

Experiment:01

Name of Experiment: Study of the transient response of 1st and 2nd order circuit

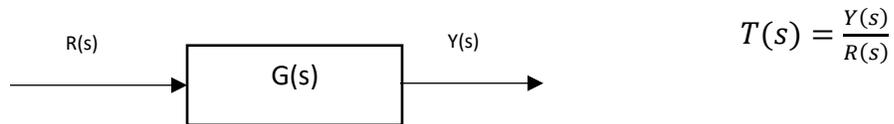
Objective: i) Observe the time response of 1st and 2nd order circuit.

ii) Calculate time constant, τ , of 1st order circuit.

iii) Calculate damping constant, ζ , of 2nd order circuit under different conditions (under/over/critically damped) and obtain voltage expression.

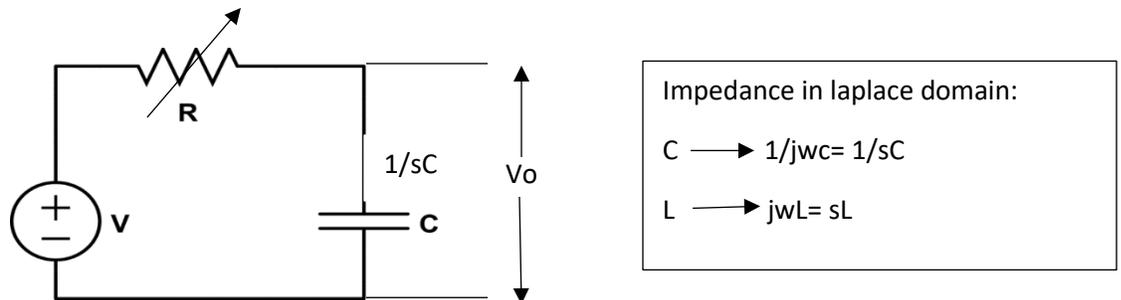
Theory:

Transfer function: It is the ratio of O/P by I/P of any system represented in laplace domain. It fully describes a control system. The order, type and frequency response can be derived from the open loop T/F (OLTF). They are used to determine the stability of the system. From the denominator of T/F, roots of the system can be obtained.



Order: The order of the system is derived from the denominator of the T/F. The number of poles (solutions of S in denominator) represents order of the system. Thus a 1st order system will have one pole, a 2nd order will have 2 poles and so on. Reactive components are responsible for order

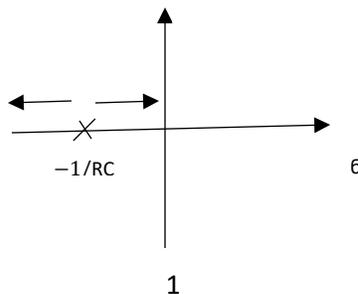
1st order circuit:



$$\frac{V_o(s)}{V_{in}(s)} = \frac{1}{RsC + 1} = \frac{\frac{1}{RC}}{s + 1/RC}$$

$j\omega_d$

Pole-Zero diagram:



For 1st order pole is in real axis. If R is varied it will move along real axis.

To obtain time response, we do inverse laplace:

If $V_{in}(t)=1; t \geq 0$ (a step input)

$$V_o(s) = \frac{\frac{1}{RC}}{s + 1/RC} * 1/s$$

Whereas $V_o(t) = 1 - e^{-t/RC}$; here time constant, $\tau = RC$

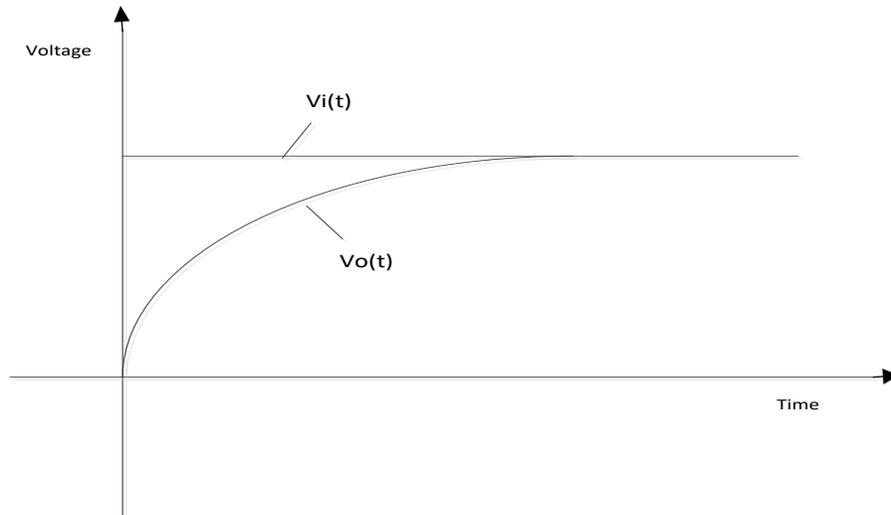


Figure: Response of 1st order circuit

Thus, if the pole is closer to origin ($R \uparrow$) T_s is higher and so curve reaches stability at a later point.

2nd order circuit:

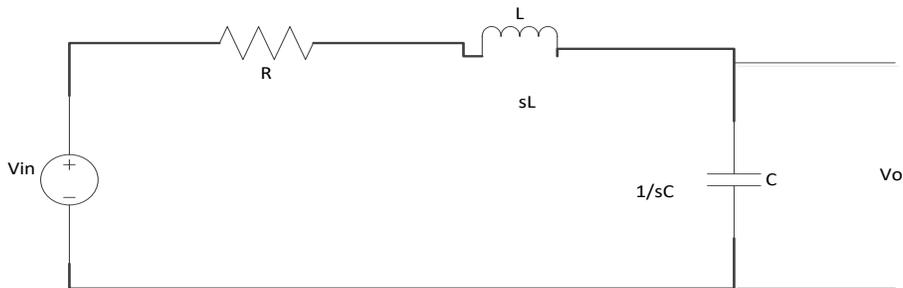


Figure: 2nd order circuit

$$\text{Transfer Function: } T(s) = \frac{\frac{1}{LC}}{s^2 + \left(\frac{R}{L}\right)s + \left(\frac{1}{LC}\right)}$$

2nd order equation can be compared with: $\frac{W_n^2}{s^2+2\zeta W_n s+W_n^2}$; where, ζ =damping factor

$W_n = \text{Natural frequency}$

comparing the transfer function with the generalized form we obtain:

$$W_n = \frac{1}{\sqrt{LC}}$$

$$\zeta = \frac{R}{2} \sqrt{C/L}$$

Thus by varying pot we can obtain different ζ

$\zeta=0$ \longrightarrow Undamped system (oscillatory)

$0 < \zeta < 1$ \longrightarrow Underdamped System

$\zeta=1$ \longrightarrow Critically Damped System

$\zeta > 1$ \longrightarrow Overdamped System

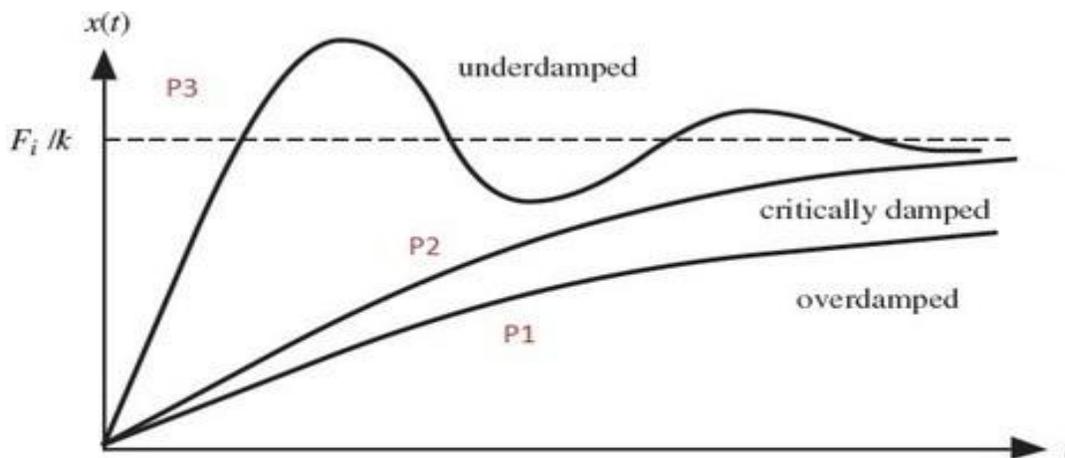


Figure : System under different dampings

For critically damped system roots are real & equal.

For underdamped system roots are complex and for overdamped roots are real and equal.

Setup circuit:

1st order:

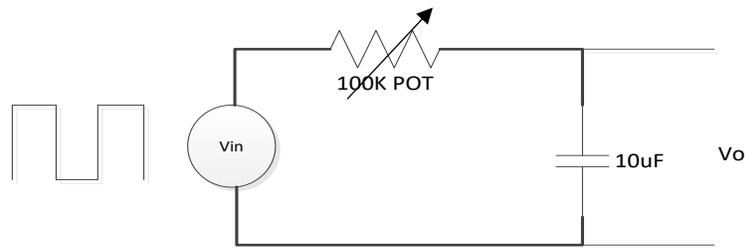


Figure :Setup 1

2nd order:

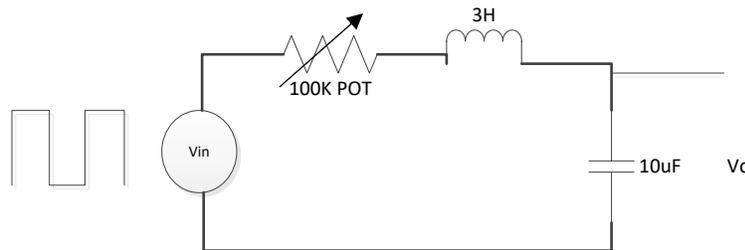


Figure :Setup 2

Procedure:

Setup 1: i) Setup the circuit in a bread board, provide input, ground and oscilloscope connections from Analog discovery 2 device.

ii) Vary the POT and for a fixed value of R measure the settling time from the output window.

iii) Measure the respective R and compare it with the calculated R from $T_s=5\tau$

Setup 2: For underdamped and overdamped:

i) Setup the circuit in a breadboard, providing required connections from Analog discovery 2 device.

ii) Vary the pot and obtain underdamped condition. Measure R of POT.

iii) Repeat step (ii) for overdamped condition.

For critically damped:

i) Calculate value of R from the equation : $\zeta = \frac{R}{2} \sqrt{C/L}$

ii) Adjust POT to that value.

iii) Setup circuit and observe output.

Obtained Parameter value:

1st order system:	R=		C=	
2nd order system:	Over damped	R=	C=	L=
	Under damped	R=	C=	L=
	Critical damped	R=	C=	L=

Assignment:

- 1) Obtain the time constant for your 1st order practical circuit.
- 2) Obtain the damping constant under all 3 conditions for the practical circuit.
- 3) For 1st and 2nd order circuit (all 3 conditions) give the output time response using the R,L,C values used practically.

Experiment No:2

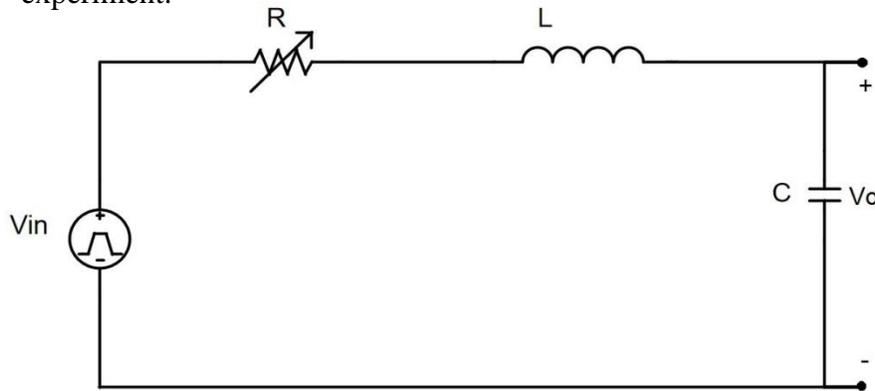
Name of the Experiment: Study of the underdamped response of a second order circuit.

Objectives:

- To observe the characteristics of the underdamped response of a second order circuit .
- To design a practical circuit using percentage overshoot and settling time (T_s).

Theory:

We have already come across a second order circuit and its transfer function in the previous experiment.



$$\frac{V_o(s)}{V_{in}(s)} = \frac{\frac{1}{LC}}{s^2 + \frac{R}{L}s + \frac{1}{LC}}$$
$$\therefore \omega_n = \frac{1}{\sqrt{LC}} \quad \& \quad \zeta = \frac{R}{2} \sqrt{\frac{C}{L}}$$

Figure 2.1: A second order (RLC) circuit.

In an underdamped system, the range of damping factor: $0 < \zeta < 1$

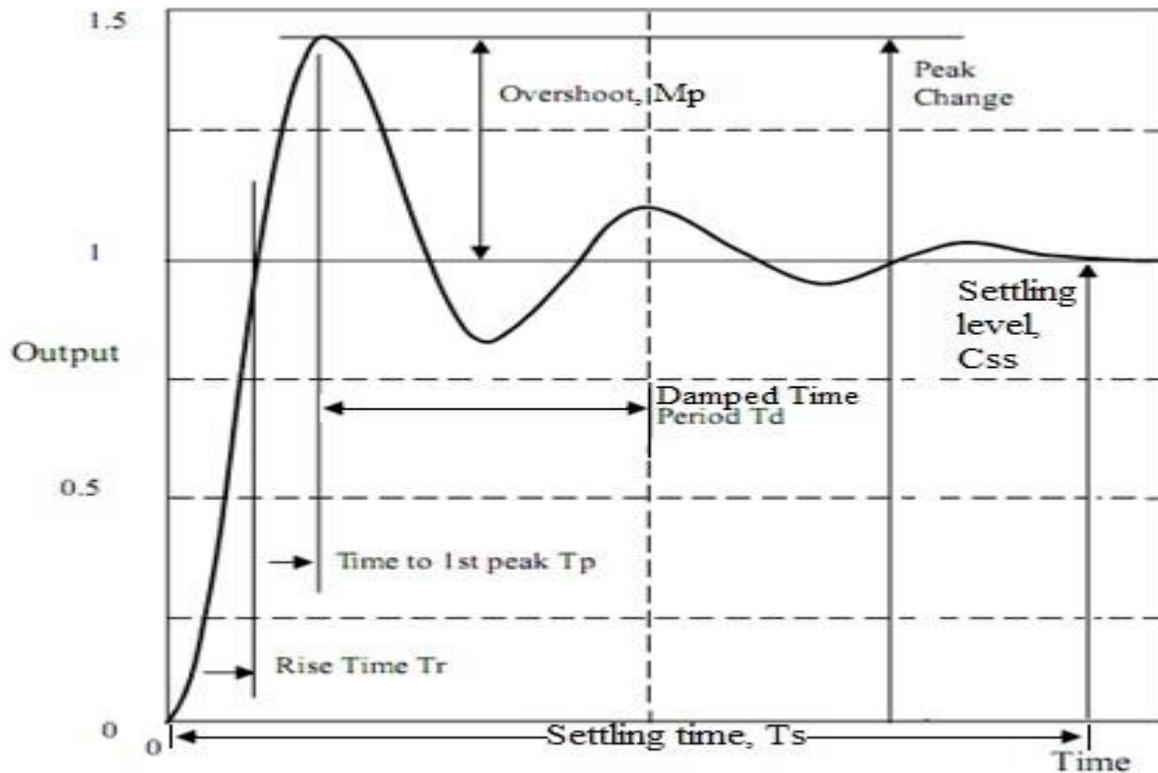


Figure 2.2: Parameters to describe a second order behavior.

From the underdamped response, we can define different parameters, such as

- i) **Rise time, T_r** : Time for the response to rise from 10%-90% (for overdamped system) or 0%-100% (for underdamped system) of the final value.

$$T_r \cong \frac{2.16\zeta + 0.6}{\omega_n}$$

- ii) **Peak, C_p** : It is the value of the first peak of the response.

$$C_p = 1 + e^{-\pi\zeta / \sqrt{1-\zeta^2}}$$

- iii) **Peak time, T_p** : Time required for the response to reach the first peak of the overshoot.

$$T_p = \frac{\pi}{\omega_n \sqrt{1-\zeta^2}}$$

- iv) **Settling level, C_{SS}** : The value at which the response settles down.

- v) **Overshoot, M_p** : Overshoot is the amount by which a response goes beyond the steady state value before settling down. Overshoot can be measured from the step response. It is the ratio of the difference between peak and settling value to the settling value.

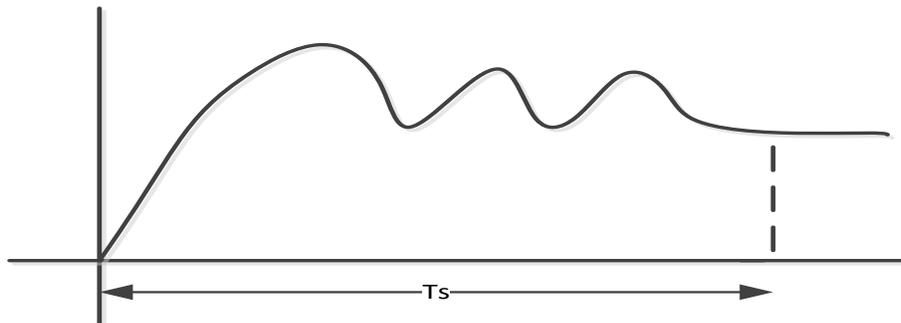
$$\therefore M_p = \frac{C_p - C_{SS}}{C_{SS}}$$

$$\text{and Percentage Overshoot, } \%M_p = \frac{C_p - C_{SS}}{C_{SS}} \times 100\%$$

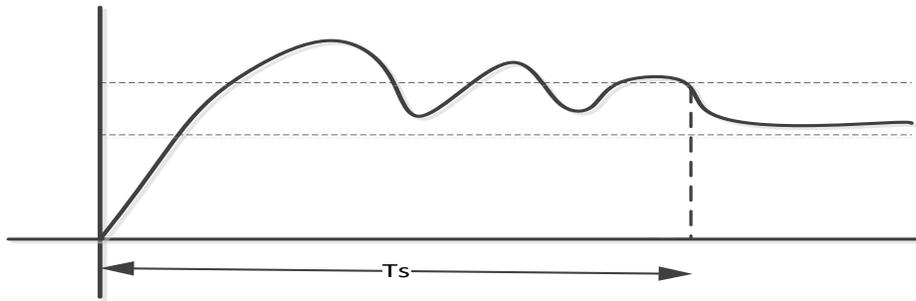
- vi) **Damped Time Period, T_d** : Difference in time between two successive positive peaks in damped region.

