

2117073 Tomos Rhys Williams

by Rhys Williams

Submission date: 09-Apr-2025 10:47PM (UTC+0100)

Submission ID: 255282686

File name: 17432_Rhys_Williams_2117073_Tomos_Rhys_Williams_183785_1088765742.pdf (2.9M)

Word count: 16666

Character count: 87411

Course: HNC Electrical/Electronic Engineering

Module: SGEE 5002 Electrical Principles, Methods and Simulation

Assignment: Analysis, Simulation and Evaluation of Circuits and Systems

Name: Tomos Rhys Williams

University Number: 2117073

Date Submitted: 09/04/2025

Word Count: 9788

Summary

The tasks featured in this assignment focus on the design, building, testing and analysis of multiple different electrical circuits and systems such as RC circuits and a DC Motor Model. Different forms of analysis will be used throughout the assignment such as transient analysis and frequency performance analysis. In this assignment, the software packages that will be utilised are Matlab and Simulink which are both quality engineering software packages that work to a professional standard, so it can be said that the results gathered from the simulation responses are valid and accurate. Throughout the assignment, crucial component values and parameters are required, some are provided and some need to be calculated through the utilisation of advanced mathematical methods and formula. Some of these calculated values were then critically analysed and compared against values gathered from simulation responses to assist in drawing conclusions to correlate the results to electrical principle theory. In other cases  transfer functions were developed through these calculated values and were then utilised to create the systems or to assist in drawing suitable conclusions.

This assignment involved conducting transient analysis of a circuit for a section of the tasks where the behaviour and response of the circuits was analysed in respect to time and this was conducted on First Order RL and RC circuits and on a Second  Order Series RLC circuit. Values gathered from the responses were compared against calculated values to see if the circuits functioned as required in regard to their transient performance. Some other tasks in this assignment will conduct frequency analysis on First Order RL and RC circuits and on a Second Order Series RLC circuit. These analysis will focus on the response of the circuits in the frequency domain and values gathered from the responses were compared against calculated values to see if the circuits functioned as required in regard to frequency. The phase shift of the systems was also gathered when the simulated responses were provided, and this could be utilised to determine the stability of each system.

Another section of the assignment focuses on the creation and development of a DC Motor Model. Once the model was created, a closed loop system was then created with a PID controller in order to help the system become as accurate and efficient as possible. The development of this system was discussed over several tasks with the function of Proportional, Integral and Derivative control discussed.

The utilisation of Matlab and Simulink to gather the visual simulation responses helped make the analysis of each task very extensive and assisted in meeting the criteria for each task as specified in the assignment brief.

Table of Contents

Summary	2
Table of Figures	4
Table of Tables	6
Introduction	7
Basic Definitions	7
Task 1 – Transient Analysis of a First Order RL Electrical Circuit	9
Task 2 – Transient Analysis of a First Order RC Electrical Circuit	14
Task 3 – Frequency Performance Analysis of a First Order RL Electrical Circuit	18
Task 4 – Frequency Performance Analysis of a First Order RC Electrical Circuit	22
Task 5 – Transient Performance Analysis of a Second Order Series Electrical Circuit	26
Task 6 – Frequency Performance Analysis of a Second Order Series Electrical Circuit	43
Task 7 – Frequency Performance Analysis of a First Order Electrical System	61
Task 8 – Frequency Performance Analysis of a First Order Electrical System (Continued)	65
Task 9 – First Order Control System Analysis	69
Task 10 – DC Motor Model Analysis	83
Task 11 – Proportional Speed Control System	91
Task 12 – Proportional-Integral Speed Control System	96
Task 13 – Proportional-Integral-Derivative Speed Control System	101
Task 14 – Controller Comparisons	106
Task 15 – Overall Conclusions and Recommendations	110

Table of Figures

Figure 1 - Multisim circuit model of the first order RL electrical system in Task 1	10
Figure 2 - Matlab Code for Task 1	11
Figure 3 - Matlab Response for Task 1	12
Figure 4 - Multisim circuit model of the first order RC electrical system in Task 2	14
Figure 5 - Matlab Code for Task 2	15
Figure 6 - Matlab Response for Task 2	16
Figure 7 - Multisim circuit model of first order RL electrical system in Task 3	19
Figure 8 - Matlab Code for Task 3	20
Figure 9 - Matlab Response for Task 3	20
Figure 10 - Multisim circuit model of the first order RC electrical system in Task 4	23
Figure 11 - Matlab Code for Task 4	24
Figure 12 - Matlab Response for task 4	24
Figure 13 - Multisim circuit model of the second order series RLC electrical system for an overdamped response in Task 5	28
Figure 14 - Matlab Code for Overdamped Response in Task 5	29
Figure 15 - Underdamped Response in matlab for Task 5	30
Figure 16 - Multisim circuit model of the second order series RLC electrical system for an underdamped response in Task 5	31
Figure 17 - Matlab Code for Underdamped Response for Task 5	32
Figure 18 - Underdamped Response in Matlab for Task 5	33
Figure 19 - Multisim circuit model of the second order series RLC electrical system for a critically damped response in Task 5	34
Figure 20 - Matlab Code for a Critically Damped Response in Task 5	36
Figure 21 - Critically Damped Response in matlab for Task 5	36
Figure 22 - Multisim circuit model of the second order series RLC electrical system for a no damping response in Task 5	37
Figure 23 - Multisim circuit model of the second order series RLC electrical system for an even worse no damping response in Task 5	37
Figure 24 - Matlab Code for No Damped Response for Task 5	38
Figure 25 - No Damped response in matlab for Task 5	39
Figure 26 - Matlab Code for even Worse No Damped Response for Task 5	41
Figure 27 - Even worse No damped Response in Matlab For Task 5	41
Figure 28 - Multisim circuit model of the second order series RLC electrical system for an overdamped response in Task 6	45
Figure 29 - Multisim circuit model of the second order series RLC electrical system for a critically damped response in Task 6	47
Figure 30 - Multisim circuit model of the second order series RLC electrical system for the underdamped 1 response in Task 6	50
Figure 31 - Multisim circuit model of the second order series RLC electrical system for the underdamped 2 response in Task 6	52
Figure 32 - Multisim circuit model of the second order series RLC electrical system for the underdamped 3 response in Task 6	54
Figure 33 - Multisim circuit model of the second order series RLC electrical system for the underdamped 4 response in Task 6	56
Figure 34 - Matlab Code for the Responses in Task 6	58
Figure 35 - Bode Diagram For Task 6	59
Figure 36 - Zoomed in View of the Bode Diagram for Task 6	59
Figure 37 - Multisim circuit model for the active Low-pass filter electrical system	61
Figure 38 - Matlab Code for Task 7	63
Figure 39 - Matlab Response for Task 7	63

Figure 40 - Multisim circuit model for the active High-pass filter electrical system	65
Figure 41 - Matlab Code for task 8	67
Figure 42 - matlab Response for Task 8.....	67
Figure 43 - Open Loop Control System for Task 9	69
Figure 44 - Step Response for the Open Loop System in Task 9.....	69
Figure 45 - Matlab Code for the Argand Diagram for the Open Loop System in Task 9.....	70
Figure 46 - Argand Diagram for the Open Loop System in Task 9	70
Figure 47 - Closed Loop Control System Model for Task 9	71
Figure 48 - Step Response for the Closed Loop System in Task 9.....	72
Figure 49 - Matlab Code for the Argand Diagram for the Closed Loop System in Task 9.....	73
Figure 50 -Argand Diagram for the Closed Loop System in Task 9	73
Figure 51 - Matlab Code for the Root Locus Plot for the Closed Loop System in Task 9.....	75
Figure 52 - Root Locus Plot Showing the minimum value of K Required for Stable System in Task 9.....	75
Figure 53 - Closed Loop System Control Model with the Minimum K Value Required for a Stable System in Task 9.....	76
Figure 54 - Step Response for the Minimum Value of K Required for a Stable System in Task 9	77
Figure 55 - Closed Loop System Control Model with a Gain Greater than the Minimum K Value Required for a Stable System in Task 9	78
Figure 56 - Step Response for a System with a Gain Greater than the Minimum Value of K Required for a Stable System in Task 9	79
Figure 57 - Closed Loop System Model with a Gain of 0 in Task 9	80
Figure 58 - Step Response for a System with a Gain of 0 in Task 9	81
Figure 59 - DC Motor Model Built in Simulink	84
Figure 60 - Simulink Graphical Response for Speed with Load	86
Figure 61 - Simulink Graphical Response for Current with Load	87
Figure 62 - Simulink Graphical Response for Torque with Load	87
Figure 63 - Simulink Graphical Response for Speed with No Load	88
Figure 64 - Simulink Graphical Response for Current with No Load.....	89
Figure 65 - Simulink Graphical Response for torque with No Load	89
Figure 66 - Auto Tune Feature in Simulink Showing the Optimum Kp Value	91
Figure 67 -Subsystems with P Controller built in Simulink	92
Figure 68 - Simulink Responses for the Three Systems with No Load	93
Figure 69 - Simulink Responses for the Three Systems with Load	93
Figure 70 - Zoomed in Simulink Responses for the Three Systems with Load	94
Figure 71 - Auto Tune Feature in Simulink Showing the Optimum Kp and Ki Values.....	96
Figure 72 - Subsystems with PI Controllers Built in Simulink.....	97
Figure 73 - Simulink Responses for the Three Systems with Load	98
Figure 74 - Zoomed in Simulink Responses for the Three Systems with Load	98
Figure 75 - Simulink Responses for the Three Systems with No Load	99
Figure 76 - Autotune Feature in Simulink Showing the Optimum Values of Kp, Ki and Kd	101
Figure 77 - Subsystems with PID Controllers Built in Simulink	102
Figure 78 - Simulink Response for Three Systems with Load	103
Figure 79 - Zoomed in Simulink Response of the Three systems with load	103
Figure 80 - Simulink Response for the Three Systems with No Load	104
Figure 81 - Subsystems with P, PI and PID controllers built in Simulink.....	106
Figure 82 - Simulink Response for the Three Subsystems with Load	107
Figure 83 - Zoomed in Simulink Response for the Three Systems with Load	107
Figure 84 - Simulink Response for the Three Systems without Load.....	108

Table of Tables

Table 1 – Comparison of calculated and simulated values of Tau in Task 1.....	12
Table 2 - Comparison of Calculated and Simulated values of Tau in Task 2.....	16
Table 3 - Comparison of Calculated and Simulated values of Cut Off Frequencies in Task 3	21
Table 4 - Comparison of calculated and Simulated Values of Cut Off Frequency in Task 4	25
Table 5 - Comparison of Damping Factor and Response Depending on Resistance in Task 5	42
Table 6 - Comparison of Quality Factor, Damping Factor and Frequency for Maximum Amplitude Depending on the Response in Task 6	60
Table 7 – Comparison of calculated and simulated values of Frequency, Magnitude and Phase shift in Task 7. 64	
Table 8 – Comparison of calculated and simulated values of Frequency, Magnitude and Phase shift in Task 8. 68	
Table 9 - Comparison of the Values of the Poles and the Responses Achieved when Altering the Value of K	82
Table 10 - Comparison of Calculated and Simulated Values of Speed	90
Table 11 - Comparison of the Variables when Altering the Value of Kp in each Subsystem.....	95
Table 12 - Comparison of the Variables when Altering the Value of Ki in each system.....	100
Table 13 - Comparison of the Variables when Altering the Value of Kd in each system.....	105
Table 14 - Comparison of the Variables in P, PI and PID Controllers.....	109

Introduction

In this assignment, the overall aim is to gain a greater understanding of the advanced electrical principles discussed in this assignment. These principles are related to control systems and electrical and electronic engineering. Several methods of modelling, simulation and analysis are to be utilised and discussed in order to describe and predict the performance of the electrical circuits and systems that are designed and discussed in this assignment. Appropriate analysis must be drawn for each task utilising electrical principle theory and the simulated responses of each circuit and system that is developed.

Additionally, the understanding of the background electrical principle theory must be critical in order to develop, analyse and evaluate each circuit and system in order to draw suitable conclusions to meet the desired criteria for each task.

Basic Definitions

During this assignment there are several examples of electrical terminology that is regularly repeated, and an understanding of these terms is required to develop a basic understanding of each task.

First Order Circuit

A circuit only containing one component that stores energy which in these examples can be a capacitor or an inductor.

Second Order Circuit

A circuit containing two energy storing components.

Cut-Off Frequency

The point at which the output frequency has dropped to approximately 70.7% of its maximum value. This value can assist you in understanding which frequencies are attenuated and which frequencies can pass.

Bode Plot

A magnitude plot versus frequency and a phase plot versus frequency displaying how the system reacts under the set parameters.

Phase Angle

A value that can be used to determine if a system is leading or lagging.

Argand Diagram

Graphical representation of complex numbers where the horizontal axis represents the real part, and the vertical axis represents the imaginary part.

Root Locus Plot

Graphical method for examining how the roots of a system change with variation of a certain system parameter, commonly a gain within a feedback system.

Pole Zero Plot

Graphical representation of a transfer function in the complex plane, where poles ("x") and zeros ("o") are plotted to visualize system behaviour and stability.



Task 1 – Transient Analysis of a First Order RL Electrical Circuit

So that I could achieve unique results, I would also choose unique component values. I chose a resistance of 650Ω and an inductance of 0.6H.



From these component values I can calculate the time constant:

$$\tau = \frac{L}{R} = \frac{0.6}{650} = 0.00092308 = 92\text{ms}$$

$$\tau = 92\text{ms}$$



The Laplace Model

This circuit can be displayed as a Laplace Model with the transfer function being in the terms of the time constant, I have already calculated.

