ELECTRICAL SIMULATION LAB
(EE431)

LABORATORY MANUAL

DEPARTMENT OF ELECTRICAL ENGINEERING

MUFFAKHAM JAH COLLEGE OF ENGINEERRING AND TECHNOLOGY

(Affiliated to Osmania University)

Banjara Hills, Hyderabad

2016

Prepared by K. Mahammad Rafi, Asst. Prof., EED

Revised by J.E.Muralidhar, Assoc.Prof.,EED

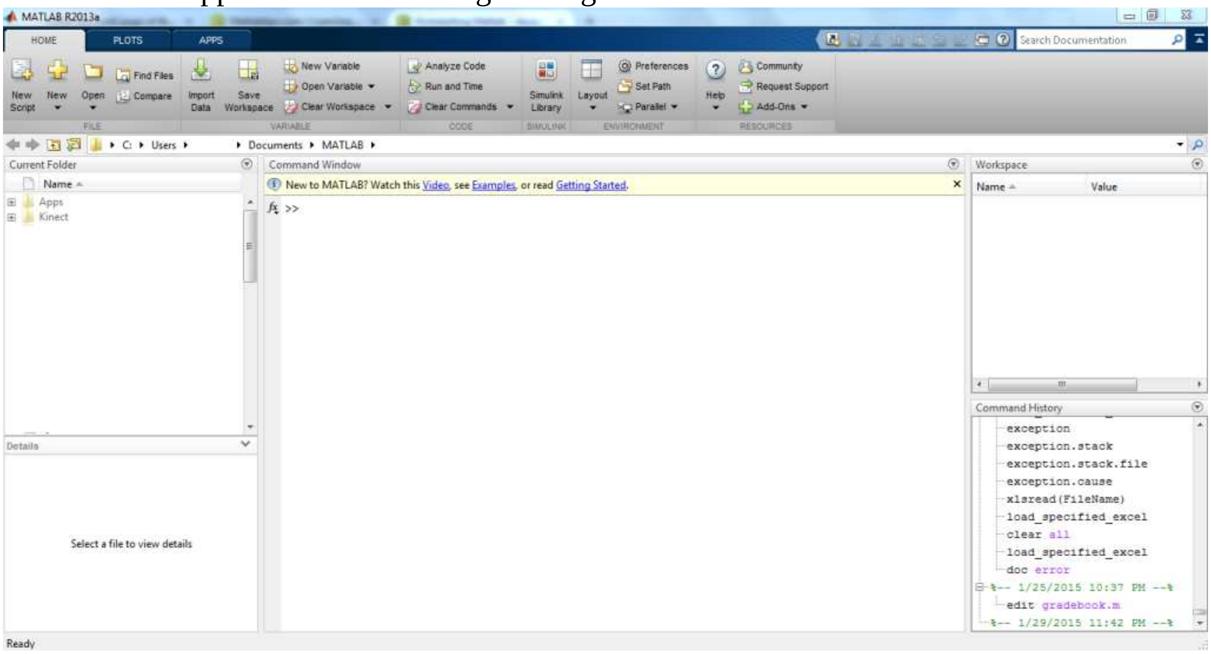
Srujana R. U, Asst. Prof., EED

LIST OF EXPERIMENTS IN ELECTRICAL SIMULATION LAB

1. Verification of Network Theorems
   i) Superposition theorem.
   ii) Thevenin's theorem.
   iii) Maximum power transfer theorem.
2. Transient responses of series RLC, RL, RC circuits with Sine and Step inputs.
3. Series and Parallel resonance.
4. Bode plot, Root-locus plot and Nyquist plot.
5. Transfer function analysis of
   i) Time response of step input
   ii) Frequency response for sinusoidal input.
6. Design of lag, lead and lag-lead compensators.
7. Load flow studies.
8. Fault analysis.
9. Transient stability studies.
10. Economic power scheduling

INTRODUCTION TO MATLAB

The name MATLAB stands for MATrix LABoratory. MATLAB was written originally to provide easy access to matrix software developed by the LINPACK (linear system package) and EISPACK (Eigen system package) projects.
MATLAB is a high-performance language for technical computing. It integrates computation, visualization, and programming environment. Furthermore, MATLAB is a modern programming language environment: it has sophisticated data structures, contains built-in editing and debugging tools, and supports object-oriented programming. These factors make MATLAB an excellent tool for teaching and research. MATLAB has many advantages compared to conventional computer languages (e.g., C, FORTRAN) for solving technical problems. MATLAB is an interactive system whose basic data element is an array that does not require dimensioning. The software package has been commercially available since 1984 and is now considered as a standard tool at most universities and industries worldwide.
It has powerful built-in routines that enable a very wide variety of computations. It also has easy to use graphics commands that make the visualization of results immediately available. Specific applications are collected in packages referred to as toolbox. There are toolboxes for signal processing, symbolic computation, control theory, simulation, optimization, and several other fields of applied science and engineering.



This is the default layout of MATLAB version used in our laboratory.
The main window is the Command Window. You can type in there any command that is available in MATLAB.
The second window in importance is the workspace. This is the current state of memory in MATLAB. The entire variables that are being used go there. The command history and the current folder are just useful tool that you can use but they are not essential to understand MATLAB.

Using MATLAB as a calculator:

As an example of a simple interactive calculation, just type the expression you want to evaluate. Let's start at the very beginning. For example, let's suppose you want to calculate the expression, 1 + 2 × 3. You type it at the prompt command (>>) as follows,

>> 1+2*3

ans = 7

You will have noticed that if you do not specify an output variable, MATLAB uses a default variable ans, short for answer, to store the results of the current calculation. Note that the

variable ans is created (or overwritten, if it is already existed). To avoid this, you may assign a value to a variable or output argument name.
For example,
>> x = 1+2*3
x = 7 will result in x being given the value 1 + 2 × 3 = 7. This variable name can always be used to refer to the results of the previous computations. Therefore, computing 4x will result in
>> 4*x
ans = 28.0000

Basic arithmetic operators

SYMBOL	OPERATION	EXAMPLE
+	Addition	$2+3$
−	Subtraction	$2-3$
*	Multiplication	$2*3$
/	Division	$2/3$

Elementary functions

cos(x)	Cosine	abs(x)	Absolute value
sin(x)	Sine	sign(x)	Signum function
tan(x)	Tangent	max(x)	Maximum value
acos(x)	Arc cosine	min(x)	Minimum value
asin(x)	Arc sine	ceil(x)	Round towards $+\infty$
atan(x)	Arc tangent	floor(x)	Round towards $-\infty$
exp(x)	Exponential	round(x)	Round to nearest integer
sqrt(x)	Square root	rem(x)	Remainder after division
log(x)	Natural logarithm	angle(x)	Phase angle
log10(x)	Common logarithm	conj(x)	Complex conjugate

Predefined constant values

pi	The π number, $\pi = 3.14159\ldots$
i,j	The imaginary unit i, $\sqrt{-1}$
Inf	The infinity, ∞
NaN	Not a number

MATLAB by default displays only 4 decimals in the result of the calculations, for example −163.6667, as shown in above examples. However, MATLAB does numerical calculations in double precision, which is 15 digits. The command format controls how the results of computations are displayed. Here are some examples of the different formats together with the resulting outputs.
>> format short
>> x=-163.6667
If we want to see all 15 digits, we use the command format long
>> format long
>> x= -1.636666666666667e+002
To return to the standard format, enter format short, or simply format. There are several other formats. For more details, see the MATLAB documentation, or type help format.

Managing the workspace:
The contents of the workspace persist between the executions of separate commands. Therefore, it is possible for the results of one problem to have an effect on the next one. To avoid this possibility, it is a good idea to issue a clear command at the start of each new independent calculation.

>> clear

The command clear or clear all removes all variables from the workspace. This frees up system memory.

In order to display a list of the variables currently in the memory, type

>> who

while, whos will give more details which include size, space allocation, and class of the variables.

Here are few additional useful commands:

- To clear the Command Window, type clc
- To abort a MATLAB computation, type ctrl-c
- To continue a line, type . . .

HELP:

To view the online documentation, select MATLAB Help from Help menu or MATLAB Help directly in the Command Window. The preferred method is to use the Help Browser. The Help Browser can be started by selecting the ? icon from the desktop toolbar. On the other hand, information about any command is available by typing

>> help Command

EXPERIMENT NO: 1

VERIFICATION OF NETWORK THEOREMS

I) SUPERPOSITION THEOREM.

II) THEVENIN'S THEOREM.

III) MAXIMUM POWER TRANSFER THEOREM.

AIM: To verify Superposition theorem, Thevenin's theorem, Norton's theorem and Maximum power Transfer theorem.

SOFTWARE USED : MULTISIM / MATLAB Simulink

SUPERPOSITION THEOREM:

"In a linear network with several independent sources which include equivalent sources due to initial conditions, and linear dependent sources, the overall response in any part of the network is equal to the sum of individual responses due to each independent source, considered separately, with all other independent sources reduced to zero".

Procedure:

Step 1:

1. Make the connections as shown in the circuit diagram by using MULTISIM/MATLAB Simulink.
2. Measure the response 'I' in the load resistor by considering all the sources 10V, 15V and 8V in the network.

Step 2:

1. Replace the sources 15V and 8V with their internal impedances (short circuited).
2. Measure the response 'I1' in the load resistor by considering 10V source in the network.

Step 3:

1. Replace the sources 10V and 8V with their internal impedances (short circuited).
2. Measure the response 'I2' in the load resistor by considering 15V source in the network.

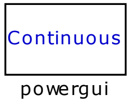
Step 4:

1. Replace the sources 10V and 15V with their internal impedances (short circuited).
2. Measure the response 'I3' in the load resistor by considering 8V source in the network.

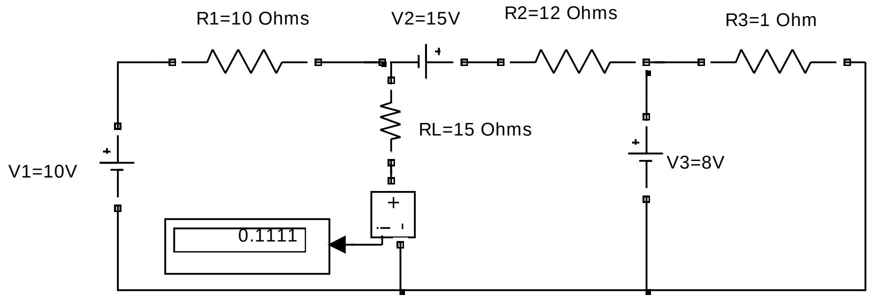
The responses obtained in step 1 should be equal to the sum of the responses obtained in step 2, 3 and 4.

$$I=I1+I2+I3$$

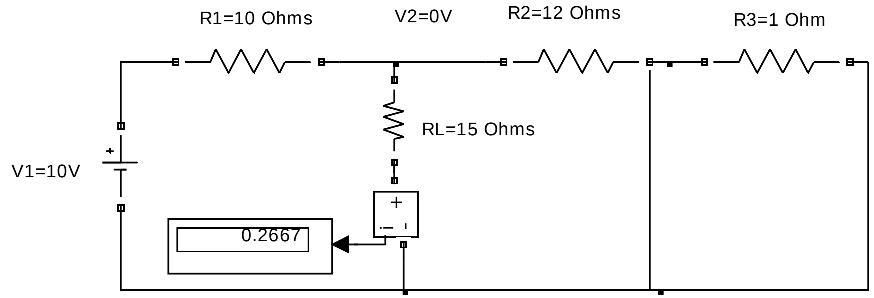
Hence Superposition Theorem is verified.



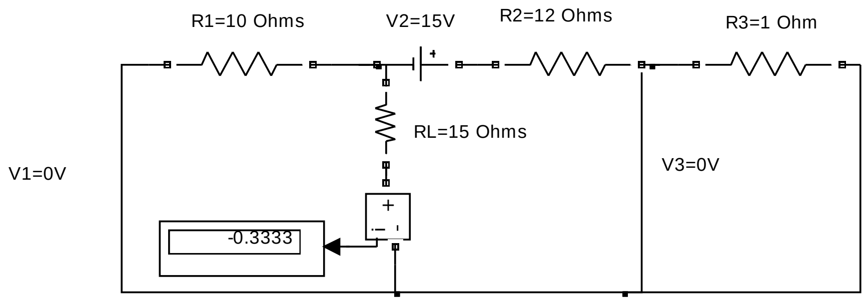
Step 1 : By Considering All Sources In The Network



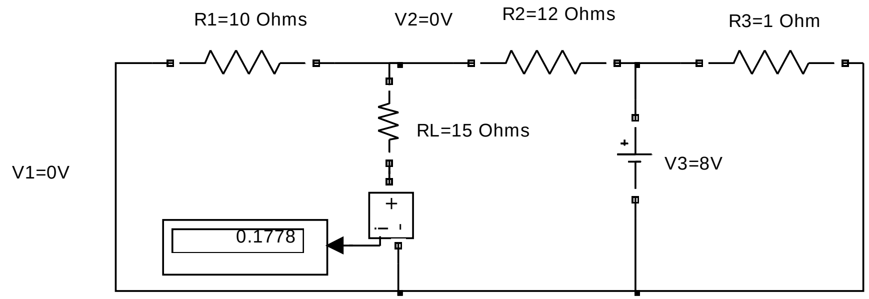
Step 2 : By Considering 10 V Sources In The Network



Step 3 : By Considering 15 V Sources In The Network



Step 4 : By Considering 8V Sources In The Network



Current through Load Resistor 15 Ohms :

Considerning 10V Source I1: 0.2667A
Considering 15V Source I2 : - 0.3333A
Considering 8V Source I3 : 0.1778A

Total Current : I1+I2+I3=0.2667-0.3333+0.1778
=0.1112A

With all the sources in the network I = 0.1111A
I=I1+I2+I3

Hence SuperPosition Theorem is Verified.

THEVENIN'S THEOREM:

"Any two terminal network consisting of linear impedances and generators may be replaced at the two terminals by a single voltage source acting in series with an impedance. The voltage of the equivalent source is the open circuit voltage measured at the terminals of the network and the impedance, known as Thevenin's equivalent impedance, Z_{TH}, is the impedance measured at the terminals with all the independent sources in the network reduced to zero ".

Procedure:

Step 1:

1. Make the connections as shown in the circuit diagram by using MULTISIM/MATLAB Simulink.
2. Measure the response 'I' in the load resistor by considering all the sources in the network.

Step 2: Finding Thevenin's Resistance(R_{TH})

1. Open the load terminals and replace all the sources with their internal impedances.
2. Measure the impedance across the open circuited terminal which is known as Thevenin's Resistance.

Step 3: Finding Thevenin's Voltage(V_{TH})

1. Open the load terminals and measure the voltage across the open circuited terminals.
2. Measured voltage will be known as Thevenin's Voltage.

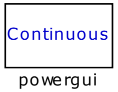
Step 4: Thevenin's Equivalent Circuit

1. V_{TH} and R_{TH} are connected in series with the load.
2. Measure the current through the load resistor $I_L = \frac{V_{TH}}{R_{TH}+R_L}$.

Current measured from Thevenin's Equivalent Circuit should be same as current obtained from the actual circuit.

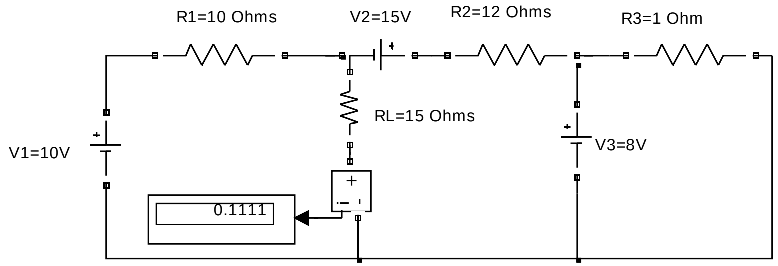
$$I = I_L.$$

Hence Thevenin's Theorem is Verified.

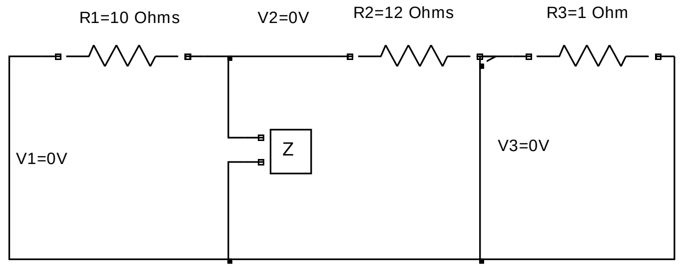


THEVENIN'S THEOREM

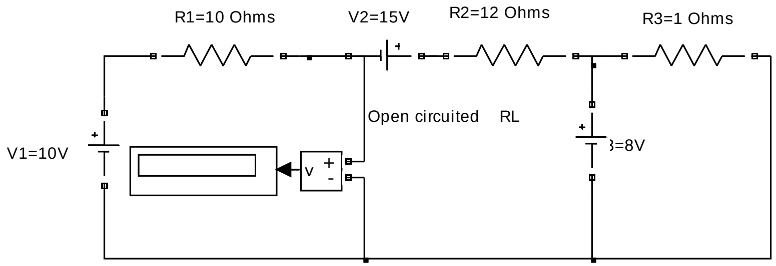
Step 1 : By Considering All Sources In The Network



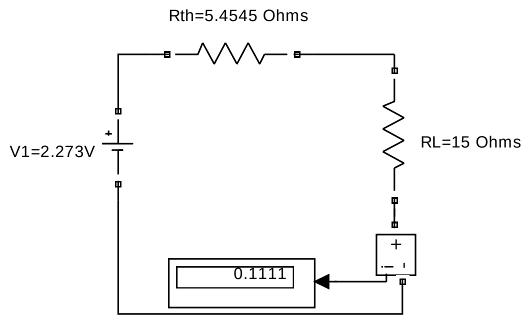
Step 2: Finding Thevenin's Resistance



Step 3 : Finding Thevenin's Voltage



Step 4 : Thevenin's Equivalent Network



Open Circuit Voltage Vth = 2.273V
Thevenin's Resistance = 5.4545Ohms
Current through Load Resistor 15 Ohms IL = 0.1111A

With all the sources in the network Current through Load Resistor 15 Ohms : I=0.1111A

I=IL

Hence Thevenin's Theorem is Verified.

NORTON'S THEOREM:

"Any two terminal network consisting of linear impedances and generators may be replaced at its two terminals, by an equivalent network consisting of a single current source in parallel with an impedance. The equivalent current source is the short circuit current measured at the terminals and the equivalent impedance is same as the Thevenin's equivalent impedance".