- vii) **Settling time, T_s** : It is the time required for the response to settle down to the settling level. It can be measured in two ways:

1. **Response Settling Time**: It is the time when the response settles down to a fixed value.



2. **Envelope Settling Time**: An envelope is defined and the time at which the response enters the envelope and never leaves again, is called the Envelope Settling time.



*We often work for 2% and 5% envelope settling time, which are actually tolerance level. That is, if the steady state value is '1', then for 2%, an envelope is defined where lower and upper limits will be at 0.98 and 1.02 respectively. So on for 5%.

$$T_s = \frac{\text{number of time constants}}{\zeta \omega_n} \approx \frac{4}{\zeta \omega_n} \text{ (2\% error)} \approx \frac{3}{\zeta \omega_n} \text{ (5\% error)}$$

- viii) **Damping Factor:** In the equations describing system behavior, overshoot is not an easy parameter to handle. Another parameter called Damping Factor is used and this gives an indication of the amount of overshoot in a system. Damping Factor has the symbol ζ (Zeta). ζ has a value of 1 when the system is critically damped, less than 1 when underdamped and greater than 1 when overdamped. The objective of a control system design is often to achieve a fast response without any overshoot or with just a little overshoot. Systems are usually designed for ζ in the range 0.7 to 1. The amount of overshoot is wholly dependent on the Damping Factor. Measuring overshoot allows the damping factor to be calculated and knowing ζ allows the overshoot to be calculated.

They are linked by the equations:

$$\zeta = \frac{1}{\sqrt{1 + \left[\frac{\pi}{\ln(\text{overshoot ratio})} \right]^2}}$$

$$\text{So, Overshoot ratio} = e^{-\pi\zeta / \sqrt{1-\zeta^2}} \Rightarrow \% \text{Overshoot ratio} = 100 \times e^{-\pi\zeta / \sqrt{1-\zeta^2}}$$

- ix) **Damped Frequency:** The frequency at which an underdamped system oscillates is called the Damped Frequency, ω_d . The inverse of the period of a cycle is its frequency in Hertz:

$$f_d = \frac{1}{T_d} \Rightarrow \omega_d = 2\pi f_d = \frac{2\pi}{T_d}, \text{ where } T_d \text{ is the period of the oscillation.}$$

The time to the first peak, T_p , is half the period. The damped frequency can then be found by measuring the time to the first peak:

$$\omega_d = \frac{\pi}{T_p}$$

- x) **Natural Frequency:** If there was no damping at all ($\zeta = 0$), the system would continuously oscillate at a frequency which is called the Natural Frequency of the system. This is given the symbol ω_n . The relationship between Natural and Damped frequencies is:

$$\omega_n = \frac{\omega_d}{\sqrt{1 - \zeta^2}} \Rightarrow \omega_d = \omega_n \sqrt{1 - \zeta^2}$$

Circuit diagram:

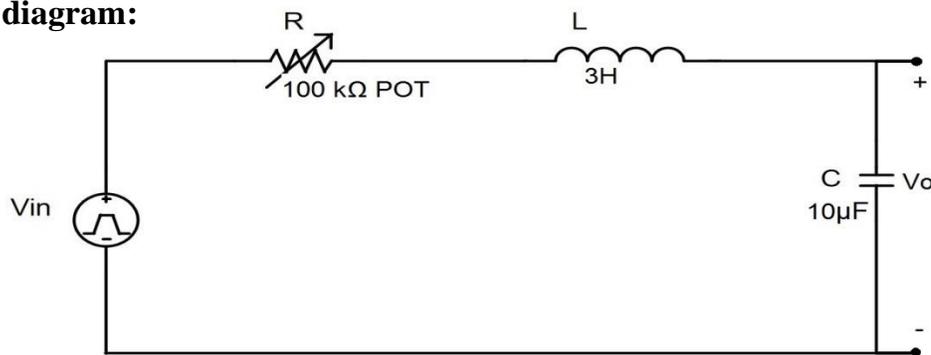


Figure 2.3: A second order (series RLC) circuit for setup.

Procedures:

To design a system having 40% overshoot .

- 1) Set up the circuit shown in figure 2.3. Provide the input, ground and scope connection from **Digilent Analog Discovery-2** device.
- 2) Keeping the value of L and C fixed, calculate R. Vary the POT to that value.
- 3) Open the **Waveforms** software on your PC and select **Wavegen** from the Welcome window.
- 4) Provide a square wave input from the **Wavegen** window.
- 5) Observe the scope and see whether the specified design criteria have been fulfilled or not.

Data:

R=	L=
C=	% Overshoot(from figure)=

Assignment:

1. Design a second order circuit using Opamp, R, L and C. You must show the circuit diagram and the transfer function.
2. You are asked to design a R-L-C circuit that will provide a rise time of 30ms. The inductor and capacitor are fixed at 5 H and 20uF. What will be the value of R.
3. Design a second order circuit that will provide a overshoot of 60% and settling time of 10s (2% method).

Experiment No: 3

Name of the Experiment: Study of the frequency response of first and second order systems by Bode Plot method and stability analysis.

Objectives:

- c. To observe the frequency response of first and second order systems using practical circuits.
- d. To observe the frequency response of first and second order systems using Simulink.
- e. To analyse the stability of the systems using the concept of gain margin and phase margin.

Theory:

The idea of frequency response originates when we excite a system with unit magnitude sine wave and the sine wave input produces sine wave output (provided system is linear). Frequency response describes how the **magnitude** and **phase** of the output sine wave varies as a function of the frequency of the input sine wave.

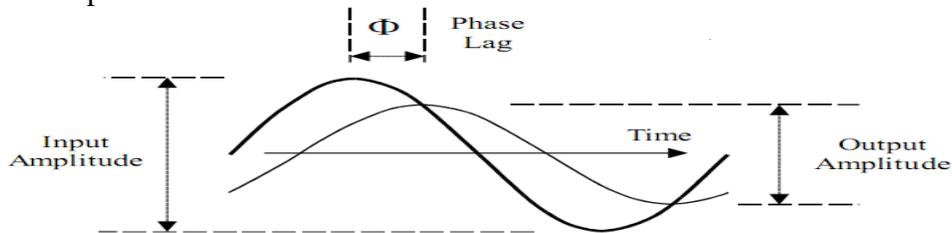


Fig. 3.1: Relationships between input and output sinusoids

The frequency response of a system is defined as the steady-state response of the system to a sinusoidal input signal. The sinusoid is a unique input signal, and the resulting output signal for a linear system, as well as signals throughout the system, is sinusoidal in the steady state; it differs from the input waveform only in amplitude and phase angle.

In the study of control system, we mainly deal with two types of frequency response plots, namely-

- i) Polar plot or Nyquist plot
- ii) Logarithmic plot or Bode plot.

In this experiment, we will study only Bode plot. The introduction of **logarithmic plots**, often called **Bode plots**, simplifies the determination of the graphical portrayal of the frequency response. The logarithmic plots are called Bode plots in honor of Engr. Hendry Wade Bode, who used them extensively in his studies of feedback amplifiers. The logarithmic gain in dB and the angle $\Phi(\omega)$ when plotted versus the frequency ω is known as Bode plot.

Bode plot of a first order system:

Let us consider an RC filter circuit that we have used in the first experiment as a first order system.

The transfer function of the system is, $T(s) = \frac{V_o(s)}{V_{in}(s)} = \frac{1}{sRC+1}$.

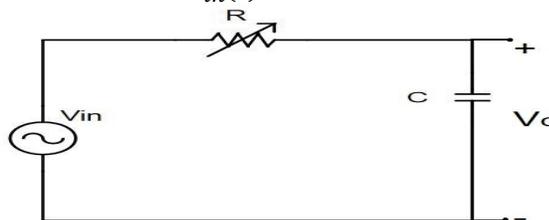


Fig. 3.2: An RC filter circuit (a first order system)

The sinusoidal steady-state transfer function is $T(j\omega) = \frac{1}{j\omega RC + 1}$.

Since $T(j\omega)$ is a complex quantity, it has a magnitude and a phase.

$$\therefore |T(j\omega)| = \frac{1}{\sqrt{(\omega RC)^2 + 1}} \text{ and } \angle T(j\omega) = \tan^{-1} 0 - \tan^{-1} \frac{\omega RC}{1} = \tan^{-1} \omega RC$$

The logarithmic gain is

$$20 \log |T(j\omega)| = 20 \log 1 - 20 \log (\sqrt{(\omega RC)^2 + 1}) = 0 - 20 \log (\sqrt{(\omega RC)^2 + 1}) \\ = -20 \log (\sqrt{(\omega RC)^2 + 1})$$

For small frequencies—that is, $\omega \ll \frac{1}{RC}$, $20 \log |T(j\omega)| \approx 0 \text{ dB}$

For large frequencies—that is, $\omega \gg \frac{1}{RC}$, $20 \log |T(j\omega)| \approx -20 \log \omega RC \text{ dB}$

We call the straight-line approximations *asymptotes*. The low-frequency approximation is called the *low-frequency asymptote*, and the high-frequency approximation is called the *high-frequency asymptote*. The frequency $\omega = \frac{1}{RC}$, for this system, is called **the corner frequency or break frequency** of the system, since it is the break between the low- and the high-frequency asymptotes. At this frequency, the intersection of the two asymptotes occur as

$$-20 \log 1 = 0 \text{ dB} = -20 \log(\omega RC)$$

The actual logarithmic gain at this frequency, is actually $20 \log |T(j\omega)| = -20 \log \sqrt{2} \text{ dB} = -3 \text{ dB}$.

The frequencies equal to or below $\frac{1}{10}$ th of the corner frequency can be termed as small frequencies and the frequencies equal to or above 10 times the corner frequency can be termed as large frequencies. From these, the asymptotic logarithmic gain and phase angle can be determined as follows:

ω	$20 \log T(j\omega) $	$\angle T(j\omega)$
$\omega = \frac{1}{10RC}$	0 dB	0°
$\omega = \frac{1}{RC}$	0dB	-45°
$\omega = \frac{10}{RC}$	-20 dB	-90°

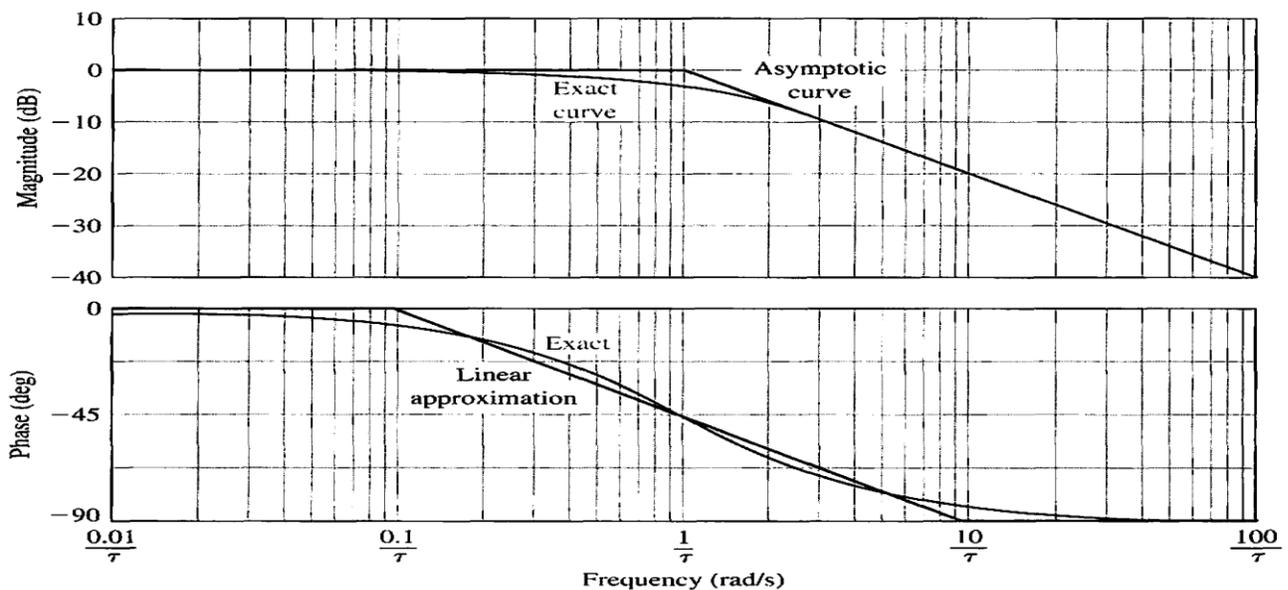


Fig. 3.3: Bode Plot of an RC filter circuit (a first order system)

From fig. 3.3, it is observed that, for a first order system, the logarithmic gain plot will be 0 dB upto corner frequency and will have a slope of -20db/decade from the corner frequency. The phase plot will vary from 0° to -90° with the cutoff/corner frequency residing at -45°.

Bode plot of a second order system:

Let us consider a transfer function $T(s) = \frac{1}{s^2+2s+1} = \frac{1}{(s+1)(s+1)}$

Evidently, the transfer function represents a second order system and the roots of the denominator are real and equal. Such type of system is a critically damped system. The sinusoidal steady-state transfer function is $T(j\omega) = \frac{1}{(j\omega+1)(j\omega+1)}$

Like the previous system, $T(j\omega)$ is a complex quantity, and it has a magnitude and a phase.

$\therefore |T(j\omega)| = \frac{1}{(\sqrt{\omega^2+1})(\sqrt{\omega^2+1})}$ and $\angle T(j\omega) = \tan^{-1} \frac{0}{1} - \tan^{-1} \frac{\omega}{1} - \tan^{-1} \frac{\omega}{1} = -2 \tan^{-1} \omega$

The logarithmic gain is

$20 \log |T(j\omega)| = 20 \log 1 - 20 \log (\sqrt{\omega^2 + 1}) - 20 \log (\sqrt{\omega^2 + 1}) = 0 - 40 \log (\sqrt{\omega^2 + 1})$
 $= -40 \log (\sqrt{\omega^2 + 1})$

For small frequencies—that is, $\omega \ll 1$, $20 \log |T(j\omega)| \approx 0$ dB

For large frequencies—that is, $\omega \gg 1$, $20 \log |T(j\omega)| \approx -40 \log \omega$ dB. Here, $\omega=1$ is the corner frequency. The actual logarithmic gain at this frequency, is actually $20 \log |T(j\omega)| = -40 \log \sqrt{2}$ dB = -6 dB. The asymptotic logarithmic gain and phase angle can be determined as follows:

ω	$20 \log T(j\omega) $	$\angle T(j\omega)$
$\omega = \frac{1}{10}$	0 dB	0°
$\omega = 1$	0dB	-90°
$\omega = 10$	-40 dB	-180°

We can see that the bode plots for this system will be quite similar to that of a first order circuit, except the fact that the gain plot will have a slope of -40 dB/decade and the phase will vary from 0° to -180° with the cutoff/corner frequency residing at -90°. The peak at the actual logarithmic gain plot and the slope of the actual phase plot will vary with the variation of damping factor, ζ . We can see this from figure 3.6. We will have a peak overshoot for underdamped system. This is due to an inherent phenomenon of a series RLC circuit, known as resonance. For the circuit shown in figure 3.4, we will obtain the plots shown in figure 3.5.

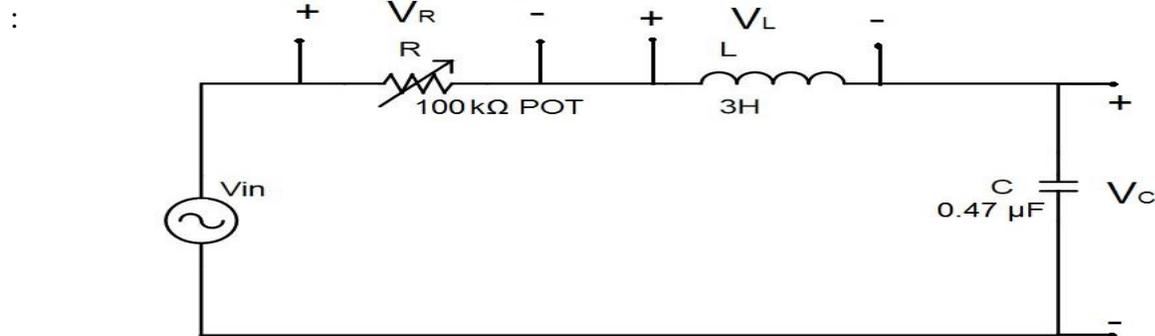


Fig. 3.4: An RLC circuit (a second order system)

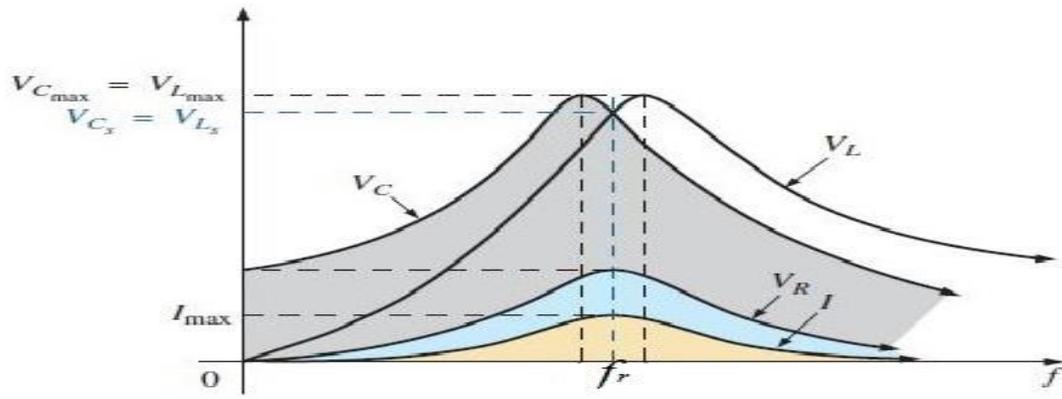


Fig. 3.5: V_R , V_L , V_C , and I versus frequency for a series resonant circuit.