Potential Divider - $Y(s) = \frac{SL}{SL+R} \times U(S)$

Step Input - $U(S) = \frac{1}{s}$

Therefore - $Y(S) = \frac{SL}{SL+R} \times \frac{1}{s}$

This can be rewritten as - $Y(S) = \frac{SL}{s(SL+R)}$

Simplified by dividing by 'S' - $Y(S) = \frac{L}{SL+R}$

Where 'A' = R/L - $\frac{1}{s+A} = \frac{1}{s+\frac{R}{L}}$

Inverse Laplace - $l^{-1} \left\{ \frac{1}{s+\frac{R}{L}} \right\}$
 $Y(t) = e^{-at}$

Where $\tau = L/R$ - $e^{-\frac{Rt}{L}}$

Transfer function in terms of τ - $\frac{Y(s)}{U(s)} = \frac{sL}{sL+R}$

This becomes - $\frac{\frac{sL}{R}}{\frac{sL}{R}+1}$



Multisim Circuit

The following circuit was constructed in NI Multisim and provides a visual interpretation of the circuit.

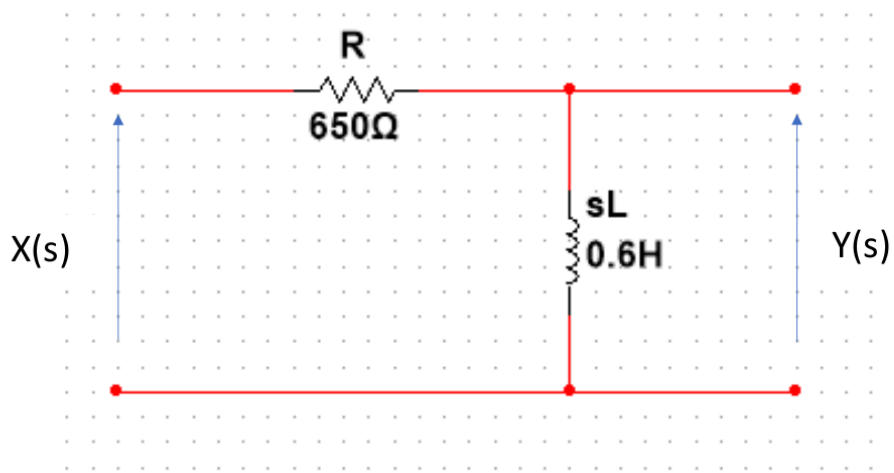


FIGURE 1 - MULTISIM CIRCUIT MODEL OF THE FIRST ORDER RL ELECTRICAL SYSTEM IN TASK 1

Matlab Model

Matlab Formula

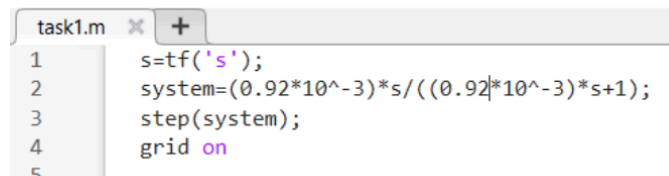
The following formula will be inputted into Matlab to graphically represent the behaviour of the RL circuit over time.

$$H(S) = \frac{\tau S}{\tau S + 1}$$

This is because $\tau = L/R$

Matlab Code

In order to obtain accurate graphical results from the Matlab model to compare against my calculated time constant, I must input the transfer function accurately into the Matlab code which will then provide me with a graph of amplitude against time. Below you can see the code I inputted into Matlab:



```
1 s=tf('s');  
2 system=(0.92*10^-3)*s/((0.92*10^-3)*s+1);  
3 step(system);  
4 grid on
```

FIGURE 2 - MATLAB CODE FOR TASK 1

Matlab Graph

The graph for this code is shown below:

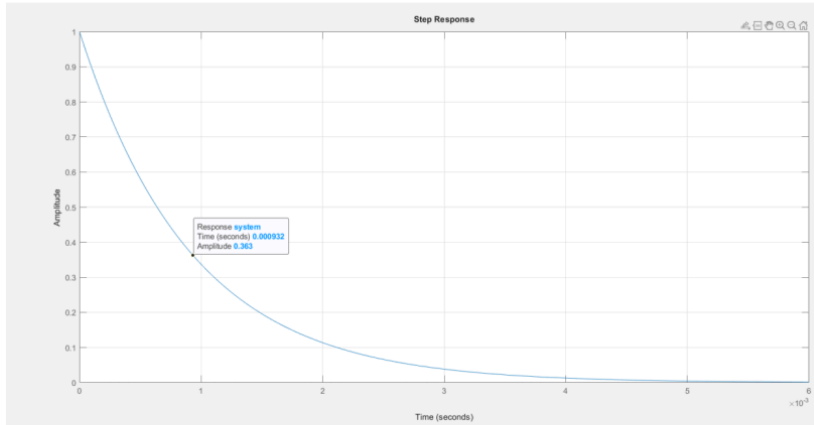


FIGURE 3 - MATLAB RESPONSE FOR TASK 1

Comparison of Results

As I now have two sets of results for this electrical circuit, I can compare them. The table of results shown below provides the calculated time constant, the simulated time constant and the error between these two results.

Calculated Value of τ	Simulated Value of τ	Error
0.00092308ms	0.000932ms	0.00000892ms

TABLE 1 – COMPARISON OF CALCULATED AND SIMULATED VALUES OF TAU IN TASK 1

Analysis

The results that I achieved with my Matlab model simulation prove that the behaviour of the circuit was as expected from a Resistor- Inductor circuit with the graph showing an exponential decay curve in the amplitude over time. This happens from when the voltage source is removed which then causes the amplitude to decrease rapidly then the decay to slow down over time, and this is because of there would be less energy to support the current which results in a smaller decay over time. The time constant (τ) is the time taken for the amplitude to reach 36.8% of its maximum which is 0.368. e^{-1} should equate to 0.368 as this is a first order RL circuit and this can be shown using the formula: $Y(t) = e^{-\frac{t}{\tau}}$. In this formula $e^{-\frac{t}{\tau}}$ represents the exponential decay over time to 0. When t is equal to τ , e^{-1} equates to 0.368 and the graph shows that the time constant can be found at 0.363. The miniscule error between the simulated values and calculated values I achieved could possibly be put down to the rounding of the values inputted into Matlab.



Task 2 – Transient Analysis of a First Order RC Electrical Circuit

So that I could achieve unique results, I would also choose a unique component value for the capacitor. I chose a capacitance of 4nF to accompany the 650Ω resistor.

From these component values I can calculate the time constant:

$$\tau = RC = 650 \times 4 \times 10^{-9} = 2.6 \times 10^{-6} = 2.6\mu\text{s}$$

$$\tau = 2.6\mu\text{s}$$

The transfer function for an RC circuit is shown below:

$$H(s) = \frac{1}{(RC)s + 1}$$



Multisim Circuit

The following circuit was constructed in NI Multisim and provides a visual interpretation of the circuit.

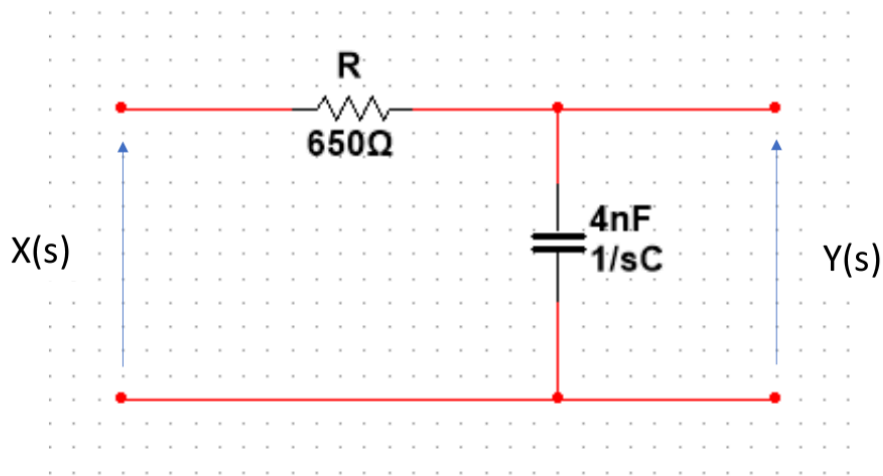


FIGURE 4 - MULTISIM CIRCUIT MODEL OF THE FIRST ORDER RC ELECTRICAL SYSTEM IN TASK 2

Matlab Model

Matlab Formula

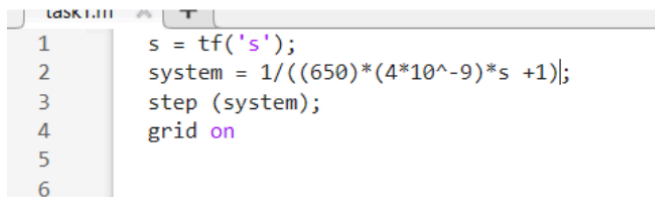
The following formula will be inputted into Matlab to graphically represent the behaviour of the RC circuit over time.

$$H(s) = \frac{1}{\tau s + 1}$$

This is because $\tau = RC$

Matlab Code

In order to obtain accurate graphical results from the Matlab model to compare against my calculated time constant, I must input the transfer function accurately into the Matlab code which will then provide me with a graph of amplitude against time. Below you can see the code I inputted into Matlab:



```
1 s = tf('s');
2 system = 1/((650)*(4*10^-9)*s +1);
3 step (system);
4 grid on
5
6
```

FIGURE 5 - MATLAB CODE FOR TASK 2

Matlab Graph

The graph for this code is shown below:

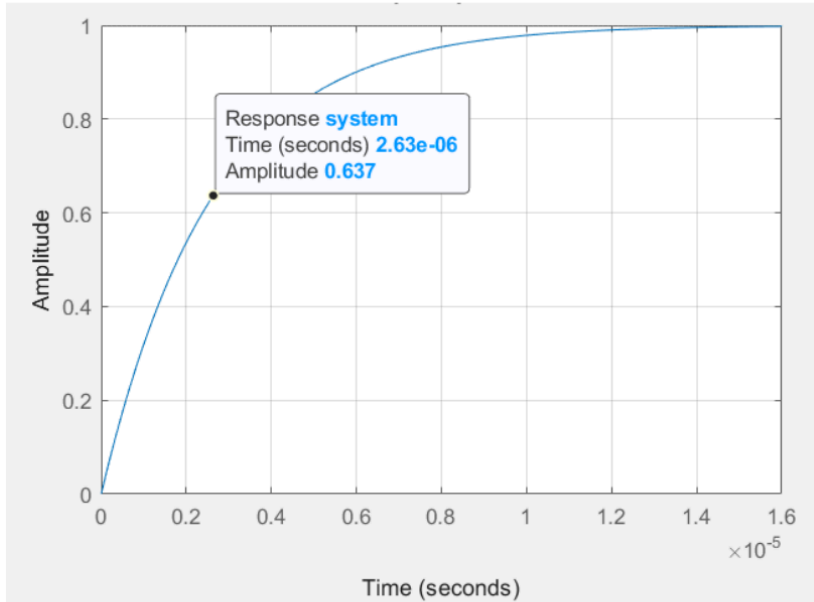


FIGURE 6 - MATLAB RESPONSE FOR TASK 2

Comparison of Results

As I now have two sets of results for this electrical circuit, I can compare them. The table of results shown below provides the calculated time constant, the simulated time constant and the error between these two results.

Calculated Value of τ	Simulated Value of τ	Error
2.6 μ s	2.63 μ s	0.03 μ s

TABLE 2 - COMPARISON OF CALCULATED AND SIMULATED VALUES OF TAU IN TASK 2

Analysis

The results that I achieved with my Matlab model simulation prove that the behaviour of the circuit was as expected from a Resistor-Capacitor circuit with the graph showing an exponential growth curve in the amplitude over time. This exponential growth is caused by the capacitor continuing to charge as time elapses from the voltage source being applied, and this would cause the current to decrease because of the voltage rise across the capacitor. In the Matlab simulation above, the cursor has been placed approximately on one time constant which gives the simulated value of $2.63\mu\text{s}$ to compare against the calculated value of τ . One time constant is 63.2% of the maximum value and this is because the rate of change of a variable can be calculated using the formula: $Y(t) = 1 - e^{-\frac{t}{\tau}}$. When t (time) is equal to τ , the formula would be: $Y(t) = 1 - e^{-1}$. This would mean that $Y(t)$ is equal to 0.632. The miniscule error between the simulated values and calculated values I achieved seems to have been caused by the inability to place the cursor directly on the one time constant value of 0.632 as I could only achieve a value of 0.637 which causes my simulated value to be slightly above the calculated value with a tiny error.



Task 3 – Frequency Performance Analysis of a First Order RL Electrical Circuit

In order to obtain the cut off frequency in a RL circuit you use the formula:

$$\omega c = \frac{R}{L}$$



I used the same component values as the RL circuit in Task 1, so a resistance of 650Ω and an inductance of 0.6H. The cut off frequency (radians per second) can then be calculated as:

$$\omega c = \frac{R}{L} = \frac{650}{0.6}$$
$$\omega c = 1083.3 \text{ rad/s}$$

The initial transfer function equation is:

$$H(s) = \frac{sL}{sL + R}$$

This can then be simplified by dividing the top and bottom by L:

$$H(s) = \frac{s}{s + \frac{R}{L}}$$

As $\omega c = R/L$ this can be rewritten as:

$$H(s) = \frac{s}{s + \omega c}$$

This is your final transfer function:

$$H(s) = \frac{s}{s + 1083.3}$$

Multisim Circuit

The following circuit was constructed in NI Multisim and provides a visual interpretation of the circuit.

