is also one time constant of the circuit. It is used in the following formulae to find out the voltage across the capacitor in relation to time

- When charging towards the applied voltage from zero voltage(V0) : V(t)=V0(1-e-t/τ)

  Where τ is RC time constant

- Discharging towards zero voltage(V0) : V(t)=V0(e-t/τ)

The RC Time Constant

In an RC circuit, the resistance responds almost instantly to any change in voltage applied to the circuit. However, a resistor does not store energy. It is a passive device that dissipates energy in the form of heat energy. On the other hand, a capacitor can store energy in the form of an electrostatic field. It comprises two electrodes that are plates made of conducting material. An insulating material separates these plates. This material is dielectric and can store electrostatic energy.

The capacitor, unlike the resistor, cannot respond instantaneously to the changes in the voltage applied to the circuit. There will always be a short period of time immediately after the voltage is firstly applied for the circuit current and voltage across the capacitor to change state.This means that there will be a certain delay in the capacitor changing its stored energy within its electric field. This will hold true both for when the energy has to be increased and when the energy has to be decreased.

The amount of time required by a circuit to respond to changes is found to always be in multiples of the product of the resistance and the capacitance of the circuit. It is the product of ohms and farads written in seconds. The following equation expresses the current that is in the capacitor.

$i_C = C(dv/dt)$

Here dv is the change in voltage and dt is the change in time.

STEADY STATE AND TRANSIENT STATE RESPONSE FOR DC AND AC EXCITATION

TRANSIENT RESPONSE FOR DC AND AC EXCITATION:-

After applying an input to an electric circuit, the output takes certain time to reach steady state. So, the output will be in transient state till it goes to a steady state. Therefore, the response of the electric circuit during the transient state is known as transient response.

The transient response will be zero for large values of 't'. Ideally, this value of 't' should be infinity. But, practically five time constants are sufficient.

Presence or Absence of Transients

Transients occur in the response due to sudden change in the sources that are applied to the electric circuit and / or due to switching action. There are two possible switching actions. Those are opening switch and closing switch.

- The transient part will not present in the response of an electrical circuit or network, if it contains only resistances. Because resistor is having the ability to adjust any amount of voltage and current.
- The transient part occurs in the response of an electrical circuit or network due to the presence of energy storing elements such as inductor and capacitor. Because they can't change the energy stored in those elements instantly.

Inductor Behavior

Assume the switching action takes place at t = 0. Inductor current does not change instantaneously, when the switching action takes place. That means, the value of inductor current just after the switching action will be same as that of just before the switching action.

Mathematically, it can be represented as

$$i_L(0^+) = i_L(0^-)$$

STEADY STATE RESPONSE FOR DC AND AC EXCITATION:-

The part of the time response that remains even after the transient response has become zero value for large values of't' is known as steady state response. This means, there won't be any transient part in the response during steady state.

Inductor Behavior

If the independent source is connected to the electric circuit or network having one or more inductors and resistors (optional) for a long time, then that electric circuit or network is said to be in steady state. Therefore, the energy stored in the inductor(s) of that electric circuit is of maximum and constant.

Mathematically, it can be represented as

$$W_L = \frac{L{i_L}^2}{2} = \quad \text{Maximum \& constant}$$

$$\Rightarrow i_L = \quad \text{Maximum \& constant}$$

Therefore, inductor acts as a constant current source in steady state.

The voltage across inductor will be

V_L=$L di_L dt$=0V

Capacitor Behavior

The capacitor voltage does not change instantaneously similar to the inductor current, when the switching action takes place. That means, the value of capacitor voltage just after the switching action will be same as that of just before the switching action.

Mathematically, it can be represented as

$$v_c(0^+) = v_c(0^-)$$

Steady state Response

The part of the time response that remains even after the transient response has become zero value for large values of 't' is known as **steady state response**. This means, there won't be any transient part in the response during steady state.

Inductor Behavior

If the independent source is connected to the electric circuit or network having one or more inductors and resistors (optional) for a long time, then that electric circuit or network is said to be in steady state. Therefore, the energy stored in the inductor(s) of that electric

circuit is of maximum and constant

.

Mathematically, it can be represented as

$$W_L = \frac{L{i_L}^2}{2} = \text{Maximum \& constant}$$

$$\Rightarrow i_L = \text{Maximum \& constant}$$

Therefore, inductor acts as a constant current source in steady state.

The voltage across inductor will be

$$V_L = L\frac{di_L}{dt} = 0V$$

Capacitor Behaviour

If the independent source is connected to the electric circuit or network having one or more capacitors and resistors (optional) for a long time, then that electric circuit or network is said to be in steady state. Therefore, the energy stored in the capacitor(s) of that electric circuit is of maximum and constant.

Mathematically, it can be represented as

$$W_c = \frac{C{v_c}^2}{2} = \text{Maximum \& constant}$$

$$\Rightarrow v_c = \text{Maximum \& constant}$$

Therefore, capacitor acts as a **constant voltage source** in steady state.

The current flowing through the capacitor will be

$$i_c = C\frac{dv_c}{dt} = 0A$$

UNIT-III

SINUSOIDAL STEADY STATE ANALYSIS

REPRESENTATION OF SINE FUNCTION AS ROTATING PHASOR

It is the product of the rms voltage phasor and the complex conjugate of the rms current phasor.

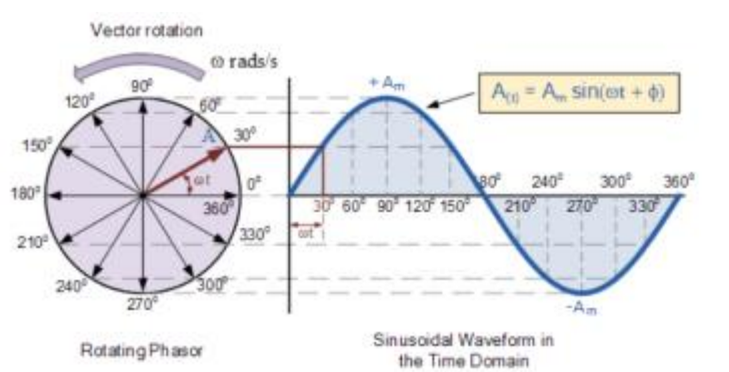
Given the phasor form V = Vm< Ѳv and I = Im<Ѳi of voltage v (t) and current i (t)

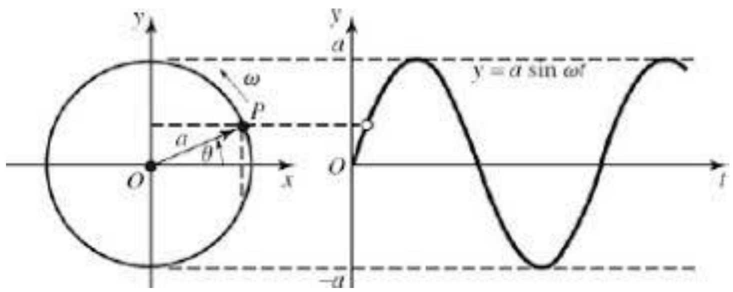
The complex power S absorbed by the AC load is the product of the voltage and the complex conjugate of the current.

Imagine a rotating phasor centred on the origin and initially aligned along the x-direction such that the angle from the positive x-axis is zero.

Now, imagine this rotating phasor (rotating in the anti-clockwise direction) moving along the positive x-axis.

The point on the circle at the tip of the phasor will trace out a sine wave





PHASOR DIAGRAMS OF STEADY STATE ANALYSIS

Steady-state sinusoidal analysis using phasors is a powerful technique used in electrical engineering to analyse the behaviour of circuits driven by a sinusoidal voltage or current sources. This method is particularly useful when dealing with circuits that contain multiple frequency components, as it simplifies the analysis process and allows for quick and accurate results. In this **Steady State Sinusoidal Analysis Using Phasors** article, we will explore the principles of steady-state sinusoidal analysis using phasors, including how to represent sinusoidal signals as phasors, how to perform phasor arithmetic, and how to use phasors to analyse AC circuits.

Phasors are mathematical tools used in electrical engineering to represent sinusoidal signals in a compact and convenient way. A phasor is a vector that represents the amplitude and phase of a sinusoidal signal at a particular frequency. By representing sinusoidal signals as phasors, we can perform algebraic operations such as addition, subtraction, multiplication, and division, which are much simpler than working with complex sinusoidal functions. Phasors are particularly useful when dealing with AC circuits, where multiple sinusoidal sources with different frequencies and phases may be present.

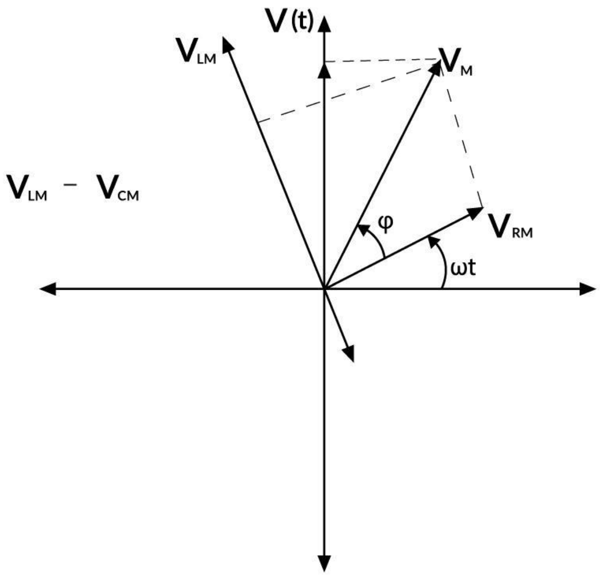
Steady State Sinusoidal Analysis Using Phasors

In the steady-state sinusoidal analysis using phasors, we assume that the circuit has reached a steady state, where all the voltages and currents have settled to their steady-state sinusoidal values. This means that we can represent all the sinusoidal signals in the circuit as phasors, which simplifies the analysis process. By using phasors, we can transform the circuit analysis problem from a time-domain problem to a phasor-domain problem, which is much simpler to solve.

The principles of steady-state sinusoidal analysis using phasors are based on the concept of impedance, which is the AC equivalent of resistance. Impedance is a complex quantity that represents the opposite of a circuit element to the flow of AC current. By representing circuit elements as impedances, we can perform phasor analysis using Ohm's law and Kirchhoff's laws, just as we would in DC circuits. By combining phasors using algebraic operations, we can calculate the phasor voltage and current values at any point in the circuit, which allows us to analyze the behaviour of the circuit under different conditions.