Procedure:

Step 1:

1. Make the connections as shown in the circuit diagram by using MULTISIM/MATLAB Simulink.
2. Measure the response 'I' in the load resistor by considering all the sources in the network.

Step 2: Finding Norton's Resistance(R_N)

1. Open the load terminals and replace all the sources with their internal impedances.
2. Measure the impedance across the open circuited terminal which is known as Norton's Resistance.

Step 3: Finding Norton's Current(I_N)

1. Short the load terminals and measure the current through the short circuited terminals.
2. Measured current is be known as Norton's Current.

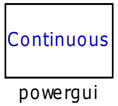
Step 4: Norton's Equivalent Circuit

1. R_N and I_N are connected in parallel to the load.
2. Measure the current through the load resistor $I_L = \frac{I_N R_N}{R_N + R_L}$.

Current measured from Norton's Equivalent Circuit should be same as current obtained from the actual circuit.

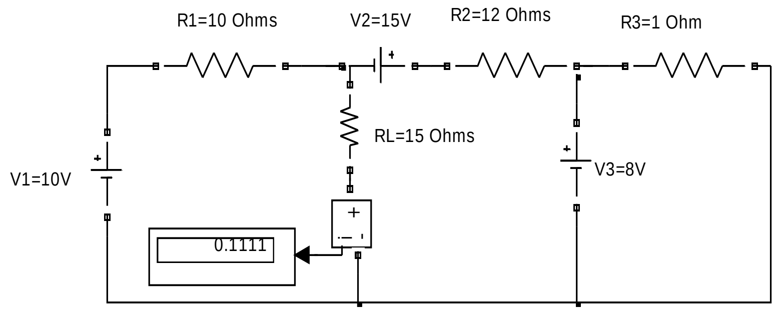
$I = I_L$.

Hence Norton's Theorem is Verified.

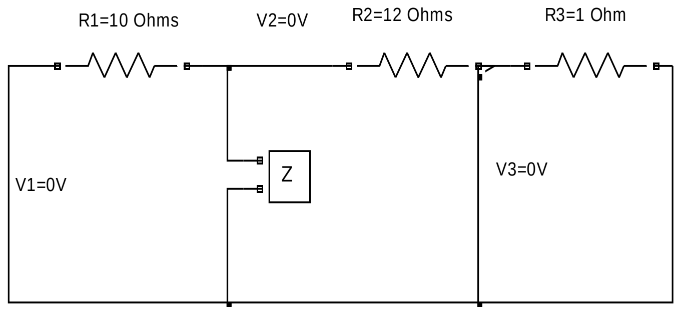


NORTON'S THEOREM

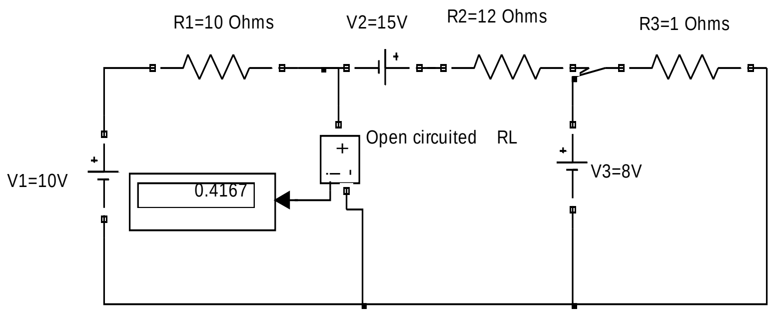
Step 1 : By Considering All Sources In The Network



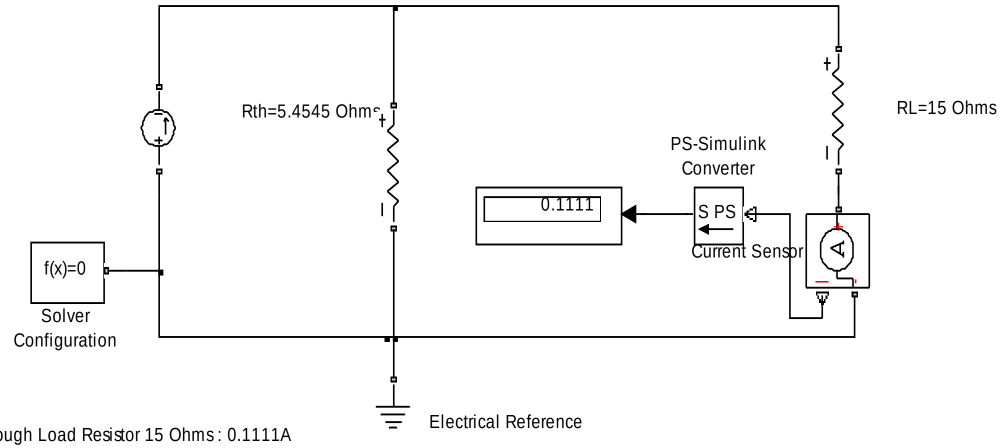
Step 2: Finding Norton's Resistance



Step 3 : Finding Norton's Current



Step 4 : Norton's Equivalent Network



Norton's Current = 0.4167 A
Norton's Resistance = 5.4545Ohms
Current through Load Resistor 15 Ohms = 0.1111A

With all the sources in the network Current through Load Resistor 15 Ohms : 0.1111A

Hence Norton's Theorem is Verified.

MAXIMUM POWER TRANSFER THEOREM:

"In any circuit the maximum power is transferred to the load when the load resistance is equal to the source resistance. The source resistance is equal to the Thevenin's equal resistance ".

Procedure:

Step 1:

1. Make the connections as shown in the circuit diagram by using Multisim/MATLAB Simulink.
2. Measure the Power across the load resistor by considering all the sources in the network.

Step 2: Finding Thevenin's Resistance(R_{TH})

1. Open the load terminals and replace all the sources with their internal impedances.
2. Measure the impedance across the open circuited terminal which is known as Thevenin's Resistance.

Step 3: Finding Thevenin's Voltage(V_{TH})

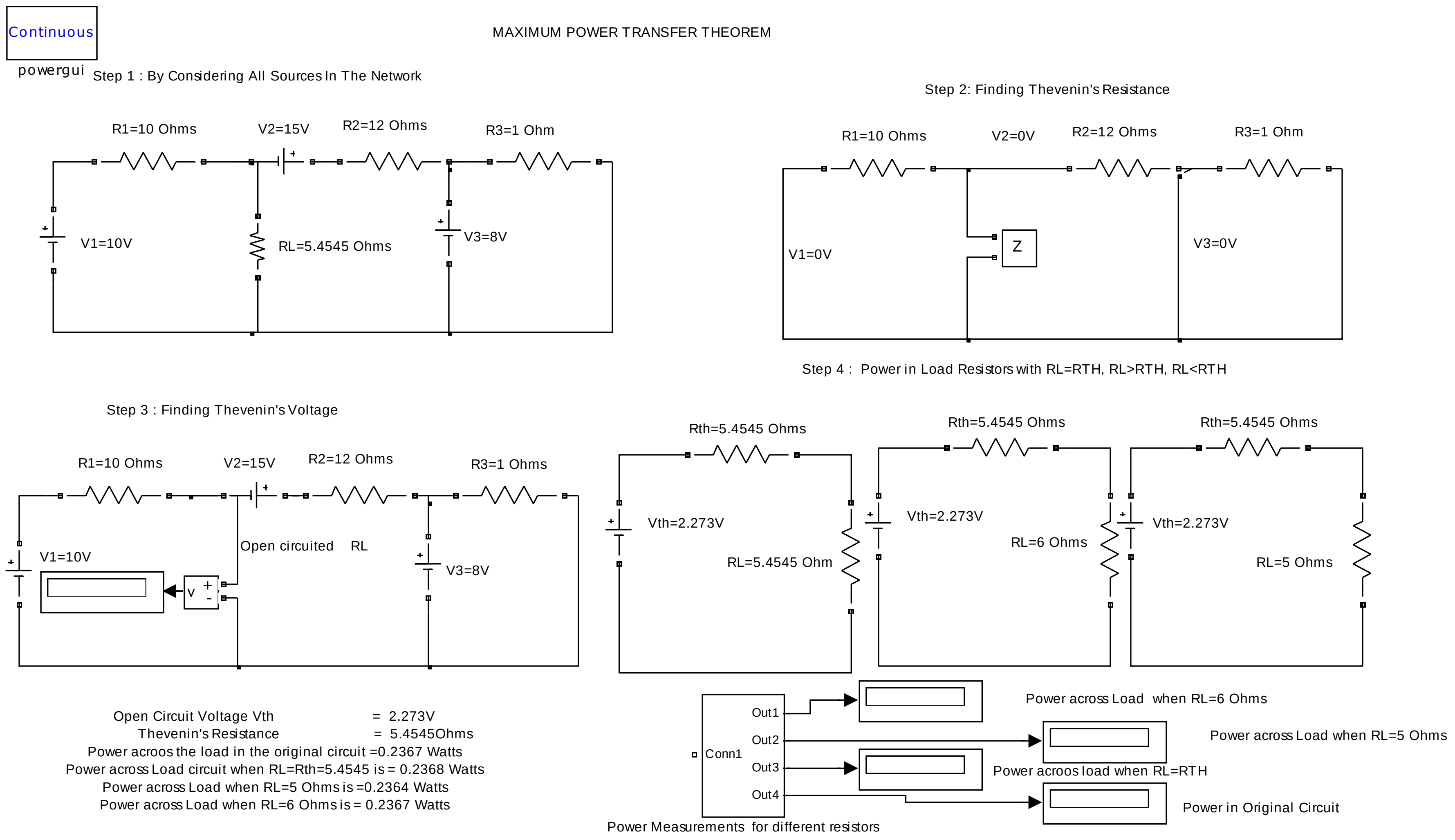
1. Open the load terminals and measure the voltage across the open circuited terminals.
2. Measured voltage will be known as Thevenin's Voltage.

Step 4: Measuring Power for different Load Resistors

1. V_{TH} and R_{TH} are connected in series with the load.
2. Measure power across the load by considering $R_L=R_{TH}$.
3. Measure power by using $P = \frac{V_{TH}^2}{4R_L}$.
4. Verify the power for different values of load resistors(i.e. $R_L>R_{TH}$ and $R_L<R_{TH}$)

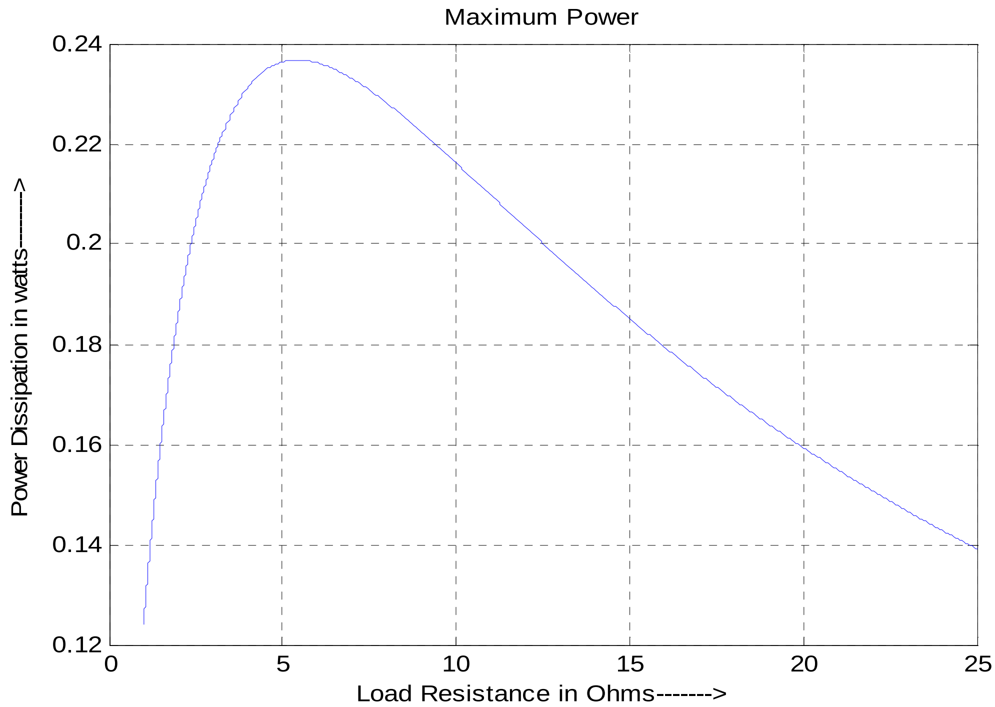
Power measured from the above steps results in maximum power dissipation when $R_L=R_{TH}$.

Hence Maximum Power Transfer Theorem is verified.


Continuous
powergui
MAXIMUM POWER TRANSFER THEOREM
Step 1 : By Considering All Sources In The Network
R1=10 Ohms
V2=15V
R2=12 Ohms
R3=1 Ohm
V1=10V
RL=5.4545 Ohms
V3=8V
Step 2: Finding Thevenin's Resistance
R1=10 Ohms
V2=0V
R2=12 Ohms
R3=1 Ohm
V1=0V
Z
V3=0V
Step 3 : Finding Thevenin's Voltage
R1=10 Ohms
V2=15V
R2=12 Ohms
R3=1 Ohms
V1=10V
Open circuited RL
V3=8V
Step 4 : Power in Load Resistors with RL=RTH, RL>RTH, RL<RTH
Rth=5.4545 Ohms
Vth=2.273V
RL=5.4545 Ohm
Rth=5.4545 Ohms
Vth=2.273V
RL=6 Ohms
Rth=5.4545 Ohms
Vth=2.273V
RL=5 Ohms
Conn1
Out1
Out2
Out3
Out4
Power across Load when RL=6 Ohms
Power across Load when RL=5 Ohms
Power acroos load when RL=RTH
Power in Original Circuit
Power Measurements for different resistors
Open Circuit Voltage Vth = 2.273V
Thevenin's Resistance = 5.4545Ohms
Power acroos the load in the original circuit =0.2367 Watts
Power across Load circuit when RL=Rth=5.4545 is = 0.2368 Watts
Power across Load when RL=5 Ohms is =0.2364 Watts
Power across Load when RL=6 Ohms is = 0.2367 Watts

M-File Program for Maximum Power Transfer Theorem:

```
clc;
close all;
clear all;

v=input('Enter the Voltage in Volts :');
rth=input('Enter the value of Thevenins Resistance:');
rl=1:0.0001:12;
i=v./(rth+rl);
p=i.^2.*rl;
plot(rl,p);
grid;
title('Maximum Power');
xlabel('Load Resistance in Ohms------->');
ylabel('Power Dissipation in watts-------->');
```



Results and Discussions: Super Position Theorem, Thevenin's Theorem, Norton's Theorem and Maximum Power Transfer Theorem are verified by using MATLAB Simulink /MULTISIM.

- The various circuit components are identified and circuits are formed in simulation environment.
- Use of network theorem in analysis can be demonstrated in this simulation exercise.

EXPERIMENT NO: 2

TRANSIENT RESPONSES OF SERIES RLC, RL, AND RC CIRCUITS WITH SINE AND STEP INPUTS

AIM: To study the transient analysis of RLC, RL and RC circuits for sinusoidal and step inputs.

SOFTWARES USED: MATLAB Simulink / MULTISIM

THEORY:

The transient response is the fluctuation in current and voltage in a circuit (after the application of a step voltage or current) before it settles down to its steady state. This lab will focus on simulation of series RL (resistor-inductor), RC (resistor-capacitor), and RLC (resistor inductor-capacitor) circuits to demonstrate transient analysis.

Transient Response of Circuit Elements:

A. Resistors: As has been studied before, the application of a voltage V to a resistor (with resistance R ohms), results in a current I, according to the formula:

   $I = V/R$

   The current response to voltage change is instantaneous; a resistor has no transient response.

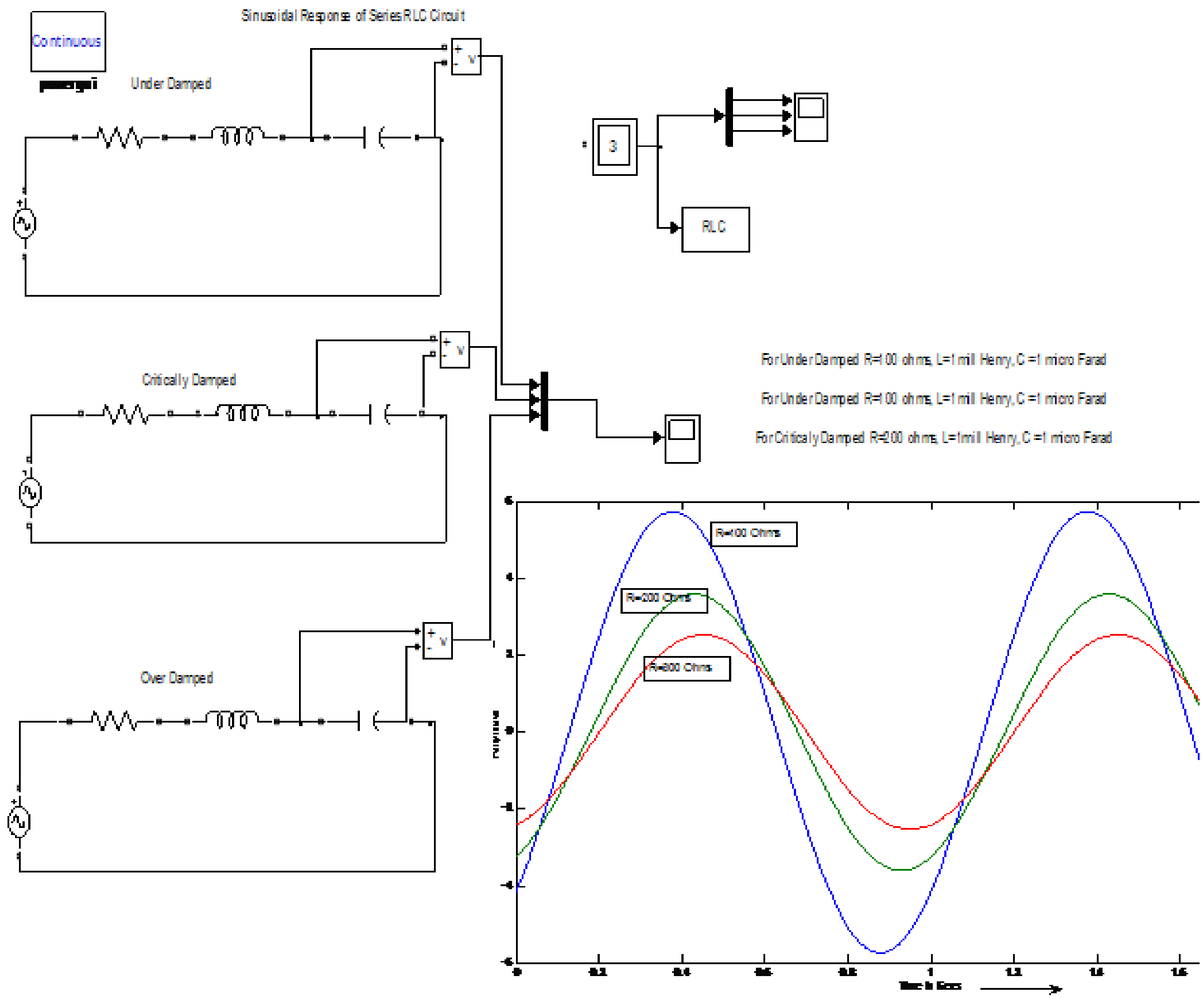
B. Inductors: A change in voltage across an inductor (with inductance L Henrys) does not result in an instantaneous change in the current through it. The i-v relationship is described with the equation: $v=L\, di/dt$