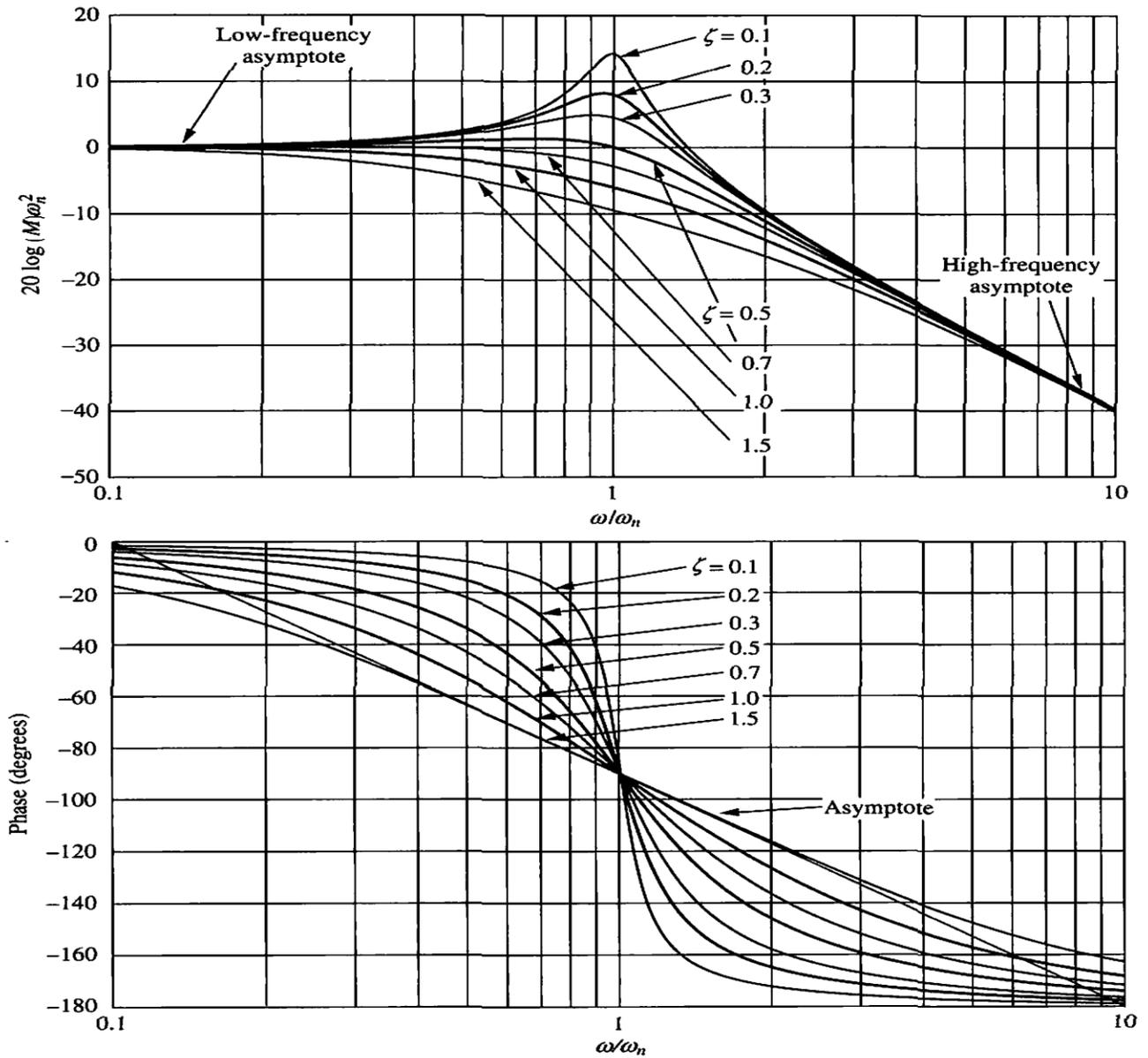


Fig. 3.6: Bode plot of second order system with varying damping factor

In figure 3.5, the peak V_C and V_L is obtained such as the value of R makes the system response underdamped. The peak V_C will be on capacitive region and V_L will be on inductive region. By varying the POT (R), we can vary damping factor and observe the change in the peak. The resonance frequency, $\omega_r = \frac{1}{\sqrt{LC}}$ will remain unchanged. Thus we conclude that, for increase in order of the system, the slope of the logarithmic gain will decline further by -20 dB/decade and phase will vary extra 90° with cut-off being at the middle.

Stability Analysis from Bode plot:

In Bode plot, the 0 dB is the reference line for analysis in dB plot and -180° is the reference line in phase plot.

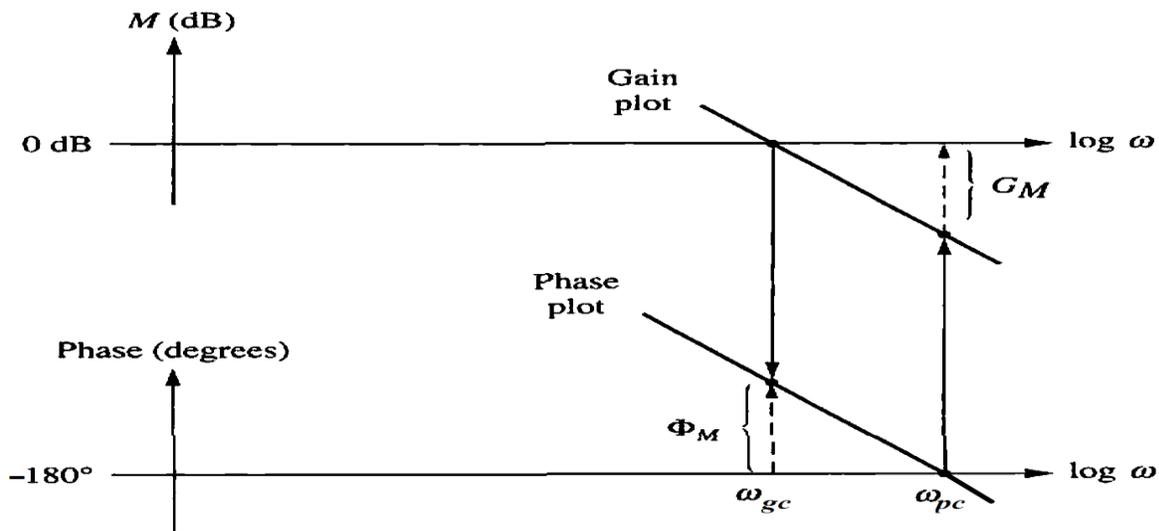


Fig. 3.7: Gain and phase margin of second order system

Frequency response specifications:

1. **Gain cross over frequency, ω_{gc}** = It is the frequency at which the logarithmic magnitude plot intersects 0 dB point.

2. **Phase cross over frequency ω_{pc}** = It is the frequency at which the phase angle plot intersects -180° point.

3. **Gain margin, GM** = It is defined as the margin in gain allowable by which gain can be increased till system reaches on the verge of instability. Mathematically it is defined as the reciprocal of the magnitude of the $T(j\omega)$ measured at phase cross over frequency.

GM is positive if above 0 dB line, GM is negative if below 0 dB line.

4. **Phase margin, Φ_M** = Amount of additional phase lag which can be introduced in the system till system reaches on the verge of instability. Mathematically it can be defined as the difference between the phase angle ($\angle T(j\omega)$) at gain cross over frequency and -180°

Φ_M is positive if above -180° line, Φ_M is negative if below -180° line.

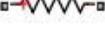
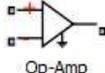
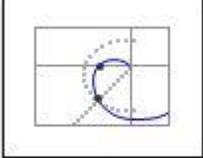
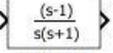
If any of the margins is negative, then the system is unstable.

When, $\omega_{gc} < \omega_{pc}$, then the system is stable.

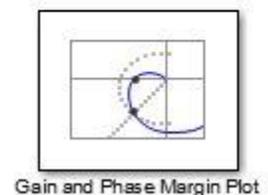
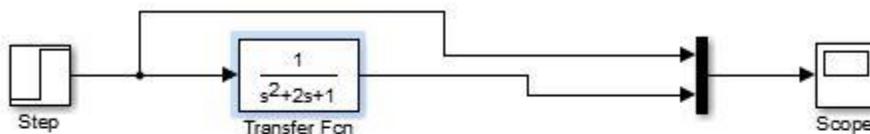
$\omega_{gc} = \omega_{pc}$, then the system is oscillatory/ marginally stable.

$\omega_{gc} > \omega_{pc}$, then the system is unstable.

Simulink library:

Sl.	Block Name:	Symbol:	Library:
1.	Resistance, Inductor, Capacitor, Electrical Reference, Opamp	 Capacitor  Inductor  Resistor  Electrical Reference  Op-Amp	Simscape/Foundation Library/Electrical/Electrical Elements
2.	Controlled Voltage Source	 Controlled Voltage Source	Simscape/Foundation Library/Electrical/Electrical Sources
3.	Voltage Sensor	 Voltage Sensor	Simscape/Foundation Library/Electrical/Electrical Sensors
4.	PS-Simulink Converter, Simulink-PS Converter, Solver Configuration	 PS-Simulink Converter  Simulink-PS Converter  Solver Configuration	Simscape/Utilities
5.	Step, Ramp	 Step  Ramp	Simulink/Sources
6.	Scope	 Scope	Simulink/Sinks
7.	Mux	 Mux	Simulink/Signal Routing
8.	Gain and Phase Margin Plot	 Gain and Phase Margin Plot	Simulink Control Design/ Linear Analysis Plots
9.	Integrator, Zero-pole, Transfer Fcn	 Integrator  Zero-Pole  Transfer Fcn	Simulink/Continuous
10.	Sum	 Sum	Simulink/Math Operation

Simulink Block Diagram Model:



Procedures for Simulink:

- i) Enter the Simulink library by pressing the Simulink tab from “Home” window in MATLAB or by writing “Simulink” in command window.
- ii) To start building a new model, press “New Model”  on Simulink Library Browser.
- iii) Build the block diagram model as shown.
- iv) To observe time response run simulation and double press “scope” to observe.
- v) To observe bode plot-
 - a. Double click on “gain & phase margin block”.
 - b. To set input and output lines, press “+” sign on the window that appears to extend it.
 - c. From the model, select the output line by pressing mouse on the line.
 - d. Press  on the gain & phase margin window to take the line under linearization.
 - e. Repeat the same procedure for input line.
 - f. Under “Configuration”, select “Input perturbation” for input line and “Open loop output” for output line.
 - g. Select plot type as “Bode” and then press “show plot”. A blank window appears. Press the run button to generate the plot.
 - h. To observe stability, right click the mouse on graph and under “characteristics”, select “All stability margins”.

Practical Circuit Diagram:

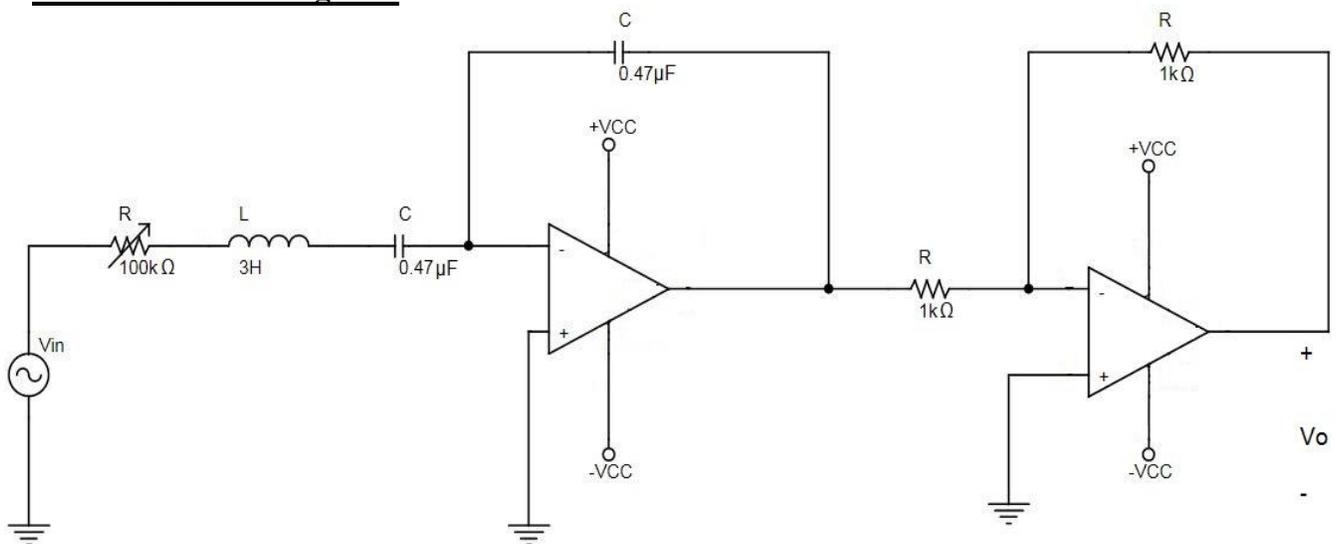


Fig. 3.6: A second order system with OPAMP.

Procedures for Practical Circuit Diagram:

- i) Setup the circuit in a breadboard, provide input, ground and scope connection from Analog Discovery-2 device.
- ii) Go to “Wavegen” window and provide a sine wave input of 1V and 1Hz.

- iii) Open the Waveforms software and go to “Network” window from Welcome menu.
- iv) Set the start frequency at a value less than $\frac{f_c}{10}$ and the end value at greater than $10f_c$.
- v) Run the program.
- vi) Observe the bode plot, check the slope and phase angle at corner frequency and note down.

Assignments:

1. Provide the stability margins for a series RC and series RLC circuit used in the previous experiments. (Use practical values)
2. For the following systems, plot the Bode plot in Simulink and obtain the stability margins”
 - a) $T(s) = \frac{21}{s^2+10s+21}$
 - b) $T(s) = \frac{10}{s^3+3s^2+3s+1}$
 - c) $T(s) = \frac{1}{s^3+3s^2+3s+1}$

Experiment No: 4

Name of the Experiment: Study of the frequency response of first and second order circuits by Nyquist/Polar Plot method and stability analysis.

Objectives:

- f. To observe the frequency response by polar plot of first and second order circuit using practical circuits.
- g. To observe the frequency response of first and second order circuit using Simulink.
- h. To analyse the stability of the system from Nyquist plot using the concept of gain margin and phase margin.

Theory:

The Nyquist criterion relates the stability of a closed-loop system to the open-loop frequency response and open-loop pole location. Thus, knowledge of the open loop system's frequency response yields information about the stability of the closed-loop system. The Nyquist Stability criterion was discovered by the Swedish-American electrical engineer Harry Nyquist at Bell Telephone Laboratories in 1932.

Procedures for finding the Frequency Response by Nyquist/ Polar plot:

1. At first find the transfer function, $T(s)$ of the system
2. Substitute $s=j\omega$ in the transfer function, $T(s)$. So, $T(s)|_{s=j\omega} = T(j\omega)$
3. Write out expressions for magnitude and phase of transfer function

$$T(j\omega) = R(\omega) + jX(\omega) = |T(j\omega)| \angle \Phi(j\omega)$$

$$\text{where, } |T(j\omega)| = \sqrt{|R(\omega)|^2 + |X(\omega)|^2}, \text{ and } \Phi(j\omega) = \tan^{-1} \frac{X(\omega)}{R(\omega)}$$

4. Determine the magnitude and phase as ω goes to zero.
5. Determine the magnitude and phase as ω goes to infinity.
6. Determine magnitude and frequency when phase takes key values, e.g. at corner frequency, when phase is -180° , etc.

Nyquist plot of a first order system:

Let us consider an RC filter circuit that we have used in the first experiment as a first order system.

The transfer function of the system is, $T(s) = \frac{V_o(s)}{V_{in}(s)} = \frac{1}{sRC + 1}$.

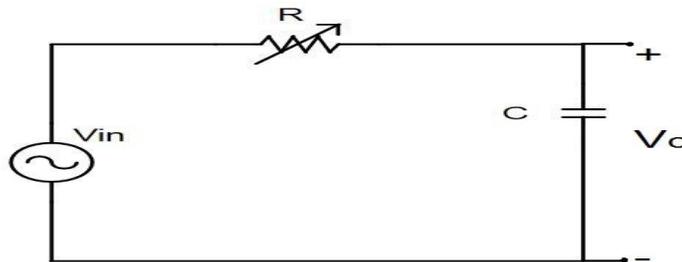


Fig. 4.1: An RC filter circuit (a first order system)

The sinusoidal steady-state transfer function is $T(j\omega) = \frac{1}{j\omega RC + 1}$.

Since $T(j\omega)$ is a complex quantity, it has a magnitude and a phase.

$$\therefore |T(j\omega)| = \frac{1}{\sqrt{(\omega RC)^2 + 1}} \text{ and } \angle \Phi(j\omega) = \tan^{-1} \frac{0}{1} - \tan^{-1} \frac{\omega RC}{1} = -\tan^{-1} \omega RC$$

The corner frequency or break frequency of the system is $\omega = \frac{1}{RC}$.