Phasors

Phasors are mathematical tools used in electrical engineering to represent sinusoidal signals in a compact and convenient way. A phasor is a vector that represents the amplitude and phase of a sinusoidal signal at a particular frequency. By representing sinusoidal signals as phasors, engineers can perform algebraic operations such as addition, subtraction, multiplication, and division, which are much simpler than working with complex sinusoidal functions. Phasors are particularly useful when dealing with AC circuits, where multiple sinusoidal sources with different frequencies and phases may be present. The use of phasors is a fundamental concept in electrical engineering, enabling engineers to simplify the analysis of complex circuits and quickly obtain accurate results.



IMPEDANCE AND ADMITTANCE OF STEADY STATE ANALYSIS:-

ADMITTANCE

admittance is a measure of how easily a circuit or device will allow a current to flow. It is defined as the reciprocal of impedance, analogous to how conductance & resistance are defined. The SI unit of admittance is the siemens (symbol S); the older, synonymous unit is mho, and its symbol is ℧ (an upside-down uppercase omega Ω). Oliver Heaviside coined the term admittance in December 1887.[1] Heaviside used Y to represent the magnitude of admittance, but it quickly became the conventional symbol for admittance itself through the publications of Charles Proteus Steinmetz. Heaviside probably chose Y simply because it is next to Z in the alphabet, the conventional symbol for impedance.[2]

Admittance is defined as

Where

Y=1/Z

Y is the admittance, measured in Siemens
Z is the impedance, measured in ohms

Resistance is a measure of the opposition of a circuit to the flow of a steady current, while impedance takes into account not only the resistance but also dynamic effects (known as reactance). Likewise, admittance is not only a measure of the ease with which a steady current can flow, but also the dynamic effects of the material's susceptance to polarization:

Where

Y=G+jB

- Y is the admittance, measured in Siemens.
- G is the conductance, measured in Siemens.
- B is the susceptance, measured in Siemens.

- The dynamic effects of the material's susceptance relate to the universal dielectric response, the power law scaling of a system's admittance with frequency under alternating current conditions.

IMPEDANCE

Parts of this article or section rely on the reader's knowledge of the complex impedance representation of capacitors and inductors and on knowledge of the frequency domain representation of signals.

The impedance, Z, is composed of real and imaginary parts,

where

- *R* is the resistance, measured in ohms
- *X* is the reactance, measured in ohms

Admittance, just like impedance, is a complex number, made up of a real part (the conductance, *G*), and an imaginary part (the susceptance, *B*), thus:

where *G* (conductance) and *B* (susceptance) are given by:

The magnitude and phase of the admittance are given by:

where

- *G* is the conductance, measured in Siemens
- *B* is the susceptance, also measured in Siemens

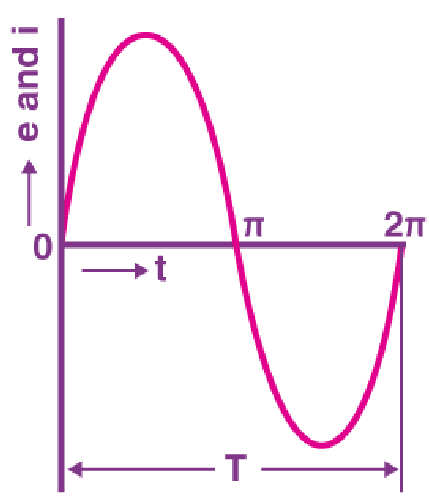
Note that (as shown above) the signs of reactance become reversed in the admittance domain; i.e. capacitive susceptance is positive and inductive susceptance is negative.

AC CIRCUIT ANALYSIS:

The current in a simple circuit is calculated by dividing the voltage by the resistance. The peak current (derived by dividing the peak voltage by the resistance), the angular frequency, and the time are used to compute the ac current.

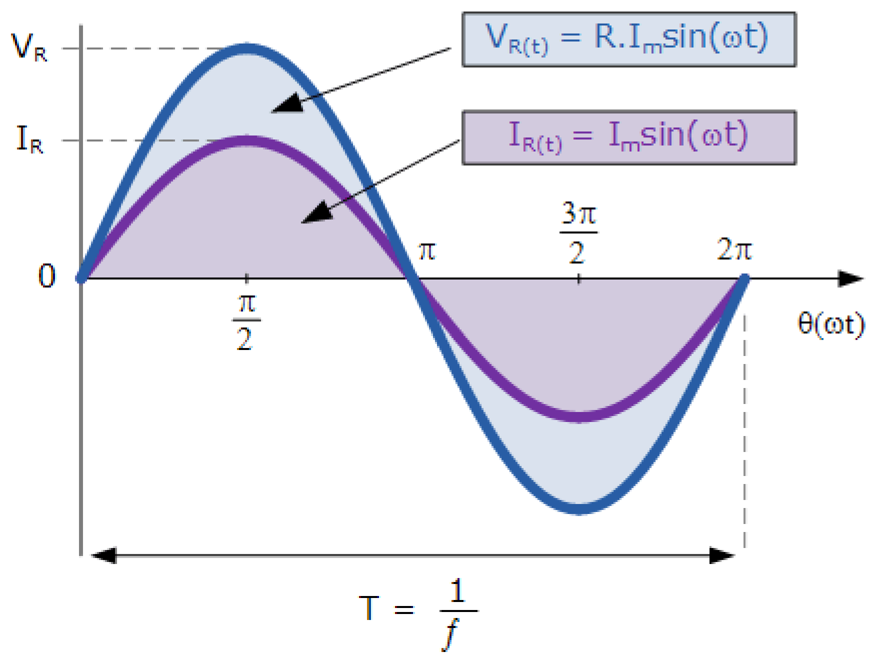
An alternating current (AC) is an electrical current that regularly reverses direction and changes its value constantly with time, contrary to DC current, which travels only in a single direction.

For many decades of electric power, the sinusoidal current and voltage have been used in power businesses and homes.



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.

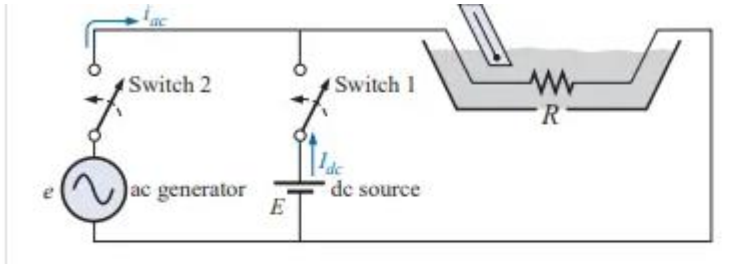


This "in-phase" effect can also be represented by a phasor diagram. In the complex domain, resistance is a real number only meaning that there is no "j" or imaginary component. Therefore, as the voltage and current are both in-phase with each other, there will be no phase difference ($\theta = 0$) between them, so the vectors of each quantity are drawn super-imposed upon one another along the same reference axis

EFFECTIVE OR RMS VALUES

This section will begin to relate dc and ac quantities with respect to the power delivered to a load. It will help us determine the amplitude of a sinusoidal ac current required to deliver the same power as a particular dc current. The question frequently arises, How is it possible for a sinusoidal ac quantity to deliver a net power if, over a full cycle, the net current in any one direction is zero (average value = 0)? It would almost appear that the power delivered during the positive portion of the sinusoidal waveform is withdrawn during the negative portion, and since the two are equal in magnitude, the net power delivered is zero.

However, understand that irrespective of direction, current of any magnitude through a resistor will deliver power to that resistor. In other words, during the positive or negative portions of a sinusoidal ac current, power is being delivered at each instant of time to the resistor. The power delivered at each instant will, of course, vary with the magnitude of the sinusoidal ac current, but there will be a net flow during either the positive or the negative pulses with a net flow over the full cycle. The net power flow will equal twice that delivered by either the positive or the negative regions of sinusoidal quantity.



A fixed relationship between ac and dc voltages and currents can be derived from the experimental setup shown in Fig. 1. A resistor in a water bath is connected by switches to a dc and an ac supply. If switch 1 is closed, a dc current I, determined by the resistance R and battery voltage E, will be established through the resistor R. The temperature reached by the water is determined by the dc power dissipated in the form of heat by the resistor.

The temperature reached by the water is now determined by the ac power dissipated in the form of heat by the resistor. The ac input is varied until the temperature is the same as that reached with the dc input. When this is accomplished, the average electrical power delivered to the resistor R by the ac source is the same as that delivered by the dc source.
The power delivered by the ac supply at any instant of time,

$$\begin{aligned} P_{ac} &= (i_{ac})^2 R = (I_m \sin wt)^2 R \\ &= I_m^2 \sin^2 wt\, R \end{aligned}$$

$$\sin^2 wt = \frac{1}{2}(1 - \cos 2wt$$

$$P_{ac} = I_m^2 R \frac{1}{2}(1 - \cos 2wt)$$

$$\boxed{P_{ac} = \frac{I_m^2 R}{2} - \frac{I_m^2 R}{2} \cos 2wt}$$

The average power delivered by the ac source is just the first term, since the average value of a cosine wave is zero even though the wave may have twice the frequency of the original input current waveform. Equating the average power delivered by the ac generator to that delivered by the dc source,

$$P_{av(ac)} = P_{dc}$$

$$\frac{I_m^2 R}{2} = I_{dc}^2 R$$

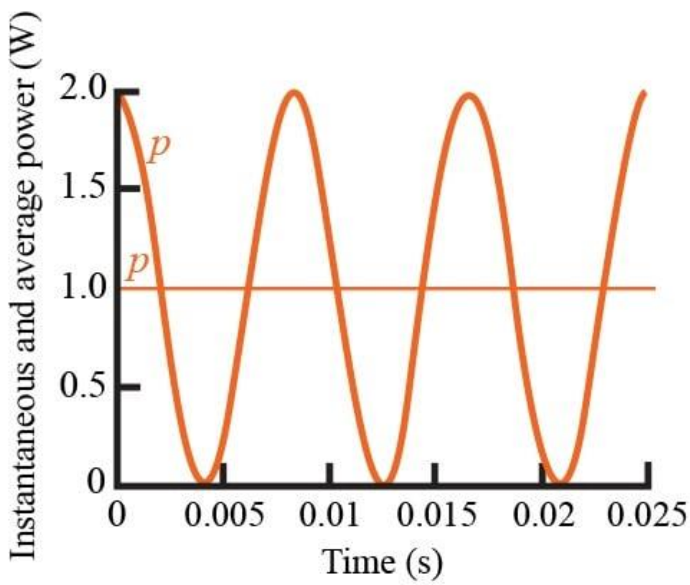
$$I_m = \sqrt{2} I_{dc}$$

$$\boxed{I_{dc} = \frac{I_m}{\sqrt{2}} = 0.707 I_m}$$

AVERAGE POWER AND COMPLEX POWER

AVERAGE POWER:-

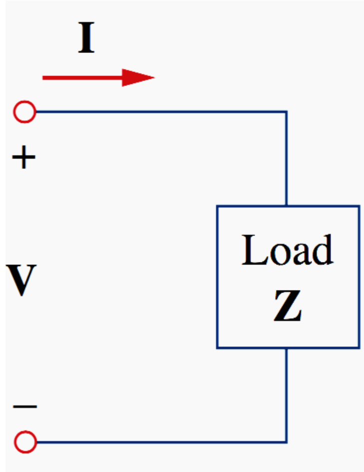
The average power, P, is the average of p, over one period. This can be seen by looking at the graph where P=1 for the circuit. Instantaneous real power can never be negative; in other words, power cannot be removed from a purely resistive network.