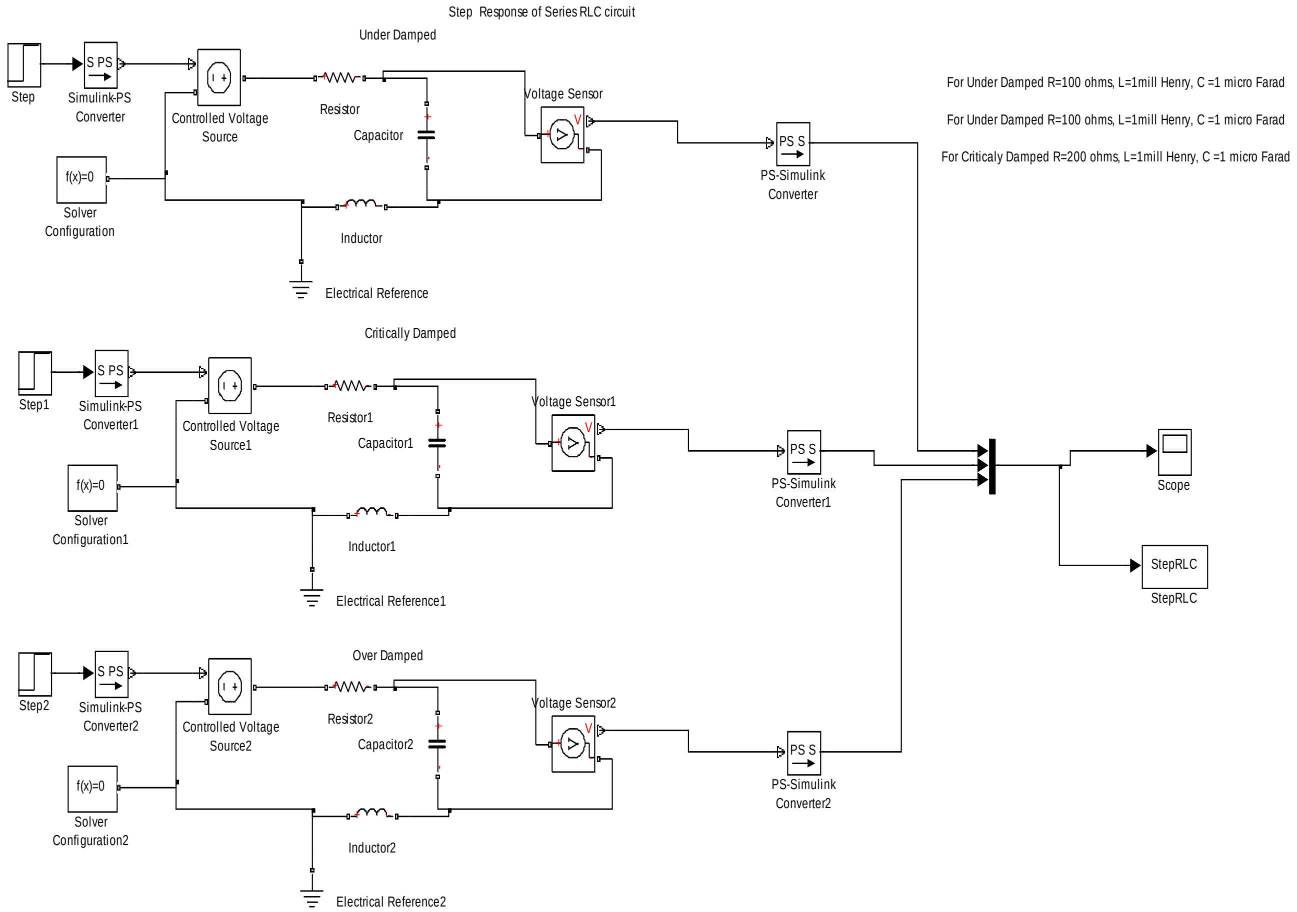
   This relationship implies that the voltage across an inductor approaches zero as the current in the circuit reaches a steady value. This means that in a DC circuit, an inductor will eventually act like a short circuit.

C. Capacitors: The transient response of a capacitor is such that it resists instantaneous change in the voltage across it. Its i-v relationship is described by: $i=C\, dv/dt$

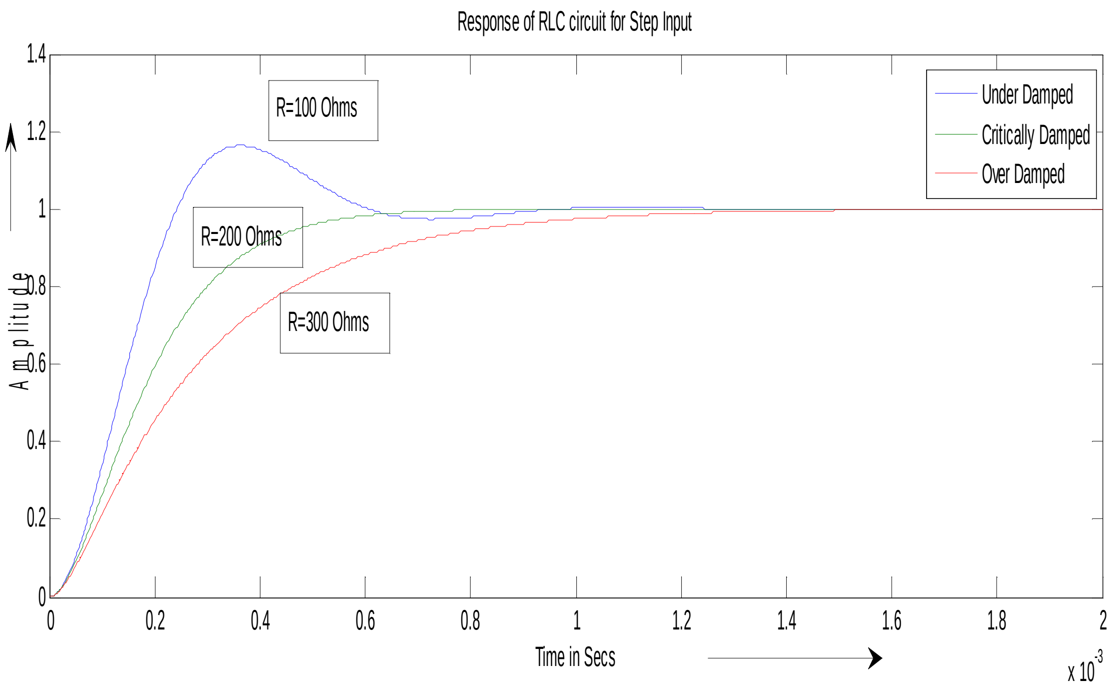
   This implies that as the voltage across the capacitor reaches a steady value, the current through it approaches zero. In other words, a capacitor eventually acts like an open circuit in a DC circuit.

Series Combinations of Circuit Elements: Solving the circuits involves the solution of first and second order differential equations.

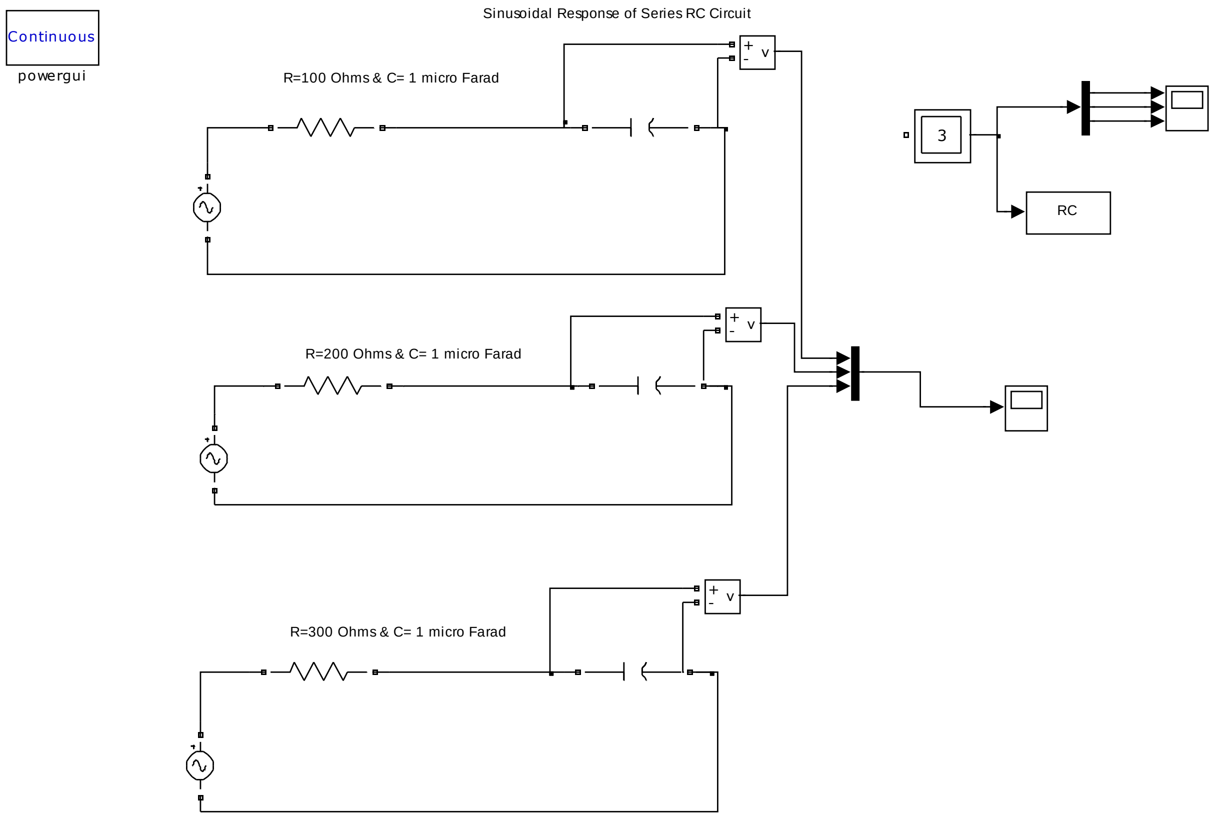


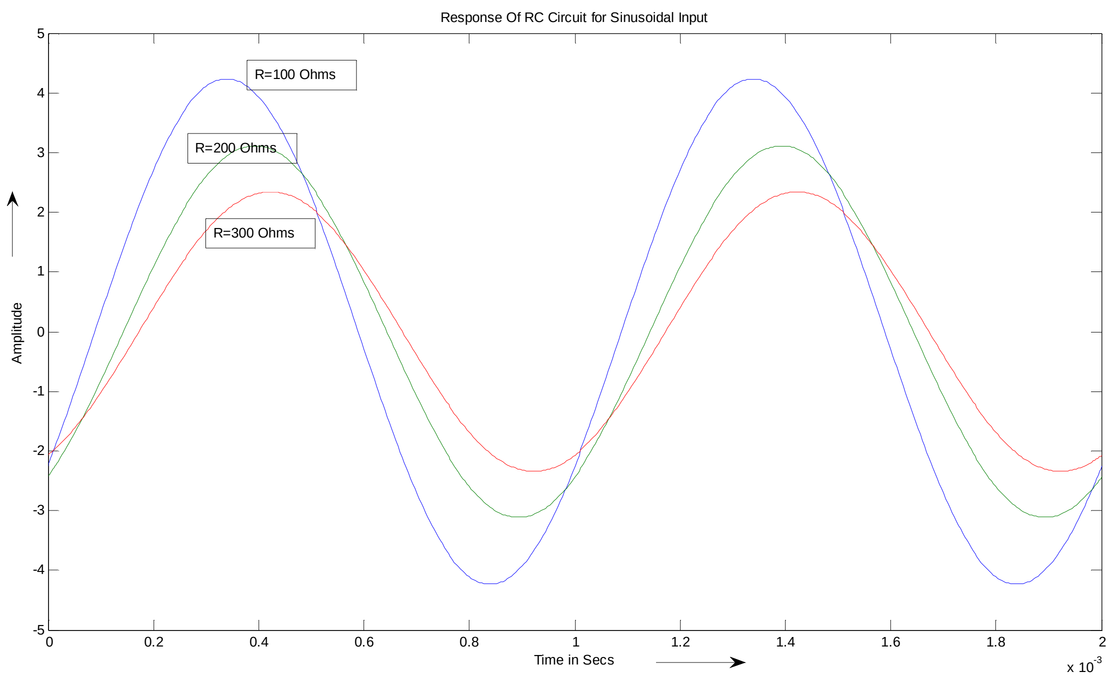


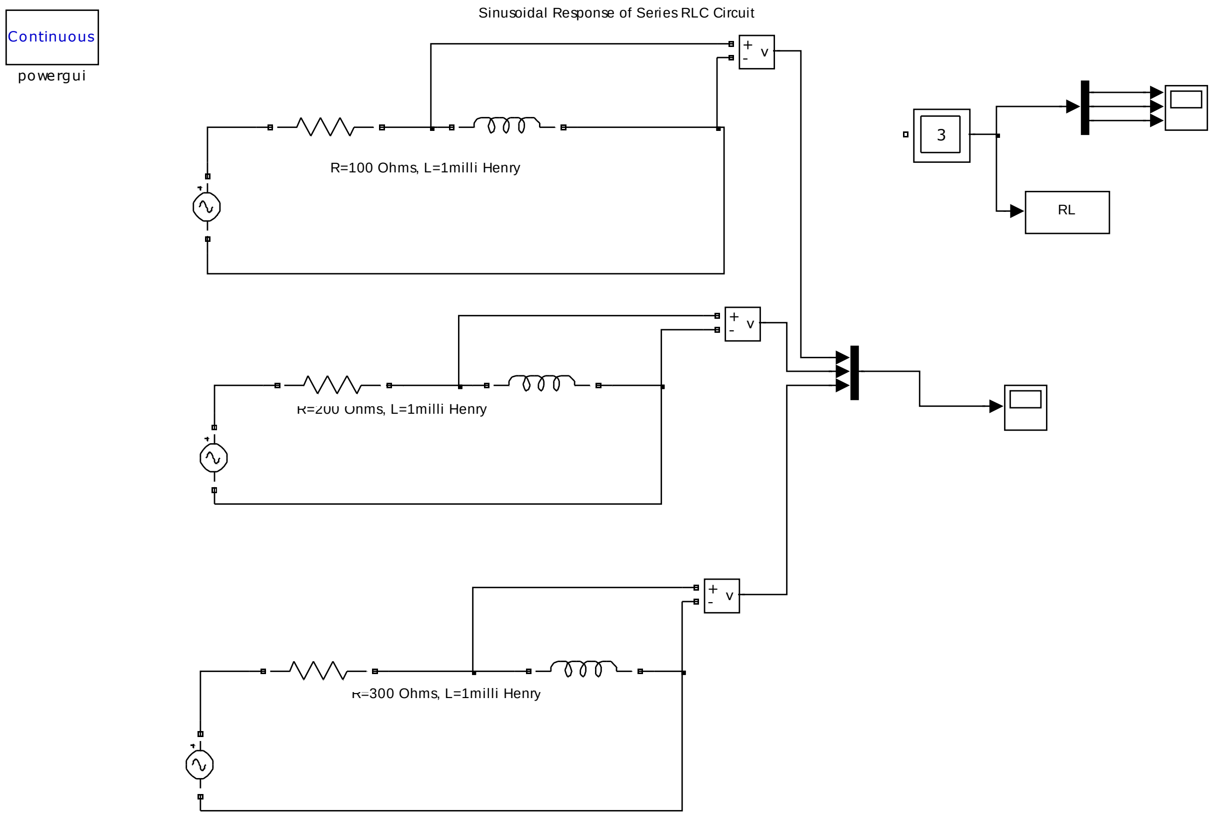
Step Response of Series RLC circuit
Under Damped
Step
Simulink-PS Converter
Controlled Voltage Source
Resistor
Capacitor
Voltage Sensor
f(x)=0
Solver Configuration
Inductor
Electrical Reference
PS-Simulink Converter
For Under Damped R=100 ohms, L=1mill Henry, C =1 micro Farad
For Under Damped R=100 ohms, L=1mill Henry, C =1 micro Farad
For Criticaly Damped R=200 ohms, L=1mill Henry, C =1 micro Farad
Critically Damped
Step1
Simulink-PS Converter1
Controlled Voltage Source1
Resistor1
Capacitor1
Voltage Sensor1
Solver Configuration1
Inductor1
Electrical Reference1
PS-Simulink Converter1
Scope
StepRLC
Over Damped
Step2
Simulink-PS Converter2
Controlled Voltage Source2
Resistor2
Capacitor2
Voltage Sensor2
Solver Configuration2
Inductor2
Electrical Reference2
PS-Simulink Converter2