From these, the magnitude and phase angle for polar plot can be determined as follows:

ω	$ T(j\omega) $	$\angle \Phi(j\omega)$
$\omega=0$	1	0°
$\omega = \infty$	0	-90°
$\omega = \frac{1}{RC}$	0.707	-45°

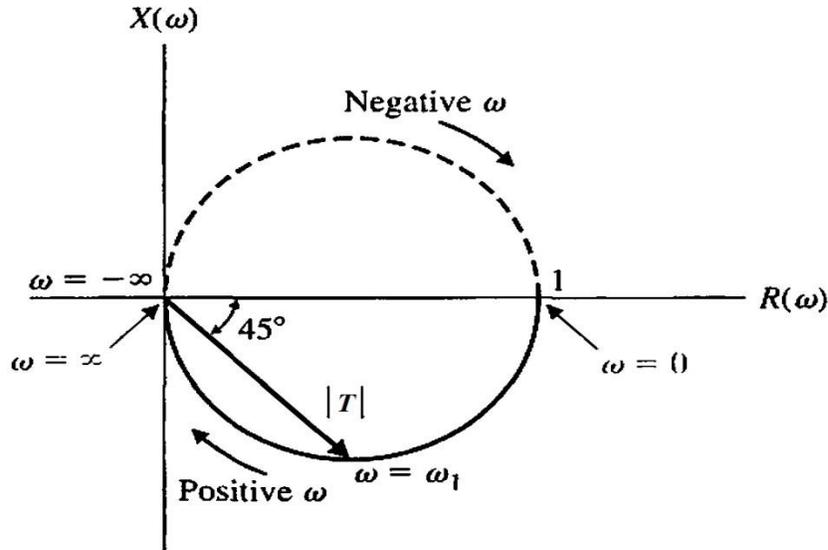


Fig. 4.2: Nyquist Plot of an RC filter circuit (a first order system)

Nyquist plot of a second order system:

Let us consider a transfer function $T(s) = \frac{21}{s^2+10s+21} = \frac{21}{(s+3)(s+7)}$

Evidently, the transfer function represents a second order system and the roots of the denominator are real and but not equal. Such type of system is an over damped system. The sinusoidal steady-state transfer function is $T(j\omega) = \frac{1}{(j\omega+3)(j\omega+7)}$

Like the previous system, $T(j\omega)$ is a complex quantity, and it has a magnitude and a phase.

$$\therefore |T(j\omega)| = \frac{1}{(\sqrt{\omega^2+3^2})(\sqrt{\omega^2+7^2})} \text{ and } \angle \Phi(j\omega) = \tan^{-1} \frac{0}{21} - \tan^{-1} \frac{\omega}{3} - \tan^{-1} \frac{\omega}{7} = -\tan^{-1} \frac{\omega}{3} - \tan^{-1} \frac{\omega}{7}$$

The corner frequencies of the system are $\omega = 3, 7$. From these, the magnitude and phase angle for polar plot can be determined as follows:

ω	$ T(j\omega) $	$\angle \Phi(j\omega)$
$\omega=0$	1	0°
$\omega = \infty$	0	-180°
$\omega = 3$	0.65	-68.2°
$\omega = 7$	0.27	-111.9°

*For negative frequencies, we will have a mirror plot.
 *Phase angle varies $0^\circ \sim -180^\circ$ for 2nd order system, $0^\circ \sim -90^\circ$ for 1st order system, $0^\circ \sim -270^\circ$ for 3rd order system and so on.

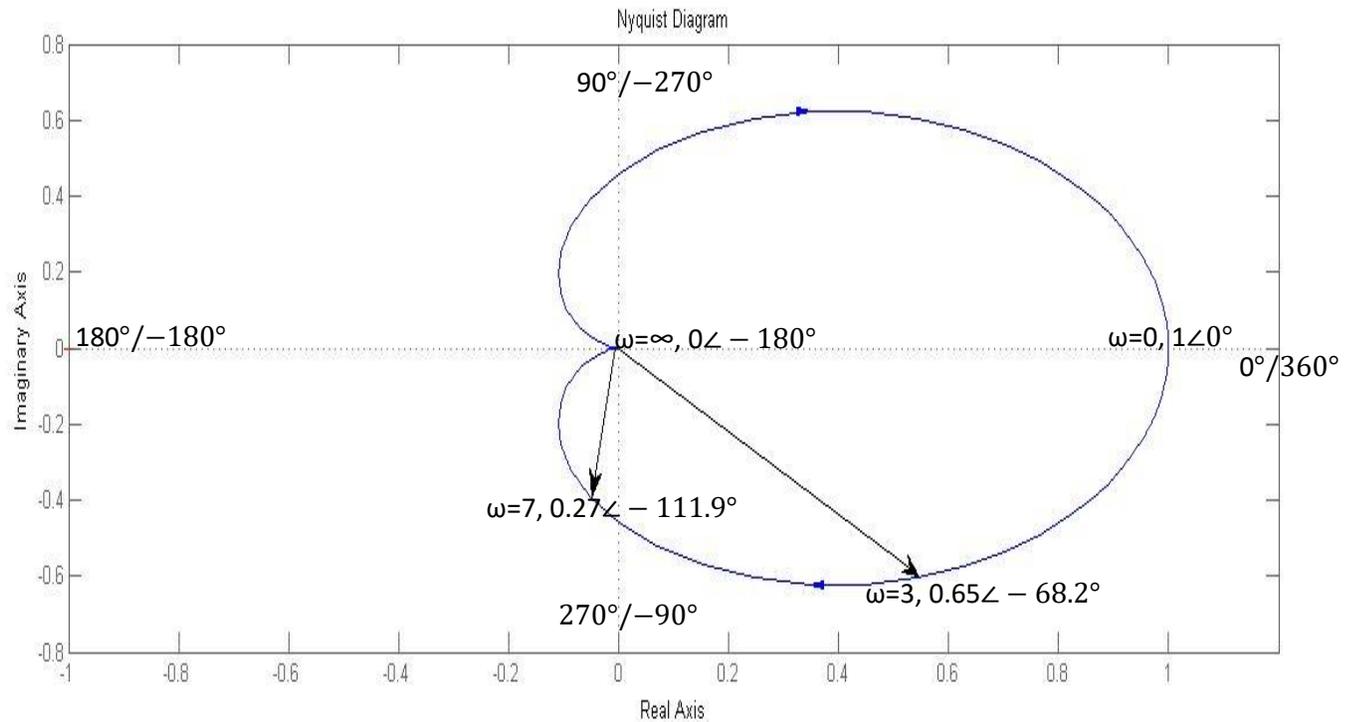


Fig. 4.3: Nyquist Plot of a second order system.

Stability Analysis from Nyquist plot:

In Nyquist plot, the reference line for stability analysis is 1 (=0 dB) and -180° .

Consider a 3rd order system having a transfer function, $T(s) = \frac{10}{s^3+3s^2+3s+1}$.

Its Nyquist plot will be as follows

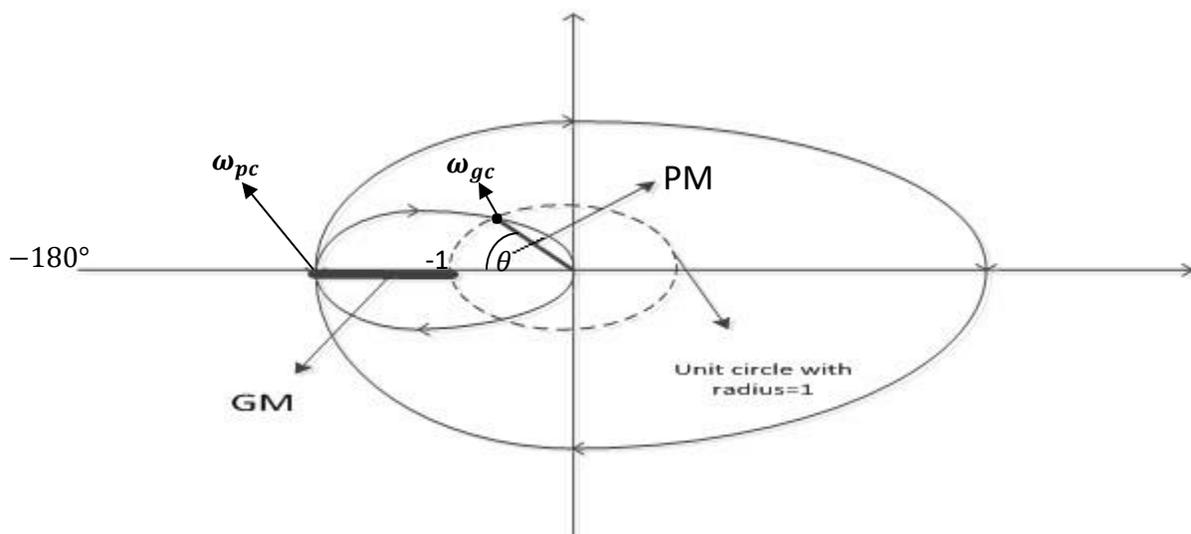


Fig. 4.4: Gain and phase margin of a third order system

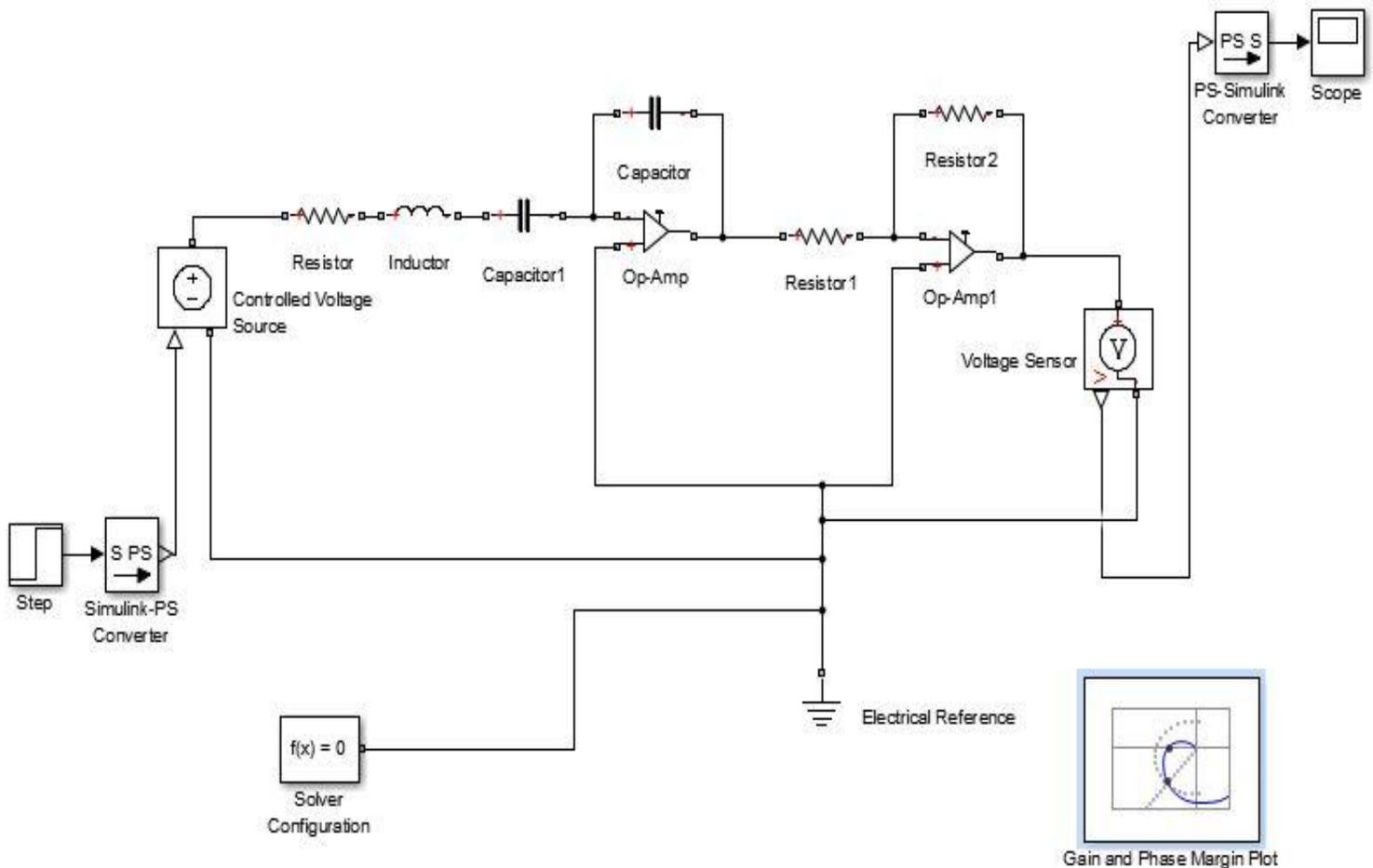
Since at 0 dB=1 crossing point we obtain gain cross over frequency, so we draw a circle with radius 1.

* **To obtain Gain margin, G.M:** The -180° axis crossing point frequency is the phase cross over frequency point, ω_{pc} . The difference between the ω_{pc} point and the unity circle gives G.M. Gain Margin is positive if the ω_{pc} point $<$ unity circle and negative if ω_{pc} point $>$ unity circle.

* **To obtain Phase margin, P.M.:** We draw a straight line to ω_{gc} point from the center of the unit circle and the angle from -180° axis is the phase margin, P.M. Phase margin is positive if below -180° axis and negative if above -180° axis.

If any of the margins is negative, then the system is unstable.

Simulink Physical System Design:



Procedures for Simulink:

- vi) Enter the Simulink library by pressing the Simulink tab from “Home” window in MATLAB or by writing “Simulink” in command window.
- vii) To start building a new model, press “New Model”  on Simulink Library Browser.
- viii) Build the physical as shown.

- ix) To observe time response run simulation and double press “scope” to observe.
- x) To observe bode plot-
 - i. Double click on “gain & phase margin block”.
 - j. To set input and output lines, press ‘+’ sign on the window that appears to extend it.
 - k. From the model, select the output line by pressing mouse on the line.
 - l. Press  on the gain & phase margin window to take the line under linearization.
 - m. Repeat the same procedure for input line.
 - n. Under “Configuration”, select “Input perturbation” for input line and “Open loop output” for output line.
 - o. Select plot type as “Nyquist” and then press “show plot”. A blank window appears. Press the run button to generate the plot.
 - p. To observe stability, right click the mouse on graph and under “characteristics”, select “All stability margins”.

Practical Circuit Diagram:

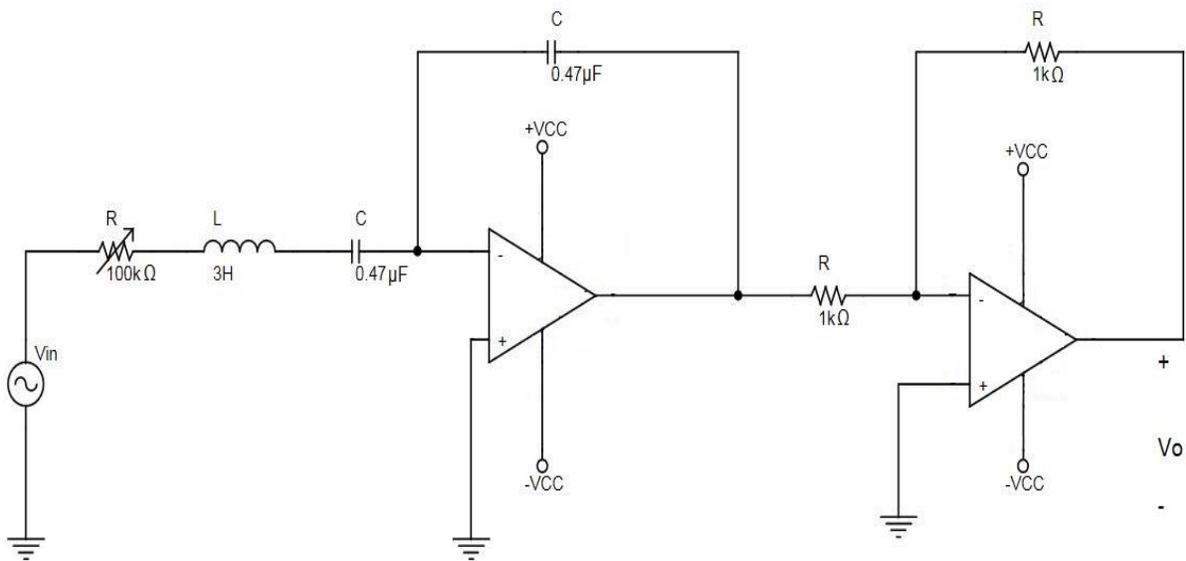


Fig. 4.5: A second order system with OPAMP.

Procedures for Practical Circuit Diagram:

- vii) Setup the circuit in a breadboard, provide input, ground and scope connection from Analog Discovery-2 device.
- viii) Go to “Wavegen” window and provide a sine wave input of 1V and 1Hz.
- ix) Open the Waveforms software and go to “Network” window from Welcome menu.
- x) Set the start frequency at a value less than $\frac{f_C}{10}$ and the end value at greater than $10f_C$.
- xi) Run the program.

- xii) Observe the bode plot, check the slope and phase angle at corner frequency and note down.
- xiii) Click the “View” tab on the Network window, and then select “Nyquist”. The Nyquist plot of the system will appear in a window beside the Bode plot. Remember, since there is no such thing as negative frequency in real life, you will see only the positive frequency part of the Nyquist plot

Assignments:

3. For the following systems, obtain the Nyquist plot in Simulink by physical system design and obtain the stability margins:

d) $T(s) = \frac{21}{s^2+10s+21}$

e) $T(s) = \frac{10}{s^3+3s^2+3s+1}$

f) $T(s) = \frac{1}{s^3+3s^2+3s+1}$

g) $T(s) = \frac{1}{s^2+2s+1}$

h) $T(s) = \frac{1}{s^2+11s+30}$

4. Also draw the Nyquist plot fro theoretical calculations for the above mentioned systems.

Experiment No: 5

Name of the Experiment: Study of steady state error analysis of different Types of system.

Objectives:

- i. To determine steady state error for different **Types** of systems using different input signals.
- j. To analyse when the steady state error is reduced or nullified.
- k. To verify the steady state error analysis using Simulink and practical circuits.

Theory:

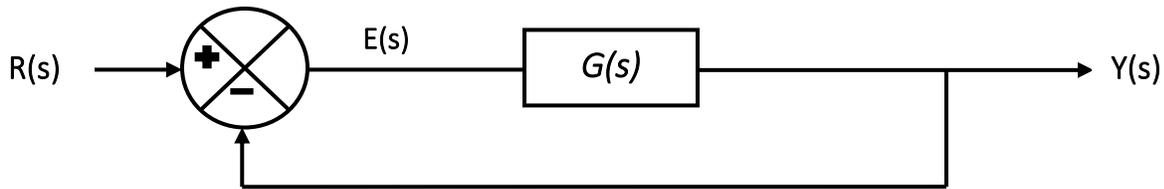
In many control system designs, we are specifically interested in the final, or steady state value of the output. This is known as steady state accuracy. Ideally, in the steady state, the output, $y(t)$, equals the command signal, $r(t)$, and the error is zero. This ideal situation is rarely met, and so we need to be able to determine the steady state error for any system. *Steady-state error* is the difference between the input and the output for a prescribed test input as $t \rightarrow \infty$. The steady state error is defined as:

$$e_{SS} = \lim_{t \rightarrow \infty} e(t) = \lim_{t \rightarrow \infty} [r(t) - y(t)]$$

Only for unity feedback systems, the error is the comparator output signal and we can determine the steady state error by examining the open loop transfer function $G(s)$. Since error in the final value can be termed as steady state error, by using “Final Value Theorem” we get,

$$E_{SS} = \lim_{s \rightarrow 0} sE(s) \dots \dots \dots (1)$$

Let us consider the following closed loop system with unity feedback.