COMPLEX POWER:-

Considerable effort has been expended over the years **to express power relations** as simply as possible. Power engineers have coined the term complex power, which they use to find the total effect of parallel loads.

Complex power is important in power analysis because it contains all the information pertaining to the power absorbed by a given load.



Consider the AC load in Figure 1 above. Given the phasor form **V = Vm∠θv** and **I = Im∠θi** of voltage v(t) and current i(t), the **complex power S** absorbed by the AC load is the product of the voltage and the complex conjugate of the current.

the voltage and the complex conjugate of the current, or:

$$\mathbf{S} = \frac{1}{2}\mathbf{V}\mathbf{I}^* \tag{1.10}$$

assuming the passive sign convention (see Figure 1). In terms of the rms values:

$$\mathbf{S} = \mathbf{V}_{\text{rms}}\mathbf{I}^*_{\text{rms}} \tag{1.11}$$

where

$$\mathbf{V}_{\text{rms}} = \frac{\mathbf{V}}{\sqrt{2}} = V_{\text{rms}}\underline{/\theta_v} \tag{1.12}$$

and

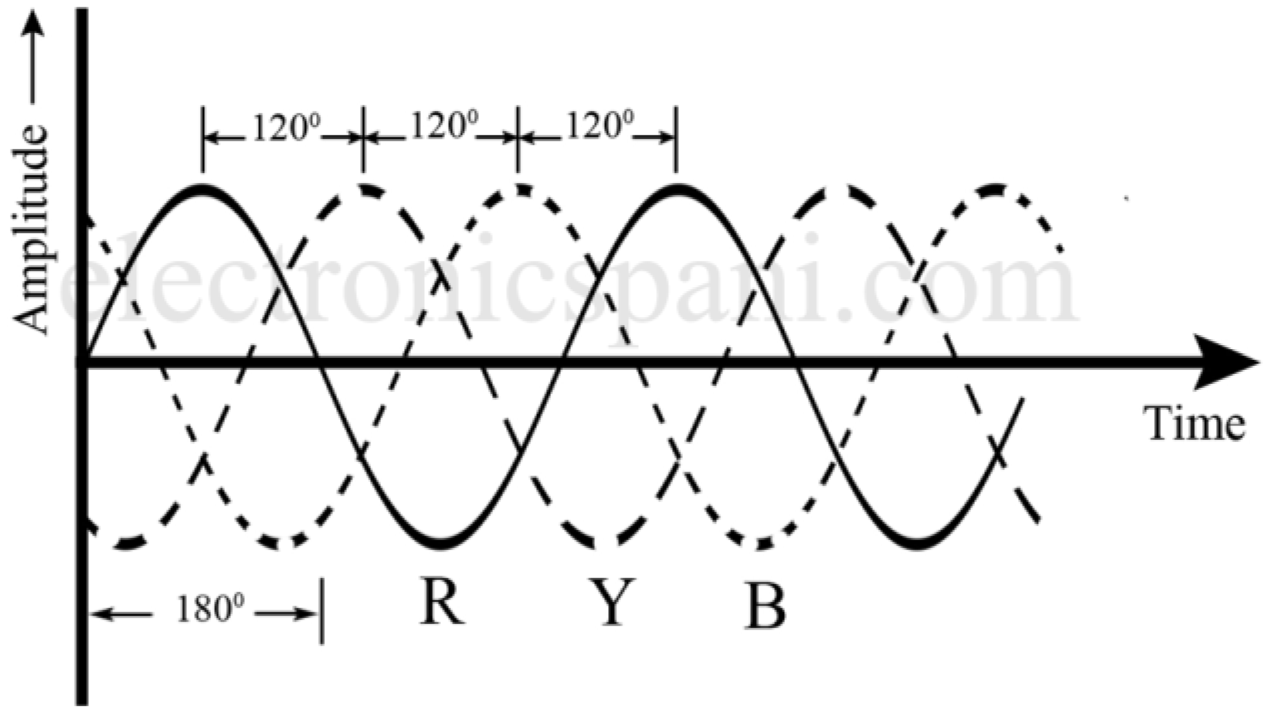
$$\mathbf{I}_{\text{rms}} = \frac{\mathbf{I}}{\sqrt{2}} = I_{\text{rms}}\underline{/\theta_i} \tag{1.13}$$

Thus we may write Eq. (1.11) as:

$$\begin{aligned}\mathbf{S} &= V_{\text{rms}}I_{\text{rms}}\underline{/\theta_v - \theta_i} \\ &= V_{\text{rms}}I_{\text{rms}}\cos(\theta_v - \theta_i) + jV_{\text{rms}}I_{\text{rms}}\sin(\theta_v - \theta_i)\end{aligned} \tag{1.14}$$

THREE PHASE CIRCUITS

The three-phase system has three live wire and one returns the path. The three-phase system is used for transmitting a large amount of power. The **3 phase system** is divided mainly into two types. One is a balanced three-phase system and another one is an unbalanced three-phase system.



It is always better to solve the balanced three-phase circuits on the basis of each phase. When the three-phase supply voltage is given without reference to the line or phase value, then it is the line voltage which is taken into consideration.

The following steps are given below to solve the balanced three-phase circuits.

Step 1 – First of all draw the circuit diagram.

Step 2 – Determine XLP = XL/phase = 2πfL.

Step 3 – Determine XCP = XC/phase = 1/2πfC.

Step 4 – Determine XP = X/ phase = XL – XC

Step 5 – Determine ZP = Z/phase = √R2P + X2P

Step 6 – Determine cosϕ = RP/ZP; the power factor is lagging when XLP > XCP and it is leading when XCP > XLP.

Step 7 – Determine the V phase.

For star connection VP = VL/√3 and for delta connection VP = VL

Step 8 – Determine IP = VP/ZP.

Step 9 – Now, determine the line current IL.

For star connection IL = IP and for delta connection IL = √3 IP

Step 10 – Determine the Active, Reactive and Apparent power.

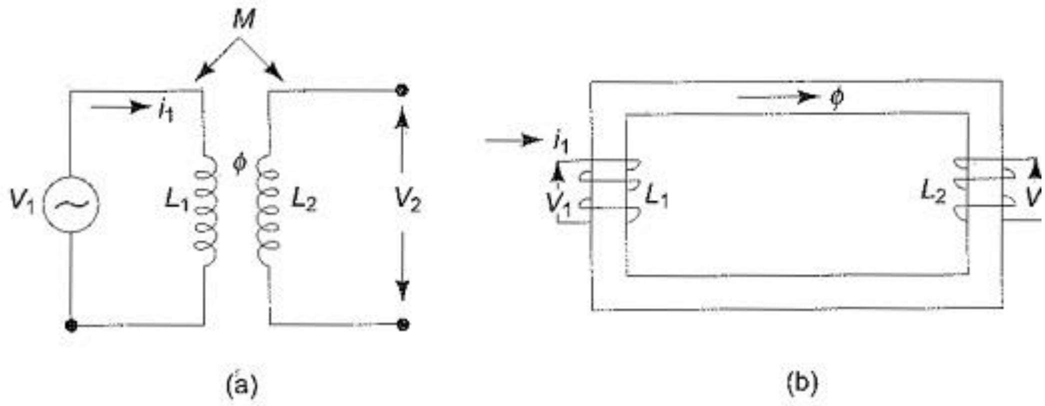
MUTUAL COUPLED CIRCUITS

By Mutual coupled circuits we mean two or more circuits, often in the form of multi-turn coils sharing a magnetic circuit, where the magnetic flux produced by the current in one coil not only links with its own winding, but also with those of the other coils. The coupling medium is the magnetic field, and as we will see the electrical effect of the coupling is manifested when the flux changes.

We know from Faraday's law that when the magnetic flux (ϕ) linking a coil changes, an e.m.f. (e) is induced in the coil, given by

i.e. the e.m.f. is proportional to the rate of change of the flux. (The minus sign indicates that if the induced e.m.f. is allowed to drive a current, the m.m.f. produced by the current will be in opposition to that which produced the original changing flux.) This equation only applies if all the flux links all N turns of the coil, the situation most commonly approached in transformer windings that share a common magnetic circuit, and are thus very tightly coupled.

We have seen that windings for induction motors are distributed, and the flux wave produced by the current in the winding is also distributed around the air-gap. As a result not all of the flux produced by one winding links with all of its turns, and we have to perform a summation (integration) of all the 'turns times flux that links them' contributions to find the 'total effective self flux linkage', which we denote by the symbol psi (ψ). The e.m.f. induced when the self-produced flux linkage changes in, say, a stator winding (subscript S) is then given by

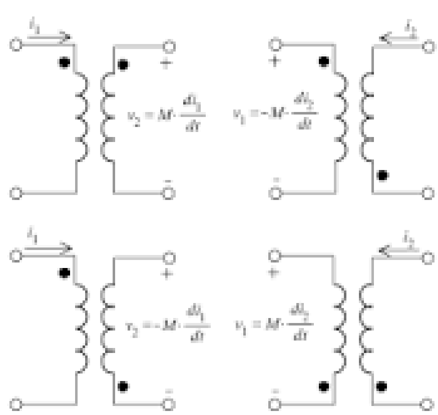


In an induction motor there are three distributed windings on the stator, and either a cage or three more distributed windings on the rotor, and some of the flux produced by current in any one of the windings will link all of the others. We term this 'mutual flux linkage', and often denote it by a double suffix: for example, the symbol ψSR is the mutual flux linkage between a stator winding and a rotor winding.

In the same way that an e.m.f. is induced in a winding when its self-produced flux changes, so also are e.m.f.s induced in all other windings that are mutually coupled to it. For example, if the flux produced by the stator winding changes, the e.m.f. in the rotor (subscript R).

DOT CONVENTION IN COUPLED CIRCUITS

If the current enters the dotted terminal of one coil, the voltage will be positive at the dot on the second coil. Similarly, the voltage of the second coil will be negative if the current leaves the dotted terminal of the first coil.