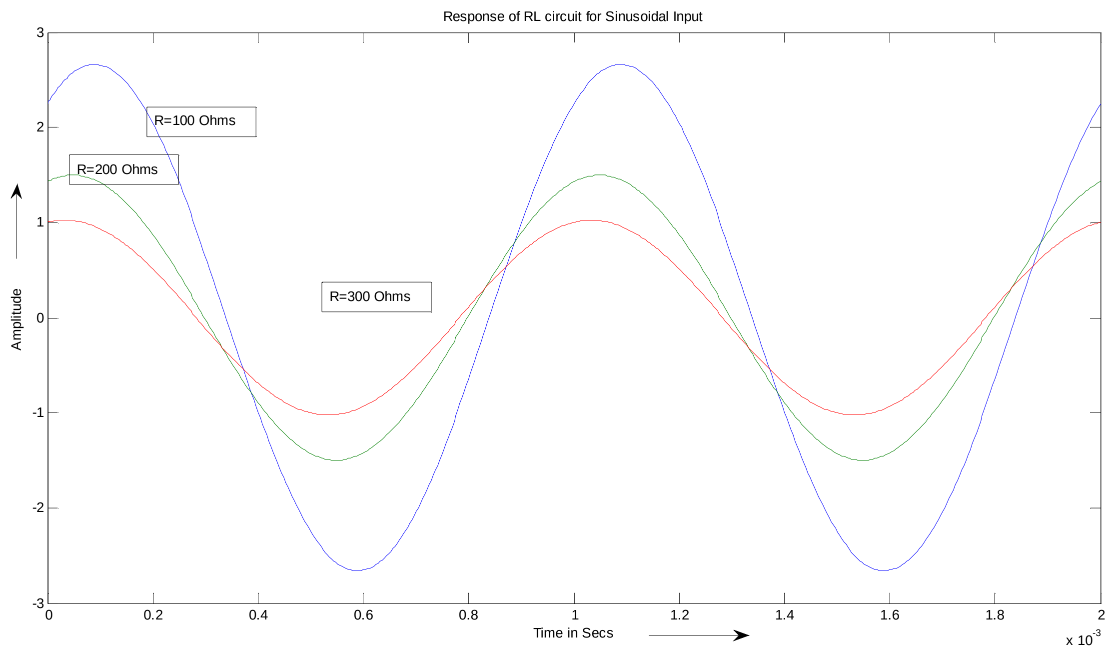


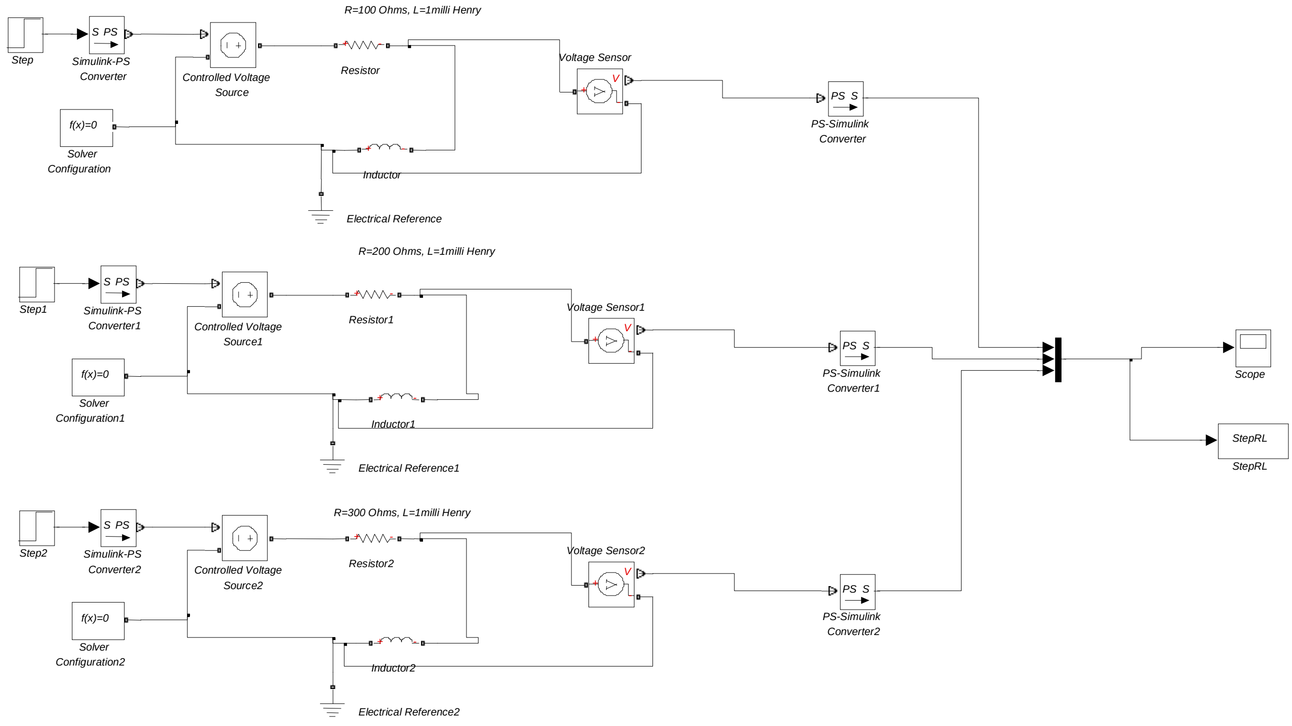
Response of RLC circuit for Step Input
R=100 Ohms
R=200 Ohms
R=300 Ohms
Under Damped
Critically Damped
Over Damped
Amplitude
Time in Secs
x 10^-3
0
0.2
0.4
0.6
0.8
1
1.2
1.4
1.6
1.8
2

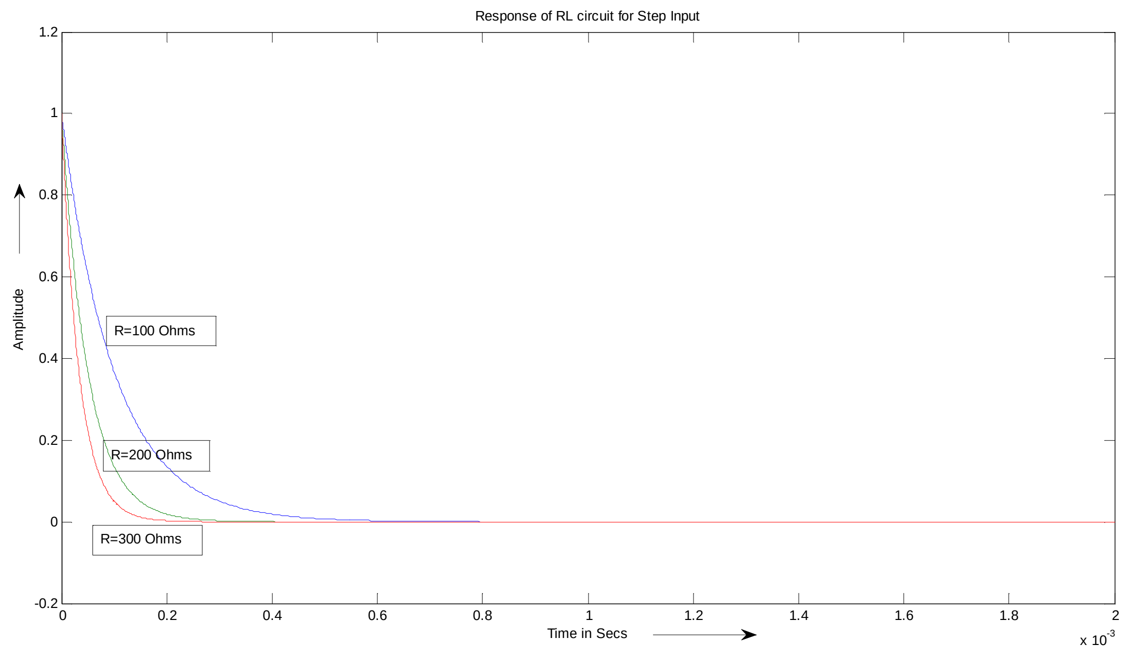

Sinusoidal Response of Series RC Circuit
Continuous
powergui
R=100 Ohms & C= 1 micro Farad
R=200 Ohms & C= 1 micro Farad
R=300 Ohms & C= 1 micro Farad
3
RC


Response Of RC Circuit for Sinusoidal Input
R=100 Ohms
R=200 Ohms
R=300 Ohms
Amplitude
Time in Secs
x 10^-3


Continuous
powergui
Sinusoidal Response of Series RLC Circuit
R=100 Ohms, L=1milli Henry
R=200 Ohms, L=1milli Henry
R=300 Ohms, L=1milli Henry
3
RL


Response of RL circuit for Sinusoidal Input
R=100 Ohms
R=200 Ohms
R=300 Ohms
Amplitude
Time in Secs
x 10^-3


R=100 Ohms, L=1milli Henry
Step
Simulink-PS Converter
Controlled Voltage Source
Resistor
Voltage Sensor
PS-Simulink Converter
f(x)=0
Solver Configuration
Inductor
Electrical Reference
R=200 Ohms, L=1milli Henry
Step1
Simulink-PS Converter1
Controlled Voltage Source1
Resistor1
Voltage Sensor1
PS-Simulink Converter1
Solver Configuration1
Inductor1
Electrical Reference1
Scope
StepRL
R=300 Ohms, L=1milli Henry
Step2
Simulink-PS Converter2
Controlled Voltage Source2
Resistor2
Voltage Sensor2
PS-Simulink Converter2
Solver Configuration2
Inductor2
Electrical Reference2


Response of RL circuit for Step Input
Amplitude
Time in Secs
x 10^-3
R=100 Ohms
R=200 Ohms
R=300 Ohms

PROCEDURE:

1. Make the connections as shown in connection diagram.
2. Observe the output waveforms across a) RLC b) RC c) RL.
3. Change the value of resistance such that the output obtained at each oscilloscope is
   - i) Critically damped.
   - ii) Under damped.
   - iii) Over damped.

RESULTS & DISCUSSIONS: The critically damped, under damped, damped response is observed for an RLC network in the simulation environment.

- The response to various inputs can be simulated .
- The response of any system designed can be simulated to verify its performance and design.

EXPERIMENT NO: 03

SERIES AND PARALLEL RESONANCE

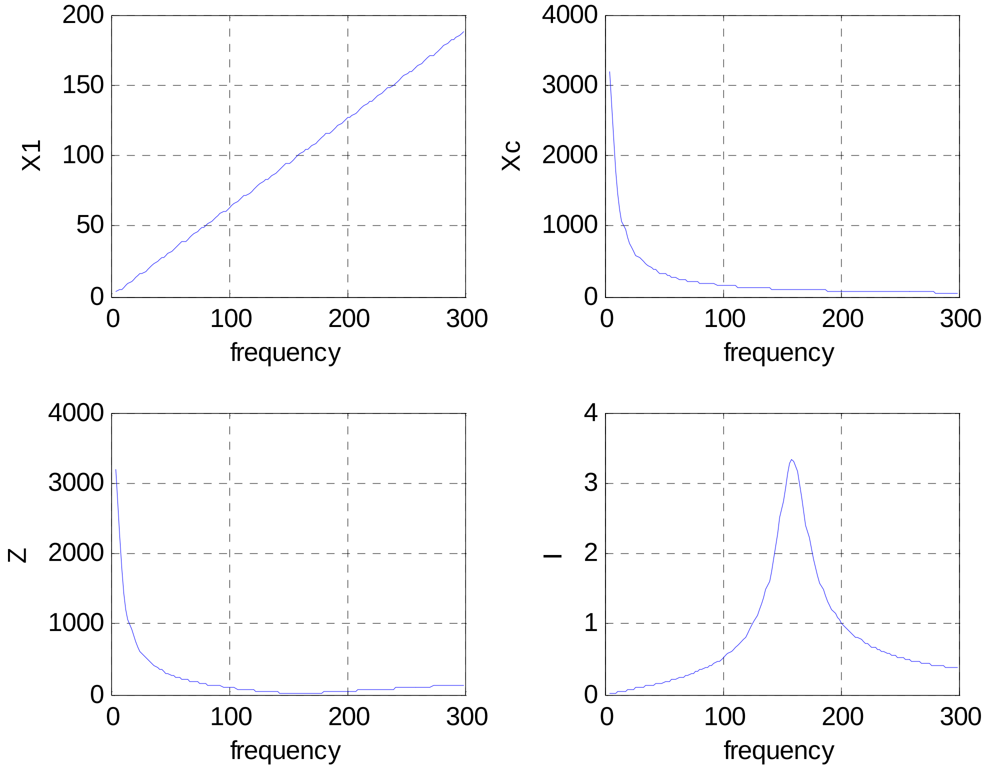
I) SERIES RESONANCE:

Aim: - To obtain the plot of of frequency vs. X_L, frequency vs. X_C , frequency vs. impedance and frequency vs. current for the given series RLC circuit and determine the resonant frequency and check by theoretical calculations.
R = 15 Ω, C = 10 μF, L = 0.1 H, V = 50V vary frequency in steps of 1 Hz using Matlab.

```
%Program to find the Parallel Resonance
clc;
clear all;
close all;
r=input('enter the resistance value----->');
l=input('enter the inductance value------>');
c=input('enter the capacitance value----->');
v=input('enter the input voltage------->');
f=5:2:300;
xl=2*pi*f*l;
xc=(1./(2*pi*f*c));
x=xl-xc;
z=sqrt((r^2)+(x.^2));
i=v./z;
%plotting the graph
subplot(2,2,1);
plot(f,xl);
grid;
xlabel('frequency');
ylabel('X1');
subplot(2,2,2);
plot(f,xc);
grid;
xlabel('frequency');
ylabel('Xc');
subplot(2,2,3);
plot(f,z);
grid;
xlabel('frequency');
ylabel('Z');
subplot(2,2,4);
plot(f,i);
grid;
xlabel('frequency');
ylabel('I');
```

PROGRAM RESULT:

```
enter the resistance value----->15
enter the inductance value------>0.1
enter the capacitance value----->10*10^-6
enter the input voltage------->50
```



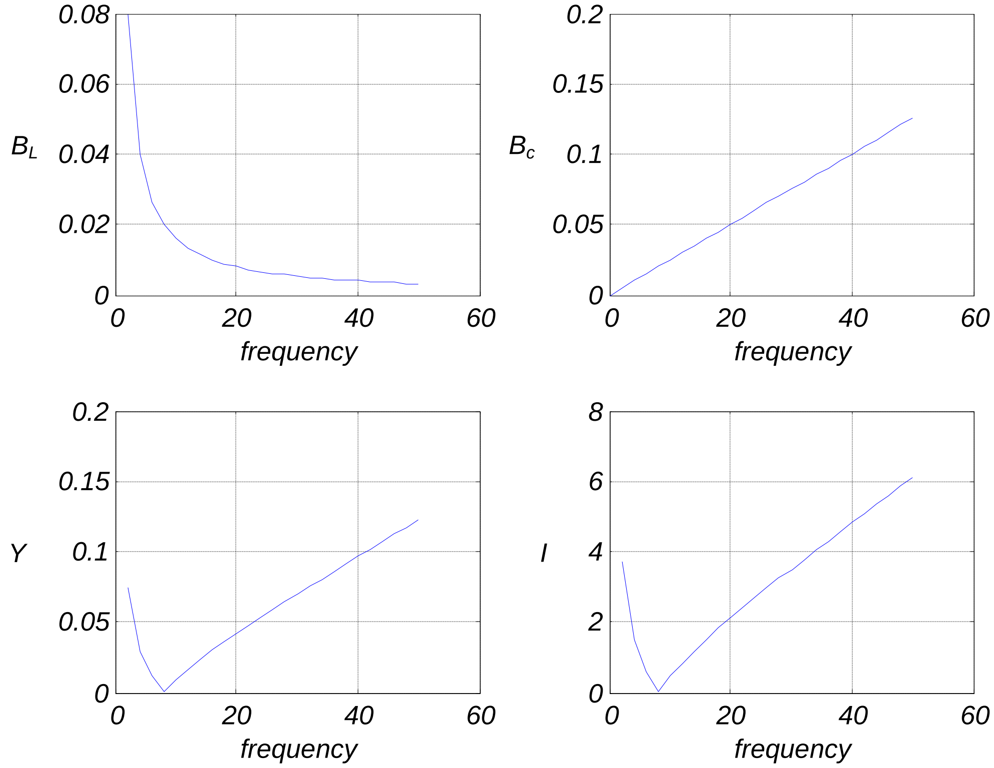
II) PARALLEL RESONANCE(Ideal Circuit):- To obtain the graphs of frequency vs. B_L , frequency vs. B_C , frequency vs. admittance and frequency vs. current vary frequency in steps for the given circuit and find the resonant frequency and check by theoretical calculations.
R = 1000 Ω, C = 400 μF, L = 1 H, V = 50V vary frequency in steps of 1 Hz using Matlab.

```
%Program to find the Parallel Resonance
clc;
clear all;
close all;
r=input('enter the resistance value----->');
l=input('enter the inductance value------>');
c=input('enter the capacitance value----->');
v=input('enter the input voltage------->');
f=0:2:50;
xl=2*pi*f*l;
xc=(1./(2*pi*f*c));
b1=1./xl;
bc=1./xc;
b=b1-bc;
```

```
g=1/r;
y=sqrt((g^2)+(b.^2));
i=v*y;
%plotting the graph
subplot(2,2,1);
plot(f,b1);
grid;
xlabel('frequency');
ylabel('B1');
subplot(2,2,2);
plot(f,bc);
grid;
xlabel('frequency');
ylabel('Bc');
subplot(2,2,3);
plot(f,y);
grid;
xlabel('frequency');
ylabel('Y');
subplot(2,2,4);
plot(f,i);
grid;
xlabel('frequency');
ylabel('I');
```

PROGRAM RESULT:

```
enter the resistance value----->1000
enter the inductance value------>1
enter the capacitance value----->400*10^-6
enter the input voltage------->50
```



RESULTS & DISCUSSIONS: Resonance phenomena for series and parallel circuits were simulated using MATLAB m-programming.