From the block diagram it is seen that,

$$E(s) = R(s) - Y(s) = R(s) - G(s) E(s)$$
$$\therefore E(s) = \frac{R(s)}{1 + G(s)} \dots \dots \dots (2)$$

From equations (1) and (2) it can be written that

$$E_{SS} = \lim_{s \rightarrow 0} \frac{s}{1 + G(s)} R(s) \dots \dots \dots (3)$$

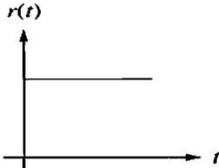
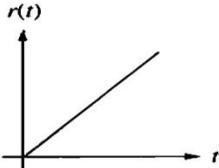
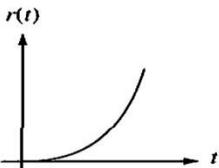
Another term that will be frequently used in the analysis of steady state error is Type of a system. Type of a system simply means how many poles the system has in the origin. A Type-0 system has no pole in the origin, a Type-1 system has one pole in the origin, a Type-2 system has two and so on. The type of a system can be increased by simply adding an integrator ($\frac{1}{s}$). We will see how the Type of a system affects its steady state error.

Because the actual input signals to a system is usually unknown, standard test input signals are employed to determine time domain performance specifications. We are interested in the steady state error for step, ramp, parabolic, and higher order polynomial inputs, i.e

$$r(t) = \frac{t^n}{n!} \rightarrow R(s) = \frac{1}{s^{n+1}}, n= 0, 1, 2, \dots$$

Test inputs that will be used for steady-state error analysis and design are summarized in Table 5.1.

TABLE 5.1 Test waveforms for evaluating steady-state errors of position control systems

Waveform	Name	Physical interpretation	Time function	Laplace transform
	Step	Constant position	1	$\frac{1}{s}$
	Ramp	Constant velocity	t	$\frac{1}{s^2}$
	Parabola	Constant acceleration	$\frac{1}{2}t^2$	$\frac{1}{s^3}$

For Step Input: $R(s) = \frac{1}{s}$

Putting this in equation (3) we get

$$E_{SS} = \lim_{s \rightarrow 0} \frac{s}{1 + G(s)} \cdot \frac{1}{s}$$

$$\Rightarrow E_{SS} = \lim_{s \rightarrow 0} \frac{1}{1 + G(s)} \dots \dots \dots (4)$$

In equation (4), only the term $G(s)$ in the denominator will have effect of putting $s \rightarrow 0$. $G(s)$ is the open loop transfer function of the system, which can be expressed in terms of zeroes and poles. Let us denote the poles by 'p' and zeroes by 'z', then $G(s)$ for a system having 'n' no. of zeroes and poles can be expressed as

$$G(s) = \frac{(s + z_1)(s + z_2) \dots \dots \dots (s + z_n)}{(s + p_1)(s + p_2) \dots \dots \dots (s + p_n)} \dots \dots \dots (5)$$

$$\text{When } s \rightarrow 0, G(s) = \frac{z_1 z_2 \dots z_n}{p_1 p_2 \dots p_n} = K_p$$

where, K_p is called position constant. So equation (4) can be written as

$$E_{SS} = \frac{1}{1 + K_p}, \text{ which is actually a constant.}$$

We can see from equation (5) that the system has no pole at the origin. So, from the above discussion, it can be inferred that for a Type-0 system, the steady state error has a constant value for Step Input. What should be done to make this value 0?

If we consider a Type-1 system, i.e. a system with one pole at the origin, then $G(s)$ can be expressed as

$$G(s) = \frac{(s + z_1)(s + z_2) \dots \dots \dots (s + z_n)}{s(s + p_1)(s + p_2) \dots \dots \dots (s + p_n)} \dots \dots \dots (6)$$

When $s \rightarrow 0$, $G(s) = \infty$. So equation (4) can be written as

$$E_{ss} = \frac{1}{1 + \infty} = 0$$

So, it can be further inferred that for a Type-1 system, the steady state error becomes 0 for Step Input. Similarly it can be shown that for a Type-2 system, the steady state error becomes 0 for Step Input. Similarly, steady state error for other two standard input signals can be determined can be determined for different *types* of system.

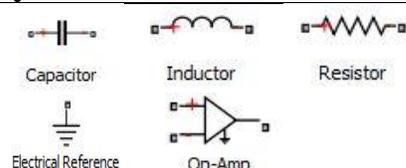
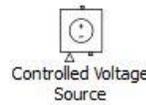
Following table shows the steady state errors for Type 0, 1, and 2 systems for the standard input signals.

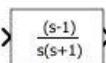
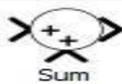
Steady State Errors for Polynomial Inputs			
System Type → Polynomial Degree ↓	0	1	2
Step Input	$\frac{1}{1 + K_p}$	0	0
Ramp Input	∞	$\frac{1}{K_v}$	0
Parabolic Input	∞	∞	$\frac{1}{K_a}$

Because we deal with many electromechanical systems, control engineers also define position, velocity and acceleration error constants as follows.

$$K_p = G(0), K_v = \lim_{s \rightarrow 0} s G(s), K_a = \lim_{s \rightarrow 0} s^2 G(s)$$

Simulink library:

Sl	Block Name:	Symbol:	Library:
1.	Resistance, Inductor, Capacitor, Electrical Reference, Opamp		Simscape/Foundation Library/Electrical/Electrical Elements
2.	Controlled Voltage Source		Simscape/Foundation Library/Electrical/Electrical Sources

3.	Voltage Sensor	 Voltage Sensor	Simscape/Foundation Library/Electrical/Electrical Sensors
4.	PS-Simulink Converter, Simulink-PS Converter,Solver Configuration	 PS-Simulink Converter  Simulink-PS Converter  f(x) = 0 Solver Configuration	Simscape/Utilities
5.	Step, Ramp	 Step  Ramp	Simulink/Sources
6.	Scope, Display	 Scope  Display	Simulink/Sinks
7.	Integrator, Zero-pole, Transfer Fcn	 $\frac{1}{s}$ Integrator  $\frac{(s-1)}{s(s+1)}$ Zero-Pole  $\frac{1}{s+1}$ Transfer Fcn	Simulink/Continuous
8.	Sum	 Sum	Simulink/Math Operation

Setup Circuit:

Type 0 System:

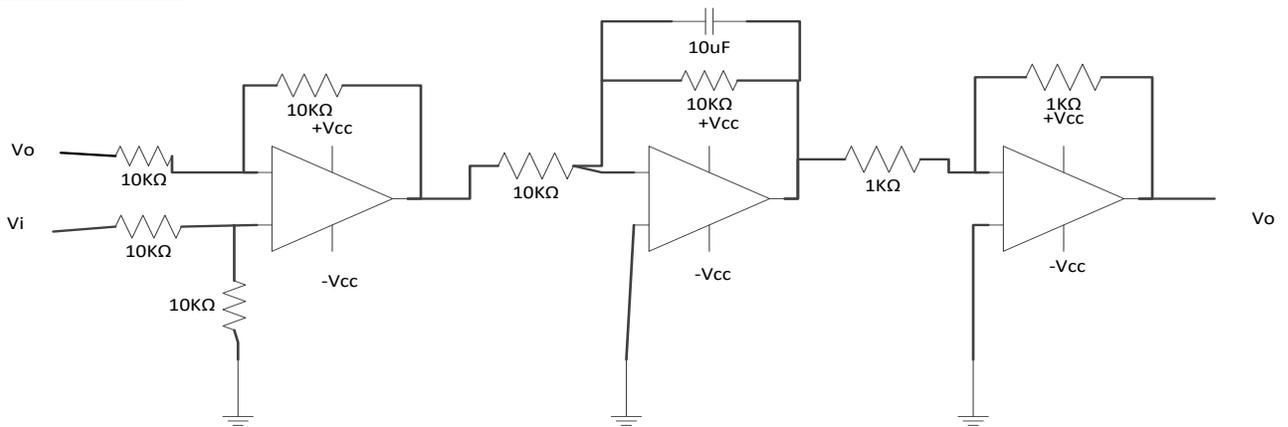


Figure 1: Type 0 system

Type 1 System

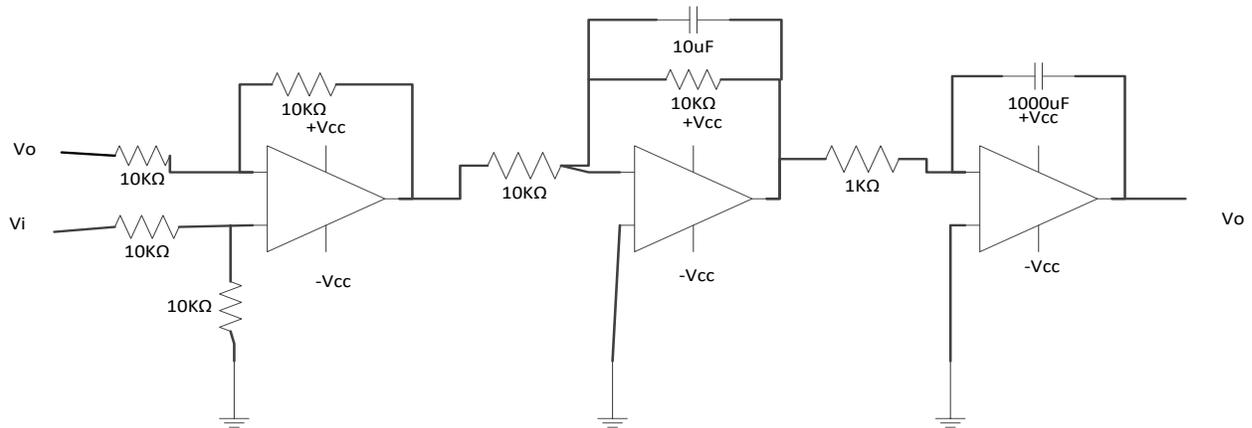


Figure 2: Type 1 System

Procedure:

- 1) Setup the circuit of figure-1 in a breadboard.
- 2) Connect analog discovery to the circuit in order to provide input, scope, Ground and +/- Vcc.
- 3) Provide step input from analog discovery with 1V & 1 Hz. Measure the difference between input and output from scope window. This value is the Ess.
- 4) Provide ramp and sine wave sequentially and observe the output.
- 5) Follow similar procedure for the second circuit and this time measure the difference when ramp input is given.
- 6) Build the same circuits in Simulink environment and compare the results.

Results:

Practical Circuit:

	Step	Ramp	Parabolic
Type 0	Ess=	Ess=	Ess=
Type 1	Ess=	Ess=	Ess=

Simulink:

	Step	Ramp	Parabolic
Type 0	Ess=	Ess=	Ess=
Type 1	Ess=	Ess=	Ess=
Type 2	Ess=	Ess=	Ess=

Assignment:

- 1) Design all the circuits (type 0/1/2) in Simulink environment and show output for each of the 3 input types.

2) For the following Open loop TF, design the model in Simulink and show outputs for each of the 3 inputs (step/ramp/parabolic):

i) $G(s)H(s) = \frac{1}{(s+4)(s+6)}$

ii) $G(s)H(s) = \frac{1}{s(s+10)}$

iii) $G(s)H(s) = \frac{10}{s^2(s+4)(s+6)}$

Experiment No. : 6

Name of the Experiment: Study of the Root Locus of a System simulated on PC using the “MATLAB” software.

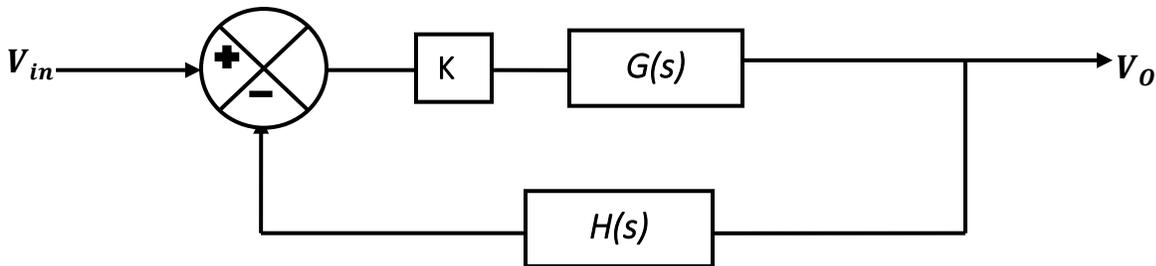
Objectives:

- l. To plot the root locus of a given system using SISOTOOL GUI of MATLAB.
- m. To see the changes in the step response by varying the controller gain.
- n. To modify root locus to meet some particular system criteria.

Theory:

Root Locus can be referred to as the location of roots of a closed loop system while varying a system parameter, like gain. It is the path of the roots of the characteristic equation traced out in the s-plane as a system parameter varies from zero to infinity. The root locus method was invented by **Walter Richard Evans**, a noted American control theorist, in 1948 and has been developed and utilized extensively in control engineering practice. It is a powerful method of analysis and design for stability and transient response. The root locus can be used to describe qualitatively the performance of a system as various parameters are changed. For example, the effect of varying gain upon percent overshoot, settling time, and peak time can be vividly displayed. The qualitative description can then be verified with quantitative analysis. Besides transient response, the root locus also gives a graphical representation of a system's stability. We can clearly see ranges of stability, ranges of instability, and the conditions that cause a system to break into oscillation.

Let us consider the following closed loop system.



The closed loop transfer function can be written as

$$CLTF = \frac{V_o}{V_{in}} = \frac{K G(s)}{1 + K G(s)H(s)} \dots \dots \dots (1)$$

And the open loop transfer function can be written as

$$OLTF = \frac{V_o}{V_{in}} = K G(s)H(s) \dots \dots \dots (2)$$

The closed loop transfer function can also be written as

$$CLTF = \frac{V_o}{V_{in}} = \frac{K \frac{N1}{D1}}{1 + K \frac{N1 N2}{D1 D2}} \dots \dots \dots (3)$$

Where, N1, N2, D1, D2 are the Numerators and Denominators of G(s) and H(s) respectively.

From equation (3), we can write,

$$CLTF = \frac{V_o}{V_{in}} = \frac{K N_1 D_2}{D_1 D_2 + K N_1 N_2} \dots \dots \dots (4)$$

The characteristic equation of the system is $1 + G(s) H(s) = 0$. It is called so because the solution of this equation provides an idea about the characteristics of the system. From equation (1) and (4) we can write the characteristic equation as

$$D_1 D_2 + K N_1 N_2 = 0 \dots \dots \dots (5)$$

Root locus technique is a graphical method for sketching the locus of the roots of the characteristic equation in the s-plane as the parameter K is varied. Actually the roots of the characteristic equation are the poles of the CLTF.

When $K=0$, equation (5) becomes

$$D_1 D_2 = 0 \dots \dots \dots (6)$$

The solution of equation (6) gives us the open-loop poles of the system [refer to equation (2)]. So, we can say that when the controller gain, $K = 0$, the open loop poles and the closed loop poles of a system are at the same position.

Again, when $K = \infty$, equation (5) can be written as

$$\begin{aligned} K \left(\frac{D_1 D_2}{K} + N_1 N_2 \right) &= 0 \\ \Rightarrow \frac{D_1 D_2}{\infty} + N_1 N_2 &= 0 \\ \Rightarrow N_1 N_2 &= 0 \dots \dots \dots (7) \end{aligned}$$

The solution of equation (7) gives us the open-loop zeroes of the system [refer to equation (2)]. So, we can say that when the controller gain, $K = \infty$, the open loop zeroes and the closed loop poles of a system are at the same position.

From the aforesaid discussion, we see that as the controller gain, K is varied from 0 to ∞ , the closed loop poles of the system moves from the position of open loop poles to that of open loop zeroes. This path of movement of the closed loop poles in the s-plane is actually the root locus of the system. So, it can be inferred that, **the root locus of a system begins at Open Loop Poles (or Closed Loop Poles when $K=0$) and ends at Open Loop Zeroes (or Closed Loop Poles when $K=\infty$) of that system.**

If the no. of open loop poles and open loop zeroes of a system are equal, each root locus will find a definite ending point. If they are not equal, then one or more locus/loci will end at infinity (finding open loop zero(es) at infinity). If there is no open loop zero, all the paths originating from open loop poles will leave for infinity.

Let us consider a unity feedback system, where $G(s) = \frac{1}{s(s+4)}$, $H(s) = 1$. The open loop poles of the system are at 0 and -4. But there is no open loop zero in the system. So how will the root locus look like? From our above discussion, we can visualize that as the gain, K is varied, the two open loop poles will look for open loop zeroes and move towards each other, meet at a point on the real axis, and then leave for infinity in the search of open loop zeroes. We will verify this through SISOTOOL GUI of MATLAB.