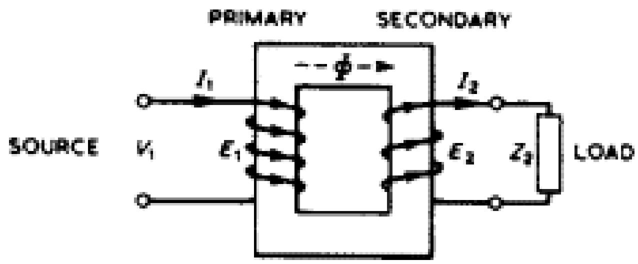


This voltage is the voltage induced by the coupled current. A transformer can have current entering from both the first and second coils. The voltage across each coil will be dependent on the current through this coil and the induced voltage from the other coil. Suppose the dots are on the same end, and both currents enter at the dots.

$$v_1 = L_1 \cdot \frac{d\,i_1}{dt} + M \cdot \frac{d\,i_2}{dt} \qquad \text{and} \qquad v_2 = L_2 \cdot \frac{d\,i_2}{dt} + M \cdot \frac{d\,i_1}{dt}$$

IDEAL TRANSFORMER

An ideal transformer comprises two resistance less coils embracing a common magnetic circuit of infinite permeability and zero core loss .The coils produce no leakage flux: i.e. the whole flux of the magnetic circuit completely links both coils. When the primary coil is energised by an alternating voltage *V*1, a corresponding flux of peak value Φm is developed, inducing in the *N*1-turn primary coil an e.m.f. *E*1 = *V*1. At the same time an e.m.f. *E*2 is induced in the *N*2-turn secondary coil. If the terminals of the secondary coil are connected to a load taking a current *I*2, the primary coil must accept a balancing current *I*1 such that *I*1N1 = I2N2, as the core requires zero excitation. The operating conditions are therefore



$$N_1 \,/\, N_2 = E_1 \,/\, E_2 = I_2 \,/\, I_1; \text{and} E_1 I_1 = E_2 I_2$$

The secondary load impedance *Z*2 = *E*2/*I*2 is reflected into the primary to give the impedance *Z*1 = *E*1/*I*1 such That

$$Z_1 = (N_1 \,/\, N_2)^2 Z_2$$

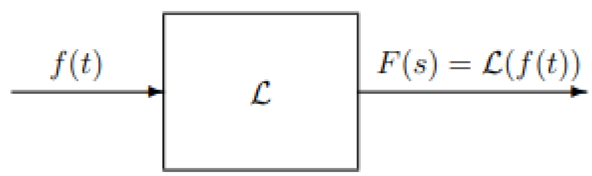
A practical power transformer differs from the ideal in that its core is not infinitely permeable and demands an excitation *N*1*I*0 = *N*1*I*1 − *N*2*I*2; the primary and secondary coils have both resistance and magnetic leakage; and core losses occur. By treating these effects separately, a practical transformer may be considered as an ideal transformer connected into an external network to account for the defects.

UNIT-IV

ELECRICAL CIRCUIT ANALYSIS USING LAPLACE TRANSFORMS

REVIEW OF LAPLACE TRANSFORM

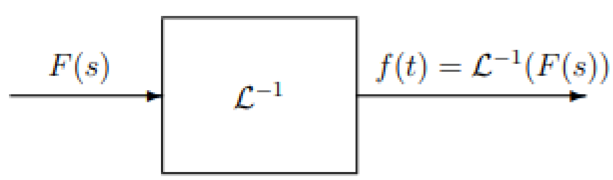
The Laplace transform is an operator that transforms a function of time, f(t), into a new function of complex variable, F(s), where s = σ+jω, as illustrated in Figure 1. The operator L denotes that the time function f(t) has been transformed to its Laplace transform, denoted F(s). The Laplace transform is very useful in solving linear differential equations and hence



in analyzing control systems. To obtain the Laplace transform of the given function of time, f(t), 1. multiply f(t) by a converging factor e −st. This is a factor that decreases to zero as time increases to infinity; 2. Integrate f(t)e −st with respect to time between the limits 0− and ∞ to obtain the Laplace transform of f(t),

$$\boxed{F(s) = \mathcal{L}(f(t)) = \int_{0^-}^{\infty} f(t)e^{-st}dt}$$

The lower limit of integration is 0−, rather than 0, to account for the effect of "instantaneous energy transfer".



The above definition of the Laplace transform is also referred to as the one-sided or unilateral Laplace transform. In the two-sided, or bilateral, Laplace transform, the lower limit is −∞. For our purposes the one-sided Laplace transform is sufficient. If we want to reverse the operation and take the inverse transform, back to the time domain, we write L −1 (F(s)) = f(t). Taking the inverse Laplace transform is illustrated .Because we are using the one-sided Laplace transform, we define all functions, whose Laplace transforms we compute, to be zero for t < 0 −.
To proceed, we recall the definition of the unit step function,
u(t), u(t) = (1 if t ≥ 0 0 if t < 0.

LAPLACE TRANSFORM FOR STANDARD INPUTS

The Laplace transform can be used to solve the different circuit problems. In order to solve the circuit problems, first the differential equations of the circuits are to be written and then these differential equations are solved by using the Laplace transform. Also, the circuit itself may be converted into *s*-domain using Laplace transform and then the algebraic equations corresponding to the circuit can be written and solved.

The electrical circuits can have three circuit elements viz. resistor (R), inductor (L) and capacitor (C).

Pure Resistive Circuit

A circuit consisting of pure resistive element is shown in Figure-1.

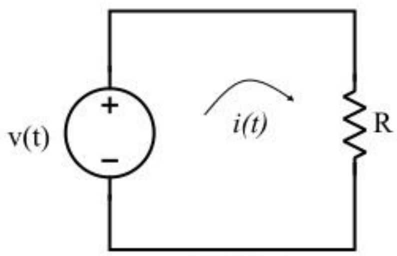


Figure-1

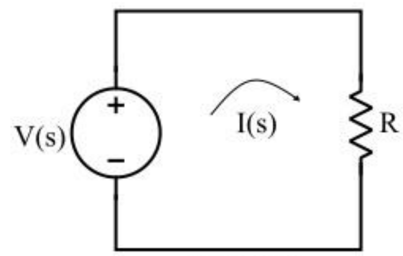


Figure-2

By applying KVL in this circuit, we can write,

$$v(t)=Ri(t)$$

Therefore, the Laplace transform of this equation is given by,

$$V(s)=RI(s)$$

Here, it is noted that the resistance R in t-domain remains R in *s*-domain. The Laplace transformed version of the circuit.

Pure Inductive Circuit

An electrical circuit consisting of pure inductor is shown .

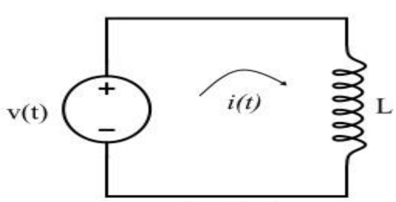


Figure-3

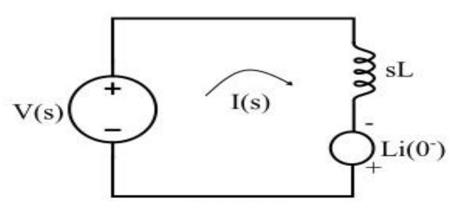


Figure-4

The voltage equation for the pure inductive element is given by,

$$v(t)=L\frac{di(t)}{dt}$$

Taking the Laplace transform of this equation on both sides, we get,

$$V(s)=\left[sI(s)-i\left(0^{-}\right)\right]L$$

$$\Rightarrow V(s)=sLI(s)-Li\left(0^{-}\right)$$

CONVOLUTION INTEGRAL

A convolution is an integral that expresses the amount of overlap of one function as it is shifted over another function . It therefore "blends" one function with another. For example, in synthesis imaging, the measured dirty map is a convolution of the "true" CLEAN map with the dirty beam (the Fourier transform of the sampling distribution). The convolution is sometimes also known by its German name, faltung ("folding").

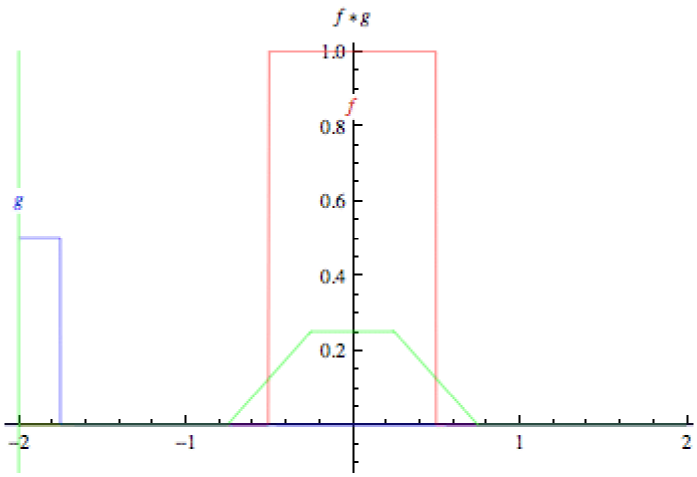
Convolution is implemented in the Wolfram Language as Convolve[f, g, x, y] and DiscreteConvolve[f, g, n, m].

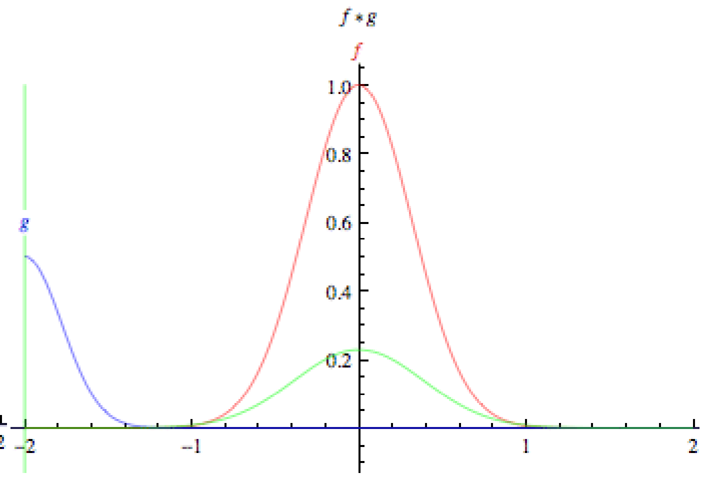
$$[f*g](t)\equiv\int_0^t f(\tau)\,g(t-\tau)\,d\tau,$$

where the symbol $[f*g](t)$ denotes convolution of f and g.