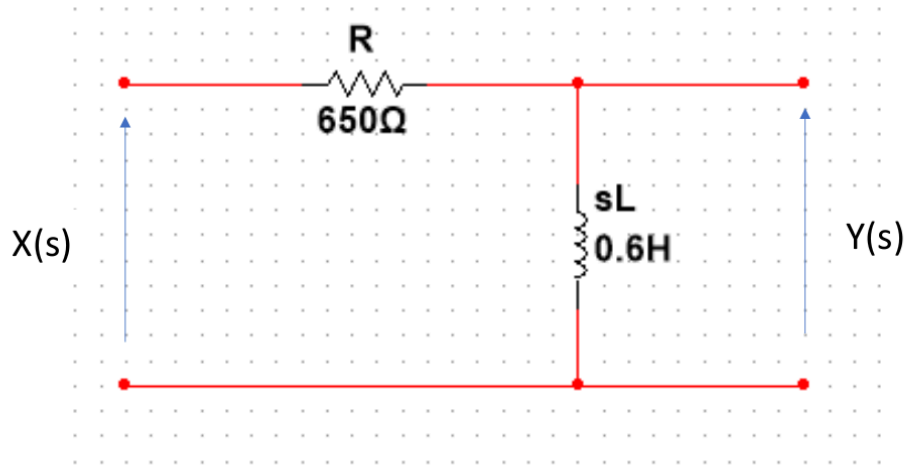


FIGURE 7 - MULTISIM CIRCUIT MODEL OF FIRST ORDER RL ELECTRICAL SYSTEM IN TASK 3

Matlab Model

Matlab Code

In order to obtain accurate graphical results from the Matlab model to compare against my calculated cut-off frequency, I must input the transfer function accurately into the Matlab code which will then provide me with two graphs. One graph of frequency against magnitude and one graph of frequency against phase angle. Below you can see the code I inputted into Matlab:

```
task1.m x +
1      s = tf('s');
2      system = s/(s+1083.3);
3      bode(system);
4      grid on
5      |
```

FIGURE 8 - MATLAB CODE FOR TASK 3

Matlab Graphs

The graphs for this code is shown below:

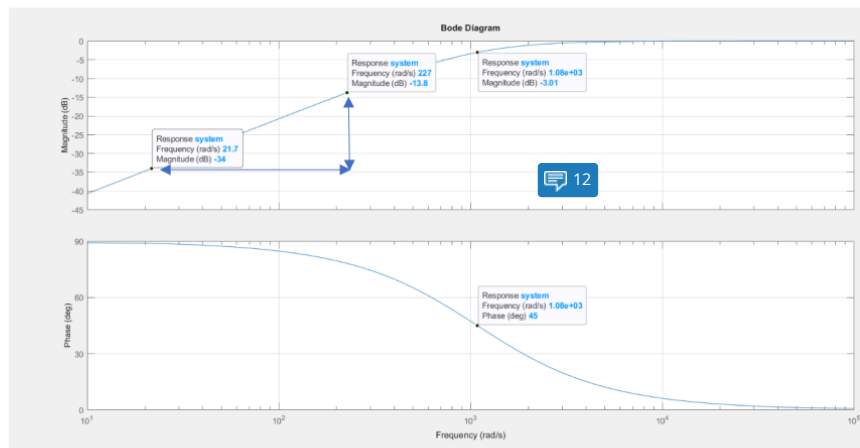


FIGURE 9 - MATLAB RESPONSE FOR TASK 3

Comparison of Results

As I now have two sets of results for this electrical circuit, I can compare them. The table of results shown below provides the calculated cut off frequency, the simulated cut off frequency and the error between these two results.

Calculated cut off frequency	Simulated cut off frequency	Error
1083.3 rad/s	1080 rad/s	3.3 rad/s

TABLE 3 - COMPARISON OF CALCULATED AND SIMULATED VALUES OF CUT OFF FREQUENCIES IN TASK 3



Analysis

This RL circuit functions as a high pass filter, this is explained by the fact that at greater frequencies the inductors impedance increases which allows the high frequencies to pass while the unwanted low frequencies are attenuated. The point at which the stopband transitions into the passband is known as the cut off frequency and this will approximately be at -3dB. In the Matlab simulation shown above the magnitude of this cut off frequency is at 3.01dB which is very accurate which only causes a very minimal error of 3.3 rad/s between by calculated cut off frequency value and my simulated cut off frequency value.

At this frequency the phase shift is 45° as this is the point in the circuit where the resistance and inductive reactance are equal in magnitude, and the output voltage is loading at this point. Additionally, data on the simulation graph shows that on the roll-off of the graph over the course of a decade, the magnitude changes by approximately 20dB and this proves that the filter attenuates the undesirable low frequencies effectively and ensures the transition over the passband will be swift.



Task 4 – Frequency Performance Analysis of a First Order RC Electrical Circuit

As this is an RC circuit, ωc will be the inverse of an RL circuit so $\omega c = \frac{1}{RC}$

I used the same component values as the RC circuit in Task 2, so a resistance of 650Ω and a capacitance of 4nf . The cut off frequency (radians per second) can then be calculated as:

$$\omega c = \frac{1}{RC} = \frac{1}{650 \times 4 \times 10^{-9}}$$
$$\omega c = 384615.3846 \text{ rad/s}$$

The transfer function in the Laplace domain for an RC circuit is:

$$H(s) = \frac{1}{RC \times s + 1} = \frac{1}{\frac{s}{\omega c} + 1}$$

The transfer function in terms of the cutoff frequency is:

$$H(s) = \frac{\omega c}{s + \omega c}$$

This is your final transfer function:

$$H(s) = \frac{384615.3846}{s + 384615.3846}$$

Multisim Circuit

The following circuit was constructed in NI Multisim and provides a visual interpretation of the circuit.

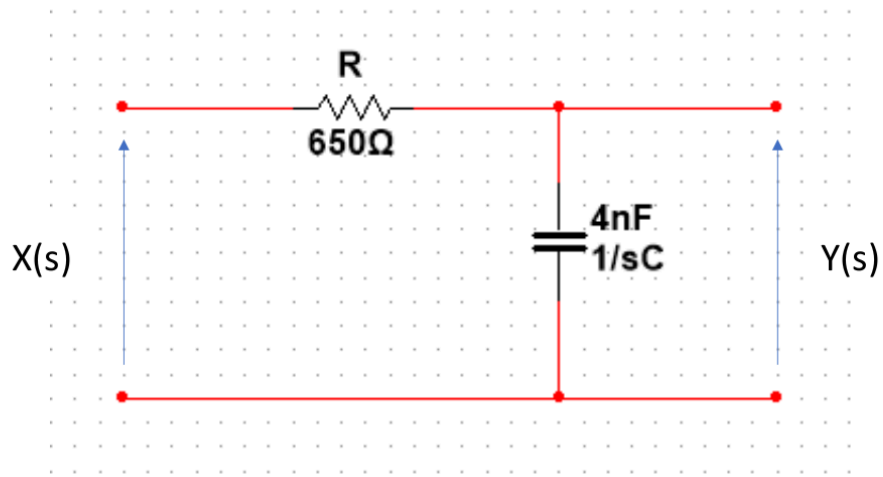


FIGURE 10 - MULTISIM CIRCUIT MODEL OF THE FIRST ORDER RC ELECTRICAL SYSTEM IN TASK 4

Matlab Model

Matlab Code

In order to obtain accurate graphical results from the Matlab model to compare against my calculated cut-off frequency, I must input the transfer function accurately into the Matlab code which will then provide me with two graphs. One graph of frequency against magnitude and one graph of frequency against phase angle. Below you can see the code I inputted into Matlab:

```
task1.m x +
1 s = tf('s');
2 system = 384615.3846/(s + 384615.3846);
3 bode (system);
4 grid on
5
```

FIGURE 11 - MATLAB CODE FOR TASK 4

Matlab Graphs

The graphs for this code is shown below:

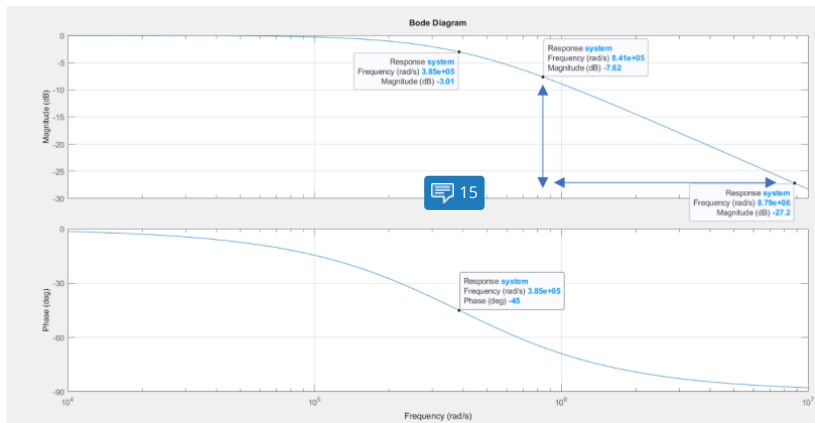


FIGURE 12 - MATLAB RESPONSE FOR TASK 4

Comparison of Results

As I now have two sets of results for this electrical circuit, I can compare them. The table of results shown below provides the calculated cut off frequency, the simulated cut off frequency and the error between these two results.

Calculated cut off frequency	Simulated cut off frequency	Error
384615.3846 rad/s	385000 rad/s	384.6164 rad/s

TABLE 4 - COMPARISON OF CALCULATED AND SIMULATED VALUES OF CUT OFF FREQUENCY IN TASK 4

Analysis

This RC circuit functions as a low pass filter, this is explained by the fact that at smaller frequencies the capacitor behaves like an open circuit thus having a high impedance allowing the signals to pass, while at greater frequencies the same capacitor behaves like a short circuit thus having a low impedance which filters these signals away. The point at which the stopband transitions into the passband is known as the cut off frequency and this will approximately be at -3dB. In the Matlab simulation shown above the magnitude of this cut off frequency is at 3.01dB which is very accurate which only caused an error of 384.6164 rad/s between by calculated cut off frequency value and my simulated cut off frequency value.

At this frequency the phase shift is -45° as this is the point in the circuit where the resistance and inductive reactance are equal in magnitude, as the output voltage is lagging at this point. Additionally, data on the simulation graph shows that on the roll-off of the graph over the course of a decade, the magnitude changes by approximately 20dB and this proves that the filter attenuates the unwanted high frequencies successfully and ensures the transition from the stopband and into the passband will be without delay.



Task 5 – Transient Performance Analysis of a Second Order Series Electrical Circuit

So that I could achieve unique results, I would also choose unique component values. I chose an inductance of 0.3H and a capacitance of 25uF. The resistance will have a direct effect on the damping factor so the resistance must be changed to achieve each desired damping response. I will use 650Ω to achieve an overdamped response, 54Ω to achieve an underdamped response, 219.089023Ω to achieve the critically damped response and then 0.00001Ω and 0.0000001Ω for the no damping responses.

The transfer function formula can be shown through the following formulas:

Potential divider rule:

$$Y(s) = \frac{\frac{1}{sC}}{R + sL + \frac{1}{sC}} X(s)$$

Multiplying the top and bottom by S/L:

$$\frac{Y(s)}{X(s)} = \frac{\frac{1}{sC} \times \frac{S}{L}}{\left(R \times \frac{S}{L}\right) + \left(sL \times \frac{S}{L}\right) + \left(\frac{1}{sC} \times \frac{S}{L}\right)}$$

This can then be simplified to find the Laplace formula:

$$H(s) = \frac{\frac{1}{LC}}{s^2 + \left(\left(\frac{R}{L}\right) \times s\right) + \frac{1}{LC}}$$

18

The Laplace can then be compared to the 2nd Order standard form equation:

$$\frac{kw_n^2}{s^2 + (2)(\zeta)(w_n)(s) + w_n^2}$$

where:

w_n = natural frequency of oscillation

ζ = damping coefficient

k = dc gain

To calculate these figures in an RLC circuit we use the following equations:

$$w_n = \frac{1}{\sqrt{LC}} \quad \text{this is because } w_n^2 = \frac{1}{LC} \quad \text{therefore } w_n = \frac{1}{\sqrt{LC}}$$

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

$$k = 1$$

As previously discussed, the resistance directly affects the damping factor which causes the four different damping responses:

- $\zeta < 1$ underdamped response with oscillations
- $\zeta > 1$ overdamped response with no oscillations
- $\zeta = 1$ critically damped response
- $\zeta = 0$ no damping

Overdamped

I chose a resistance of 650Ω and an inductance of 0.3H and a capacitance of $25\mu\text{F}$.

Multisim Circuit

The following circuit was constructed in NI Multisim and provides a visual interpretation of the circuit.

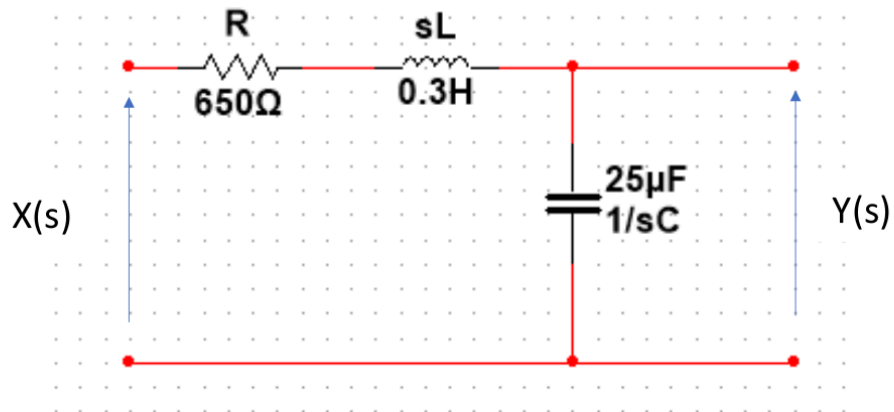


FIGURE 13 - MULTISIM CIRCUIT MODEL OF THE SECOND ORDER SERIES RLC ELECTRICAL SYSTEM FOR AN OVERDAMPED RESPONSE IN TASK 5

Transfer Function equations for an Overdamped Response

For an overdamped response you require the damping factor to be greater than 1.