Basic Rules for Sketching the Root Locus

- **Number of branches:** The number of branches of the root locus equals the number of closed-loop poles.
- **Symmetry:** The root locus is symmetrical about the real axis.
- **Real-axis segments:** On the real axis, for $K > 0$ the root locus exists to the left of an odd number of real-axis, finite open-loop poles and/or finite open-loop zeros.
- **Starting and ending points:** The root locus begins at the finite and infinite poles of $G(s)H(s)$ and ends at the finite and infinite zeros of $G(s)H(s)$.
- **Behavior at infinity:** The root locus approaches straight lines as asymptotes as the locus approaches infinity. Further, the equations of the asymptotes are given by the real-axis intercept and angle in radians as follows:

$$\sigma_a = \frac{\sum \text{finite poles} - \sum \text{finite zeroes}}{\# \text{finite poles} - \# \text{finite zeroes}}$$

$$\theta_a = \frac{(2k + 1)\pi}{\# \text{finite poles} - \# \text{finite zeroes}}, \text{ where } k = 0, \pm 1, \pm 2, \pm 3, \dots$$

Additional Rules for Refining the Sketch

- **Real-axis breakaway and break-in points:** The root locus breaks away from the real axis at a point where the gain is maximum and breaks into the real axis at a point where the gain is minimum.
- **Calculation of $j\omega$ -axis crossings:** The root locus crosses the $j\omega$ -axis at the point where $\angle G(s)H(s) = (2k + 1)180^\circ$. Routh-Hurwitz or a search of the $j\omega$ -axis for $(2k + 1)180^\circ$ can be used to find the $j\omega$ -axis crossing.
- **Angles of departure and arrival:** The root locus departs from complex, open-loop poles and arrives at complex, open-loop zeros at angles that can be calculated as follows. Assume a point ϵ close to the complex pole or zero. Add all angles drawn from all open-loop poles and zeros to this point. The sum equals $(2k + 1)180^\circ$. The only unknown angle is that drawn from the ϵ close pole or zero, since the vectors drawn from all other poles and zeros can be considered drawn to the complex pole or zero that is ϵ close to the point. Solving for the unknown angle yields the angle of departure or arrival.
- **Plotting and calibrating the root locus:** All points on the root locus satisfy the relationship $\angle G(s)H(s) = (2k + 1)180^\circ$. The gain, K , at any point on the root locus is given by $K = \frac{1}{|G(s)H(s)|}$

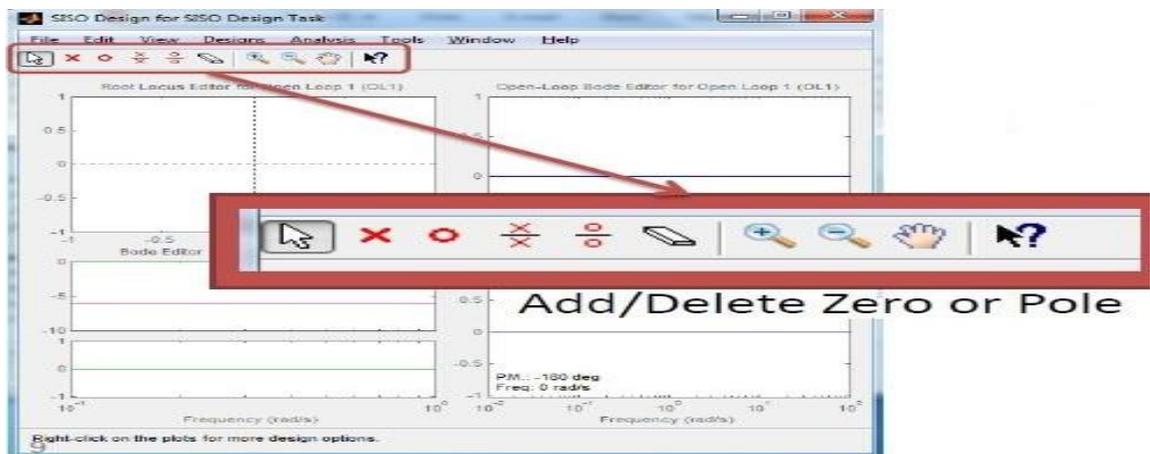
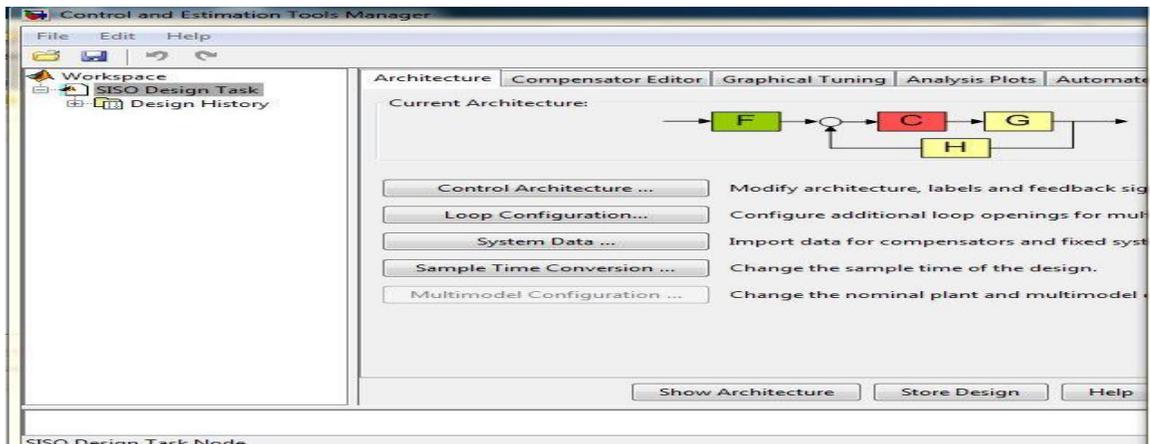
Guidelines for MATLAB's SISO Design Tool GUI

The **SISO Design Tool** is a graphical user interface (GUI) that facilitates the design of compensators for single-input, single-output feedback loops. The SISO Design Tool allows you to iterate rapidly on your designs and perform the following tasks:

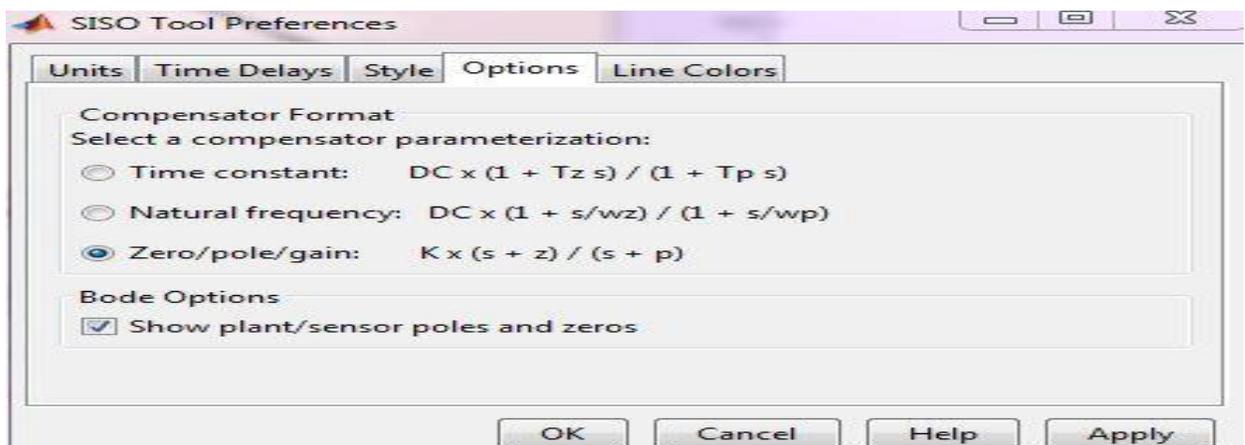
- | | |
|---|--|
| *Manipulate closed-loop dynamics using root locus techniques. | *Inspect closed-loop responses (using the LTI Viewer). |
| *Shape open-loop Bode responses. | *Adjust phase and gain margins. |
| *Add compensator poles and zeros. | *Convert models between discrete and continuous time. |
| *Add and tune lead/lag networks and notch filters. | |

Procedures:

1. Write **sisotool** in the command window of MATLAB, press enter, SISOTOOL Design GUI opens. The following two windows will appear on your screen. The window below in the figure is “**SISO Design for SISO Design Task**”, or **SISO Design tool window**. Real and complex poles and zeroes in the system can be added/deleted using this tool.

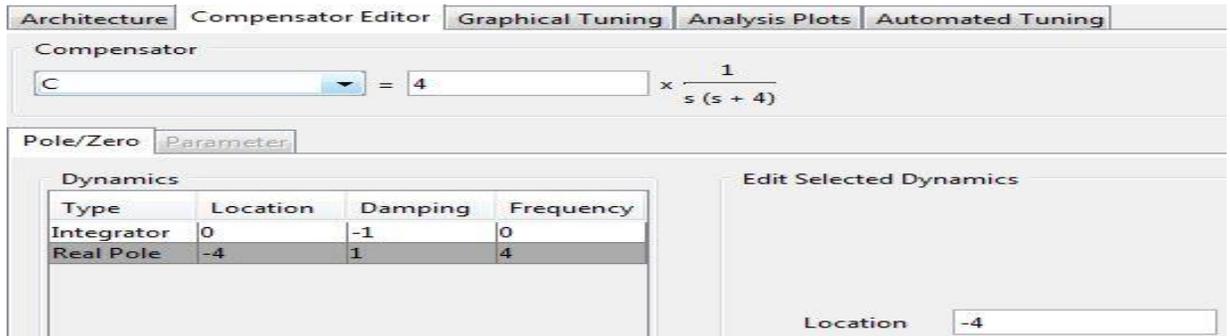


2. On **SISO Design tool window**, click on **Edit>SISO Tool Preferences**. Select the **Options** tab. Select “**Zero/pole/gain**” in the **Compensator Format** and **Apply** it.

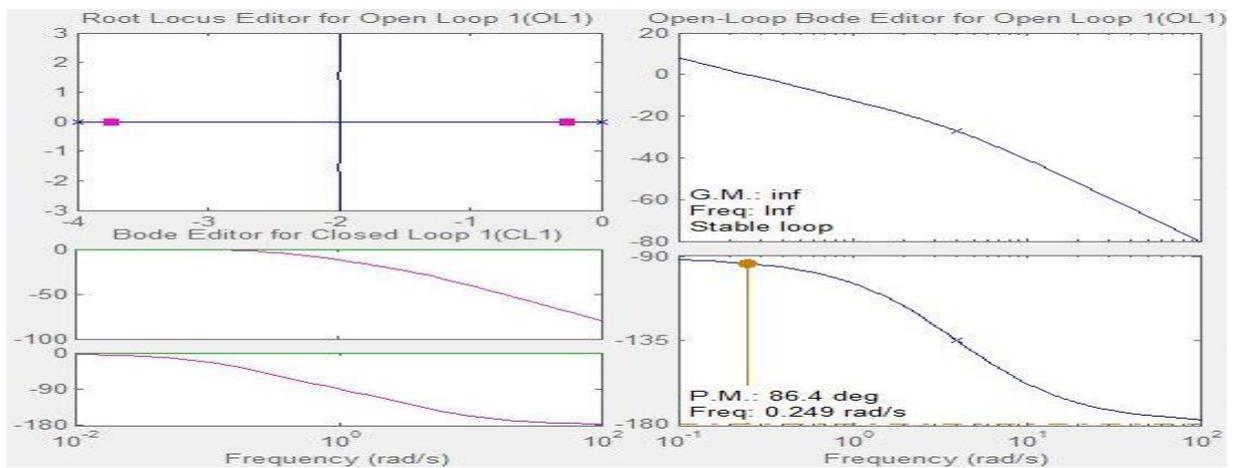


3. On **SISO Design tool window**, click on **Design >Edit Compensator**. **Right click inside the Dynamics box**; the options of adding real and complex poles/zeros etc. will appear. Add two

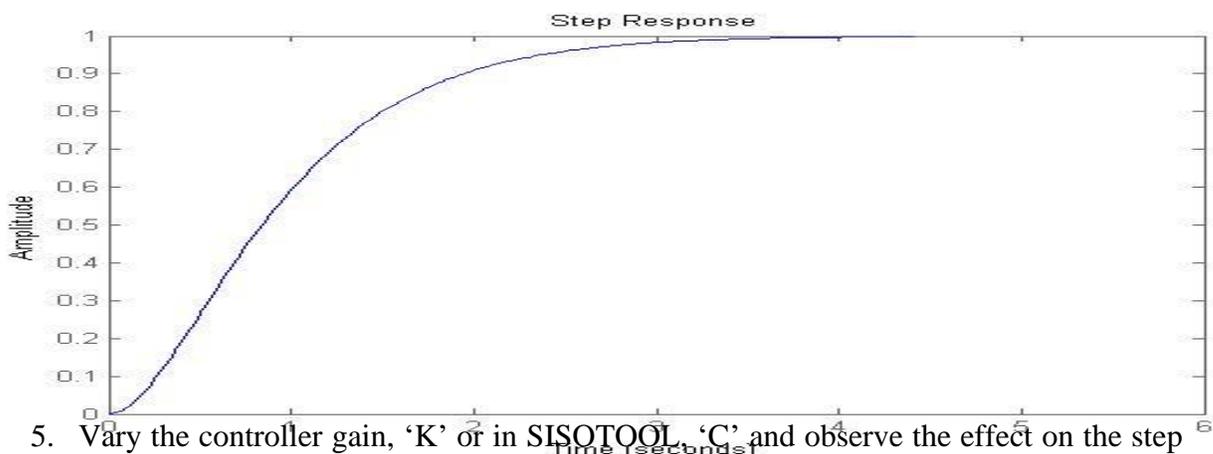
real poles of OLTF, $G(s)H(s) = 1/s(s + 4)$. Select a pole added on the **Dynamics** box, and then write the pole-position on **Location** box on the right-side under **Edit Selected Dynamics**.



The SISO Design tool window now looks as follows:

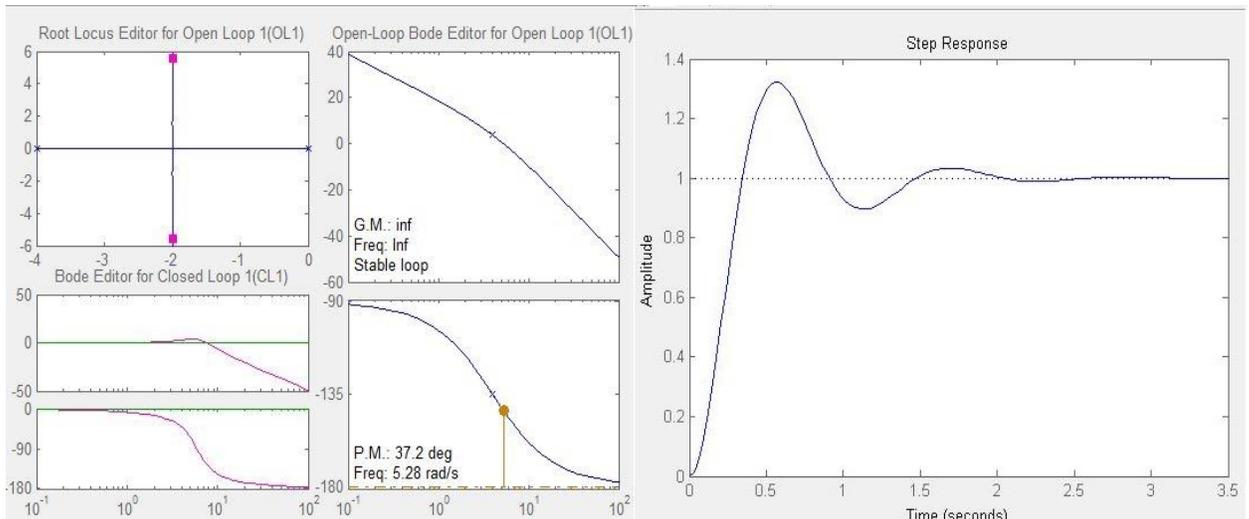


- To see the corresponding step response of the system in a separate window, click on **Analysis**, and then select **“Response to Step Command”**. Evidently it is an over-damped response for the system chosen. Right click on the response, select **“Systems”**, two tick-marks can be seen on **“Closed loop r to y (blue)”** & on **“Closed loop r to u (green)”**. Un-tick **“Closed loop r to u (green)”**, because we want to see the input to output closed loop response, and in the architecture, the input node is named as r and the output node is named as y.

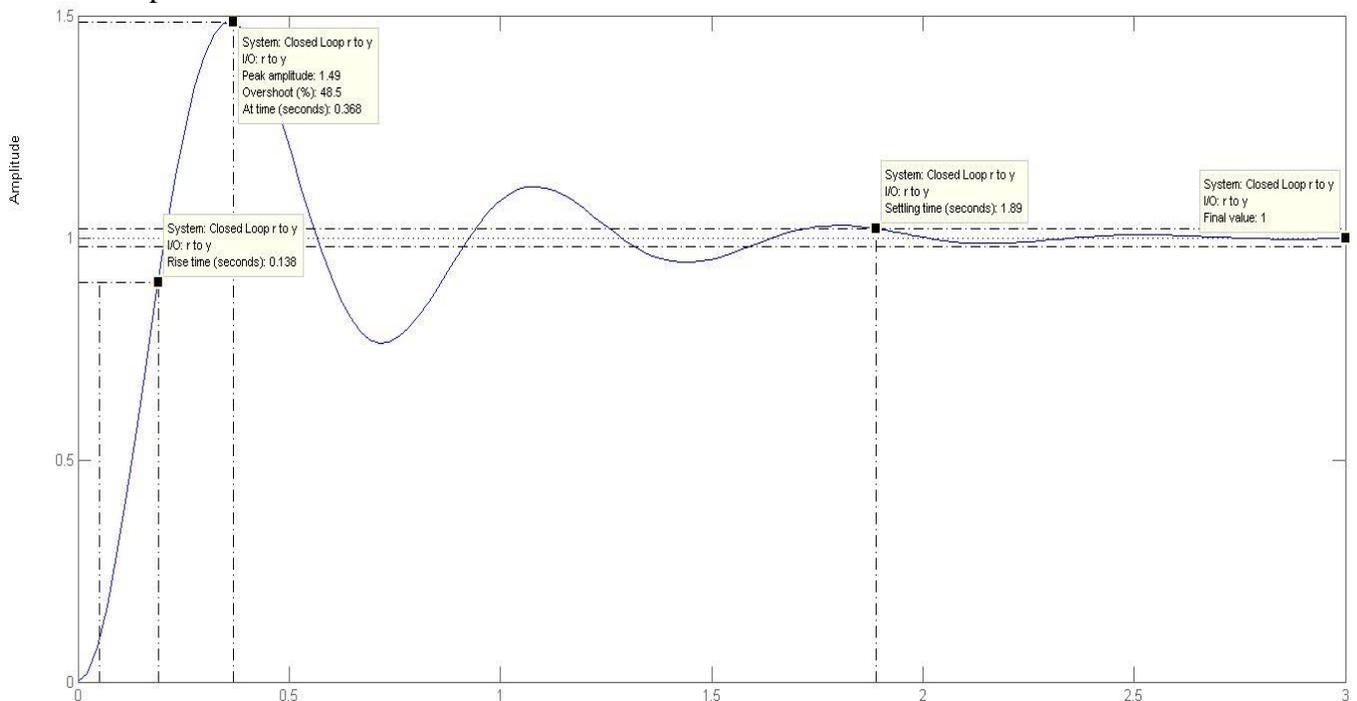


- Vary the controller gain, ‘K’ or in SIMULINK ‘C’ and observe the effect on the step response. **Notice the two pink colored squares on the root locus. Left-click on any them, hold and drag to vary the gain.** Overshoot starts to appear on the step response when the boxes are on the imaginary axis. When on imaginary axis, click & hold on

any of the pink squares, useful informations like current location of the poles, natural frequency and damping can be seen. Now keep the cursor on anywhere of the root locus plot, the value of gain at that point can be seen.