- MATLAB m- programming allows customizing to our simulation requirement and required results/graphs can be studied and analyzed.
- Effect of resonance on current and other quantities can be seen
- Effect of L,C parameters on resonant frequency can be seen from the simulation.
- Current amplification for series circuit are observed

EXPERIMENT – 4

ROOT LOCUS, BODE AND NYQUIST PLOT

ROOT LOCUS:

AIM: To obtain the root locus of the system whose transfer function is defined by

$$G(S)= \frac{(S+5)}{S^2+7S+25}$$

PROCEDURE:

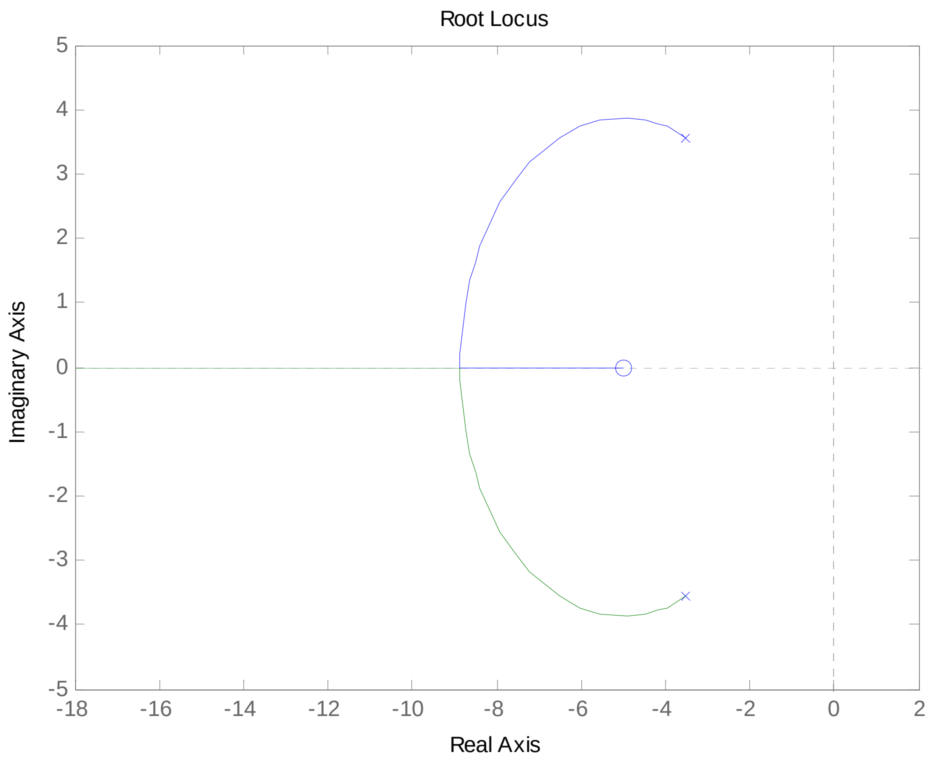
1. Input the numerator and denominator co-efficient.
2. Formulate the transfer function using the numerator and denominators co-efficient with the help of function T = t_f (num, den)
3. Plot the root locus of the above transfer function using rlocus(t).

PROGRAM:

```
%Program to find the root locus of transfer function%
     s+5)
% -----------
% s^2+7s+25

clc;
clear all;
close all;
% initialzations
num=input('enter the numerator coefficients---->');
den=input('enter the denominator coefficients---->');
%Transfer function
sys=tf(num,den);
rlocus(sys);
```

PROGRAM RESULT:

```
enter the numerator coefficients---->[1 5]
enter the denominator coefficients---->[1 7 25]
```



BODE PLOT:

THEORY: The gain margin is defined as the change in open loop gain required to make the system unstable. Systems with greater gain margins can withstand greater changes in system parameters before becoming unstable in closed loop. Keep in mind that unity gain in magnitude is equal to a gain of zero in dB.

The phase margin is defined as the change in open loop phase shift required to make a closed loop system unstable.

The phase margin is the difference in phase between the phase curve and -180 deg at the point corresponding to the frequency that gives us a gain of 0dB (the gain cross over frequency, Wgc).

Likewise, the gain margin is the difference between the magnitude curve and 0dB at the point corresponding to the frequency that gives us a phase of -180 deg (the phase cross over frequency, Wpc).

AIM: To obtain the bode plot and to calculate the phase margin, gain margin, phase cross over and gain cross over frequency for the systems whose open loop transfer function is given as follows.

$$G(s) = \frac{25(S+1)(S+7)}{S(S+2)(S+4)(S+8)}$$

PROCEDURE:

1. Input the zeroes, poles and gain of the given system.
2. Formulate the transfer function from zeroes, poles and gain of the system.
3. Plot the bode plot using function bode (t).

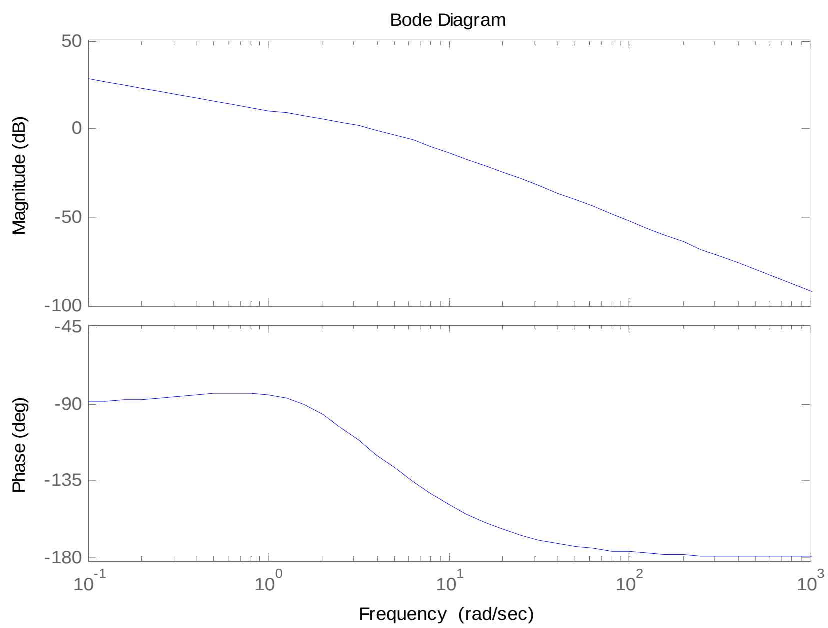
4. Estimate PM,GM, W_{PC} , and W_{GC}. Using function margin.

PROGRAM:

```
%Program to find Bode Plot
%  25(s+1)(s+7)
% -------------
% s(s+2)(s+4)(s+8)
clc;
clear all;
close all;
% initialzations
k=input('enter the gain---->');
z=input('enter the zeros---->');
p=input('enter the ploes---->');
t=zpk(z,p,k);
bode(t);
[Gm,Pm,Wcg,Wcp]=margin(t);
disp(Gm);
disp(Pm);
disp(Wgc);
disp(Wpc);
```

PROGRAM RESULT:

```
enter the gain---->25
enter the zeros---->[-1 -7]
enter the ploes---->[0 -2 -4 -8]

Gm= Inf
Pm= 63.1105
Wgc= Inf
Wpc= 3.7440
```



NYQUIST PLOT:

AIM:

To obtain the Nyquist plot and to calculate the phase margin, gain margin, phase cross over and gain cross over frequency for the systems whose open loop transfer function is given as follows.

$$G(S) = \frac{50(S+1)}{S(S+3)(S+5)}$$

PROCEDURE:

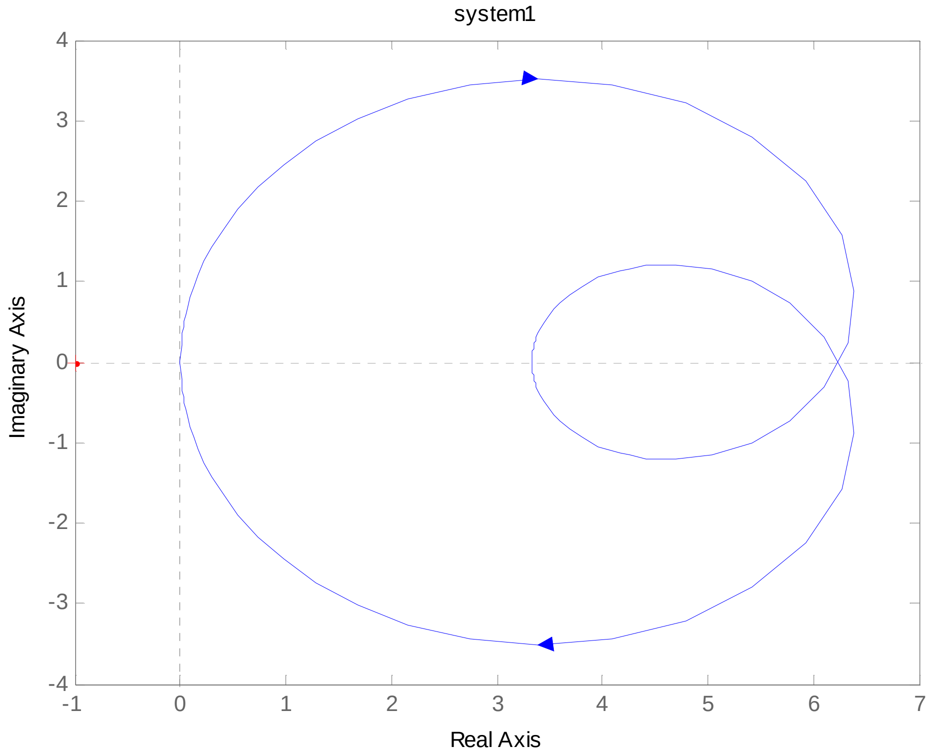
1. Input the zeroes, poles and gain of the given system.
2. Formulate the transfer function from zeroes, poles and gain of the system.
3. Plot the nyquist plot using function nyquist(t).
4. Estimate PM,GM, W_{PC} , and W_{GC}. Using function margin.

PROGRAM:

```
%Program to find the Nyquist Plot

%  50(s+1)
% --------
% s(s+3)(s+5)

clc;
clear all;
close all;
% initialzations
num=input('enter the numerator coefficients---->');
den=input('enter the denominator coefficients---->');
sys=tf(num,den);
nyquist(sys);
title('system1');
[Gm,Pm,Wcg,Wcp]=margin(sys);
disp(Gm);
disp(Pm);
disp(Wgc);
disp(Wpc);
```

PROGRAM RESULT:

```
enter the numerator coefficients---->[50 50]
enter the denominator coefficients---->[1 8 15]

Gm= Inf
Pm= 98.0516
Wgc=Inf
Wpc=49.6681
```



RESULTS & DISCUSSIONS: Root Locus, Bode plot and Nyquist plot determined using the built-in functions of MATLAB.

- They are a powerful tool to design systems to required performance.
- In order to determine the stability of the system using the **root locus technique** we find the range of values of k for which the complete performance of the system will be satisfactory and the operation is stable.
- **Bode plots** provides relative stability in terms of **gain margin** and **phase margin**

EXPERIMENT – 5

TRANSFER FUNCTION ANALYSIS OF I) TIME RESPONSE FOR STEP INPUT II) FREQUENCY RESPONSE FOR SINUSOIDAL INPUT.

AIM: To find the I) Time response for step input II) Frequency response for sinusoidal input.

I .TIME RESPONSE FOR STEP INPUT:

SOFTWARES USED: MATLAB

THEORY: The general expression of transfer function of a second order control system is given as

$$\frac{C(s)}{R(s)} = \frac{\omega_n^2}{s^2 + 2\zeta\omega_n s + \omega_n^2}$$

Here, ζ and ω_n are damping ratio and natural frequency of the system respectively
There are number of common terms in transient response characteristics and which are

1. **Delay time** (t_d) is the time required to reach at 50% of its final value by a time response signal during its first cycle of oscillation. $T_d = \frac{1+0.7\zeta}{\omega_n}$

2. **Rise time** (t_r) is the time required to reach at final value by a under damped time response signal during its first cycle of oscillation. If the signal is over damped, then rise time is counted as the time required by the response to rise from 10% to 90% of its final value. $\beta = \tan^{-1}\left(\frac{\sqrt{1-\zeta^2}}{\zeta}\right)$

$$T_r = \frac{1}{\omega_d}\tan^{-1}\left(-\frac{\sqrt{1-\zeta^2}}{\zeta}\right) = \frac{\pi - \beta}{\omega_d}$$

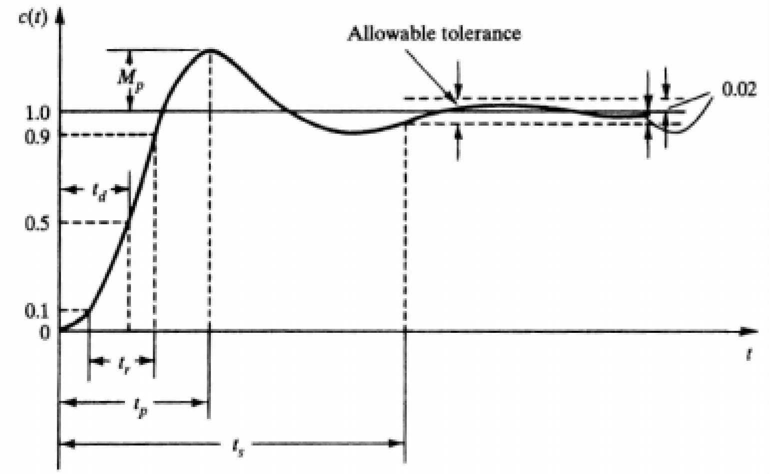
3. **Peak time** (t_p) is simply the time required by response to reach its first peak i.e. the peak of first cycle of oscillation, or first overshoot. $T_p = \frac{\pi}{\omega_d} = \frac{\pi}{\omega_n\sqrt{1-\zeta^2}}$

4. **Maximum overshoot** (M_p) is straight way difference between the magnitude of the highest peak of time response and magnitude of its steady state. Maximum overshoot is expressed in term of percentage of steady-state value of the response. As the first peak of response is normally maximum in magnitude, maximum overshoot is simply normalized difference between first peak and steady-state value of a response. $M_p = e^{-\pi\zeta/\sqrt{1-\zeta^2}}$

$$M_p\% = e^{-\pi\zeta/\sqrt{1-\zeta^2}} \times 100\%$$

5. **Settling time** (t_s) is the time required for a response to become steady. It is defined as the time required by the response to reach and steady within specified range of 2 % to 5 % of its final value. $T_s = \frac{4}{\zeta\omega_n}$ (2% Criterion)

6. **Steady-state error** (e_{ss}) is the difference between actual output and desired output at the infinite range of time. $e_{ss} = \lim_{t\to\infty} [r(t) - c(t)]$



PROBLEM STATEMENT: For the closed loop system defined by

$$\frac{C(S)}{R(S)} = \frac{100}{S^2 + 12S + 100}$$

Plot the unit step response curve and find time domain specifications

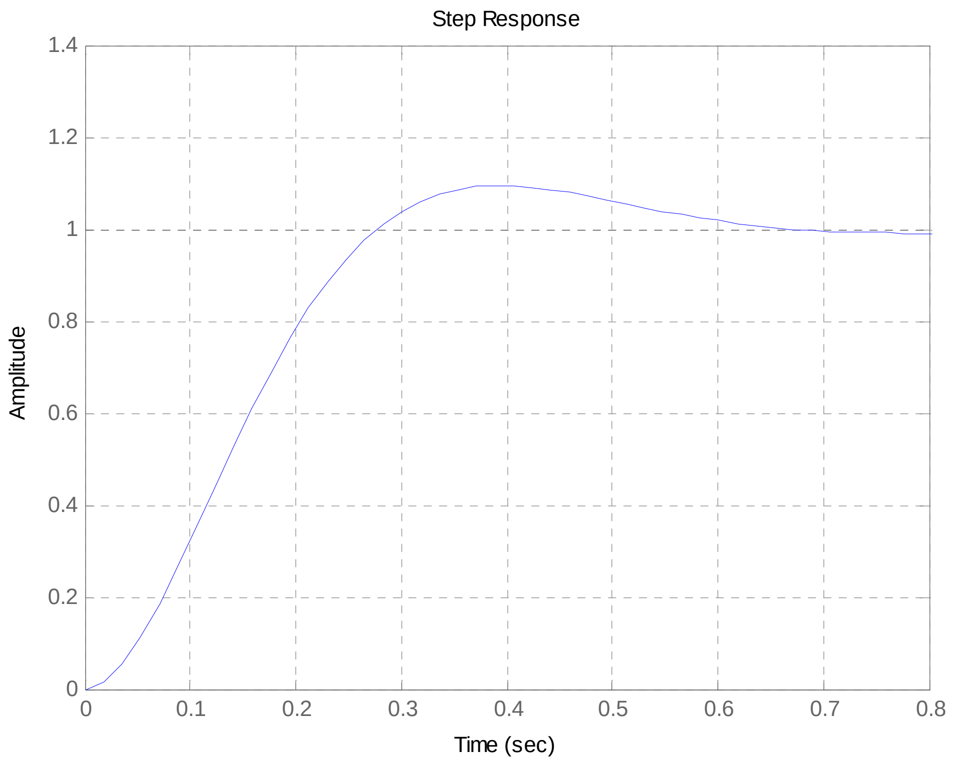
PROGRAM:

```
clc;
clear all;
close all;
num=input('enter the numerator coefficients---->');
den=input('enter the denominator coefficients---->');
system=tf(num,den);
system
step(system)
grid on;
wn=sqrt(den(1,3));
zeta= den(1,2)/(2*wn);
wd=wn*sqrt(1-zeta^2);
disp('Delay time in seconds is')
td=(1+0.7*zeta)/wd
disp('Rise time in seconds is')
theta=atan(sqrt(1-zeta^2)/zeta);
tr=(pi-theta)/wd
disp('Peak time in seconds');
tp=pi/wd
disp('Peak overshoot is');
```

```
mp=exp(-zeta*pi/sqrt(1-zeta^2))*100
disp('settling time in seconds is');
ts=4/(zeta*wn)
```

PROGRAM RESULT:

```
enter the numerator coefficients---->100
enter the denominator coefficients---->[1 12 100]

Transfer function:
      100
----------------
s^2 + 12 s + 100

Delay time in seconds is

td =

    0.1775

Rise time in seconds is

tr =

    0.2768

Peak time in seconds

tp =

    0.3927

Peak overshoot is

mp =

    9.4780

settling time in seconds is

ts =

    0.6667
```



II. FREQUENCY RESPONSE FOR SINUSOIDAL INPUT

By the term frequency response, we mean the steady-state response of a system to a sinusoidal input. Industrial control systems are often designed using frequency response methods. Many techniques are available in the frequency response methods for the analysis and design of control systems.

Consider a system with sinusoidal input $r(t) = A\sin\omega t$. The steady-state output may be written as, $c(t) = B\sin(\omega t + \phi)$. The magnitude and the phase relationship between the sinusoidal input and the steady-state output of a system is called *frequency response*. The frequency response test is performed by keeping the amplitude A fixed and determining B and Φ for a suitable range of frequencies. Whenever it is not possible to obtain the transfer function of a system through analytical techniques, frequency response test can be used to compute its transfer function.

The design and adjustment of open-loop transfer function of a system for specified closed-loop performance is carried out more easily in frequency domain. Further, the effects of noise and parameter variations are relatively easy to visualize and assess through frequency response. The Nyquist criteria is used to extract information about the stability and the relative stability of a system in frequency domain.