Damping factor (ζ):

$$\zeta = \frac{R}{2} \sqrt{\frac{C}{L}} = \frac{650}{2} \sqrt{\frac{25 \times 10^{-6}}{0.3}} = 2.96683052$$

Natural frequency of oscillation (w_n):

$$w_n = \frac{1}{\sqrt{LC}} = \frac{1}{\sqrt{0.3 \times 25 \times 10^{-6}}} = 365.1483717 \text{ rad/s}$$

DC gain (k) = 1

Transfer Function:

$$H(s) = \frac{k w_n^2}{s^2 + (2)(\zeta)(w_n)(s) + w_n^2}$$

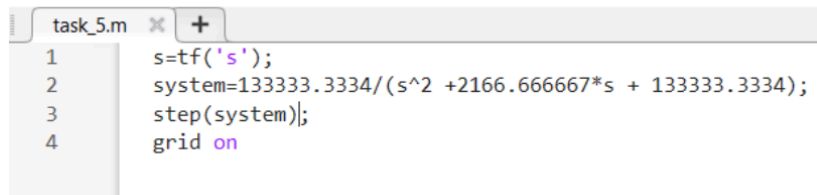
$$H(s) = \frac{365.1483717^2}{s^2 + (2)(2.96683052)(365.1483717)(s) + 365.1483717^2}$$

Final Transfer Function:

$$H(s) = \frac{133333.3334}{s^2 + (2166.666667)(s) + 133333.3334}$$

Matlab Code

In order to obtain accurate graphical results from the Matlab of the damping response, I must input the transfer function accurately into the Matlab code which will then provide me with a graph of amplitude against time. Below you can see the code I inputted into Matlab:



```
task_5.m  X  +
1      s=tf('s');
2      system=133333.3334/(s^2 +2166.666667*s + 133333.3334);
3      step(system);
4      grid on
```

FIGURE 14 - MATLAB CODE FOR OVERDAMPED RESPONSE IN TASK 5

Matlab Graph

The graph for this code is shown below:

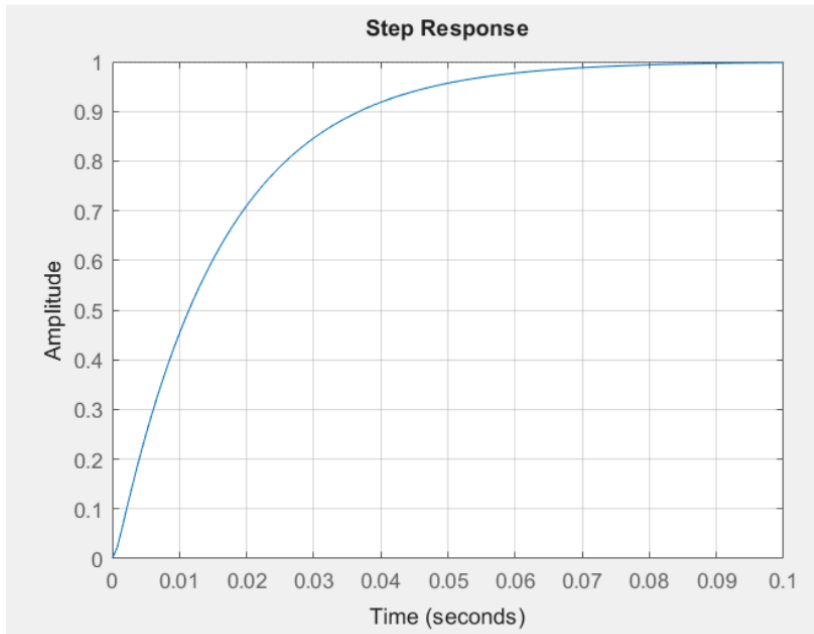


FIGURE 15 - UNDERDAMPED RESPONSE IN MATLAB FOR TASK 5

Underdamped

I chose a resistance of 54Ω and an inductance of 0.3H and a capacitance of $25\mu\text{F}$.

Multisim Circuit

The following circuit was constructed in NI Multisim and provides a visual interpretation of the circuit.

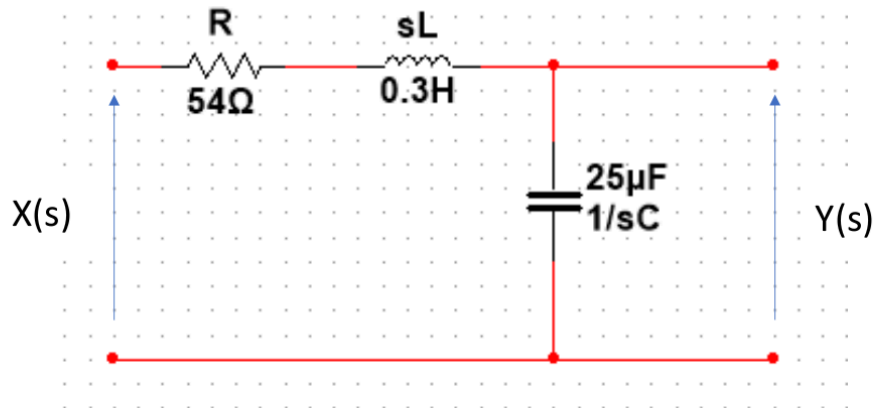


FIGURE 16 - MULTISIM CIRCUIT MODEL OF THE SECOND ORDER SERIES RLC ELECTRICAL SYSTEM FOR AN UNDERDAMPED RESPONSE IN TASK 5

Transfer Function equations for an Underdamped Response

For an underdamped response you require the damping factor to be less than 1.

Damping factor (ζ):

$$\zeta = \frac{R}{2} \sqrt{\frac{C}{L}} = \frac{54}{2} \sqrt{\frac{25 \times 10^{-6}}{0.3}} = 0.2464751509$$

Natural frequency of oscillation (w_n):

$$w_n = \frac{1}{\sqrt{LC}} = \frac{1}{\sqrt{0.3 \times 25 \times 10^{-6}}} = 365.1483717 \text{ rad/s}$$

DC gain (k) = 1

Transfer Function:

$$H(s) = \frac{k w_n^2}{s^2 + (2)(\zeta)(w_n)(s) + w_n^2}$$

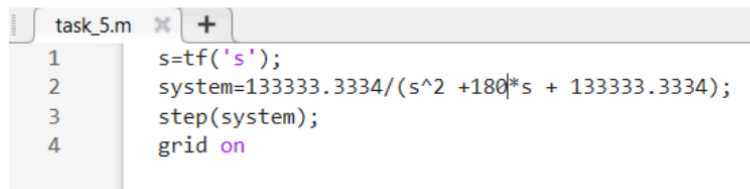
$$H(s) = \frac{365.1483717^2}{s^2 + (2)(0.246475109)(365.1483717)(s) + 365.1483717^2}$$

Final transfer function:

$$H(s) = \frac{133333.3334}{s^2 + (180)(s) + 133333.3334}$$

Matlab Code

In order to obtain accurate graphical results from the Matlab of the damping response, I must input the transfer function accurately into the Matlab code which will then provide me with a graph of amplitude against time. Below you can see the code I inputted into Matlab:



```
task_5.m  ✕  +
1      s=tf('s');
2      system=133333.3334/(s^2 +180*s + 133333.3334);
3      step(system);
4      grid on
```

FIGURE 17 - MATLAB CODE FOR UNDERDAMPED RESPONSE FOR TASK 5

Matlab Graph

The graph for this code is shown below:

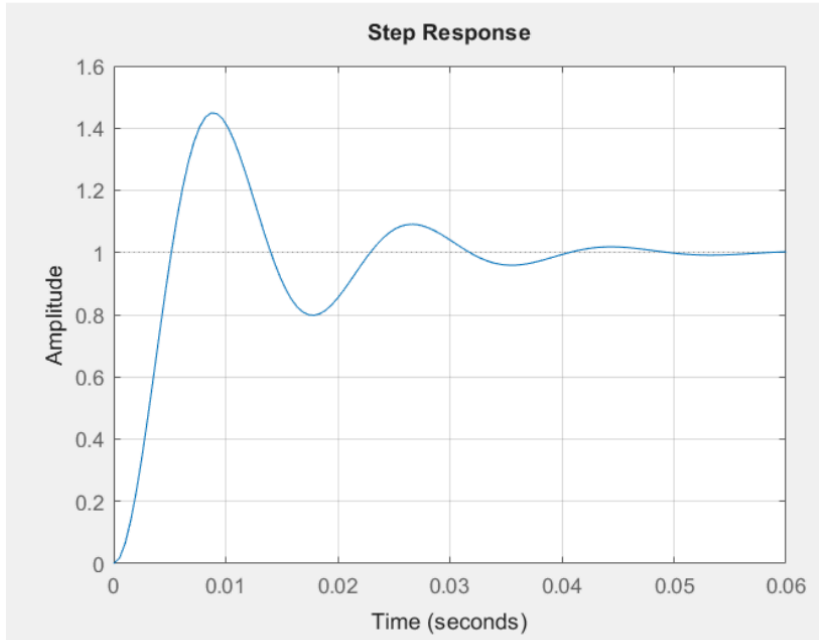


FIGURE 18 - UNDERDAMPED RESPONSE IN MATLAB FOR TASK 5

Critically Damped

I chose a resistance of 219.089023Ω and an inductance of 0.3H and a capacitance of $25\mu\text{F}$.

Multisim Circuit

The following circuit was constructed in NI Multisim and provides a visual interpretation of the circuit.

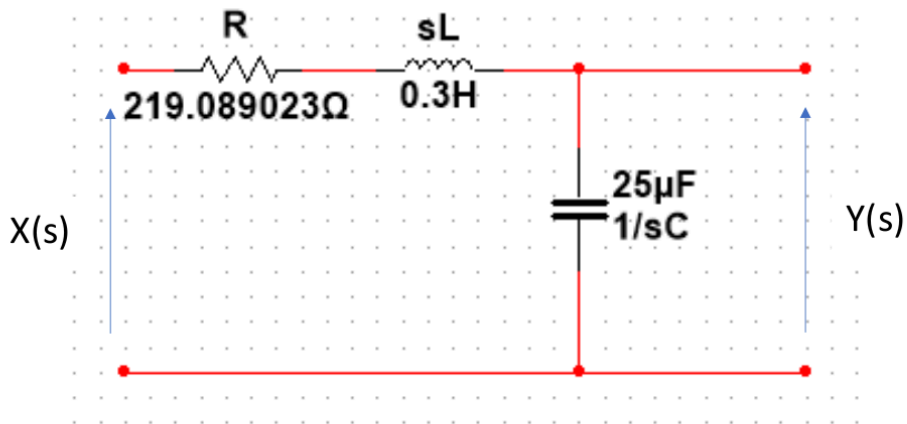


FIGURE 19 - MULTISIM CIRCUIT MODEL OF THE SECOND ORDER SERIES RLC ELECTRICAL SYSTEM FOR A CRITICALLY DAMPED RESPONSE IN TASK 5

Transfer Function equations for a Critically Damped Response

For a critically damped response you require the damping factor to be exactly equal to 1, so I rearranged the damping factor equation to find the resistance required for a damping factor of 1.

Damping factor (ζ):

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

Multiple both sides by the 2:

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

Isolate the R by taking the root to the other side:

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

Input all the values and use a damping factor of 1:

$$R = \frac{2 \times 1}{\sqrt{\frac{25 \times 10^{-6}}{0.3}}} = 219.089023 \Omega$$

Confirming the damping factor of 1:

$$\zeta = \frac{219.089023}{2} \sqrt{\frac{25 \times 10^{-6}}{0.3}} = 1$$

Natural frequency of oscillation (w_n):

$$w_n = \frac{1}{\sqrt{LC}} = \frac{1}{\sqrt{0.3 \times 25 \times 10^{-6}}} = 365.1483717 \text{ rad/s}$$

DC gain (k) = 1

Transfer Function:

$$H(s) = \frac{k w_n^2}{s^2 + (2)(\zeta)(w_n)(s) + w_n^2}$$

$$H(s) = \frac{365.1483717^2}{s^2 + (2)(1)(365.1483717)(s) + 365.1483717^2}$$

Final Transfer Function:

$$H(s) = \frac{133333.3334}{s^2 + (730.2967434)(s) + 133333.3334}$$

Matlab Code

In order to obtain accurate graphical results from the Matlab of the damping response, I must input the transfer function accurately into the Matlab code which will then provide me with a graph of amplitude against time. Below you can see the code I inputted into Matlab:

```
task_5.m x +
1 s=tf('s');
2 system=133333.3334/(s^2 +730.2967434*s + 133333.3334);
3 step(system);
4 grid on
```

FIGURE 20 - MATLAB CODE FOR A CRITICALLY DAMPED RESPONSE IN TASK 5

Matlab Graph

The graph for this code is shown below:

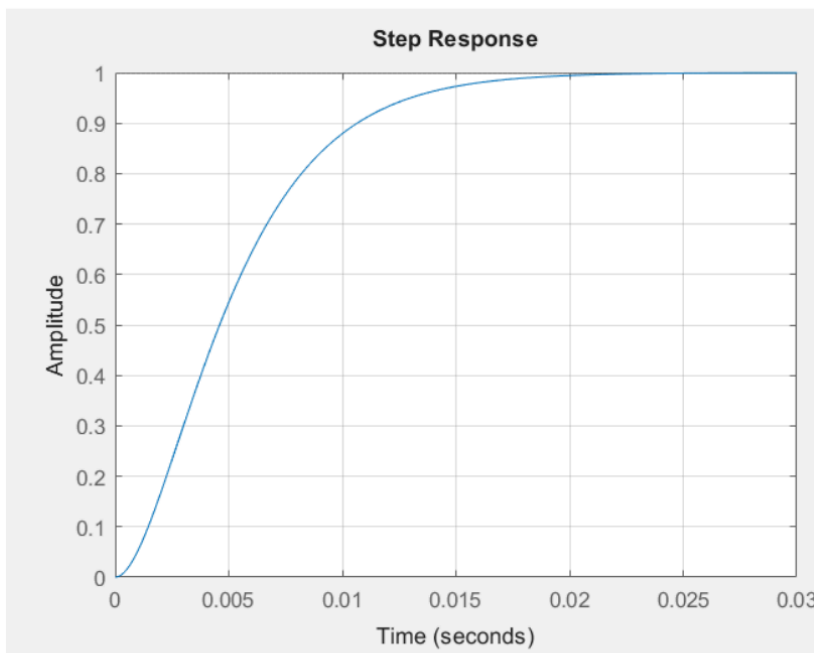


FIGURE 21 - CRITICALLY DAMPED RESPONSE IN MATLAB FOR TASK 5

No Damping

I chose a resistance of 0.00001Ω and an inductance of 0.3H and a capacitance of $25\mu\text{F}$ for the no damping response and then a resistance of 0.0000001Ω to make the response even worse.