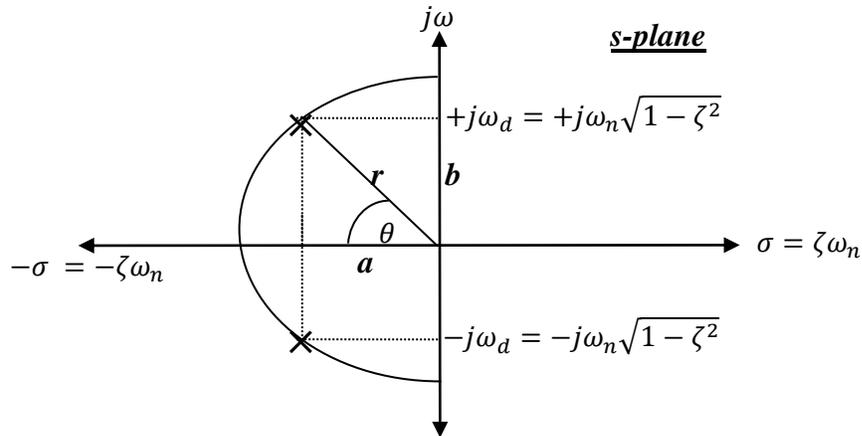
6. System parameters like peak overshoot, settling time (2% margin), rise time can be seen from the step response. **Right-click on the step response, go to characteristics, options like Peak response, settling time, rise time will appear.** Each time you can select one. Some dashed lines and dots will appear, **click on the dots** to see the system parameters.



7. To transfer the plots to another application such as MS Word, choose **Print to Figure** from the **File** menu and the **Save** it by **.jpg/.emf/.png** etc. format.
8. Observe the root locus and corresponding step response, effect of varying controller gain and the system parameters for other systems instructed by the teacher.

Effect on ζ and ω_n with respect to the pole position:

The roots of the denominator ($s^2 + 2\zeta\omega_n s + \omega_n^2$) of a general second order transfer function are, $s_{1,2} = -\zeta\omega_n \pm j\omega_n\sqrt{1-\zeta^2} = -\sigma \pm j\omega_d$. The performance of the system will depend on the values of ζ & ω_n . The poles of an underdamped system can be shown in s-plane as follows:



From the pole plot, $a = -\zeta\omega_n$, $b = \omega_n\sqrt{1-\zeta^2}$, thus 'r' can be by Pythagoras' Theorem.

$$\text{So, } r^2 = a^2 + b^2 = (-\zeta\omega_n)^2 + \omega_n^2(1-\zeta^2)$$

$$\Rightarrow r^2 = \omega_n^2 \therefore r = \omega_n$$

$$\text{again, } \cos\theta = \frac{|a|}{|r|} = \frac{\zeta\omega_n}{\omega_n} \therefore \zeta = \cos\theta$$

Therefore, it can be inferred that if a semicircle is drawn in the s-plane taking **the distance of any of the complex poles from the origin** as radius, all the points on the semicircle the natural frequency, ω_n is equal. Inside the circle ω_n is smaller and outside the circle ω_n is greater. On the other hand, straight lines drawn from the origin of the s-plane at an angle θ can represent damping ratio, ζ . So when poles are moved ζ & ω_n also changes accordingly.

*if $\theta=0$, $\zeta=1$, so the system is then critically damped.

*if we increase θ , then ζ decreases and the system becomes more underdamped.

*if r is reduced (i.e. brought closer to the origin) then natural frequency, ω_n drops.

Meeting a specific Design Requirement:

Suppose, we are asked to design a system having 40% overshoot and settling time of 5s (for 2% error). How do we design it?

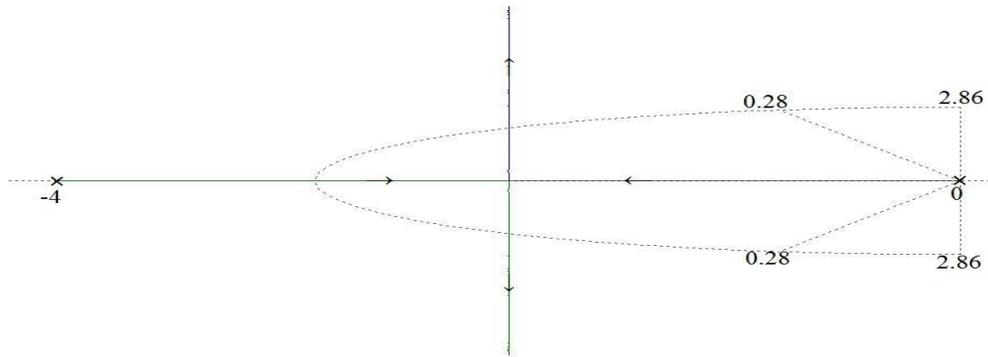
For 40% overshoot, we get ζ (damping ratio) = 0.28, from the equation

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

And settling time 5s (for 2% error) gives natural frequency, $\omega_n = 2.86 \text{ rad/s}$, from the equation

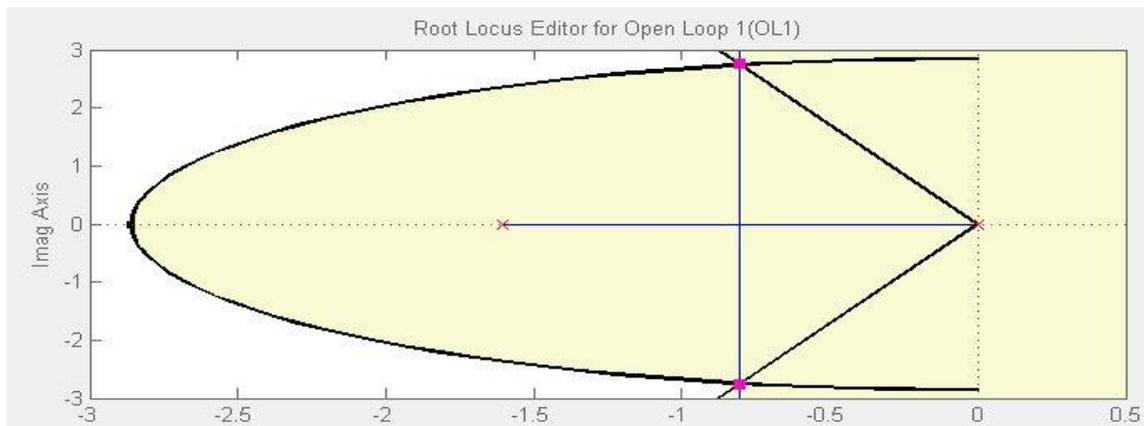
$$T_s = \frac{4}{\zeta\omega_n} \text{ (2\% error)}$$

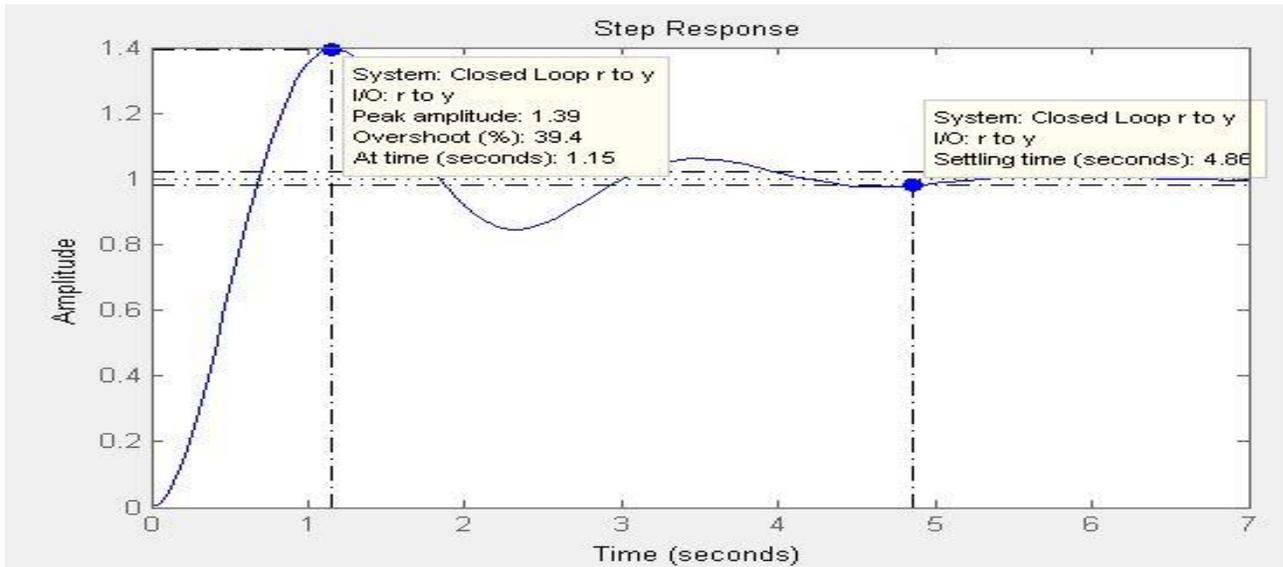
The intersecting points of “**a semicircle with radius 2.86**” and “**two straight lines drawn from origin at an angle $\theta = \cos^{-1}(0.28)$** ” [since $\cos\theta = \zeta$] on the s-plane are the points where the design requirements are fulfilled. In the following root locus plot, the two straight lines at an angle indicate pole locations with $\zeta=0.28$; in between these lines, the poles have $\zeta > 0.28$ and outside of the lines $\zeta < 0.28$. The semi circle indicates pole locations with natural frequency $\omega_n = 2.86$; inside the semicircle, $\omega_n < 2.86$ and outside the semicircle $\omega_n > 2.86$.



It is clearly seen that the root locus of the system does not pass through the desired points. So, to meet the design requirement, the poles have to be moved so that the root locus passes through the desired points. Follow the steps mentioned below.

- i) Go to the Root locus plot. Now right click-on root locus plot , go to design requirements>New. Here various design requirements can be set. Set the values of damping ratio and natural frequency as calculated. Each time we can set one design parameter. Each requirement defines an exclusion region, indicated by a yellow shaded area. To satisfy a requirement, a response plot must remain outside of the associated exclusion region. Now a semicircle and two straight lines at an angle can be seen.
- ii) Hold the pole at -4 and drag it towards right until the imaginary axis part of the root locus passes through the intersecting points of the straight lines and the semicircle. Then drag the pink square boxes until they reach the intersecting points. At this stage the requirements are met and can be seen from the step response, with slight deviations due to the rounded-off calculations. Click once on any of the pink squares and keep the cursor in the white region of the root locus plot to see the gain at the desired points at the bottom of the window. The figures are shown here.





Assignment:

- Show the root locus and the other related plots (including Step response) generated in SISOTOOL GUI in MATLAB for the following OLFs:
 - $G(s)H(s) = \frac{K(s+2)}{(s^2+4s+13)}$
 - $G(s)H(s) = \frac{K(s+2)}{(s^2+4s+13)}$
 - $G(s)H(s) = \frac{K(s^2+4s+8)}{s^2(s+7)}$
 - $G(s)H(s) = \frac{K(s+2)}{s(s+4)(s^2+4s+32)}$
 - $G(s)H(s) = \frac{K(s^2+0.105625)}{s(s^2+1)}$
 - $G(s)H(s) = \frac{s^2+6s+8}{s^2-7s+12}$
- For the OLF, $G(s)H(s) = \frac{Ks(s+3.75)}{(s^2+3.75s+13)}$, show the root locus and other other related plots(including Step response) generated in SISOTOOL GUI in MATLAB. Modify the root locus to meet a design requirement of 2s settling time (2%) and show the plots (including Step response).

Experiment No:7

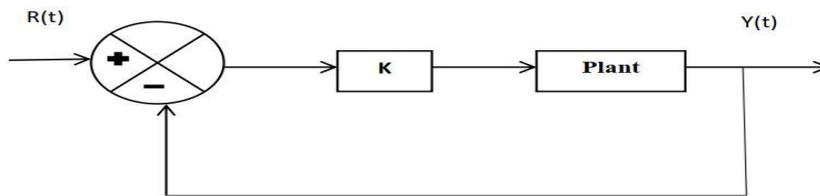
Name of the Experiment: Modelling of circuit using root locus and study of lead & lag Compensator

Objectives:

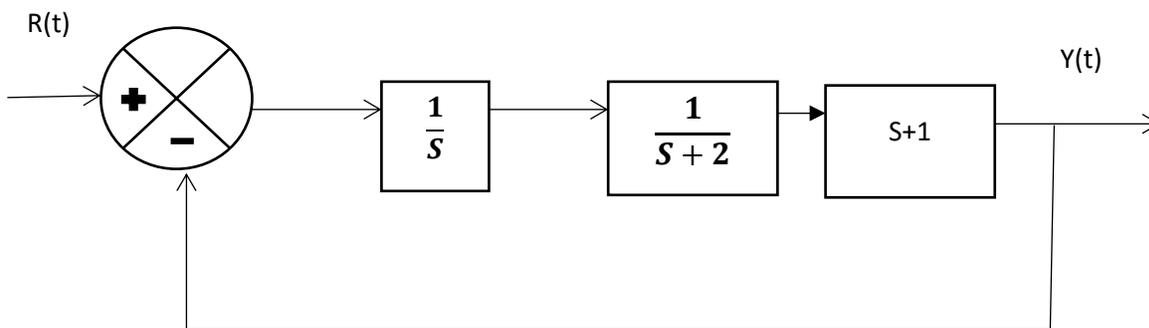
- a) Design of closed loop circuits to meet specific design criteria using root locus method.
- b) Improve the design with the help of lag/lead compensator.

Theory:

A closed loop system is given as follows:



If the system has an OLTF of $G(s)H(s) = \frac{s+1}{s(s+2)}$, it can be deduced that the system has a gain, $k=1$, zero at -1 and two poles situated at $S_1=0$ & $S_2= -2$. The system may be expressed as:



The adder block can be designed with the help of a differential amplifier, the $\frac{1}{s}$ block with an integrator, the $\frac{1}{s+2}$ with the help of an integrator having shifted pole and the $s+1$ block with a shifted zero.

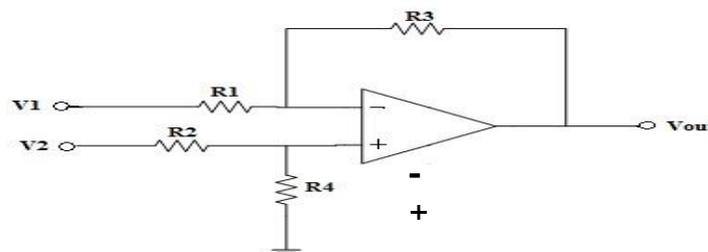


Fig.- Differential Amplifier Circuit

If $R1=R2=R3=R4$ then $V_{out}=V2-V1$.

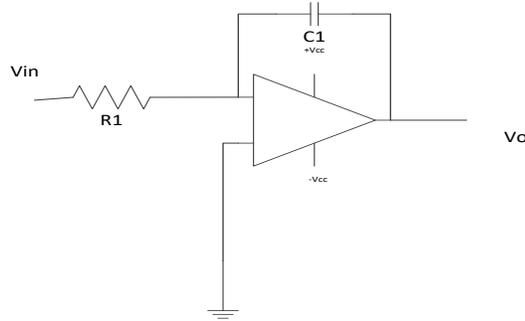


Fig. :Integrator circuit

For the integrator circuit above the transfer function is $\frac{V_o(s)}{V_{in}(s)} = \frac{1}{-R_1 C_1 S}$, Thus if we adjust the value R_1 & C_1 such that $\frac{1}{R_1 C_1} = 1$ (e.g. $R_1=100K$, $C_1=10\mu F$) then the TF becomes $-1/S$. The negative sign can be eliminated with the help of an inverting amplifier.

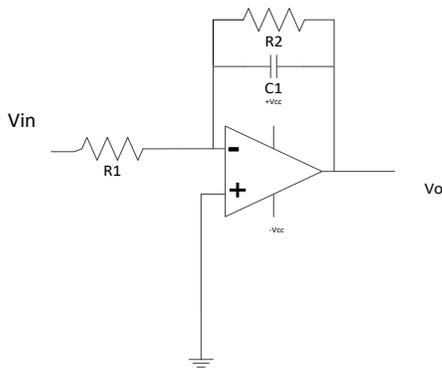


Fig. :Shifted Pole

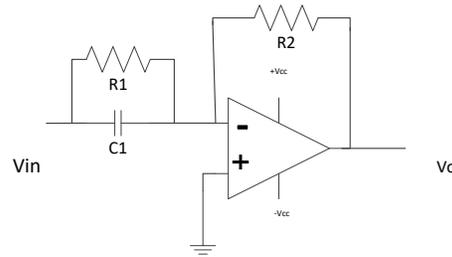


Fig. : Shifted zero

For design of $\frac{1}{s+2}$ block, the above circuit (Fig. Shifted pole) may be implemented. The TF of the circuit is $\frac{V_o(s)}{V_{in}(s)} = \frac{-R_1 C_1}{s + \frac{1}{R_2 C_1}}$. Thus by adjusting $R_1 C_1$ the numerator can be made to be 1 and by adjusting C_1 $s+2$ is attained such that $\frac{1}{R_2 C_1} = 2$.