Convolution is more often taken over an infinite range,

$$\begin{aligned} f*g &\equiv \int_{-\infty}^{\infty} f(\tau)\,g(t-\tau)\,d\tau \\ &= \int_{-\infty}^{\infty} g(\tau)\,f(t-\tau)\,d\tau \end{aligned}$$





The animations above graphically illustrate the convolution of two boxcar functions (left) and two Gaussians (right). In the plots, the green curve shows the convolution of the blue and red curves as a function of, the position indicated by the vertical green line. The gray region indicates the product as a function of, so its area as a function of is precisely the convolution. One feature to emphasize and which is not conveyed by these illustrations (since they both exclusively involve symmetric functions) is that the function must be mirrored before lagging it across and integrating.

$$\begin{aligned}\int_{-\infty}^{\infty}(f * g)\,dt &= \int_{-\infty}^{\infty}\left[\int_{-\infty}^{\infty} f(u)\,g(t-u)\,du\right]dt \\ &= \int_{-\infty}^{\infty} f(u)\left[\int_{-\infty}^{\infty} g(t-u)\,dt\right]du \\ &= \left[\int_{-\infty}^{\infty} f(u)\,du\right]\left[\int_{-\infty}^{\infty} g(t)\,dt\right].\end{aligned}$$

INVERSE LAPLACE TRANSFORM

The derivative of s is jω; when finding what X(s) equals when Y(s) equals zero, we find the inverse function (X(-s)) and remember that multiplying both sides by j gives complex conjugate. We can multiply both sides by j to get rid of the (-s).

The Inverse Laplace Transform takes the output Y(s) and finds what X(s) it is in terms of, for a given transfer function H(s).

Transfer Functions: The transfer function is simply s divided by jω. Since Laplace transforms are linear, the transfer function can be factored into a product of simpler functions.

INVERSE LAPLACE TRANSFORM TABLE

Function y(a)	Transform Y(b)	b
1	$\frac{1}{b}$	b>0
a	$\frac{1}{b^2}$	b>0
A^i, i = integer	$\frac{i!}{s(i+1)}$	b>0
exp (ta), where t = constant	$\frac{1}{(b-t)}$	b>t
cos (sa), s= constant	$\frac{b}{b^2+s^2}$	b>0
Sin (sa), s = constant	$\frac{t}{b^2+s^2}$	b>0

Example 1 Compute the inverse Laplace transform of Y (s) = $\frac{2}{3-5s}$.

Solution Adjust it as follows:

$$Y(s) = \frac{2}{3-5s} = \frac{-2}{5} . \frac{1}{s-\frac{3}{5}}$$

Thus, by linearity,

$$Y(t) = L^{-1}[\frac{-2}{5} . \frac{1}{s-\frac{3}{5}}]$$

$$= \frac{-2}{5} L^{-1}[\frac{1}{s-\frac{3}{5}}]$$

$$= \frac{-2}{5} e^{(\frac{3}{5})t}$$

TRANSFORMED NETWORK WITH INITIAL CONDITIONS

Conditions for initial value theorem are

1. If t approaches to 0+, function f(t) should exist

$$\left[i.e. \lim_{t \to 0^+} f(t) \ should \ exist\right]$$

Function f(t) and its derivative f'(t) should be laplace transformable.

Statement

Initial value theorem is given by

$$\lim_{t \to 0^+} f(t) = \lim_{s \to \infty} sF(s) = f(0^+)$$

Where F(s) is laplace transform of f(t).

PROOF

We know that,

$L[f'(t)] = s\,L[f(t)] - f(0) = sF(s) - f(0)$

$\therefore sF(s)$

$= L[f'(t)] + f(0)$

$= \int_0^\infty e^{-st} f'(t)dt + f(0)$

Taking limit as $s \to \infty$ on both sides,

we have

$$\lim_{s\to\infty} sF(s) = \lim_{s\to\infty} \left[\int_0^\infty e^{-st} f'(t)dt + f(0)\right]$$

$$= \lim_{s\to\infty} \left[\int_0^\infty e^{-st} f'(t)dt\right] + f(0)$$

$$= \int_0^\infty \lim [\, e^{-st} f'(t)]dt + f(0)$$

$$= 0 + f(0) \qquad \because e^{-\infty} = 0$$

$$= f(0)$$

$$= \lim_{t\to 0} f(t)$$

TRANSFER FUNCTION REPRESENTATION

Control System Toolbox™ software supports transfer functions that are continuous-time or discrete-time, and SISO or MIMO. You can also have time delays in your transfer function representation.

A SISO continuous-time transfer function is expressed as the ratio:

$$G(s) = \frac{N(s)}{D(s)},$$

of polynomials $N(s)$ and $D(s)$, called the numerator and denominator polynomials, respectively.

You can represent linear systems as transfer functions in polynomial or factorized (zero-pole-gain) form. For example, the polynomial-form transfer function:

$$\sum_{i=0}^{n} a_i y^{(n-i)} = \sum_{i=0}^{m} b_i x^{(m-i)}$$

$$H(s) = \frac{Y(s)}{X(s)} = \frac{\sum_{i=0}^{m} b_i s^{m-i}}{\sum_{i=0}^{n} a_i s^{n-i}} = \frac{\mathcal{L}(\text{output})}{\mathcal{L}(\text{input})}$$

$$G(s) = \frac{s^2 - 3s - 4}{s^2 + 5s + 6}$$

can be rewritten in factorized form as:

$$G(s) = \frac{(s+1)(s-4)}{(s+2)(s+3)}.$$

The tf model object represents transfer functions in polynomial form. The zpk model object represents transfer functions in factorized form.

MIMO transfer functions are arrays of SISO transfer functions. For example:

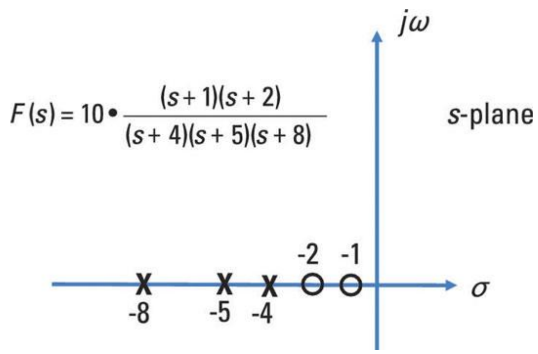
$$G(s) = \begin{bmatrix} \dfrac{s-3}{s+4} \\ \dfrac{s+1}{s+2} \end{bmatrix}$$

is a one-input, two output transfer function.

POLES AND ZEROS

Poles and Zeros of a transfer function are the frequencies for which the value of the denominator and numerator of transfer function becomes infinite and zero respectively. The values of the poles and the zeros of a system determine whether the system is stable, and how well the system performs. Control systems, in the simplest sense, can be designed simply by assigning specific values to the poles and zeros of the system.

Let's say we have a transfer function defined as a ratio of two polynomials:



Where N(s) and D(s) are simple polynomials. Zeros are the roots of N(s) (the numerator of the transfer function) obtained by setting N(s) = 0 and solving for s.

The polynomial order of a function is the value of the highest exponent in the polynomial.

Poles are the roots of D(s) (the denominator of the transfer function), obtained by setting D(s) = 0 and solving for s. Because of our restriction above, that a transfer function must not have more zeros than poles, we can state that the polynomial order of D(s) must be greater than or equal to the polynomial order of N(s).

Physically realizable control systems must have a number of poles greater than the number of zeros. Systems that satisfy this relationship are called Proper. We will elaborate on this below.

EXAMPLE:

Consider the transfer function:

$$H(s) = \frac{s+2}{s^2+0.25}$$

We define *N(s)* and *D(s)* to be the numerator and denominator polynomials, as such:

$$N(s) = s+2$$

$$D(s) = s^2+0.25$$

We set *N(s)* to zero, and solve for *s*:

$$N(s) = s+2 = 0 \to s = -2$$

So we have a zero at *s* → *-2*. Now, we set *D(s)* to zero, and solve for *s* to obtain the poles of the equation:

$$D(s) = s^2+0.25 = 0 \to s = +i\sqrt{0.25}, -i\sqrt{0.25}$$

And simplifying this gives us poles at: *-i/2* , *+i/2*. Remember, *s* is a complex variable, and it can therefore take imaginary and real values.

EFFECTS:

As *s* approaches a zero, the numerator of the transfer function (and therefore the transfer function itself) approaches the value 0. When *s* approaches a pole, the denominator of the transfer function approaches zero, and the value of the transfer function approaches infinity. An output value of infinity should raise an alarm bell for people who are familiar with BIBO stability. We will discuss this later.

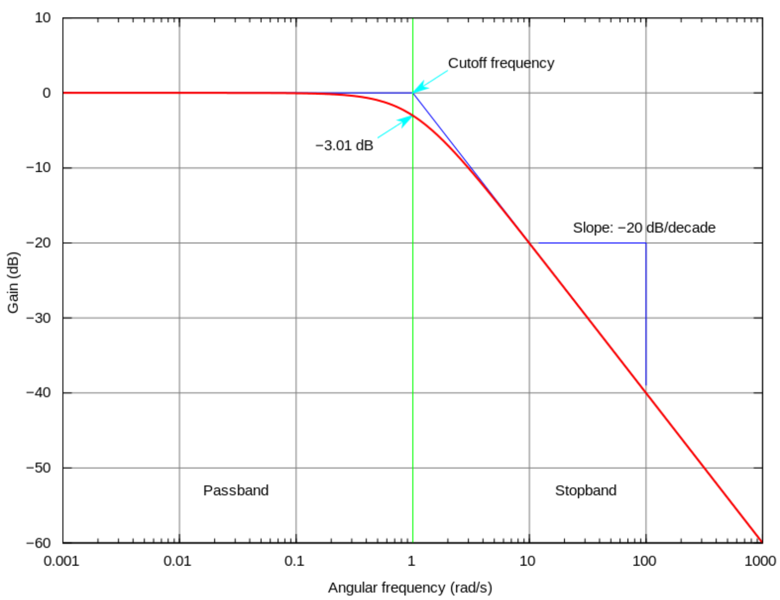
As we have seen above, the locations of the poles, and the values of the real and imaginary parts of the pole determine the response of the system. Real parts correspond to exponentials, and imaginary parts correspond to sinusoidal values.

Addition of poles to the transfer function has the effect of pulling the root locus to the right, making the system less stable. Addition of zeros to the transfer function has the effect of pulling the root locus to the left, making the system more stable.

FREQUENCY RESPONSE

In signal processing and electronics, the frequency response of a system is the quantitative measure of the magnitude and phase of the output as a function of input frequency.[1] The frequency response is widely used in the design and analysis of systems, such as audio and control systems, where they simplify mathematical analysis by converting governing differential equations into algebraic equations. In an audio system, it may be used to minimize audible distortion by designing components (such as microphones, amplifiers and loudspeakers) so that the overall response is as flat (uniform) as possible across the system's bandwidth. In control systems, such as a vehicle's cruise control, it may be used to assess system stability, often through the use of Bode plots. Systems with a specific frequency response can be designed using analog and digital filters.

The frequency response characterizes systems in the frequency domain, just as the impulse response characterizes systems in the time domain. In linear systems (or as an approximation to a real system neglecting second order non-linear properties), either response completely describes the system and thus have one-to-one correspondence: the frequency response is the Fourier transform of the impulse response. The frequency response allows simpler analysis of cascaded systems such as multistage amplifiers, as the response of the overall system can be found through multiplication of the individual stages' frequency responses (as opposed to convolution of the impulse response in the time domain). The frequency response is closely related to the transfer function in linear systems, which is the Laplace transform of the impulse response.