Multisim Circuits

The following circuits were constructed in NI Multisim and provide a visual interpretation of the circuits.

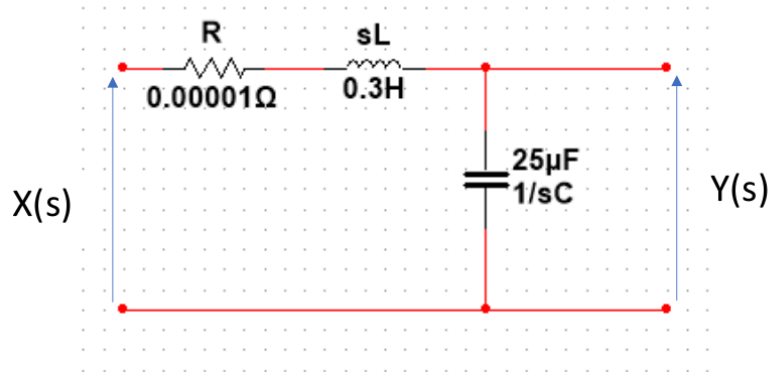


FIGURE 22 - MULTISIM CIRCUIT MODEL OF THE SECOND ORDER SERIES RLC ELECTRICAL SYSTEM FOR A NO DAMPING RESPONSE IN TASK 5

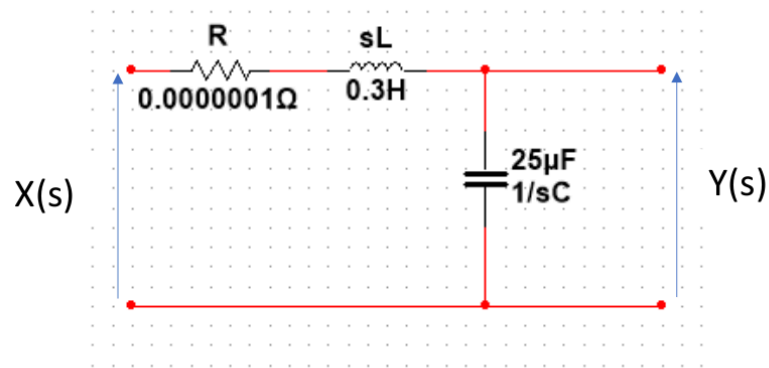


FIGURE 23 - MULTISIM CIRCUIT MODEL OF THE SECOND ORDER SERIES RLC ELECTRICAL SYSTEM FOR AN EVEN WORSE NO DAMPING RESPONSE IN TASK 5

Transfer Function equations for a No Damped Response

For a no damped response you require the damping factor to be a very small value.

Damping factor (ζ):

$$\zeta = \frac{R}{2} \sqrt{\frac{C}{L}} = \frac{0.00001}{2} \sqrt{\frac{25 \times 10^{-6}}{0.3}} = 4.564354646 \times 10^{-8}$$

Natural frequency of oscillation (w_n):

$$w_n = \frac{1}{\sqrt{LC}} = \frac{1}{\sqrt{0.3 \times 25 \times 10^{-6}}} = 365.1483717 \text{ rad/s}$$

DC gain (k) = 1

Transfer Function:

$$H(s) = \frac{k w_n^2}{s^2 + (2)(\zeta)(w_n)(s) + w_n^2}$$

$$H(s) = \frac{365.1483717^2}{s^2 + (2)(4.564354646 \times 10^{-8})(365.1483717)(s) + 365.1483717^2}$$

Final transfer function:

$$H(s) = \frac{133333.3334}{s^2 + (3.333333334 \times 10^{-5})(s) + 133333.3334}$$

Matlab Code

In order to obtain accurate graphical results from the Matlab of the damping response, I must input the transfer function accurately into the Matlab code which will then provide me with a graph of amplitude against time. Below you can see the code I inputted into Matlab:

```
task_5.m  x  +
1      s=tf('s');
2      system=133333.3334/(s^2 +3.333333334*10^-5*s + 133333.3334);
3      step(system);
4      grid on
```

FIGURE 24 - MATLAB CODE FOR NO DAMPED RESPONSE FOR TASK 5

Matlab Graph

The graph for this code is shown below:

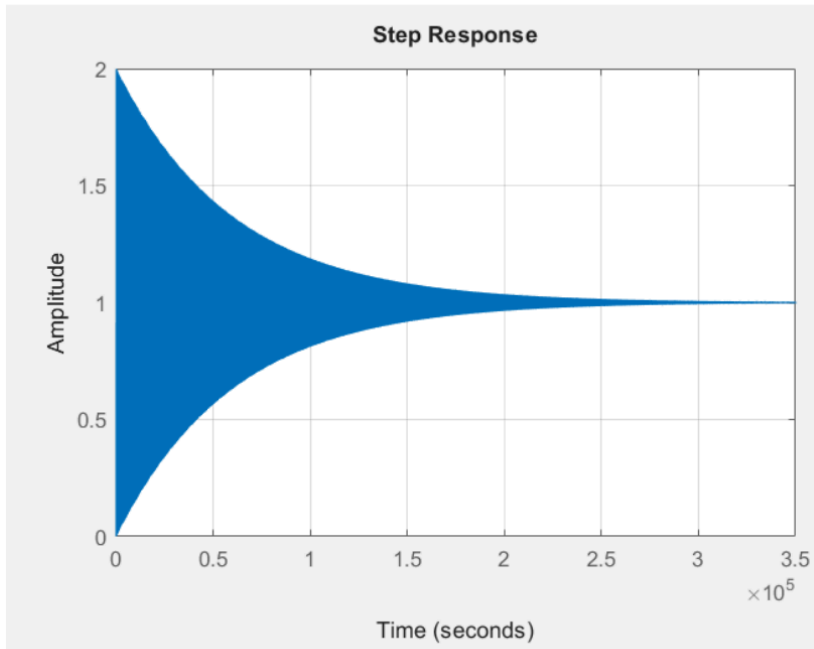


FIGURE 25 - NO DAMPED RESPONSE IN MATLAB FOR TASK 5

Transfer Function equations for an even worse No Damped Response

For a no damped response you require the damping factor to be a very small value.

Damping factor (ζ):

$$\zeta = \frac{R}{2} \sqrt{\frac{C}{L}} = \frac{0.000000001}{2} \sqrt{\frac{25 \times 10^{-6}}{0.3}} = 4.564354646 \times 10^{-12}$$

Natural frequency of oscillation (w_n):

$$w_n = \frac{1}{\sqrt{LC}} = \frac{1}{\sqrt{0.3 \times 25 \times 10^{-6}}} = 365.1483717 \text{ rad/s}$$

DC gain (k) = 1

Transfer Function:

$$H(s) = \frac{k w_n^2}{s^2 + (2)(\zeta)(w_n)(s) + w_n^2}$$

$$H(s) = \frac{365.1483717^2}{s^2 + (2)(4.564354646 \times 10^{-12})(365.1483717)(s) + 365.1483717^2}$$

Final transfer function:

$$H(s) = \frac{133333.3334}{s^2 + (3.333333334 \times 10^{-9})(s) + 133333.3334}$$

Matlab Code

In order to obtain accurate graphical results from the Matlab of the damping response, I must input the transfer function accurately into the Matlab code which will then provide me with a graph of amplitude against time. Below  20 see the code I inputted into Matlab:

```
task_5.m x +
1 s=tf('s');
2 system=133333.3334/(s^2 +3.333333334*10^-9*s + 133333.3334);
3 step(system);
4 grid on
```

FIGURE 26 - MATLAB CODE FOR EVEN WORSE NO DAMPED RESPONSE FOR TASK 5

Matlab Graph

The graph for this code is shown below:

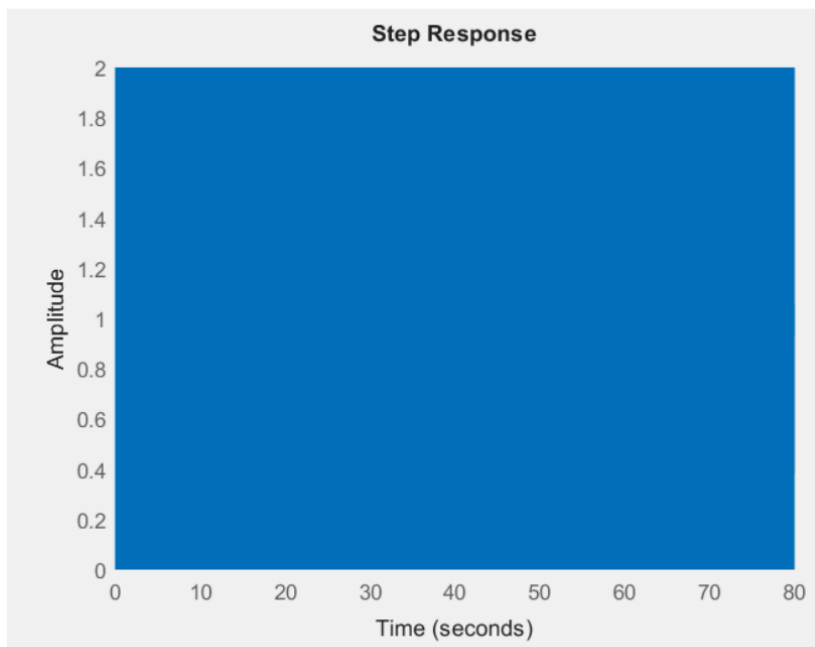


FIGURE 27 - EVEN WORSE NO DAMPED RESPONSE IN MATLAB FOR TASK 5

Comparison of Results

As I have now produced the four different types of responses, I can compare the resistances and damping factors that caused these responses.

Resistance(Ω)	Damping Factor(ζ)	Response
219.089023 Ω	1	Critically Damped
650 Ω	2.96683052	Overdamped
54 Ω	0.2464751509	Underdamped
0.00001 Ω	$4.564354646 \times 10^{-8}$	No Damping
0.000000001 Ω	$4.564354646 \times 10^{-12}$	Even worse No Damping

TABLE 5 - COMPARISON OF DAMPING FACTOR AND RESPONSE DEPENDING ON RESISTANCE IN TASK 5

Analysis

It is evident from the results provided above in Table 5 that varying the resistance and damping factor has a drastic impact on the circuits and the characteristics of their response. The resistance has a direct effect on the damping of the circuit, with a greater value of resistance causing a greater a damping factor value. A value of 1 is the minimum value of damping that can be applied to the circuit with there being no oscillations caused and this response is called critically damped, and this was true for this system as there was no oscillations. For the two circuits which had almost no damping, there were huge oscillations and the systems could not stabilise, while the underdamped circuit caused big oscillations and did eventually settle. The overdamped system  21 gely accurate with minimal oscillations but wasn't quite as perfect as the critically damped system, taking 0.07 seconds longer to settle at the desired amplitude.

Task 6 – Frequency Performance Analysis of a Second Order Series Electrical Circuit

So that I could achieve unique results, I utilised the same component values as Task 5, with an inductance of 0.3H and a capacitance of 25uF. The resistance will have a direct effect on the damping factor so the resistance must be changed to achieve each desired response. I will use resistive values of 650Ω, 219.089023Ω, 120Ω, 54Ω, 5Ω and 1Ω.

The transfer function formula is also the same as Task 5 and is shown below:

$$H(s) = \frac{\frac{1}{LC}}{s^2 + \left(\left(\frac{R}{L}\right) s\right) + \frac{1}{LC}}$$

The Laplace can then be compared to the 2nd Order standard form equation:

$$\frac{k w_n^2}{s^2 + (2)(\zeta)(w_n)(s) + w_n^2}$$

where:

w_n = natural frequency of oscillation

ζ = damping coefficient

k = dc gain

These variables and other important variables can be calculated using the following formulas:

Damping factor (ζ):

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

Natural frequency of oscillation (w_n):

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

Frequency for maximum amplitude (w_m):

$$w_m = \sqrt{\frac{1}{LC} - \frac{1}{2}\left(\frac{R}{L}\right)^2}$$

Quality Factor (Q):

$$Q = \frac{1}{R} \sqrt{\frac{L}{C}}$$

Resonant Frequency (w_o):

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

The values of resonant frequency (w_o) and natural frequency of oscillation (w_n) will not change throughout this task and will stay constant while the alterations of the resistor value will cause changes in the other variables previously discussed.

Overdamped

I chose a resistance of 650Ω and an inductance of 0.3H and a capacitance of $25\mu\text{F}$.

Multisim Circuit

The following circuit was constructed in NI Multisim and provides a visual interpretation of the circuit.