The transfer function of a standard second-order system can be written as,

$$T(s) = \frac{C(s)}{R(s)} = \frac{\omega_n^2}{s^2 + 2\zeta\omega_n s + \omega_n^2}.$$

Substituting s by $j\omega$ we obtain, $T(j\omega) = \dfrac{\omega_n^2}{(j\omega)^2 + 2\zeta\omega_n(j\omega) + \omega_n^2} = \dfrac{1}{(1-u^2) + j2\zeta u}.$

Where, $u = \omega / \omega_n$ is the normalized signal frequency. From the above equation we get,

$$|T(j\omega)| = M = \frac{1}{\sqrt{(1-u^2)^2 + (2\zeta u)^2}} \,.$$

$$\angle T(j\omega) = \phi = -\tan^{-1}[2\zeta u/(1-u^2)]$$

The steady-state output of the system for a sinusoidal input of unit magnitude and variable frequency ω is given by,

$$c(t) = \frac{1}{\sqrt{(1-u^2)^2 + (2\zeta u)^2}} \sin\left(\omega t - \tan^{-1}\frac{2\zeta u}{1-u^2}\right).$$

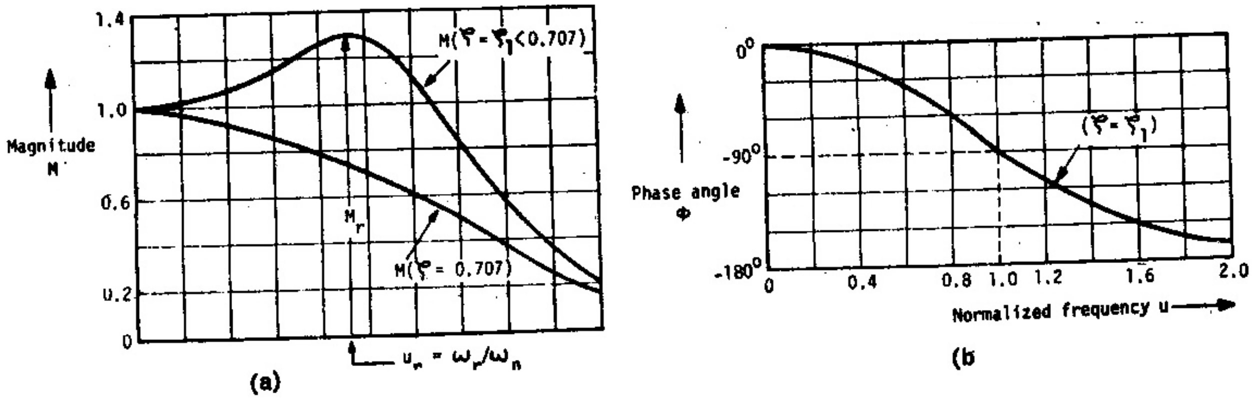
It is seen from the above equation that when,

$u = 0,\quad M = 1 \quad$ and $\quad \phi = 0$

$u = 1,\quad M = \frac{1}{2\zeta} \quad$ and $\quad \phi = -\pi/2$

$u = \infty,\quad M \to 0 \quad$ and $\quad \phi \to -\pi$

The magnitude and phase angle characteristics for normalized frequency u for certain values of ζ are shown in figure in the next page.



(a) (b

The frequency where ***M*** has a peak value is called *resonant frequency*. At this point the slope of the magnitude curve is zero. Setting $\left.\frac{dM}{du}\right|_{u=u_r} = 0$ we get, $-\frac{1}{2}\frac{\left[-4(1-u_r^2)u_r + 8\zeta^2 u_r\right]}{\left[(1-u_r^2)^2 + (2\zeta u_r)^2\right]^{3/2}} = 0.$

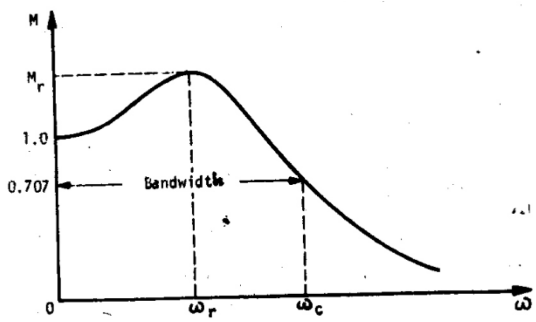
Solving, $u_r = \sqrt{1-2\zeta^2}$ or, resonant frequency $\omega_r = \omega_n\sqrt{1-2\zeta^2}$. ………… …… (01)

The *resonant peak* is given by resonant peak, $M_r = \frac{1}{2\zeta\sqrt{1-\zeta^2}}$. ……………… (02)

- For, $\zeta > \frac{1}{\sqrt{2}}\ (=0.707)$, the resonant frequency does not exist and *M* decreases monotonically with increasing *u*.
- For $0 < \zeta < \frac{1}{\sqrt{2}}$, the resonant frequency is always less than ω_n and the resonant peak has a value greater than 1.

From equation (01) and (02) it is seen that The resonant peak M_r of frequency response is indicative of **damping factor** and the resonant frequency ω_r is indicative of **natural frequency** for a given ζ and hence indicative of settling time.

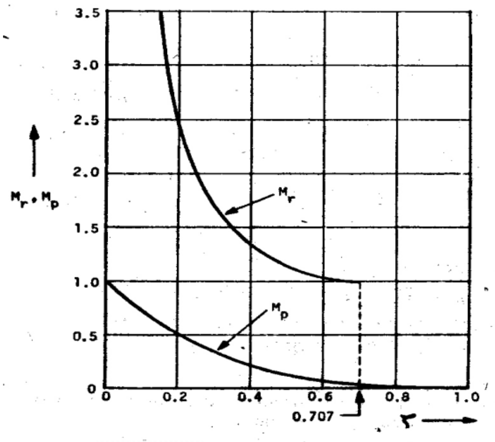
For $\omega > \omega_r$, M decreases monotonically. The frequency at which M has a value of $\frac{1}{\sqrt{2}}$ is called the cut-off frequency ω_c. The range of frequencies over which M is equal to or greater than $\frac{1}{\sqrt{2}}$ is defined as bandwidth, ω_b.



The bandwidth of a second-order system is given by,

$$\omega_b = \omega_n \left[1 - 2\zeta^2 + \sqrt{2 - 4\zeta^2 + 4\zeta^4}\right]^{1/2} \quad \text{............(03)}$$

Figure below shows the plot of resonant peak of frequency response and the peak overshoot of step response as a function of ζ.



M_r, M_p versus ζ.

It is seen that the two performance indices are correlated as both are the functions of the system damping factor ζ only. For $\zeta > \frac{1}{\sqrt{2}}$ $(= 0.707)$ the resonant peak does not exist and the correlation breaks down. For this range of ζ, M_p is hardly perceptible.

From equation (03) it is seen that the bandwidth is indicative of natural frequency and hence indicative of settling time, i.e., the speed of response for a given ζ.

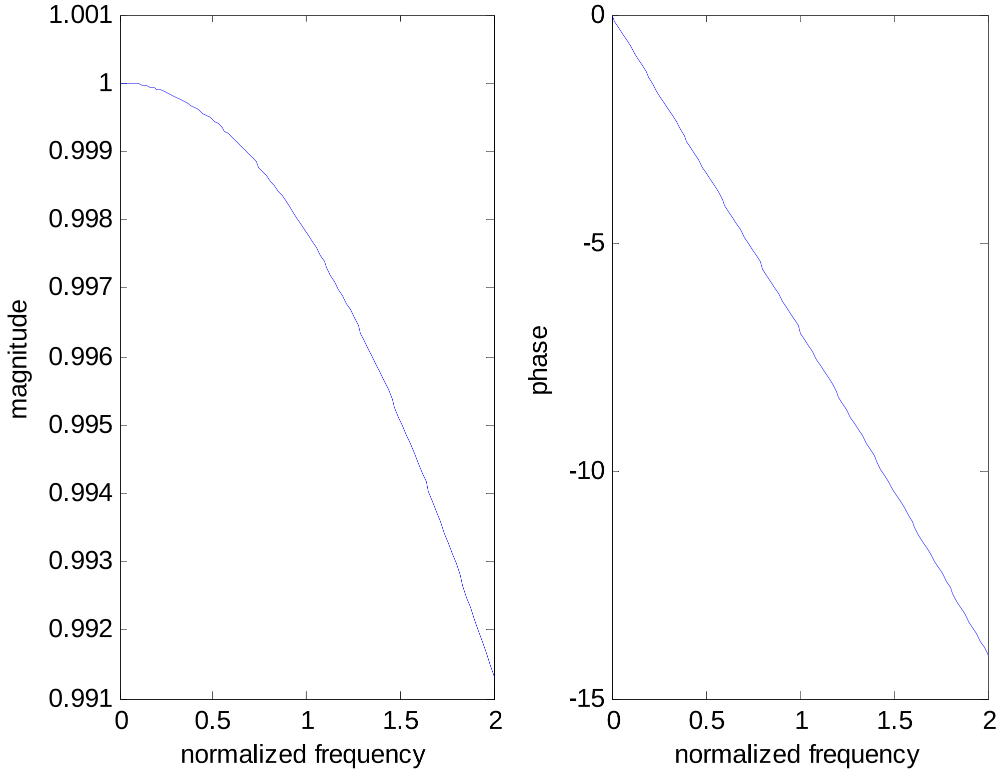
PROGRAM:

```
%Frequency Response of second order system
clc;
clear all;
close all;
num=input('enter the numerator coefficients---->');
den=input('enter the denominator coefficients---->');
%Transfer function
sys=tf(num,den);
wn=sqrt(den(1,3));
zeta= den(1,2)/(2*wn);
w=linspace(0,2);
u=w/wn;
len=length(u);
for k=1:len
    m(k)=1/(sqrt((1-u(k)^2)+(2*zeta*u(k))^2));
    phi(k)=-atan((2*zeta*u(k))/(1-u(k)^2))*180/pi;
end
subplot(1,2,1)
plot(w,m)
xlabel('normalized frequency')
ylabel('magnitude')
subplot(1,2,2)
```

```
plot(w,phi)
xlabel('normalized frequency')
ylabel('phase')
disp('resonant peak is');
mr=1/(2*zeta*sqrt(1-zeta^2))
disp('resonant frequency in rad/sec is');
wr=wn*sqrt(1-2*zeta^2)
disp('bandwidth in rad/sec is');
wb=wn*sqrt(1-2*zeta^2+sqrt(2-4*zeta^2+4*zeta^4))
disp('phase margin in degrees is')
pm=180+(atan(2*zeta/sqrt(-2*zeta^2+sqrt(4*zeta^4 +1))))*180/pi
```

PROGRAM RESULT:

```
enter the numerator coefficients---->100
enter the denominator coefficients---->[1 12 100]
resonant peak is

mr =

    1.0417
resonant frequency in rad/sec is

wr =

    5.2915

bandwidth in rad/sec is

wb =

   11.4824

phase margin in degrees is

pm =

  239.1873
```



RESULTS & DISCUSSIONS: Defining transfer functions and finding response using these transfer functions has been simulated using MATLAB.

Responses can be studied with addition of controllers and their effect on performance.

EXPERIMENT NO: 6

DESIGN OF LAG, LEAD AND LAG-LEAD COMPENSATOR

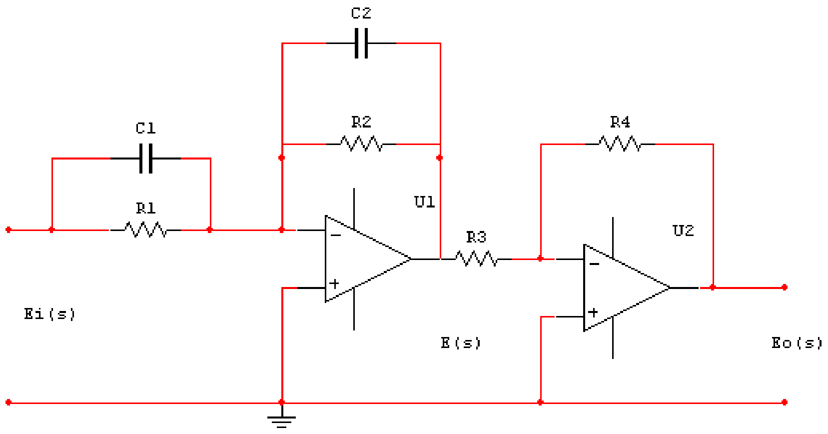
AIM: To design lag, lead compensator, lag-lead compensator

THEORY:

The primary objective of this experiment is to design the compensation of single –input-single-output linear time invariant control system.

Compensation is the modification of the system dynamics to satisfy the given specification. The compensation is done by adding some suitable device in which is called as compensator. Compensator is realized by such a way as to meet the performance specifications.

If sinusoidal input is applied to a network and if the steady state output has a phase lead, then the network is called a lead network, and if the output has a phase lag then the network is called as a phase lag network. Compensators are realized in our experiments using op-amps , electrical RC network as shown in figure.



$$\frac{Eo(s)}{Ei(s)} = \frac{R_2}{R_1}\frac{R_4}{R_3}\frac{R_1C_1s+1}{R_2C_2s+1} = \frac{R_4C_1s+\frac{1}{R_1C_1}}{R_3C_2s+\frac{1}{R_2C_2}} = K_c\alpha\frac{Ts+1}{\alpha Ts+1}$$

where

$$T = R_1C_1$$

$$\alpha T = R_2C_2$$

$$K_c = \frac{R_4C_1}{R_3C_2}$$

$$\alpha = \frac{R_2C_2}{R_1C_1}$$

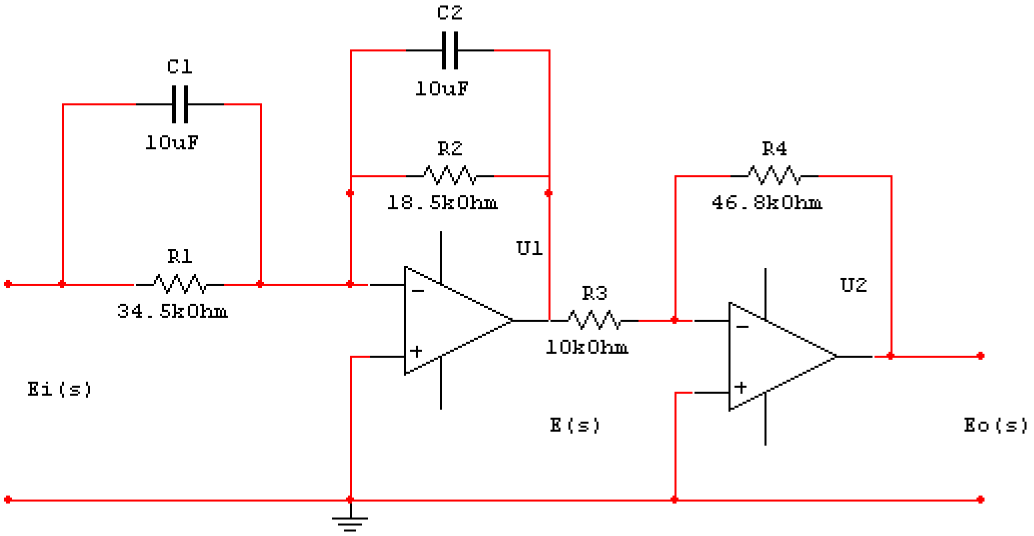
This network is a lead network if $R_1C_1 > R_2C_2$ or $\alpha<1$.
Or this is a lag network if $\alpha>1$ or $R_1C_1 < R_2C_2$

PROCEDURE

1) Consider any uncompensated system
2) Design the lead and lag compensator from the given circuit using the above equations.

3) Connect this design compensator to uncompensated system in series compensation.
4) Then find the closed loop transfer function equation for this compensated system
5) Plot the response for both uncompensated and compensated system

For Lead Compensator



The closed loop transfer function equation for the compensated system becomes

$$\frac{C(s)}{R(S)} = \frac{18.7(S+2.9)}{s(s+2)(s+5.4)+18.7(s+2.9)}$$

$$= \frac{18.7S+54.23}{s^3+7.4s^2+29.5s+54.23}$$

Hence

numc= [0 0 18.7 54.23]
denc= [1 7.4 29.5 54.23]

for the uncompensated system the closed loop transfer function is

$$\frac{C(s)}{R(S)} = \frac{4}{s^2+2s+4}$$

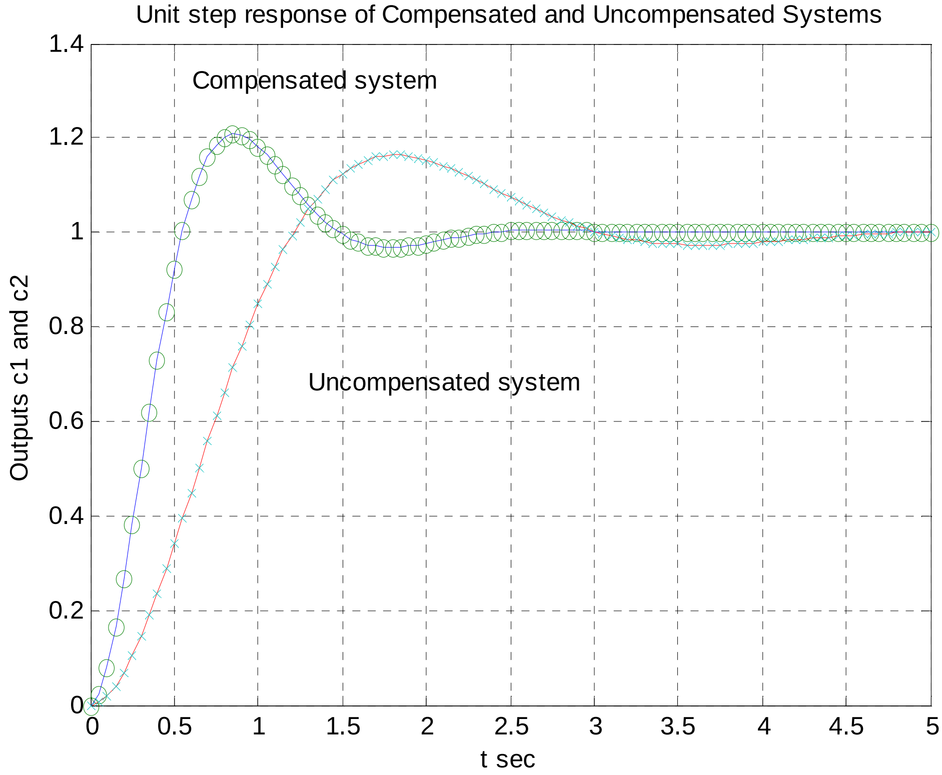
Hence

numc= [0 0 4]
denc= [1 2 4]