Several methods using different input signals may be used to measure the frequency response of a system, including:

- Applying constant amplitude sinusoids stepped through a range of frequencies and comparing the amplitude and phase shift of the output relative to the input. The frequency sweep must be slow enough for the system to reach its steady-state at each point of interest
- Applying an impulse signal and taking the Fourier transform of the system's response

- Applying a wide-sense stationary white noise signal over a long period of time and taking the Fourier transform of the system's response. With this method, the cross-spectral density (rather than the power spectral density) should be used if phase information is required

The frequency response is characterized by the magnitude, typically in decibels (dB) or as a generic amplitude of the dependent variable, and the phase, in radians or degrees, measured against frequency, in radian/s, Hertz (Hz) or as a fraction of the sampling frequency.

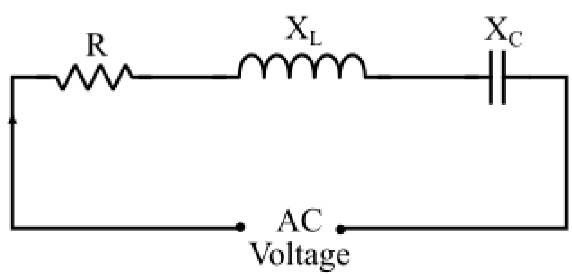
There are three common ways of plotting response measurements:

- Bode plots graph magnitude and phase against frequency on two rectangular plots
- Nyquist plots graph magnitude and phase parametrically against frequency in polar form
- Nichols plots graph magnitude and phase parametrically against frequency in rectangular form.

SERIES AND PARALLEL RESONANCE

SERIES RESONANCE

When resistor (R), inductor (L) and capacitor (C) are connected in series, and at some frequency of supply voltage, the effect of inductor and capacitor cancel each other so that the circuit behaves like a pure resistive circuit, then this condition of the series circuit is known as **series resonance**.



In series resonance, the inductive reactance (XL) and the capacitive reactance (XC) become equal, therefore, the total impedance of the series resonating circuit is equal the resistance of the circuit, i.e

$Z=R$

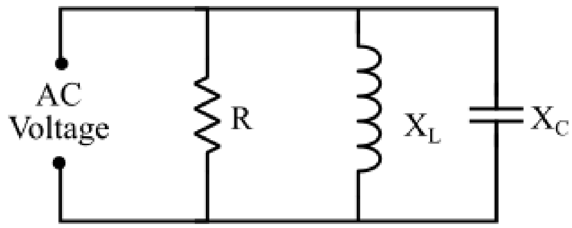
Hence, at the series resonance condition, the circuit offers minimum impedance. Consequently, the value of electric current flowing through the circuit will be maximum. The series resonance results in the maximum admittance in the series RLC circuit.

Some common applications of series resonance are −

- Oscillator circuits
- Voltage amplifiers
- High frequency filters, etc.

PARALLEL RESONANCE

When resistor (R), inductor (L) and capacitor (C) are connected in parallel and the effect of inductor cancels the effect of capacitor at a particular supply frequency, then this condition of the circuit is known as parallel resonance.



The parallel resonance causes maximum impedance in the circuit. As a result, the current flowing through the circuit at parallel resonance is minimum. As the parallel resonance eliminates the effect of capacitor and inductor from the circuit, thus the circuit behaves like a pure resistive circuit. The parallel resonance is also used in many applications such as:

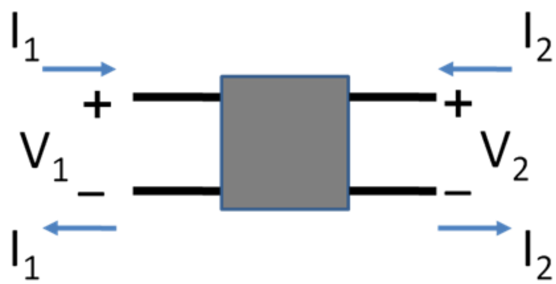
- Current amplifiers
- Filter circuits
- Radio frequency amplifiers
- Induction heating systems, etc.

UNIT-V

TWO PORT NETWORK AND NETWORK FUNCTIONS

TWO PORT NETWORKS

In electronics, a two-port network (a kind of four-terminal network or quadripole) is an electrical network (i.e. a circuit) or device with two pairs of terminals to connect to external circuits. Two terminals constitute a port if the currents applied to them satisfy the essential requirement known as the port condition: the current entering one terminal must equal the current emerging from the other terminal on the same port.



The ports constitute interfaces where the network connects to other networks, the points where signals are applied or outputs are taken. In a two-port network, often port 1 is considered the input port andport 2 is considered the output port.

It is commonly used in mathematical circuit analysis.

The two-port network model is used in mathematical circuit analysis techniques to isolate portions of larger circuits. A two-port network is regarded as a "black box" with its properties specified by a matrix of numbers. This allows the response of the network to signals applied to the ports to be calculated easily, without solving for all the internal voltages and currents in the network. It also allows similar circuits or devices to be compared easily. For example, transistors are often regarded as two-ports, characterized by their h-parameters (see below) which are listed by the manufacturer. Any linear circuit with four terminals can be regarded as a two-port network provided that it does not contain an independent source and satisfies the port conditions.

Examples of circuits analyzed as two-ports are filters, matching networks, transmission lines, transformers, and small-signal models for transistors (such as the hybrid-pi model). The analysis of passive two-port networks is an outgrowth of reciprocity theorems first derived by Lorentz.

In two-port mathematical models, the network is described by a 2 by 2 square matrix of complex numbers. The common models that are used are referred to as z-parameters, y-parameters, h-parameters, g-parameters, and ABCD-parameters, each described individually below. These are all limited to linear networks since an underlying assumption of their derivation is that any given circuit condition is a linear superposition of various short-circuit and open circuit conditions. They are usually expressed in matrix notation, and they establish relations between the variables

$V1$, voltage across port 1
$I1$, current into port 1
$V2$, voltage across port 2
$I2$, current into port 2

which are shown in figure 1. The difference between the various models lies in which of these variables are regarded as the independent variables. These current and voltage variables are most useful at low-to-moderate frequencies. At high frequencies (e.g., microwave frequencies), the use of power and energy variables is more appropriate, and the two-port current–voltage approach is replaced by an approach based upon scattering parameters.

TERMINAL PAIRS

A pair of terminals at which an electrical signal may enter or leave a network is called a **port**. The terminals or port is required for connecting input excitation to the network. It is also required for connecting some other networks such as load. The terminals are most useful for making measurements. In general, the minimum number of terminals required is two.

A network having only one pair of terminals or one port is called one port **network.** The Fig (a) shows a one port network.

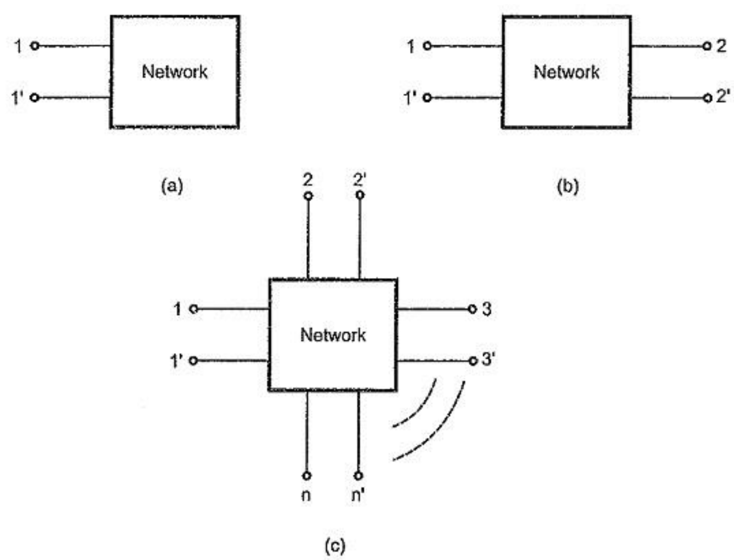


Fig. 6.1 (a) One port network (b) Two port network (c) Multiport network

A network consisting two pairs of terminals is called **two port network** as shown in the Fig. (b). The terminals are generally named as 1-1′ and 2-2′ In general, a port designated 1-1′, is connected to the driving energy source while the other port designated 2-2′ is connected to the load.

A port at which energy source is connected is called **driving point** of the network or **input port**. A port at which load is connected is called **output port**.

Fig. (c) represents a network with n-port called **n-port network**. In such networks, generally one port is connected to energy source, one port is connected to load and other ports may be connected to the different networks.

RELATIONSHIP OF TWO PORT VARIABLES

Here, we have to represent Y parameters in terms of Z parameters. So, in this case Y parameters are the desired parameters and Z parameters are the given parameters.

Step 1 − We know that the following set of two equations, which represents a two port network in terms of Y parameters.

$$I_1 = Y_{11}V_1 + Y_{12}V_2$$

$$I_2 = Y_{21}V_1 + Y_{22}V_2$$

We can represent the above two equations in matrix form as

$$\begin{bmatrix} I_1 \\ I_2 \end{bmatrix} = \begin{bmatrix} Y_{11} & Y_{12} \\ Y_{21} & Y_{22} \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix}$$

Step 2 − We know that the following set of two equations, which represents a two port network in terms of Z parameters.

$$V_1 = Z_{11}I_1 + Z_{12}I_2$$

$$V_2 = Z_{21}I_1 + Z_{22}I_2$$

We can represent the above two equations in matrix form as

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} Z_{11} & Z_{12} \\ Z_{21} & Z_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix}$$

Step 3 − We can modify it as

$$\begin{bmatrix} I_1 \\ I_2 \end{bmatrix} = \begin{bmatrix} Z_{11} & Z_{12} \\ Z_{21} & Z_{22} \end{bmatrix}^{-1} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix}$$

Step 4 − By equating Equation 1 and Equation 2, we will get

$$\begin{bmatrix} Y_{11} & Y_{12} \\ Y_{21} & Y_{22} \end{bmatrix} = \begin{bmatrix} Z_{11} & Z_{12} \\ Z_{21} & Z_{22} \end{bmatrix}^{-1}$$

$$\Rightarrow \begin{bmatrix} Y_{11} & Y_{12} \\ Y_{21} & Y_{22} \end{bmatrix} = \frac{\begin{bmatrix} Z_{22} & -Z_{12} \\ -Z_{21} & Z_{11} \end{bmatrix}}{\Delta Z}$$

Where,

$$\Delta Z = Z_{11} Z_{22} - Z_{12} Z_{21}$$

So, just by doing the inverse of Z parameters matrix, we will get Y parameters matrix.