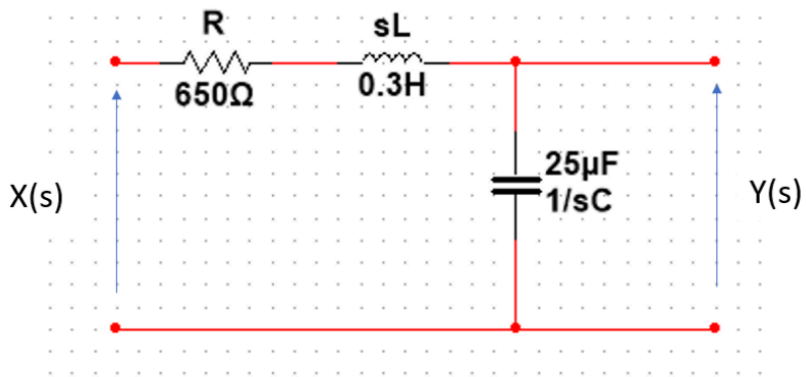


FIGURE 28 - MULTISIM CIRCUIT MODEL OF THE SECOND ORDER SERIES RLC ELECTRICAL SYSTEM FOR AN OVERDAMPED RESPONSE IN TASK 6

Transfer Function equations for an Overdamped Response

For an overdamped response you require the damping factor to be greater than 1.

Damping factor (ζ):

$$\zeta = \frac{R}{2} \sqrt{\frac{C}{L}} = \frac{650}{2} \sqrt{\frac{25 \times 10^{-6}}{0.3}} = 2.96683052$$

Natural frequency of oscillation (ω_n):

$$\omega_n = \frac{1}{\sqrt{LC}} = \frac{1}{\sqrt{0.3 \times 25 \times 10^{-6}}} = 365.1483717 \text{ rad/s}$$

Resonant Frequency (ω_o):

$$\omega_o = \frac{1}{\sqrt{LC}} = \frac{1}{\sqrt{0.3 \times 25 \times 10^{-6}}} = 365.1483717 \text{ rad/s}$$

Quality Factor (Q):

$$Q = \frac{1}{R} \sqrt{\frac{L}{C}} = \frac{1}{650} \sqrt{\frac{0.3}{25 \times 10^{-6}}} = 0.1685300177$$

Frequency for maximum amplitude (ω_m):

$$\omega_m = \sqrt{\frac{1}{LC} - \frac{1}{2} \left(\frac{R}{L} \right)^2} = \sqrt{\frac{1}{0.3 \times 25 \times 10^{-6}} - \frac{1}{2} \left(\frac{650}{0.3} \right)^2} = -1487.914275 \text{ rad/s}$$

Transfer Function:

$$H(s) = \frac{\frac{1}{LC}}{s^2 + \left(\frac{R}{L} \right)(s) + \frac{1}{LC}} = \frac{\frac{1}{0.3 \times 25 \times 10^{-6}}}{s^2 + \left(\frac{650}{0.3} \right)(s) + \frac{1}{0.3 \times 25 \times 10^{-6}}}$$

Final Transfer Function:

$$H(s) = \frac{133333.3}{s^2 + (2166.6)(s) + 133333.3}$$

Critically Damped

I chose a resistance of 219.089023Ω and an inductance of 0.3H and a capacitance of $25\mu\text{F}$.

Multisim Circuit

The following circuit was constructed in NI Multisim and provides a visual interpretation of the circuit.

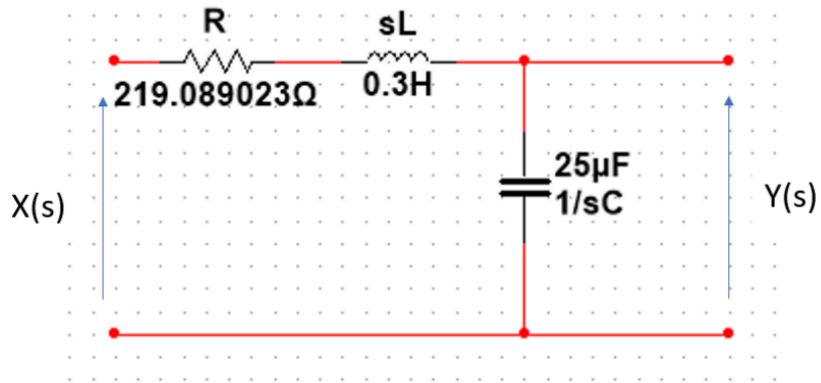


FIGURE 29 - MULTISIM CIRCUIT MODEL OF THE SECOND ORDER SERIES RLC ELECTRICAL SYSTEM FOR A CRITICALLY DAMPED RESPONSE IN TASK 6

Transfer Function equations for a Critically Damped Response

For a critically damped response you require the damping factor to be exactly equal to 1, so I rearranged the damping factor equation to find the resistance required for a damping factor of 1.

Damping factor (ζ):

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

Multiple both sides by the 2:

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

Isolate the R by taking the root to the other side:

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

Input all the values and use a damping factor of 1:

$$R = \frac{2 \times 1}{\sqrt{\frac{25 \times 10^{-6}}{0.3}}} = 219.089023 \Omega$$

Confirming the damping factor of 1:

$$\zeta = \frac{219.089023}{2} \sqrt{\frac{25 \times 10^{-6}}{0.3}} = 1$$

Natural frequency of oscillation (w_n):

$$w_n = \frac{1}{\sqrt{LC}} = \frac{1}{\sqrt{0.3 \times 25 \times 10^{-6}}} = 365.1483717 \text{ rad/s}$$

Resonant frequency (w_o):

$$w_o = \frac{1}{\sqrt{LC}} = \frac{1}{\sqrt{0.3 \times 25 \times 10^{-6}}} = 365.1483717 \text{ rad/s}$$

Quality Factor (Q):

$$Q = \frac{1}{R} \sqrt{\frac{L}{C}} = \frac{1}{219.089023} \sqrt{\frac{0.3}{25 \times 10^{-6}}} = 0.5$$

Frequency for maximum amplitude (w_m):

$$w_m = \sqrt{\frac{1}{LC} - \frac{1}{2} \left(\frac{R}{L} \right)^2} = \sqrt{\frac{1}{0.3 \times 25 \times 10^{-6}} - \frac{1}{2} \left(\frac{219.089023}{0.3} \right)^2} = -365.1483718 \text{ rad/s}$$

Transfer Function:

$$H(s) = \frac{\frac{1}{LC}}{s^2 + (\frac{R}{L})(s) + \frac{1}{LC}} = \frac{\frac{1}{0.3 \times 25 \times 10^{-6}}}{s^2 + (\frac{219.089023}{0.3})(s) + \frac{1}{0.3 \times 25 \times 10^{-6}}}$$

Final Transfer Function:

$$H(s) = \frac{133333.3}{s^2 + (730.2967433)(s) + 133333.3}$$

Underdamped 1

I chose a resistance of 120Ω and an inductance of 0.3H and a capacitance of $25\mu\text{F}$.

Multisim Circuit

The following circuit was constructed in NI Multisim and provides a visual interpretation of the circuit.

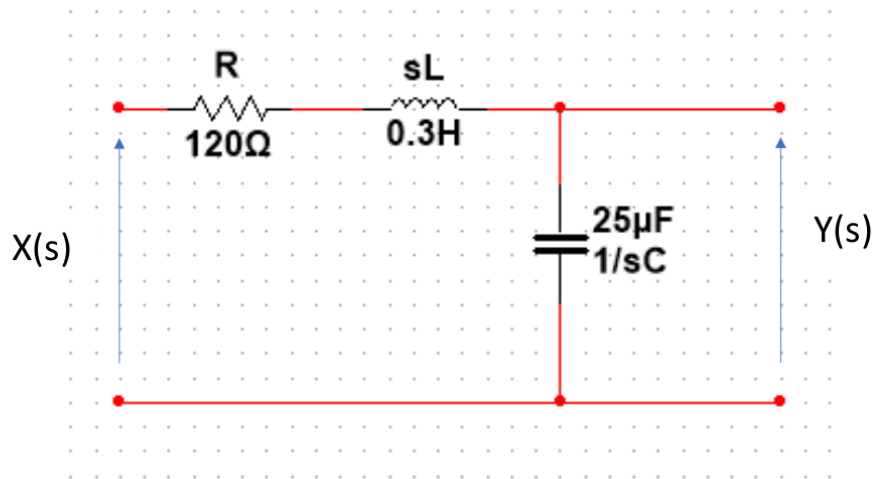


FIGURE 30 - MULTISIM CIRCUIT MODEL OF THE SECOND ORDER SERIES RLC ELECTRICAL SYSTEM FOR THE UNDERDAMPED 1 RESPONSE IN TASK 6

Transfer Function equations for an Underdamped Response

For an underdamped response you require the damping factor to be less than 1.

Damping factor (ζ):

$$\zeta = \frac{R}{2} \sqrt{\frac{C}{L}} = \frac{120}{2} \sqrt{\frac{25 \times 10^{-6}}{0.3}} = 0.5477225575$$

Natural frequency of oscillation (ω_n):

$$\omega_n = \frac{1}{\sqrt{LC}} = \frac{1}{\sqrt{0.3 \times 25 \times 10^{-6}}} = 365.1483717 \text{ rad/s}$$

Resonant frequency (ω_o):

$$\omega_o = \frac{1}{\sqrt{LC}} = \frac{1}{\sqrt{0.3 \times 25 \times 10^{-6}}} = 365.1483717 \text{ rad/s}$$

Quality Factor (Q):

$$Q = \frac{1}{R} \sqrt{\frac{L}{C}} = \frac{1}{120} \sqrt{\frac{0.3}{25 \times 10^{-6}}} = 0.9128709292$$

Frequency for maximum amplitude (ω_m):

$$\omega_m = \sqrt{\frac{1}{LC} - \frac{1}{2} \left(\frac{R}{L} \right)^2} = \sqrt{\frac{1}{0.3 \times 25 \times 10^{-6}} - \frac{1}{2} \left(\frac{120}{0.3} \right)^2} = 230.9401077 \text{ rad/s}$$

Transfer Function:

$$H(s) = \frac{\frac{1}{LC}}{s^2 + \left(\frac{R}{L} \right)(s) + \frac{1}{LC}} = \frac{\frac{1}{0.3 \times 25 \times 10^{-6}}}{s^2 + \left(\frac{120}{0.3} \right)(s) + \frac{1}{0.3 \times 25 \times 10^{-6}}}$$

Final Transfer Function:

$$H(s) = \frac{133333.3}{s^2 + (400)(s) + 133333.3}$$

Underdamped 2

I chose a resistance of 54Ω and an inductance of 0.3H and a capacitance of $25\mu\text{F}$.

Multisim Circuit

The following circuit was constructed in NI Multisim and provides a visual interpretation of the circuit.

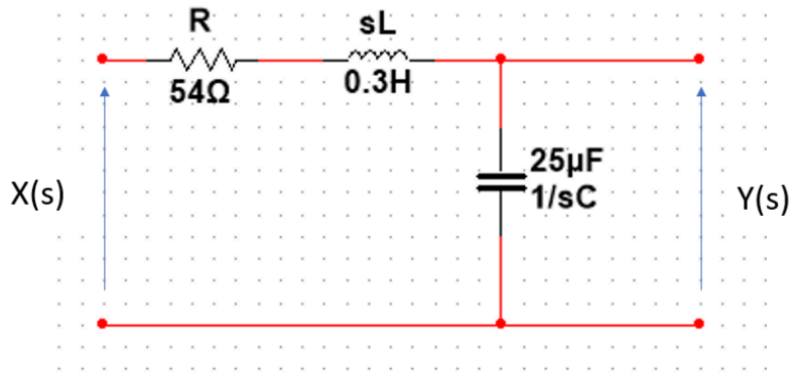


FIGURE 31 - MULTISIM CIRCUIT MODEL OF THE SECOND ORDER SERIES RLC ELECTRICAL SYSTEM FOR THE UNDERDAMPED 2 RESPONSE IN TASK 6

Transfer Function equations for an Underdamped Response

For an underdamped response you require the damping factor to be less than 1.

Damping factor (ζ):

$$\zeta = \frac{R}{2} \sqrt{\frac{C}{L}} = \frac{54}{2} \sqrt{\frac{25 \times 10^{-6}}{0.3}} = 0.2464751509$$

Natural frequency of oscillation (ω_n):

$$\omega_n = \frac{1}{\sqrt{LC}} = \frac{1}{\sqrt{0.3 \times 25 \times 10^{-6}}} = 365.1483717 \text{ rad/s}$$

Resonant frequency (ω_o):

$$\omega_o = \frac{1}{\sqrt{LC}} = \frac{1}{\sqrt{0.3 \times 25 \times 10^{-6}}} = 365.1483717 \text{ rad/s}$$

Quality Factor (Q):

$$Q = \frac{1}{R} \sqrt{\frac{L}{C}} = \frac{1}{54} \sqrt{\frac{0.3}{25 \times 10^{-6}}} = 2.028602065$$

Frequency for maximum amplitude (ω_m):

$$\omega_m = \sqrt{\frac{1}{LC} - \frac{1}{2} \left(\frac{R}{L} \right)^2} = \sqrt{\frac{1}{0.3 \times 25 \times 10^{-6}} - \frac{1}{2} \left(\frac{54}{0.3} \right)^2} = 342.2474738 \text{ rad/s}$$

Transfer Function:

$$H(s) = \frac{\frac{1}{LC}}{s^2 + \left(\frac{R}{L} \right)(s) + \frac{1}{LC}} = \frac{\frac{1}{0.3 \times 25 \times 10^{-6}}}{s^2 + \left(\frac{54}{0.3} \right)(s) + \frac{1}{0.3 \times 25 \times 10^{-6}}}$$

Final Transfer Function:

$$H(s) = \frac{133333.3}{s^2 + (180)(s) + 133333.3}$$

Underdamped 3

I chose a resistance of 5Ω and an inductance of 0.3H and a capacitance of $25\mu\text{F}$.

Multisim Circuit

The following circuit was constructed in NI Multisim and provides a visual interpretation of the circuit.