PROGRAM:

```
% Unit Step Response of Compensated and Uncompensated systems

numc=[0 0 18.7 54.23];
denc=[1 7.4 29.5 54.23];
num=[0 0 4];
den=[1 2 4];
t=0:0.05:5;
```

```
[c1,x1,t]=step(numc,denc,t);
[c2,x2,t]=step(num,den,t);
plot(t,c1,t,c1,'o',t,c2,t,c2,'x');
grid;
title('Unit step response of Compensated and Uncompensated Systems');
xlabel('t sec')
ylabel('Outputs c1 and c2');
text(0.6,1.32,'Compensated system');
text(1.3,0.68,'Uncompensated system');
```



For Lag Compensator

The closed loop transfer function equation for the compensated system becomes

$$\frac{C(s)}{R(S)} = \frac{1.0235(s+0.05)}{s(s+0.005)(s+1)(s+2)+1.0235(s+0.05)}$$

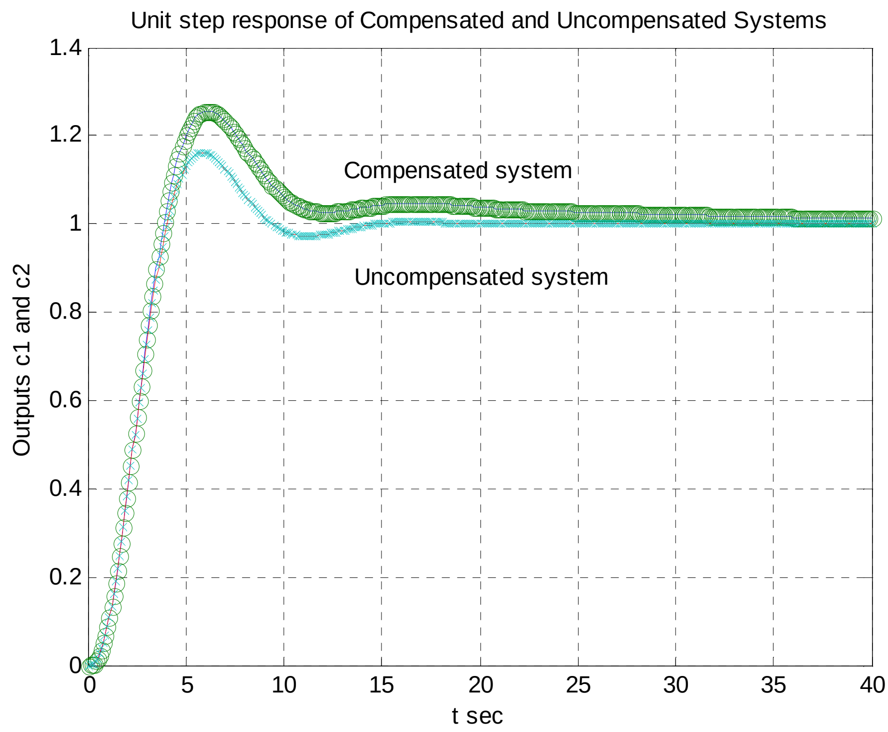
$$= \frac{1.0235s+0.0512}{s^4+3.005s^3+2.015s^2+1.0335s+0.0512}$$

for the uncompensated system the closed loop transfer function is

$$\frac{C(s)}{R(S)} = \frac{1.06}{s(s+1)(s+2)+1.06}$$

PROGRAM:

```
% Unit Step Response of Compensated and Uncompensated systems

numc=[0 0 0 1.0235 0.0512];
denc=[1 3.005 2.015 1.0335 0.0512];
num=[0 0 0 1.06];
den=[1 3 2 1.06];
t=0:0.1:40;
[c1,x1,t]=step(numc,denc,t);
[c2,x2,t]=step(num,den,t);
plot(t,c1,t,c1,'o',t,c2,t,c2,'x');
grid;
text(13,1.12,'Compensated system');
text(13.6,0.88,'Uncompensated system');
title('Unit step response of Compensated and Uncompensated Systems');
xlabel('t sec')
ylabel('Outputs c1 and c2');
```



RESULTS & DISCUSSIONS: Compensators are added to existing systems to improve their performance.

- Such compensators based on the change required in performance of the system have been designed and improvement in performance analyzed using MATLAB.
- Change in the performance of the system with compensator is observed.

EXPERIMENT – 7

LOAD FLOW STUDIES

AIM: For the given system, find load flow solution using

a) Gauss Seidel Method.

b) Newton Raphson Method.

c) Fast Decoupled Method.

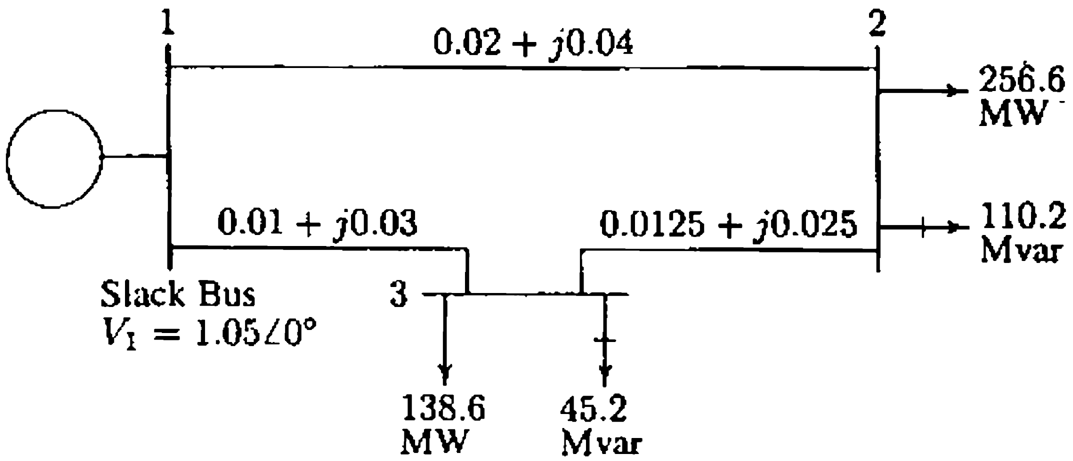
APPARATUS USED:

a) MATLAB Software.

b) Power System Functions.

PROBLEM STATEMENT:

Figure shows the one line diagram of a simple three-bus power system with generation at bus 1. The magnitude of voltage at bus 1is adjusted to 1.05 p.u.The scheduled loads at buses 2 and 3 are as marked on the diagram.Line impedances are marked in p.u. on a 100 MVA base and the line charging susceptances are neglected.



a) Using different loadflow methods, determine the phasor values of the voltage at load buses 2 and 3.

b) Find the slack bus real and reactive power.

c) Determine line flows and line losses

Bus data format:

Bus No.	Bus Code	Voltage Mag.	Angle Deg.	Load Mw Mvar	Generator Mw Mvar Qmin Qmax	Injected Mvar.

Line data format:

Bus from	Bus to	R Pu	X Pu	½ B Pu	Line Code or Tap setting

a) LOAD FLOW USING GAUSS SEIDEL METHOD

```
PROGRAM:
clear all;
clc;
basemva=100;
accuracy=0.001;
accl=1.6;
maxiter=80;
busdata=[1 1 1.05 0 0     0     0 0 0 0 0
         2 0 1.00 0 256.6 110.2 0 0 0 0 0
         3 0 1.00 0 138.6 45.2  0 0 0 0 0];
linedata=[1 2 0.02  0.04  0 1
          1 3 0.01  0.03  0 1
          2 3 0.125 0.025 0 1];
lfybus
lfgauss
lineflow
```

```
PROGRAM  RESULT:

          Line Flow and Losses

     --Line--   Power at bus & line flow    --Line loss--  Transformer
     from  to     MW       Mvar      MVA       MW       Mvar       tap

     1         414.010  191.484  456.147
           2   223.081  136.143  261.343   12.390   24.780
           3   190.972   55.392  198.843    3.586   10.759
```

```
     2        -256.600 -110.200  279.263
           1 -210.691 -111.363  238.312   12.390   24.780
           3  -45.898    1.135   45.912    2.874    0.575

     3        -138.600  -45.200  145.784
           1 -187.386  -44.633  192.628    3.586   10.759
           2   48.772   -0.560 48.775    2.874    0.575


  Total loss                            18.851   36.114
```

b) **LOAD FLOW USING NEWTON-RAPHSON METHOD**

PROGRAM:

```
clear all;
clc;
basemva=100;
accuracy=0.001;
maxiter=80;
busdata=[1 1 1.05 0 0     0     0 0 0 0 0
         2 0 1.00 0 256.6 110.2 0 0 0 0 0
         3 0 1.00 0 138.6 45.2  0 0 0 0 0];
linedata=[1 2 0.02 0.04 0 1
          1 3 0.01 0.03 0 1
          2 3 0.125 0.025 0 1];
lfybus
lfnewton
lineflow
```

PROGRAM RESULT:

```
    Line Flow and Losses

     --Line--  Power at bus & line flow    --Line loss--  Transformer
     from  to    MW        Mvar      MVA       MW       Mvar       tap

     1         413.987  191.456  456.114
           2  223.092  136.124  261.343   12.390   24.780
           3  190.957   55.388  198.828    3.586   10.757

     2        -256.600 -110.200  279.263
           1 -210.702 -111.344  238.313   12.390   24.780
           3  -45.898    1.144   45.912    2.874    0.575

     3        -138.600  -45.200  145.784
           1 -187.372  -44.630  192.614    3.586   10.757
           2   48.772   -0.570   48.775    2.874    0.575
```

```
    Total loss                            18.850   36.112
```

c) LOAD FLOW USING FAST DECOUPLED METHOD

PROGRAM:

```
clear all;
clc;
basemva=100;
accuracy=0.001;
accl=1.6;
maxiter=80;
busdata=[1 1 1.05 0 0     0     0 0 0 0 0
         2 0 1.00 0 256.6 110.2 0 0 0 0 0
         3 0 1.00 0 138.6 45.2  0 0 0 0 0];
linedata=[1 2 0.02  0.04  0 1
          1 3 0.01  0.03  0 1
          2 3 0.125 0.025 0 1];
lfybus
decouple
lineflow;
```

PROGRAM RESULT:

```
    Line Flow and Losses

     --Line--  Power at bus & line flow    --Line loss--  Transformer
     from  to    MW       Mvar      MVA       MW       Mvar       tap

     1         413.960  191.719  456.201
           2   222.724  135.280  260.589   12.319   24.637
           3   191.099   55.998  199.135    3.597   10.790

     2        -256.600 -110.200  279.263
           1  -210.405 -110.643  237.723   12.319   24.637
           3   -45.489    1.232   45.505    2.821    0.564
```

```
     3       -138.600  -45.200  145.784

           1 -187.503  -45.208  192.876    3.597   10.790

           2   48.310   -0.668   48.314    2.821    0.564


    Total loss                            18.737   35.992
```

RESULTS & DISCUSSIONS: Load flow studies using different methods have been carried out using MATLAB.

- These form tools to analyze large practical power systems using computer and simulation softwares.
- Load flow studies form basis for many power system studies. Hence they play vital role in planning, operation and control of Power Systems.
- Convergence, accuracy, time taken and number of iterations taken for solution are largely dependent on the method chosen for load flow analysis.

EXPERIMENT – 8

FAULT ANALYSIS

AIM: To find the fault current in a given power system where there is

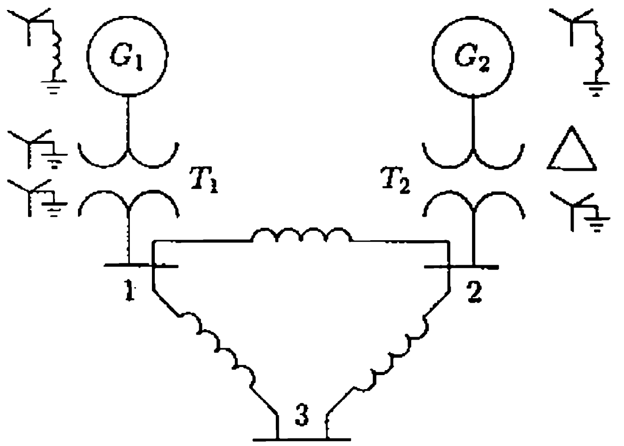
a) Balanced 3-ϕ fault. (LLL/LLLG).
b) Single line to ground fault(LG).
c) Line to line fault(LL).
d) Double line to ground fault(LLG).

SOFTWARES USED:

c) MATLAB Software
d) Power System Functions.

PROBLEM STATEMENT:

For the given power systems shown in fig, the neutral of each generator is grounded through a current limiting reactor of 0.25/3 p.u. on a 100 MVA base. The system data expressed in p.u. on a 100 MVA base is tabulated below. The generators are running on no load at their related voltage and rated frequency with their emfs in phase.



Item	Base MVA	Voltage Rating	X^1	X^2	X^0
G_1	100	20 kV	0.15	0.15	0.05
G_2	100	20 kV	0.15	0.15	0.05
T_1	100	20/220 kV	0.10	0.10	0.10
T_2	100	20/220 kV	0.10	0.10	0.10
L_{12}	100	220 kV	0.125	0.125	0.30
L_{13}	100	220 kV	0.15	0.15	0.35
L_{23}	100	220 kV	0.25	0.25	0.7125

Determine the fault current for the following details of faults.

a) A balanced 3-ϕ fault at bus 3 through a fault impedance Zf = 0.1pu.

b) A Single line to ground fault at bus 3 through a fault impedance Zf = 0.1pu.

c) A line to line fault at bus 3, fault impedance Zf = 0.1pu.

d) A double line to ground fault at bus 3 through a fault impedance Zf = 0.1pu

PROGRAM:

```
%program to find fault analysis%
 clc;
 clear all;
 close all;
 %positive sequence reactance data%
zdata1=[0 1 0 0.25
           0 2 0 0.25
           1 2 0 0.125
           1 3 0 0.15
           2 3 0 0.25];
    %zero sequence impedence data%
zdata0=[0 1 0 0.4
        0 2 0 0.1
        1 2 0 0.3
        1 3 0 0.35
        2 3 0 0.7125];
   %negative sequence reactance=positive reactance%
%    zdata2=[0 1 0 0.25
%            0 2 0 0.25
%            1 2 0 0.125
%            1 3 0 0.15
%            2 3 0 0.25];
zdata2=zdata1;
zbus1=zbuild(zdata1);
```

```
zbus0=zbuild(zdata0);
zbus2=zbus1;
symfault(zdata1,zbus1);
lgfault(zdata0,zbus0,zdata1,zbus1,zdata2,zbus2)
llfault(zdata1,zbus1,zdata2,zbus2)
dlgfault(zdata0,zbus0,zdata1,zbus1,zdata2,zbus2)
```

PROGRAM RESULT:

a) **Balanced three-phase fault(LLL/LLLG)**

```
Enter Faulted Bus No. -> 3

Enter Fault Impedance Zf = R + j*X in complex form (for bolted fault
enter 0). Zf = 0+j*0.1

Balanced three-phase fault at bus No. 3
Total fault current =   3.1250 per unit

Bus Voltages during fault in per unit

     Bus     Voltage       Angle
     No.     Magnitude     degrees
      1       0.5938       0.0000
      2       0.6250       0.0000
      3       0.3125       0.0000

Line currents for fault at bus No.  3

     From      To     Current     Angle
     Bus       Bus    Magnitude   degrees
      G         1      1.6250    -90.0000
      1         3      1.8750    -90.0000
      G         2      1.5000    -90.0000
      2         1      0.2500    -90.0000
      2         3      1.2500    -90.0000
      3         F      3.1250    -90.0000
```

b) **Single line to-ground fault(LG)**

```
Enter Faulted Bus No. -> 3

Enter Fault Impedance Zf = R + j*X in
complex form (for bolted fault enter 0). Zf = 0+j*0.1

Single line to-ground fault at bus No. 3
```

```
Total fault current =    2.7523 per unit


Bus Voltages during the fault in per unit

     Bus    -------Voltage Magnitude-------
     No.    Phase a     Phase b     Phase c
     1      0.6330      1.0046      1.0046
     2      0.7202      0.9757      0.9757
     3      0.2752      1.0647      1.0647

Line currents for fault at bus No.  3

     From      To      -----Line Current Magnitude----
     Bus       Bus     Phase a     Phase b     Phase c
      1         3      1.6514      0.0000      0.0000
      2         1      0.3761      0.1560      0.1560
      2         3      1.1009      0.0000      0.0000
      3         F      2.7523      0.0000      0.0000
```

c) **Line-to-line fault analysis (LL)**

```
Enter Faulted Bus No. -> 3

Enter Fault Impedance Zf = R + j*X in
complex form (for bolted fault enter 0). Zf = 0+j*0.1

Line-to-line fault at bus No. 3
Total fault current =    3.2075 per unit

Bus Voltages during the fault in per unit

     Bus    -------Voltage Magnitude-------
     No.    Phase a     Phase b     Phase c
     1      1.0000      0.6720      0.6720
     2      1.0000      0.6939      0.6939
     3      1.0000      0.5251      0.5251

Line currents for fault at bus No.  3

     From      To      -----Line Current Magnitude----
     Bus       Bus     Phase a     Phase b     Phase c
      1         3      0.0000      1.9245      1.9245
      2         1      0.0000      0.2566      0.2566
      2         3      0.0000      1.2830      1.2830
      3         F      0.0000      3.2075      3.2075
```

d) **Double line-to-ground fault analysis(LLG)**

```
Enter Faulted Bus No. -> 3

Enter Fault Impedance Zf = R + j*X in
complex form (for bolted fault enter 0). Zf = 0+j*0.1

Double line-to-ground fault at bus No. 3
Total fault current =    1.9737 per unit

Bus Voltages during the fault in per unit

     Bus    -------Voltage Magnitude-------
     No.    Phase a     Phase b     Phase c
     1      1.0066      0.5088      0.5088
     2      0.9638      0.5740      0.5740
     3      1.0855      0.1974      0.1974

Line currents for fault at bus No.  3

     From      To      -----Line Current Magnitude----
     Bus       Bus     Phase a     Phase b     Phase c
      1         3      0.0000      2.4350      2.4350
      2         1      0.1118      0.3682      0.3682
      2         3      0.0000      1.6233      1.6233
      3         F      0.0000      4.0583      4.0583
```

RESULTS & DISCUSSIONS: Fault studies critical to protection design have been carried out using MATLAB.

- The effect of the various power system components on the fault level can be readily seen from the above simulation results.
- Severity of fault based on type, fault location and impedance can be studied.
- Such simulations help in choosing appropriate protection system/relay co ordination.