IMPEDANCE PARAMETERS

Impedance parameters or Z-parameters (the elements of an impedance matrix or Z-matrix) are properties used in electrical engineering, electronic engineering, and communication systems engineering to describe the electrical behavior of linear electrical networks. They are also used to describe the small-signal (linearized) response of non-linear networks. They are members of a family of similar parameters used in electronic engineering, other examples being: S-parameters Y-parameters, H-parameters, T-parameters or ABCD-parameters.

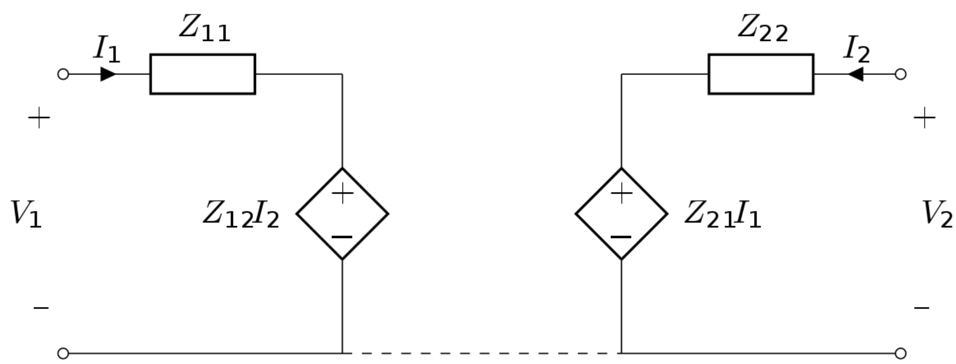
Z-parameters are also known as open-circuit impedance parameters as they are calculated under open circuit conditions. i.e., Ix=0, where x=1,2 refer to input and output currents flowing through the ports (of a two-port network in this case) respectively.

A Z-parameter matrix describes the behaviour of any linear electrical network that can be regarded as a black box with a number of ports. A port in this context is a pair of electrical terminals carrying equal and opposite currents into and out-of the network, and having a particular voltage between them. The Z-matrix gives no information about the behaviour of the network when the currents at any port are not balanced in this way (should this be possible), nor does it give any information about the voltage between terminals not belonging to the same port. Typically, it is intended that each external connection to the network is between the terminals of just one port, so that these limitations are appropriate.

For a generic multi-port network definition, it is assumed that each of the ports is allocated an integer n ranging from 1 to N, where N is the total number of ports. For port n, the associated Z-parameter definition is in terms of the port current and port voltage,and respectively.

For all ports the voltages may be defined in terms of the Z-parameter matrix and the currents by the following matrix equation:

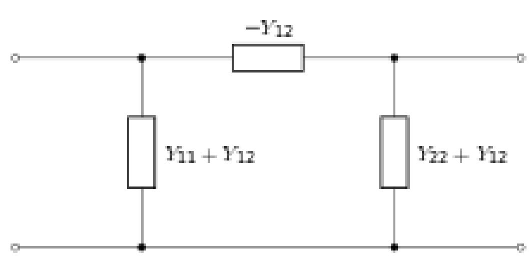
V=ZI



where Z is an N × N matrix the elements of which can be indexed using conventional matrix notation. In general the elements of the Z-parameter matrix are complex numbers and functions of frequency. For a one-port network, the Z-matrix reduces to a single element, being the ordinary impedance measured between the two terminals. The Z-parameters are also known as the open circuit parameters because they are measured or calculated by applying current to one port and determining the resulting voltages at all the ports while the undriven ports are terminated into open circuits.

ADMITTANCE PARAMETERS

Admittance parameters or Y-parameters (the elements of an admittance matrix or Y-matrix) are properties used in many areas of electrical engineering, such as power, electronics, and telecommunications. These parameters are used to describe the electrical behavior of linear electrical networks. They are also used to describe the small-signal (linearized) response of non-linear networks. Y parameters are also known as short circuited admittance parameters. They are members of a family of similar parameters used in electronic engineering, other examples being: S-parameters,[1] Z-parameters,[2] H-parameters, T-parameters or ABCD-parameters.



A Y-parameter matrix describes the behaviour of any linear electrical network that can be regarded as a black box with a number of ports. A port in this context is a pair of electrical terminals carrying equal and opposite currents into and out of the network, and having a particular voltage between them. The Y-matrix gives no information about the behaviour of the network when the currents at any port are not balanced in this way (should this be possible), nor does it give any information about the voltage between terminals not belonging to the same port. Typically, it is intended that each external connection to the network is between the terminals of just one port, so that these limitations are appropriate.

For a generic multi-port network definition, it is assumed that each of the ports is allocated an integer n ranging from 1 to N, where N is the total number of ports. For port n, the associated Y-parameter definition is in terms of the port voltage and port current, Vn and In respectively.

For all ports the currents may be defined in terms of the Y-parameter matrix and the voltages by the following matrix equation:

I=VY

where Y is an N × N matrix the elements of which can be indexed using conventional matrix notation. In general the elements of the Y-parameter matrix are complex numbers and functions of frequency. For a one-port network, the Y-matrix reduces to a single element, being the ordinary admittance measured between the two terminals.

TRANSMISSION PARAMETER

The ABCD-parameters are known variously as chain, cascade, or transmission parameters. There are a number of definitions given for ABCD parameters, the most common is'

$$\begin{bmatrix} V_1 \\ I_1 \end{bmatrix} = \begin{bmatrix} A & B \\ C & D \end{bmatrix} \begin{bmatrix} V_2 \\ -I_2 \end{bmatrix}$$

Note: Some authors chose to reverse the indicated direction of I2 and suppress the negative sign on I2.

Where

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

For reciprocal networks AD – BC = 1. For symmetrical networks A = D. For networks which are reciprocal and lossless, A and D are purely real while B and C are purely imaginary.

This representation is preferred because when the parameters are used to represent a cascade of two-ports, the matrices are written in the same order that a network diagram would be drawn, that is, left to right. However, a variant definition is also in use

$$\begin{bmatrix} V_2 \\ -I_2 \end{bmatrix} = \begin{bmatrix} A' & B' \\ C' & D' \end{bmatrix} \begin{bmatrix} V_1 \\ I_1 \end{bmatrix}$$

where

$$A' \stackrel{\text{def}}{=} \left. \frac{V_2}{V_1} \right|_{I_1=0} \qquad B' \stackrel{\text{def}}{=} \left. \frac{V_2}{I_1} \right|_{V_1=0}$$

$$C' \stackrel{\text{def}}{=} \left. -\frac{I_2}{V_1} \right|_{I_1=0} \qquad D' \stackrel{\text{def}}{=} \left. -\frac{I_2}{I_1} \right|_{V_1=0}$$

The negative sign of –I2 arises to make the output current of one cascaded stage (as it appears in the matrix) equal to the input current of the next. Without the minus sign the two currents would have opposite senses because the positive direction of current, by convention, is taken as the current entering the port. Consequently, the input voltage/current matrix vector can be directly replaced with the matrix equation of the preceding cascaded stage to form a combined A'B'C'D' matrix.

The terminology of representing the ABCD parameters as a matrix of elements designated a11 etc. as adopted by some authors and the inverse A'B'C'D' parameters as a matrix of elements designated b11 etc. is used here for both brevity and to avoid confusion with circuit elements.

$$[\mathbf{a}] = \begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix} = \begin{bmatrix} A & B \\ C & D \end{bmatrix}$$

$$[\mathbf{b}] = \begin{bmatrix} b_{11} & b_{12} \\ b_{21} & b_{22} \end{bmatrix} = \begin{bmatrix} A' & B' \\ C' & D' \end{bmatrix}$$

Transmission parameters:

Element	[a] matrix	[b] matrix	Remarks
Series impedance	$\begin{bmatrix} 1 & Z \\ 0 & 1 \end{bmatrix}$	$\begin{bmatrix} 1 & -Z \\ 0 & 1 \end{bmatrix}$	Z, impedance
Shunt admittance	$\begin{bmatrix} 1 & 0 \\ Y & 1 \end{bmatrix}$	$\begin{bmatrix} 1 & 0 \\ -Y & 1 \end{bmatrix}$	Y, admittance
Series inductor	$\begin{bmatrix} 1 & sL \\ 0 & 1 \end{bmatrix}$	$\begin{bmatrix} 1 & -sL \\ 0 & 1 \end{bmatrix}$	L, inductance s, complex angular frequency
Shunt inductor	$\begin{bmatrix} 1 & 0 \\ \frac{1}{sL} & 1 \end{bmatrix}$	$\begin{bmatrix} 1 & 0 \\ -\frac{1}{sL} & 1 \end{bmatrix}$	L, inductance s, complex angular frequency
Series capacitor	$\begin{bmatrix} 1 & \frac{1}{sC} \\ 0 & 1 \end{bmatrix}$	$\begin{bmatrix} 1 & -\frac{1}{sC} \\ 0 & 1 \end{bmatrix}$	C, capacitance s, complex angular frequency
Shunt capacitor	$\begin{bmatrix} 1 & 0 \\ sC & 1 \end{bmatrix}$	$\begin{bmatrix} 1 & 0 \\ -sC & 1 \end{bmatrix}$	C, capacitance s, complex angular frequency
Transmission line	$\begin{bmatrix} \cosh(\gamma l) & Z_0 \sinh(\gamma l) \\ \frac{1}{Z_0} \sinh(\gamma l) & \cosh(\gamma l) \end{bmatrix}$	$\begin{bmatrix} \cosh(\gamma l) & -Z_0 \sinh(\gamma l) \\ -\frac{1}{Z_0} \sinh(\gamma l) & \cosh(\gamma l) \end{bmatrix}$ [15]	Z_0, characteristic impedance γ, propagation constant ($\gamma = \alpha + i\beta$) l, length of transmission line (m)

HYBRID PARAMETERS

Hybrid parameters (also known as **h parameters**) are known as 'hybrid' parameters as they use Z parameters, Y parameters, voltage ratio, and current ratios to represent the relationship between voltage and current in a two port network. **H parameters** are useful in describing the input-output characteristics of circuits where it is hard to measure Z or Y parameters (such as in a transistor).
H parameters encapsulate all the important linear characteristics of the circuit, so they are very useful for simulation purposes. The relationship between voltages and current in h parameters can be represented as:

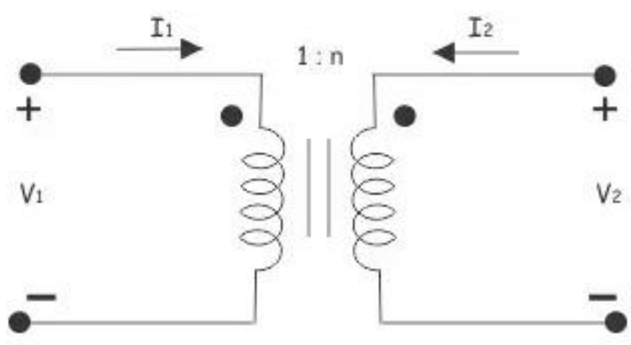
$$V_1 = h_{11} I_1 + h_{12} V_2$$
$$I_2 = h_{21} I_1 + h_{22} V_2$$

This can be represented in matrix form as:

$$\begin{bmatrix} V_1 \\ I_2 \end{bmatrix} = \begin{bmatrix} h_{11} & h_{12} \\ h_{21} & h_{22} \end{bmatrix} \begin{bmatrix} I_1 \\ V_2 \end{bmatrix}$$

To illustrate where h parameters are useful, take the case of an ideal transformer, where Z parameters cannot be used. Since here, the relations between voltages and current in that ideal transformer would be,

$$V_1 = \frac{1}{n} V_2 \; and \; I_1 = -nI_2$$



Since an ideal transformer's voltage can not be expressed in terms of current, it is impossible to analyze a transformer with Z parameters because a transformer does not have Z parameters. The problem can instead be solved by using **hybrid parameters** (i.e. **h parameters**).