For design of shifted Zeros, the circuit (Fig. Shifted zero) can be implemented. The TF of the circuit is $\frac{v_o(s)}{v_{in}(s)} = -R_2 C_1 \left(s + \frac{1}{R_1 C_1} \right)$. Here $R_2 C_1$ is adjusted to control the gain and $\frac{1}{R_1 C_1}$ to maintain the shifted zero position.

Thus the overall system can be shown as :

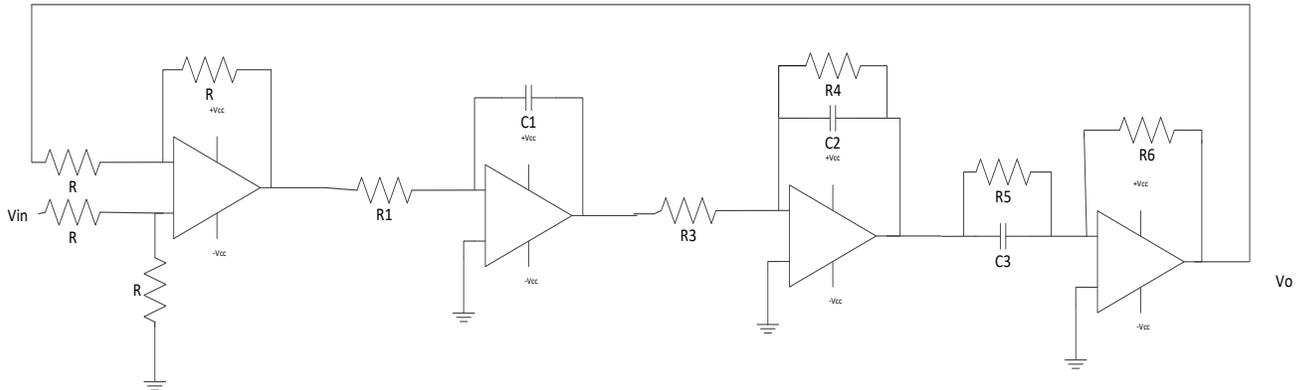


Fig. :Op-amp representation of the block diagram

Design from root locus: As seen from the previous experiment some design criteria may need to be met (e.g.40% O.S. & $T_s=5s$). Thus a second order system with O.L.T.F., $\frac{V_o}{V_i} = \frac{K}{s(s+a)}$ by obtaining 'K' and 'a' from sisotool and then design the circuit using op-amp in practical field or in Simulink environment.

Updating the design using lead/lag compensator:

Sometimes it may be necessary to update the system without changing the system parameters. This can be done with the help of compensators. For instance, the design with 40% O.S and 5s T_s have a gain factor of around $k=8$ and $a=1.6$. If this system is to be updated to 20% O.S. and 10s T_s without changing system parameters, then a zero located at -1.6 can be used to compensate the original pole and a new pole can be added and adjusted to meet the new design criteria.

Experimental Setup:

Setup 1: Design with root locus (40% O.S. $T_s=5s$)

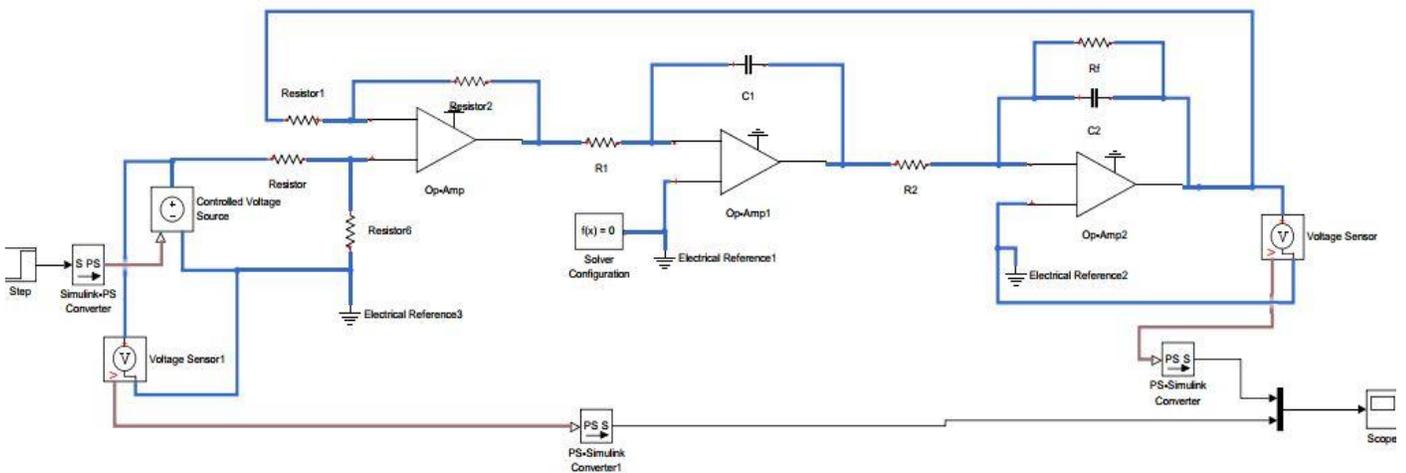


Figure: Setup 01

Setup 2: Design with compensator (20% O.S. $T_s=10s$)

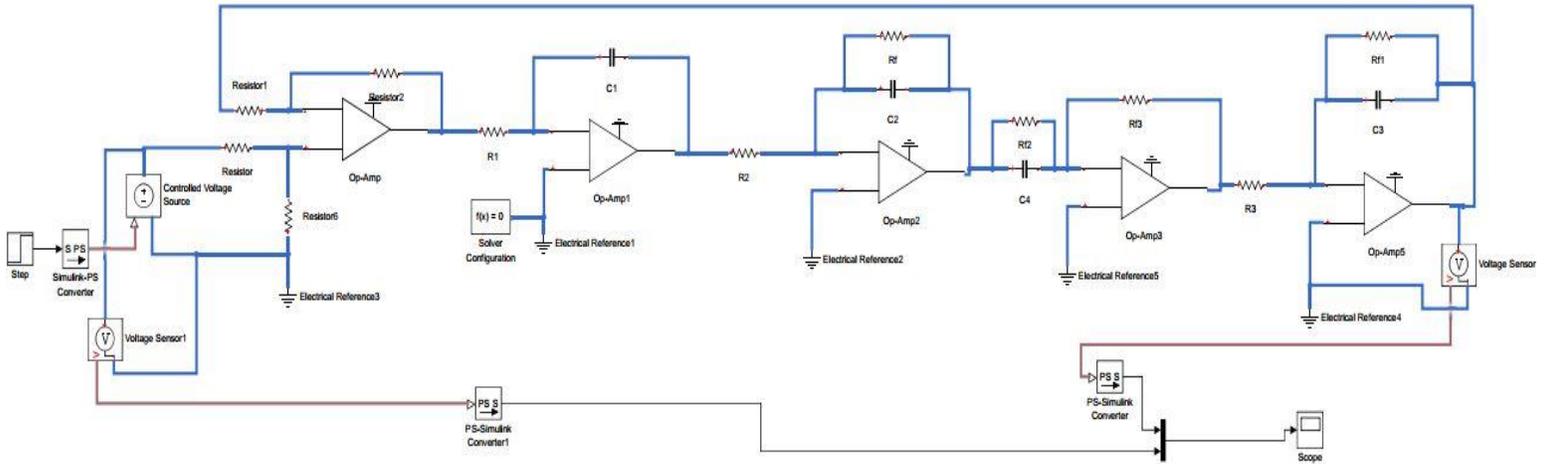


Figure : Setup 02

Data table:

Setup 1: For 40% O.S. & $T_s=5s$,Damping factor, $\zeta=$ Natural Freq. $\omega_n=$

From sisotool Gain, $K=$ shifted pole $(s+a)$, $a=$

For Simulink design calculated $R_1=$ $C_1=$ $R_2=$ $C_2=$

$R_f=$

Setup 2: For 20% O.S. & $T_s=10s$,Damping factor, $\zeta=$ Natural Freq. $\omega_n=$

From sistool Gain, $K_{new}=$ $K_{new}=K_{old} \cdot \text{new gain factor}$ shifted pole $(s+b)$, $b=$

For Simulink design calculated $R_{f2}=$ $C_4=$ $R_{f3}=$ $C_3=$

$R_{f1}=$ $R_3=$

Assignment:

1) Design a second order system using root locus such that %O.S.=30% and Rise time, $T_r=2s$ in Simulink environment.

2) Update the previous design without changing the original system so that the new % O.S= 20% and $T_r=4s$.

Experiment No: 08

Name of the Experiment: Control of a conveyor system using Programmable Logic Controller (PLC)

Introduction:

In industry, many manufacturing process demand a sequence of operation which are to be performed repetitively. Early automation systems were mechanical in design, timing and sequencing being affected by gears and cams. Slowly these design concepts were replaced by electrical drives which were controlled by relays. Even these days the relay control has not yet been obsolete rather is favorite to the engineers in many industries who understand a process and its cont better using Relay Ladder diagram.. However, the relays suffer from a number of problems viz: large size, slower operation, contact wear, inability to accept more than one input simultaneously, and the necessity of replacing the whole control panel in case another set of operation different from those for which the relay are hard wired are to be performed.

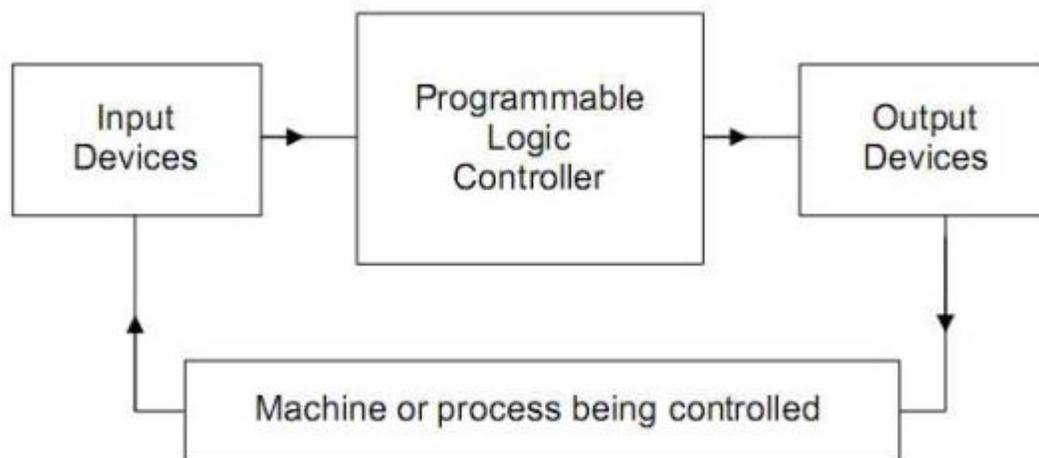
The Programmable Logic Controller

The necessity of controlling ever increasing systems, engineers turned to computers. The computer however was not suited to the industrial environment and the use of the computer on the factory floor, was not possible, unless costly interfacing filtering was used.

A programmable logic controller is a solid-state device, designed to operate in a noisy environment and perform all the logic function previously achieved using electromechanical relays, drum switches, mechanical timers and counters.

Basic PLC Operation:

Figure below shows how the PLC controls a machine or plant.



The PLC System:

The PLC, like a computer employs a microprocessor chip to do processing and memory chips to store the program.

The PLC consists of three sections:

1. A processor.
2. InputOutput (I/O).
3. A programming unit.

The inputs and outputs are connected via interface.

Basic PLC architecture:

Figure below shows the basic architecture of a PLC. It contains a processor (microprocessor chips), memory chips and an arithmetic logic unit (ALU). It also contains all the input and output interfacing. The programming device either hand held, dedicated terminal or desktop, are remote from the PLC controller.

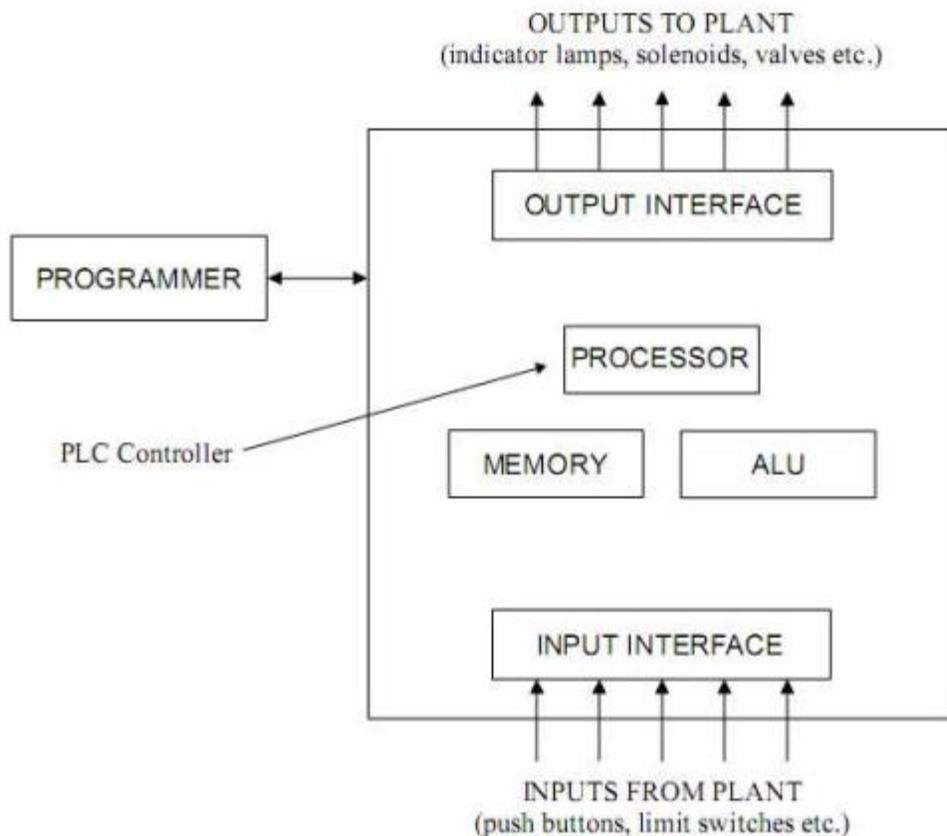


Table 1 to 3 show the connection that should be made between the PC45 trainer, MicroLogix 1500 PLC and Sequence Switch Module in order to complete the programming exercise.

Table 1:

PLC Input Addresses	PC45 Connection	Description
I0	SWG	Green Push Button
I1	SWR	Red Push Button
I2	S1	Sensor 1
I3	S2	Sensor 2
I4	R1	Micro Switch 1
I5	R2	Micro Switch 2
I6	R3	Micro Switch 3

Table 2:

PLC Input Addresses	PC45 Connection	Description
O4	E	Conveyor (Forward)
O5	R	Conveyor (Backward)
O6	LG	Green Lamp
O7	LR	Red Lamp
O8	C1	Cylinder 1
O9	C2	Cylinder 2
O10	C3	Cylinder 3

Table 3:

PLC Input Addresses	Switch Sequence Module Connection	Description
I1	S1	Switch 1
I8	S2	Switch 2
I9	S3	Switch 3

Other Connections:

DC COMMO and DCCOM1 are shorted and connected to 0 volt of upper block.

VAC VVDC4 and VAC DC5 are shorted and connection to 0 volt of lower block.

Vsw of sequence switch module to be connected to 24 volt of lower block.

Note:

When the cylinders of the PC45 Trainer are used, you will need to have the hand compressor pump connected.

As the cylinders on the PC45n Trainer are operated, air pressure drops in the hand compressor, so additional air will need to be pumped in as the system is used.

1. Short component should move the full length of the conveyor, being rejected by falling of the far end.
2. If the red push button is pressed, the conveyor should stop and the red lamp should lit.
3. Pressing green push button again should start the conveyor.

[Note: Before running the program make sure that the sensors are properly adjusted. The transmitter and receiver of the sensors should be aligned properly for accurate operation. The voltage between S1 br S2 arid 0 V should be around 23 volts if they are adjusted properly. Check the voltage of S1 and S2 in upper block and adjust the sensors if necessary.]

Procedures:

PART 1 To run the conveyor in forward and backward direction.

1. Open the RSLogix programming software by the following instruction.
Start → All Programs → Rockwell Software → RSLogix500English → RSLogix500English.
2. Open a new project. You will get a window with a list of processor name. Select a processor of Bul.1764 LSP Series C by scrolling down. A window will appear to develop the ladder logic diagram.
3. Develop the ladder logic diagram as instructed and save it properly.
4. Now you have to download the file. To download the file click on the drop-down arrow of the top-left dialogue box appearing offline and select download option. Before downloading the file make sure that the MicroLogix1500 PLC and the PC45 trainer is switched on.
5. When the program has been downloaded, you will be asked if you want to go online. Click yes to go online.
6. The top left dialogue box will change color and read REMOTE PROG. Click on the down arrow to the right of this and select Run from the drop down menu. Click on yes to change to run mode.
7. Place the green push button to run the conveyor in forward direction. Press the Red push button to stop the conveyor.

8. Place the object at the right most position of the conveyor. When sensor 2 gets the object the conveyor should run in backward direction and it will continue to run in the backward direction unless you reprogram it.
9. Modify Rung 03 of the ladder logic diagram as suggested and observe the movement of conveyor after sensing the object at sensor 2. The conveyor should run back and forth with the movement of the object in between sensor 1 and 2.
10. To bring the program out of Run mode, click on the down arrow button to the right of the top left dialogue box and select program, then click on yes to return to program mode.
11. Click on the file menu and select close to close the project. Now you are safe to exit the programming software.
12. Remove the power from MicroLogix 1500 PLC and PC45 trainer.

PART 2: To distinguish tall object from short object placed on a running conveyor. In this part the program should perform the following operation.

1. The Green push button should be used to start the conveyor in the forward, right to left direction.
2. While the conveyor is operating, the green lamp should be lit.
3. Use sensor 1 on the conveyor to determine if a component is tall or short. (Should be set to detect a tall component)
4. Sensor 2 on the conveyor is used to start timing for either a tall component or a short component. (Should be set to detect all components)
5. Cylinder 2 (the center cylinder) is operated when a tall reaches it. (Remember that micro switch 2 is activated when cylinder 2 is operated)

Report:

Explain the Logic of the ladder diagrams programmed in both parts.