EXPERIMENT – 9

TRANSIENT STABILITY STUDIES

AIM: To find the transient stability when there is sudden increase in power input and on occurrence of fault.

SOFTWARES USED:

a) MATLAB Software
b) Power System Functions.

CASE: 1

When there is a sudden increase in power input.

A synchronous generator is connected to infinite bus bars as shown in fig 1.0 (All reactances are in p.u.). It is delivering a real power of 0.6 p.u. at 0.8 pt lag at voltage of 1.0 p.u.

a) Find the maximum power that can be transmitted without the loss of synchronism.
b) Repeat a) with zero initial power input.

CASE: 2

When there is occurrence of fault.

A synchronous generator is connected to infinite bus as shown in fig 2.0 [All reactances are in p.u]. Delivery real power Po = 0.8p.u., Q=0.074p.u. at a voltage of 1.0 p.u.

a) A temporary 3-phase fault occurs at the sending end of line at point F1. When the fault is cleared both the lines are intact. Determine the critical clearing angle and critical clearing time.
b) A 3-phase fault occurs at the middle of the line, the fault is cleared and the faulty line is isolated. Determine the critical clearing angle.

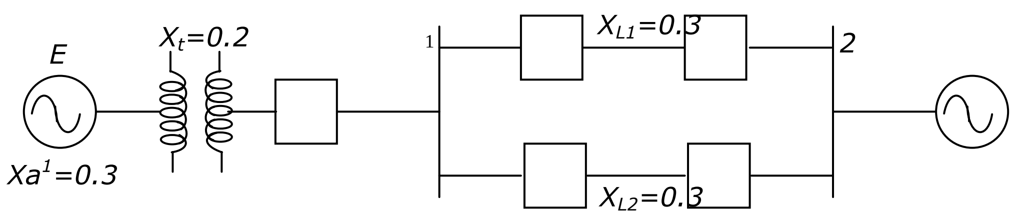


Fig 1.0

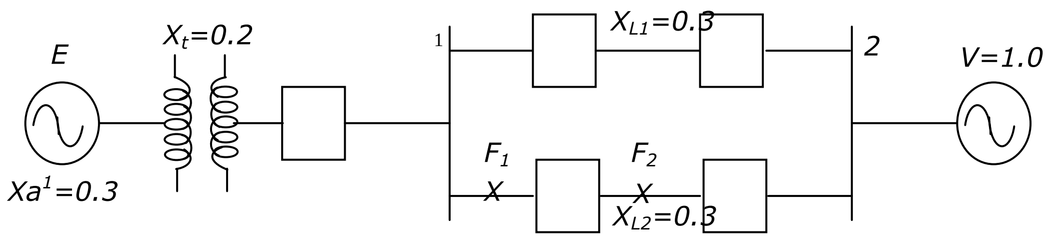
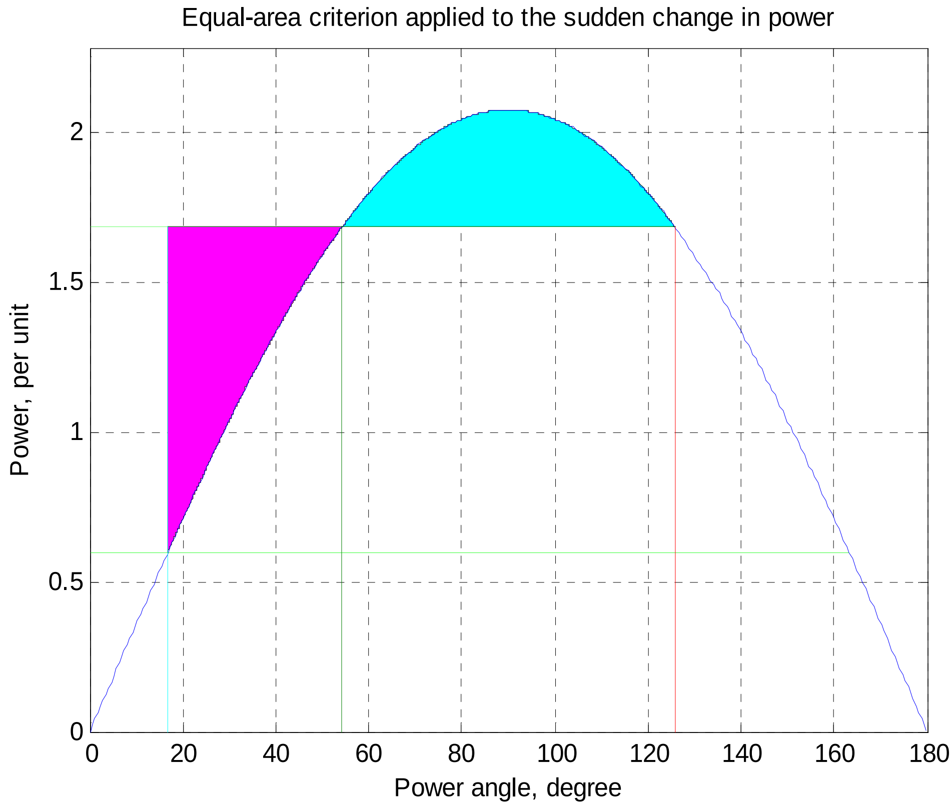


Fig 2.0

```
    %TRANSIENT STABILITY STUDIES: SUDDEN INCREASE IN POWER INPUT %

%program to calculate the critical clearing angle time when 3phase
with 0.6pu initial power%


clc;
clear all;
close all;


p0=0.6;
E=1.35;
V=1.0;
X=0.65;
eacpower(p0,E,V,X)



%program to calculate the critical clearing angle time when 3ph with
zero initial power%


pm=0.0;
E=1.35;
V=1.0;
X=0.65;
eacpower(pm,E,V,X)
```

OUTPUT OF TRANSIENT STABILITY STUDIES

I .With 0.6 p.u initial power:

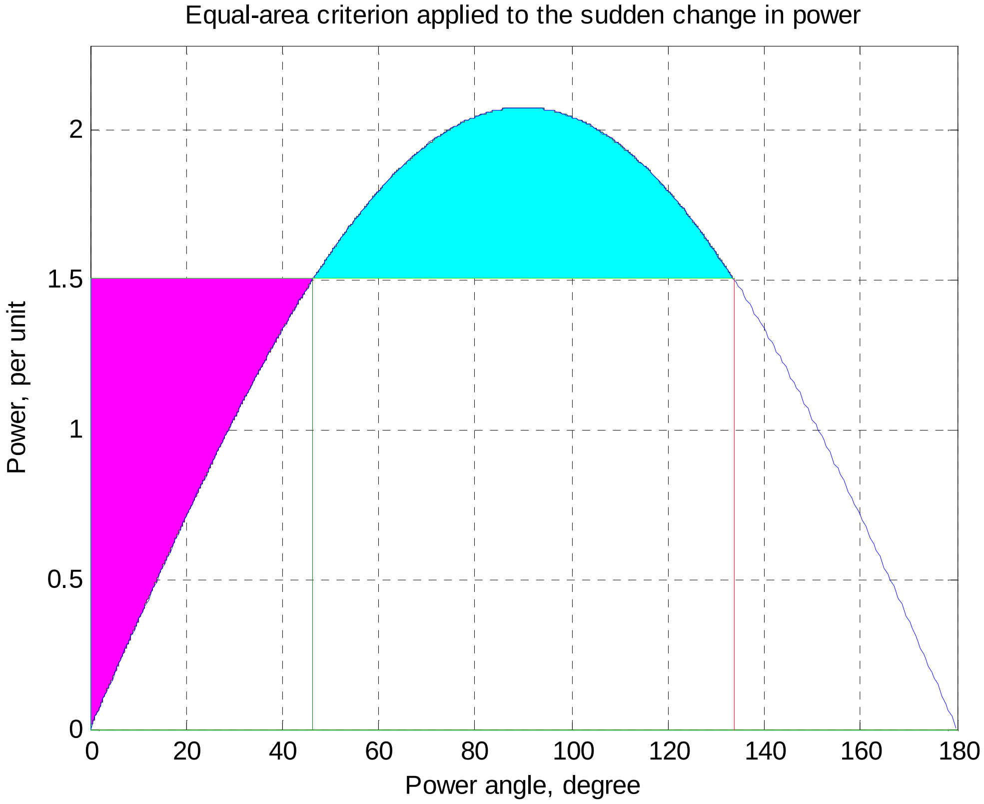


PROGRAM RESULT:

```
Initial power                      =  0.600 p.u.
Initial power angle                = 16.791 degrees
Sudden additional power            =  1.084 p.u.
Total power for critical stability =  1.684 p.u.
Maximum angle swing                =125.840 degrees
New operating angle                = 54.160 degrees
```

OUTPUT OF TRANSIENT STABILITY STUDIES

II .With Zero initial power:



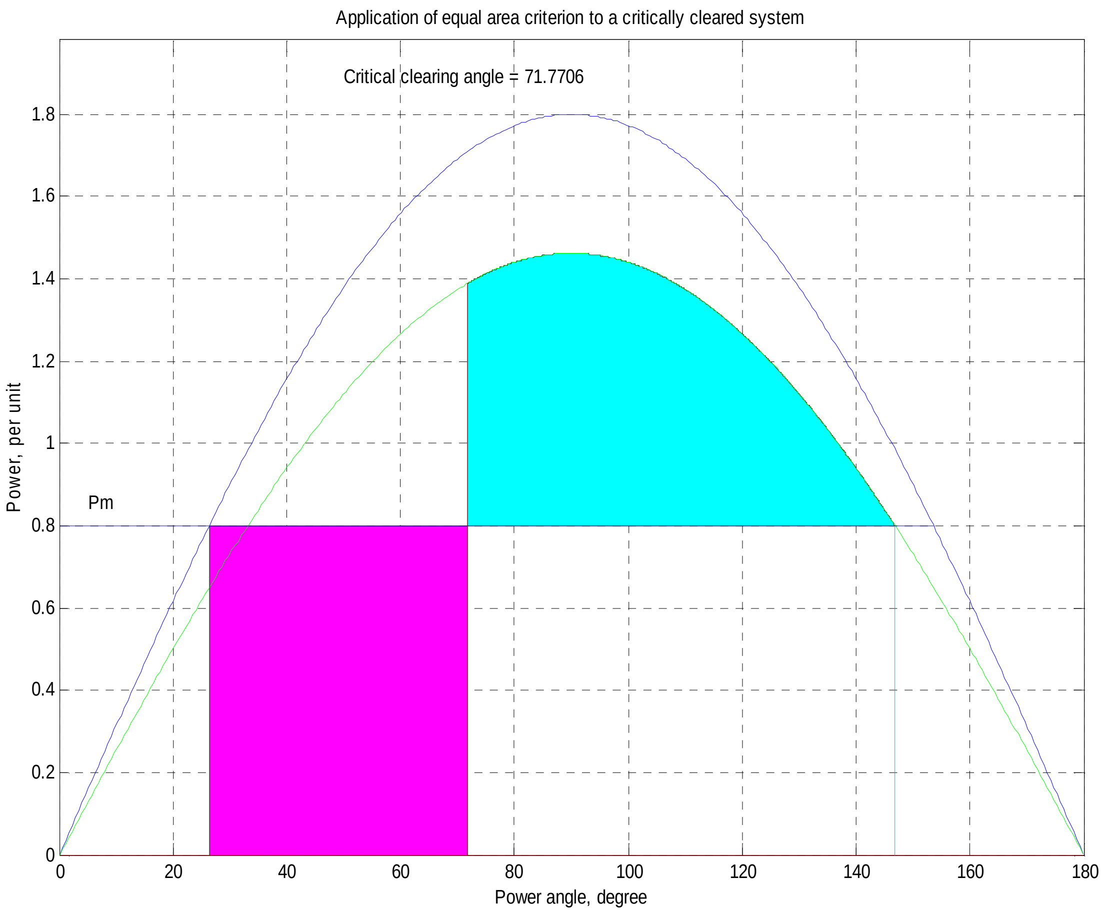
PROGRAM RESULT:

```
Initial power                      =  0.000 p.u.
Initial power angle                =  0.000 degrees
Sudden additional power            =  1.505 p.u.
Total power for critical stability =  1.505 p.u.
Maximum angle swing                =133.563 degrees
New operating angle                = 46.437 degrees
```

```
%TRANSIENT STABILITY STUDIES: FAULT%

%program to calculate the critical clearing angle time when a 3ph
fault that occurred at the sending end is cleared%

clc;
clear all;
close all;

pm=0.8;
E=1.17;
V=1.0;
X1=0.65;
X2=inf;
X3=0.8;
eacfault(pm,E,V,X1,X2,X3);

%program to calculate the critical clearing angle&time when 3ph fault
that occurred in the middle of the line(F2)is cleared and the faulty
line is isolated%

pm=0.8;
E=1.17;
V=1.0;
X1=0.65;
X2=1.8;
X3=0.8;
eacfault(pm,E,V,X1,X2,X3);
```

OUTPUT OF TRANSIENT STABILITY STUDIES

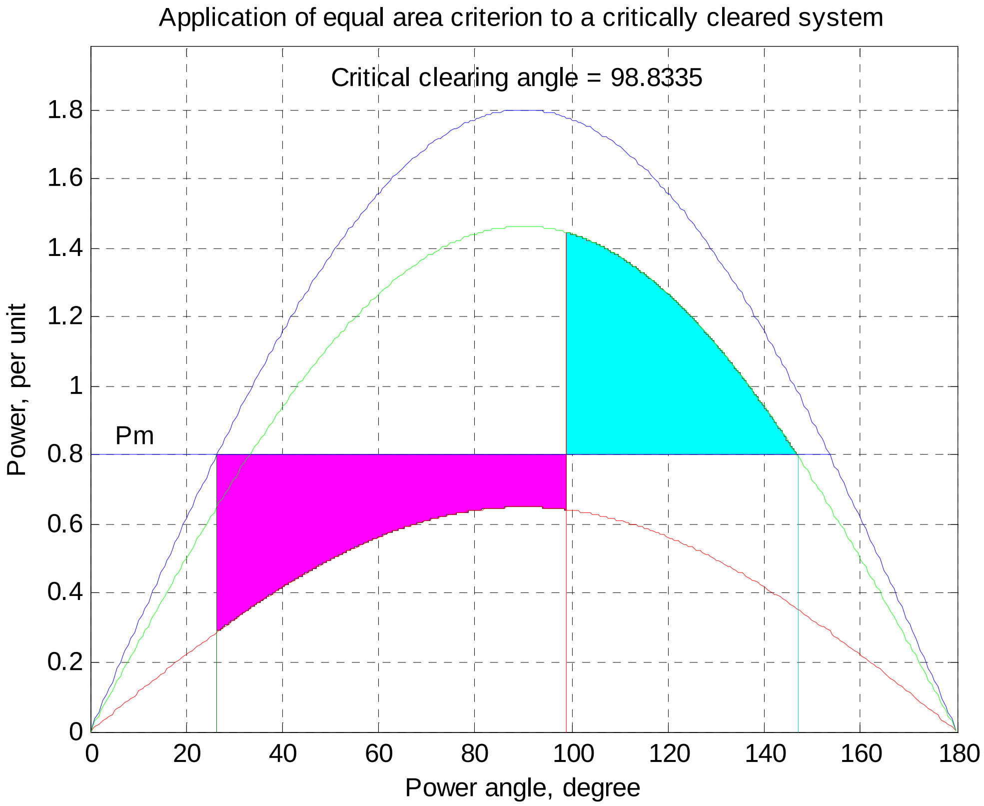
I. TRANSIENT STABILITY 3-PH FAULT AT SENDING END.



PROGRAM RESULT:

```
For this case tc can be found from analytical formula.
To find tc enter Inertia Constant H, (or 0 to skip) H = 0

Initial power angle     =  26.388
Maximum angle swing     = 146.838
Critical clearing angle =  71.771
```

II. TRANSIENT STABILITY 3-PH FAULT AT RECEIVING END



PROGRAM RESULT:

```
Initial power angle      =  26.388
Maximum angle swing      = 146.838
Critical clearing angle =  98.834
```

RESULTS & DISCUSSIONS: Transient stability analysis is key to the functioning of the dynamic nature of power system. Events like sudden load increase/ decrease and faults are common.

- Determining critical clearing angles to assess the stability margin is done using MATLAB using equal area criteria.
- Shifting of operating point (delta angle) under the two cases can be seen. This is the new stable operating point.
- Dependence of fault clearing times on stability can be appreciated

EXPERIMENT – 10

ECONOMIC POWER SCHEDULING

AIM: To find the optimal dispatch and total cost of generator

a) without line losses and generator limits
b) with line losses and generator limits

SOFTWARES USED:

a) MATLAB.
b) Power System Functions.

PROBLEM STATEMENT: The fuel cost function for three thermal plants in Rs/hr are given by

$$C1 = 500+5.3P_1+0.004P_1^2$$

$$C2 = 400+5.5P_2 + 0.006P_2^2$$

$$C3 = 500+5.3P_3+0.004P_3^2$$

where P_1, P_2, P_3 are in Mega watt. The total load is 925MW.

a) Neglecting line losses and generator limits. Find optimal dispatch and the total cost in Rs/hr.
b) With the generator limits (in Megawatts) for the 3 generators

$200<=P_1<=450$

$150<=P_2<=350$

$100<=P_3<=225$

OPTIMAL DISPATCH OF GENERATOR

PROGRAM:

```
%program to find optimal dispatch with & without generator limits and line losses%

clc;
clear all;
close all;
cost=[500 5.3 0.004
      400 5.5 0.006
      200 5.8 0.009]; %Input Data of the cost functions
```

```
mwlimits=[0 500
          0 500
          0 500]; %gen limits( give appropriate limits to check for
with and without limits cases)
dispatch;% User Defined Function to find optimal dispatch
gencost;% User defined function to calculate generation cost
```

PROGRAM RESULT:

```
Enter total demand Pdt = 975
Incremental cost of delivered power (system lambda) = 9.163158
Rs./MWh
Optimal Dispatch of Generation:

  482.8947
  305.2632
  186.8421


Total generation cost = 8228.03 Rs/hr
```

RESULTS & DISCUSSIONS: Optimal Dispatch of generation is carried out using MATLAB.

- It helps planning of generation schedule accounting for economics and constraints on the generators. Optimization applied to power system is studied.
- Lambda is the Incremental Cost of supplying one unit of electricity.
- Difference in total cost of generation with and without generation limits is studied.
- Considering the line losses, generation must equal sum of demand and losses.