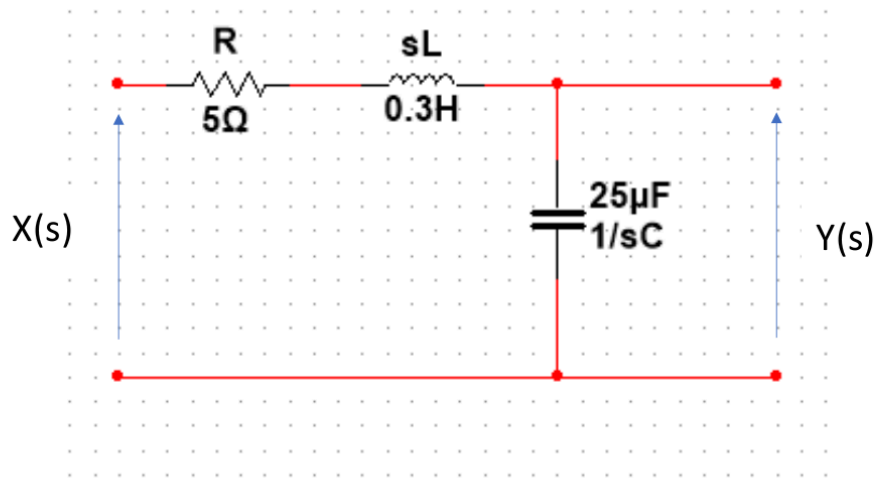


FIGURE 32 - MULTISIM CIRCUIT MODEL OF THE SECOND ORDER SERIES RLC ELECTRICAL SYSTEM FOR THE UNDERDAMPED 3 RESPONSE IN TASK 6

Transfer Function equations for an Underdamped Response

For an underdamped response you require the damping factor to be less than 1.

Damping factor (ζ):

$$\zeta = \frac{R}{2} \sqrt{\frac{C}{L}} = \frac{5}{2} \sqrt{\frac{25 \times 10^{-6}}{0.3}} = 0.02282177323$$

Natural frequency of oscillation (ω_n):

$$\omega_n = \frac{1}{\sqrt{LC}} = \frac{1}{\sqrt{0.3 \times 25 \times 10^{-6}}} = 365.1483717 \text{ rad/s}$$

Resonant frequency (ω_o):

$$\omega_o = \frac{1}{\sqrt{LC}} = \frac{1}{\sqrt{0.3 \times 25 \times 10^{-6}}} = 365.1483717 \text{ rad/s}$$

Quality Factor (Q):

$$Q = \frac{1}{R} \sqrt{\frac{L}{C}} = \frac{1}{5} \sqrt{\frac{0.3}{25 \times 10^{-6}}} = 21.9089023$$

Frequency for maximum amplitude (ω_m):

$$\omega_m = \sqrt{\frac{1}{LC} - \frac{1}{2} \left(\frac{R}{L} \right)^2} = \sqrt{\frac{1}{0.3 \times 25 \times 10^{-6}} - \frac{1}{2} \left(\frac{5}{0.3} \right)^2} = 364.9581407 \text{ rad/s}$$

Transfer Function:

$$H(s) = \frac{\frac{1}{LC}}{s^2 + \left(\frac{R}{L} \right)s + \frac{1}{LC}} = \frac{\frac{1}{0.3 \times 25 \times 10^{-6}}}{s^2 + \left(\frac{5}{0.3} \right)s + \frac{1}{0.3 \times 25 \times 10^{-6}}}$$

Final Transfer Function:

$$H(s) = \frac{133333.3}{s^2 + (16.6)s + 133333.3}$$

Underdamped 4

I chose a resistance of 1Ω and an inductance of 0.3H and a capacitance of $25\mu\text{F}$.

Multisim Circuit

The following circuit was constructed in NI Multisim and provides a visual interpretation of the circuit.

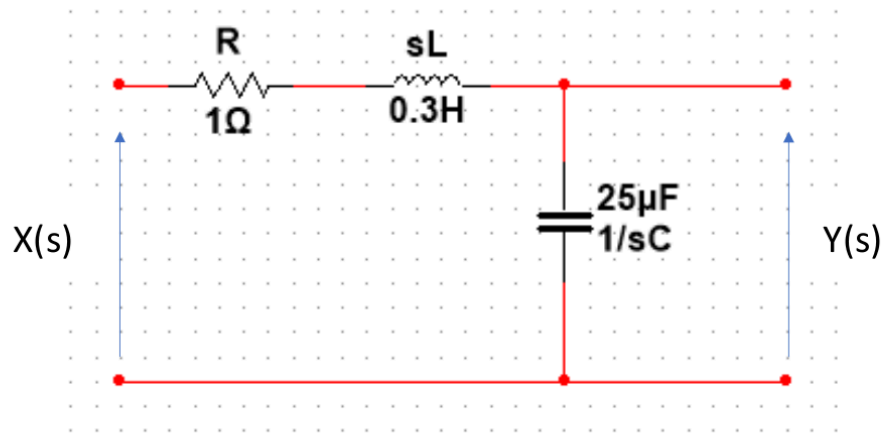


FIGURE 33 - MULTISIM CIRCUIT MODEL OF THE SECOND ORDER SERIES RLC ELECTRICAL SYSTEM FOR THE UNDERDAMPED 4 RESPONSE IN TASK 6

Transfer Function equations for an Underdamped Response

For an underdamped response you require the damping factor to be less than 1.

Damping factor (ζ):

$$\zeta = \frac{R}{2} \sqrt{\frac{C}{L}} = \frac{1}{2} \sqrt{\frac{25 \times 10^{-6}}{0.3}} = 4.564354646 \times 10^{-3}$$

Natural frequency of oscillation (ω_n):

$$\omega_n = \frac{1}{\sqrt{LC}} = \frac{1}{\sqrt{0.3 \times 25 \times 10^{-6}}} = 365.1483717 \text{ rad/s}$$

Resonant frequency (ω_o):

$$\omega_o = \frac{1}{\sqrt{LC}} = \frac{1}{\sqrt{0.3 \times 25 \times 10^{-6}}} = 365.1483717 \text{ rad/s}$$

Quality Factor (Q):

$$Q = \frac{1}{R} \sqrt{\frac{L}{C}} = \frac{1}{1} \sqrt{\frac{0.3}{25 \times 10^{-6}}} = 109.5445155$$

Frequency for maximum amplitude (ω_m):

$$\omega_m = \sqrt{\frac{1}{LC} - \frac{1}{2} \left(\frac{R}{L}\right)^2} = \sqrt{\frac{1}{0.3 \times 25 \times 10^{-6}} - \frac{1}{2} \left(\frac{1}{0.3}\right)^2} = 365.1407643 \text{ rad/s}$$

Transfer Function:

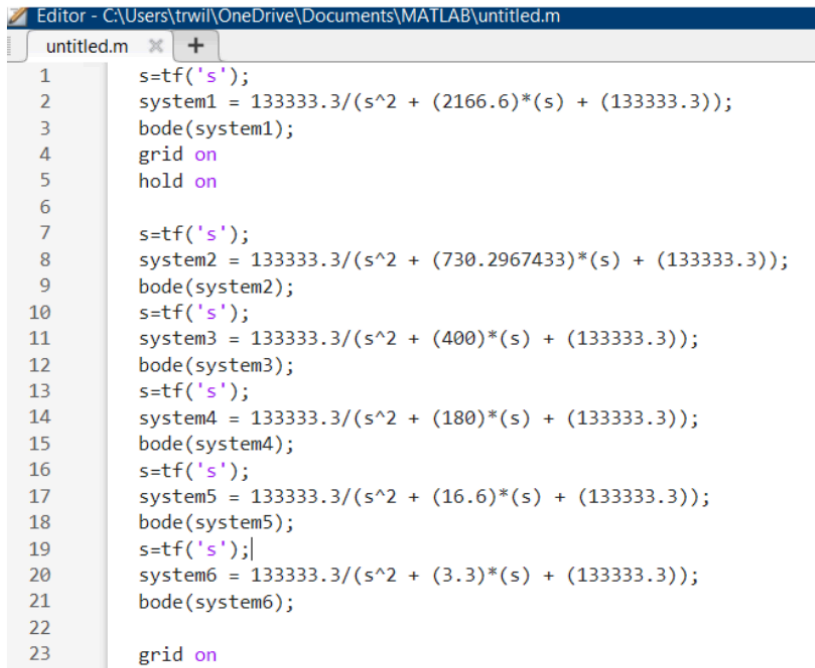
$$H(s) = \frac{\frac{1}{LC}}{s^2 + \left(\frac{R}{L}\right)s + \frac{1}{LC}} = \frac{\frac{1}{0.3 \times 25 \times 10^{-6}}}{s^2 + \left(\frac{1}{0.3}\right)s + \frac{1}{0.3 \times 25 \times 10^{-6}}}$$

Final Transfer Function:

$$H(s) = \frac{133333.3}{s^2 + (3.33)(s) + 133333.3}$$

Matlab Code

In order to obtain accurate graphical results from the Matlab of the responses, I must input the transfer functions accurately into the Matlab code which will then provide me with graphs of magnitude against frequency and phase angle against frequency. Below you can see the code I inputted into Matlab:



```
Editor - C:\Users\trwil\OneDrive\Documents\MATLAB\untitled.m
untitled.m  x  +
1      s=tf('s');
2      system1 = 133333.3/(s^2 + (2166.6)*(s) + (133333.3));
3      bode(system1);
4      grid on
5      hold on
6
7      s=tf('s');
8      system2 = 133333.3/(s^2 + (730.2967433)*(s) + (133333.3));
9      bode(system2);
10     s=tf('s');
11     system3 = 133333.3/(s^2 + (400)*(s) + (133333.3));
12     bode(system3);
13     s=tf('s');
14     system4 = 133333.3/(s^2 + (180)*(s) + (133333.3));
15     bode(system4);
16     s=tf('s');
17     system5 = 133333.3/(s^2 + (16.6)*(s) + (133333.3));
18     bode(system5);
19     s=tf('s');|
20     system6 = 133333.3/(s^2 + (3.3)*(s) + (133333.3));
21     bode(system6);
22
23     grid on
```

FIGURE 34 - MATLAB CODE FOR THE RESPONSES IN TASK 6

Matlab Graph

The graphs for this code is shown below:

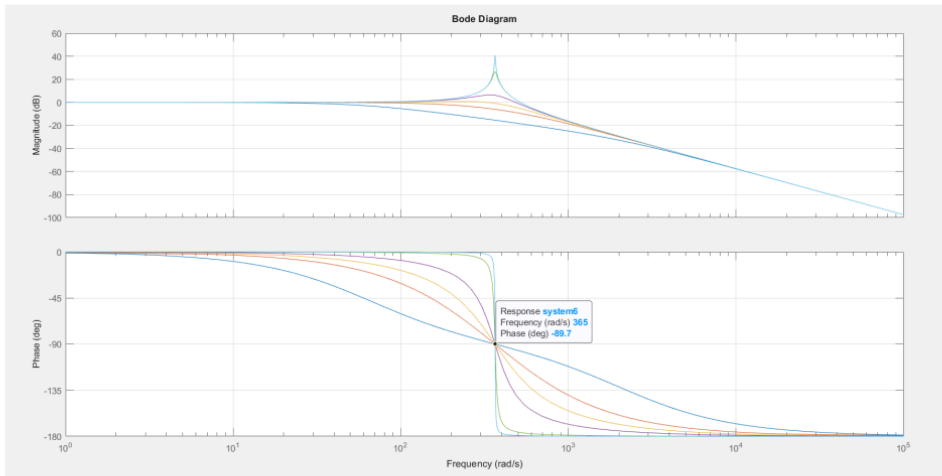


FIGURE 35 - BODE DIAGRAM FOR TASK 6

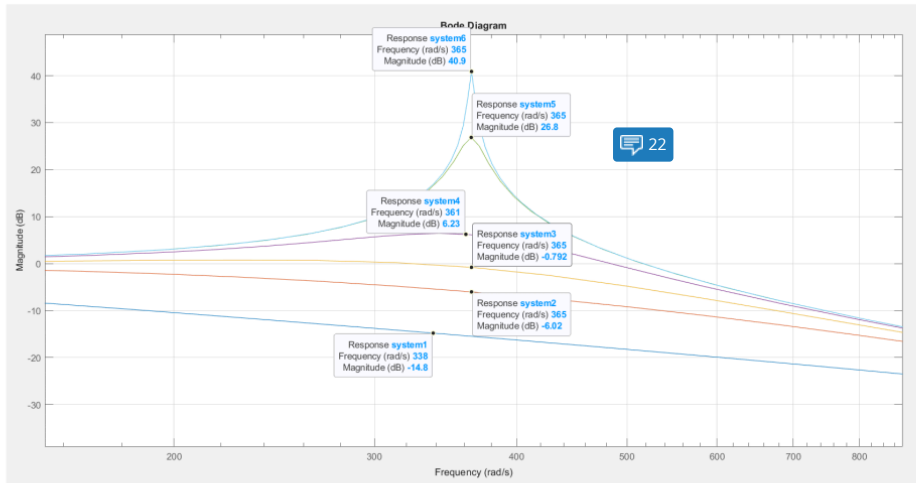


FIGURE 36 - ZOOMED IN VIEW OF THE BODE DIAGRAM FOR TASK 6

Comparison of Results

As I have now produced the six different types of responses, I can compare the quality factor (Q), the damping factor (ζ) and the frequency for maximum amplitude (ω_m) and the different types of responses.