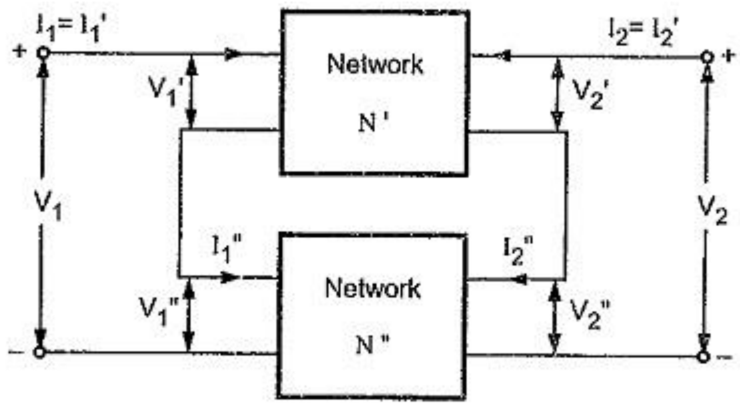
INTERCONNECTIONS OF TWO PORT NETWORKS

Various types of Interconnection of Two Port Network such as series connection of two ports, parallel connection of two ports, cascade connection of two ports etc.

Series Connection of Two Ports:

Consider two networks N' and N" are connected in series as shown in Fig. 6.8 (a). When two ports are connected in series, we can add their z parameters to get overall z-parameter of the overall series connection.

Let the z-parameters of network N' be z'11, z'12, z'21, z'22. Let the z parameters of network N" be z"11, z"12, z"21, z"22. Let the overall z-parameters of series connection be z11, z12, z21, z22.



For series connection we have,

$$V_1 = V_1' + V_1'' \quad \ldots (1)$$

$$V_2 = V_2' + V_2'' \quad \ldots (2)$$

$$I_1 = I_1' = I_1'' \quad \ldots (3)$$

$$I_2 = I_2' = I_2'' \quad \ldots (4)$$

For network N' , z-parameter equations are,

$$V_1' = z_{11}'I_1' + z_{12}'I_2'$$

$$V_2' = z_{21}'I_1' + z_{22}'I_2'$$

For network N", z-parameter equations are,

$$V_1'' = z_{11}''I_1'' + z_{12}''I_2''$$

$$V_2'' = z_{21}''I_1'' + z_{22}''I_2''$$

From equations (1) , (2) and (3) , (4) we can write,

$$V_1 = (z'_{11} + z''_{11})I_1 + (z'_{12} + z''_{12})\,I_2$$
$$V_2 = (z'_{21} + z''_{21})I_1 + (z'_{22} + z''_{22})\,I_2$$

In the Matrix form, above equations can be written as,

$$\begin{bmatrix} V_1 \\ V_2 \end{bmatrix} = \begin{bmatrix} (z'_{11} + z''_{11}) & (z'_{12} + z''_{12}) \\ (z'_{21} + z''_{21}) & (z'_{22} + z''_{22}) \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \end{bmatrix}$$

Thus, overall z-parameters are,

$$[z] = \begin{bmatrix} z'_{11} + z''_{11} & z'_{12} + z''_{12} \\ z'_{21} + z''_{21} & z'_{22} + z''_{22} \end{bmatrix}$$

Hence, the z-parameters of the series connection are the sum of z-parameters of the individual network connected in series.

Parallel Connection of Two Ports:

Consider two networks N' and N" are connected in parallel as shown in Fig. 6.8 (b). When two ports are connected in parallel, we can add their y-parameters to get overall y-parameters of the parallel connection.

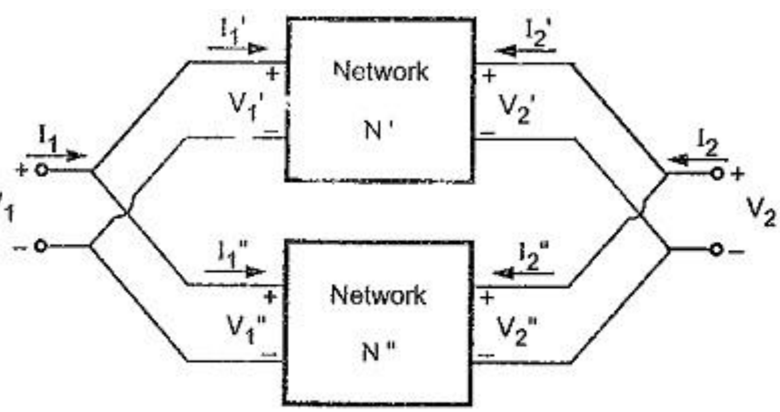


Fig. 6.8 (b) Parallel connection of 2 two port networks

Let the y-parameters of the network N' by y'11, y'12, y'21, y'22. Let the y-parameters of the network N" be y"11, y"12, y"21, y"22. Let the overall y-parameters of parallel connection be y11, y12, y21, y22.

For parallel connection we have,

$$I_1 = I_1' + I_1'' \qquad \ldots(1)$$
$$I_2 = I_2' + I_2'' \qquad \ldots(2)$$
$$V_1 = V_1' = V_1'' \qquad \ldots(3)$$
$$V_2 = V_2' = V_2'' \qquad \ldots(4)$$

For network N', the y-parameter equations are,

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

For network N" the y-parameter equations are,

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

From equations (1), (2) and (3), (4), we can write,

$$I_1 = \left(y_{11}' + y_{11}''\right) V_1 + \left(y_{12}' + y_{12}''\right) V_2$$
$$I_2 = \left(y_{21}' + y_{21}''\right) V_1 + \left(y_{22}' + y_{22}''\right) V_2$$

In matrix form, above equations can be written as,

$$\begin{bmatrix} I_1 \\ I_2 \end{bmatrix} = \begin{bmatrix} \left(y_{11}' + y_{11}''\right) & \left(y_{12}' + y_{12}''\right) \\ \left(y_{21}' + y_{21}''\right) & \left(y_{22}' + y_{22}''\right) \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \end{bmatrix}$$

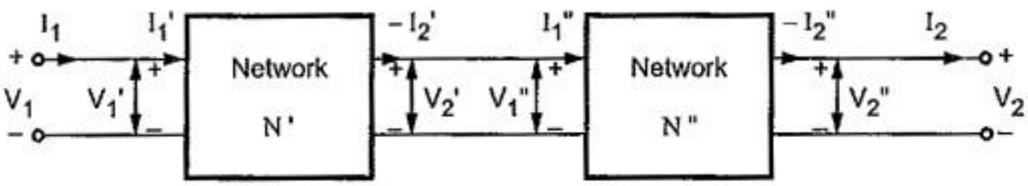
Thus, overall y-parameters are,

$$[y] = \begin{bmatrix} y_{11}' + y_{11}'' & y_{12}' + y_{12}'' \\ y_{21}' + y_{21}'' & y_{22}' + y_{22}'' \end{bmatrix}$$

Hence, the y-parameters of the parallel connection are the sum of y-parameters of the individual networks connected in parallel.

Cascade Connection of Two Ports:

The cascade connection is also called Tandem connection. Consider two networks N' and N" are connected in cascade as shown in Fig. 6.8 (c). When two ports are connected in cascade, we can multiply their individual transmission parameters to get overall transmission parameters of the cascade connection.



Let the transmission parameters of network N' be A' , B' ,C' , D'. Let the transmission parameters of network N" be A", B", C", D". Let the overall transmission parameters of cascade connection be A, B, C, D.

For cascade connection we have,

$$V_1 = V_1', \; V_2' = V_1'', \; V_2 = V_2'' \qquad \ldots(1)$$
$$I_1' = I_1, \; -I_2' = I_1'', \; I_2 = -I_2'' \qquad \ldots(2)$$

For the network N', transmission parameter equations are,

$$V_1' = A' V_2' + B' (-I_2')$$
$$I_1' = C' V_2' + D' (-I_2')$$

For the network N", transmission parameter equations are,

$$V_1'' = A'' V_2'' + B'' (-I_2'')$$
$$I_1'' = C'' V_2'' + D'' (-I_2'')$$

The overall transmission parameters of the cascade connection can be written as,

$$\begin{bmatrix} V_1 \\ I_1 \end{bmatrix} = \begin{bmatrix} V_1' \\ I_1' \end{bmatrix}$$

$$= \begin{bmatrix} A' & B' \\ C' & D' \end{bmatrix} \begin{bmatrix} V_2' \\ -I_2' \end{bmatrix}$$

$$= \begin{bmatrix} A' & B' \\ C' & D' \end{bmatrix} \begin{bmatrix} V_1'' \\ I_1'' \end{bmatrix}$$

$$= \begin{bmatrix} A' & B' \\ C' & D' \end{bmatrix} \begin{bmatrix} A'' & B'' \\ C'' & D'' \end{bmatrix} \begin{bmatrix} V_2'' \\ -I_2'' \end{bmatrix}$$

$$= \begin{bmatrix} A' & B' \\ C' & D' \end{bmatrix} \begin{bmatrix} A'' & B'' \\ C'' & D'' \end{bmatrix} \begin{bmatrix} V_2 \\ -I_2 \end{bmatrix}$$

$$\begin{bmatrix} V_1 \\ I_1 \end{bmatrix} = \begin{bmatrix} A & B \\ C & D \end{bmatrix} \begin{bmatrix} V_2 \\ -I_2 \end{bmatrix}$$

$$\begin{bmatrix} A & B \\ C & D \end{bmatrix} = \begin{bmatrix} A' & B' \\ C' & D' \end{bmatrix} \begin{bmatrix} A'' & B'' \\ C'' & D'' \end{bmatrix}$$

Hence, the transmission parameters for the cascaded two port network is simply the matrix product of the transmission parameter matrix of each individual two port network in cascade.