Response	Quality Factor (Q)	Damping Factor (ζ)	Frequency for Maximum Amplitude (ω_m)
Critically Damped	0.5	1	365.1483718 rad/s
Overdamped	0.1685300177	2.96683052	1487.914275 rad/s
Underdamped 1	0.9128709292	0.5477225575	230.9401077 rad/s
Underdamped 2	2.028602065	0.2464751509	342.2474738 rad/s
Underdamped 3	21.9089023	0.02282177323	364.9581407 rad/s
Underdamped 4	109.5445155	$4.564354646 \times 10^{-3}$	365.1407643 rad/s

TABLE 6 - COMPARISON OF QUALITY FACTOR, DAMPING FACTOR AND FREQUENCY FOR MAXIMUM AMPLITUDE DEPENDING ON THE RESPONSE IN TASK 6

Analysis

In the table above the quality factor (Q), the damping factor (ζ) and the frequency for maximum amplitude (ω_m) are compared to each different graphical dampening response when the resistance is varied. The damping factor is a device for displaying how rapidly oscillations decay with a lower value being superior while the quality factor is a ratio of how efficient the electrical circuit is performing with  23 er value being superior. The frequency in the circuits is always trying to reach resonant frequency and when suitable values are chosen the system will more easily reach resonant frequency. The graph of phase shift against frequency shows there is a phase shift of approximately -90° at resonant frequency which is anticipated and supports the theo  24

It is shown in the table above that the lower the quality factor gets and the higher the damping factor gets the higher the frequency of your maximum amplitude will be and the graphical responses on Matlab support this statement. The critically damped response is a slight outlier from this theory as the damping factor is reasonably high and the quality factor is quite low, yet there are no oscillations as it is critically damped defining that resonant frequency is reached more readily causing the value of the quality factor to be meaningless.  25

Task 7 – Frequency Performance Analysis of a First Order Electrical System

26

For this task, I will be carrying out a frequency performance analysis on an active low-pass filter electrical system. So that I could achieve unique results I chose different component values to achieve distinctive results to analyse. For the capacitor I chose a value of 340nF, for R1 a value of 1kΩ and for R2 a value of 20kΩ. As the capacitor is in parallel with R2 it will cause a lagging effect on the phase shift and the inverting amp will affect this phase shift even further.

27

Transfer Function

The transfer function for this electrical system can be shown using the formula:

$$H(s) = -k \left[\frac{wc}{s + wc} \right]$$

Multisim

The following circuit was constructed in NI Multisim and provides a visual interpretation of the circuit.

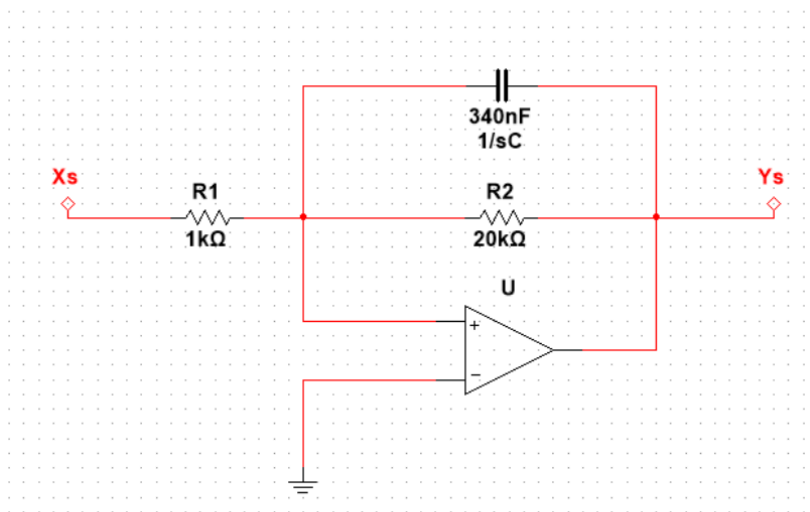


FIGURE 37 - MULTISIM CIRCUIT MODEL FOR THE ACTIVE LOW-PASS FILTER ELECTRICAL SYSTEM

Calculations

Calculations for the active low-pass filter electrical system:

Cutoff frequency (Wc):

$$W_c = \frac{1}{CR_2} = \frac{1}{340 \times 10^{-9} \times 20,000} = 147.0588235294117647 \text{ rad/s}$$

Gain (k):

$$k = \frac{R_2}{R_1} = \frac{20,000}{1000} = 20$$

Transfer Function:

$$H(s) = -k \left[\frac{wc}{s + wc} \right] = -20 \left[\frac{147.0588235294117647}{s + 147.0588235294117647} \right]$$

Decibel gain (G(dB)):

$$G(dB) = 20 \log(k) = 20 \log(20) = 26.02059991 dB$$

-3dB as it is the limit of the filters passband

$$26.02059991 - 3 = 23.02059991 dB$$

Matlab Model

Matlab Code

In order to obtain accurate graphical results from the Matlab model to compare against my calculated time constant, I must input the transfer function accurately into the Matlab code which will then provide me with a graph of magnitude against frequency and a graph of phase angle against frequency. Below you can see the code I inputted into Matlab:

```
s=tf('s');  
system = -20*(147.0588235294117647/(s + 147.0588235294117647));  
bode(system)  
grid on
```

FIGURE 38 - MATLAB CODE FOR TASK 7

Matlab Graph

The graph for this code is shown below:

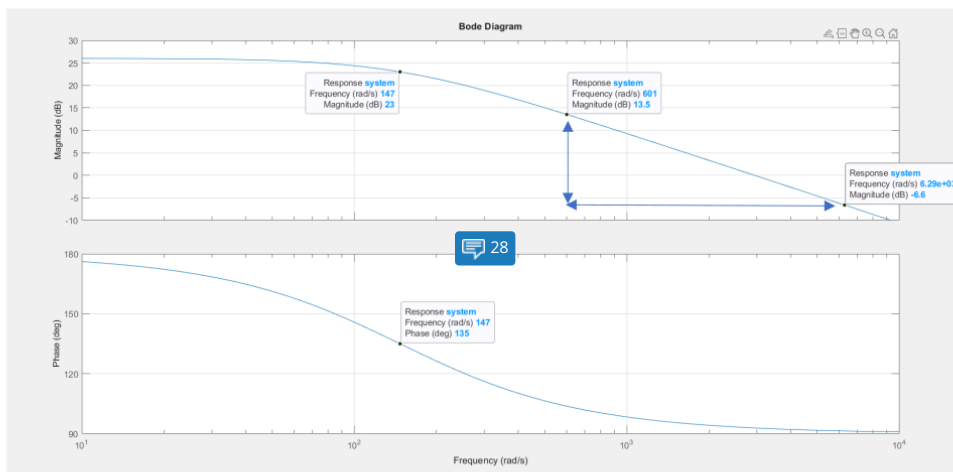


FIGURE 39 - MATLAB RESPONSE FOR TASK 7

Comparison of Results

I can now compare my calculated values of Frequency (rad/s), Magnitude (dB) and Phase Shift (deg) against the values extracted from the Matlab simulation.

	Frequency (rad/s)	Magnitude (dB)	Phase Shift (deg)
Calculated	147.0588235294117647	23.02059991	135
Simulated	147	23	135

TABLE 7 – COMPARISON OF CALCULATED AND SIMULATED VALUES OF FREQUENCY, MAGNITUDE AND PHASE SHIFT IN TASK

7

Analysis

The Matlab simulation graph proves that below the 23dB point, frequencies are allowed to pass with little attenuation as this is an active low-pass filter. This point is also known as the cutoff frequency. As the gain of this filter is so high the magnitude of the cutoff frequency has been scaled to approximately 23dB while the cutoff frequency itself has stayed the same so the -3dB attenuation will still occur at this point in the filter. Using the formula ' $20\log(k)$ ' and taking the -3dB off this value, the magnitude was calculated as 23.02059991dB with a cutoff frequency of 147.0588235294117647 rad/s and data extracted from the simulation shows there was a magnitude of 23dB and a cutoff frequency of 147 rad/s. These values are extremely accurate and show the active low-pass filter is allowing the correct frequencies to pass through the filter. The phase shift of a low pass filter is -45° at the cutoff frequency as the output voltage is lagging at this point, however, the inverting amplifier causes the value to be inverted, adding 180° meaning the final phase shift is 135° and this value can also be seen on the simulation.

Data on the simulation graph shows that on the roll-off of the graph over the course of a decade, the magnitude changes by approximately 20dB and this proves that the filter attenuates the unwanted high frequencies successfully and ensures the transition from the stopband and into the passband will be without delay.



Task 8 - Frequency Performance Analysis of a First Order Electrical System (Continued)

For this task, I will be carrying out a frequency performance analysis on an active high-pass filter electrical system. So that I could achieve unique results I chose different component values to achieve distinctive results to analyse. For the capacitor I chose a value of 34nF, for R1 a value of 1kΩ and for R2 a value of 2kΩ. As the capacitor is in series with R1 and R2 it will cause a leading effect on the phase shift and the inverting amp will affect this phase shift even further.

Transfer Function

The transfer function for this electrical system can be shown using the formula:

$$H(s) = -k \left[\frac{s}{s + wc} \right]$$

Multisim

The following circuit was constructed in NI Multisim and provides a visual interpretation of the circuit.

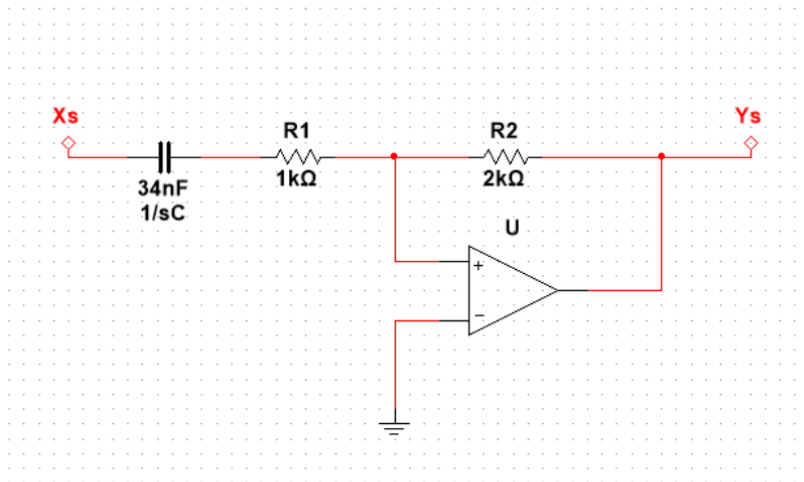


FIGURE 40 - MULTISIM CIRCUIT MODEL FOR THE ACTIVE HIGH-PASS FILTER ELECTRICAL SYSTEM

Calculations

Calculations for the active high-pass filter electrical system:

Cutoff frequency (Wc):

$$W_c = \frac{1}{CR_1} = \frac{1}{34 \times 10^{-9} \times 1000} = 29411.76471 \text{ rad/s}$$

Gain (k):

$$k = \frac{R_2}{R_1} = \frac{2000}{1000} = 2$$

Transfer Function:

$$H(s) = -k \left[\frac{s}{s + w_c} \right] = -20 \left[\frac{s}{s + 29411.76471} \right]$$

Decibel gain (G(dB)):

$$G(dB) = 20 \log(k) = 20 \log(2) = 6.020599913 dB$$

-3dB as it is the limit of the filters passband

$$6.020599913 - 3 = 3.020599913 dB$$

Matlab Model

Matlab Code

In order to obtain accurate graphical results from the Matlab model to compare against my calculated time constant, I must input the transfer function accurately into the Matlab code which will then provide me with a graph of magnitude against frequency and a graph of phase angle against frequency. Below you can see the code I inputted into Matlab:

```
s=tf('s');  
system = -2*(s/(s + 29411.76471));  
bode(system)  
grid on
```

FIGURE 41 - MATLAB CODE FOR TASK 8

Matlab Graph

The graph for this code is shown below:

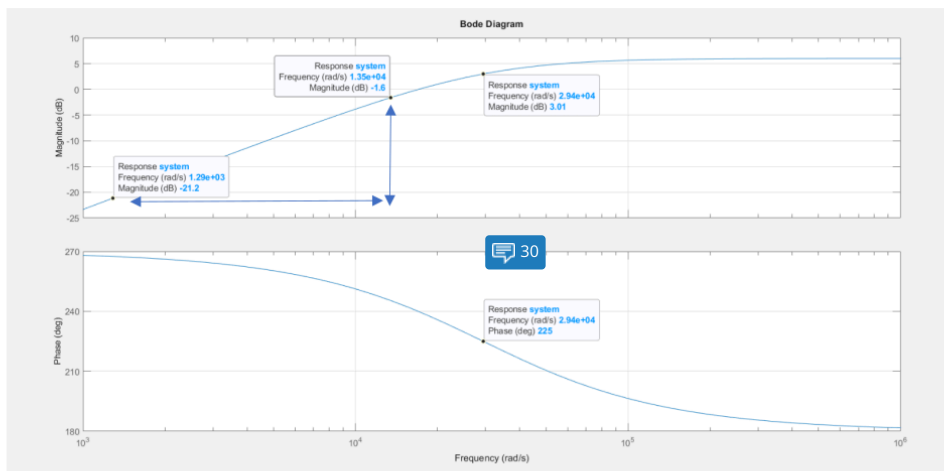


FIGURE 42 - MATLAB RESPONSE FOR TASK 8

Comparison of Results

I can now compare my calculated values of Frequency (rad/s), Magnitude (dB) and Phase Shift (deg) against the values extracted from the Matlab simulation.

	Frequency (rad/s)	Magnitude (dB)	Phase Shift (deg)
Calculated	29411.76471	3.020599913	225
Simulated	29400	3.01	225

TABLE 8 – COMPARISON OF CALCULATED AND SIMULATED VALUES OF FREQUENCY, MAGNITUDE AND PHASE SHIFT IN TASK

8

Analysis

The Matlab simulation graph proves that above the -3dB point, frequencies are allowed to pass with minimal attenuation as this is an active high-pass filter. The -3dB point is also known as the cutoff frequency. Using the formula ' $20\log(k)$ ' and taking the -3dB off this value, the magnitude was calculated as approximately 3.02dB with a cutoff frequency of 29411 rad/s and data extracted from the simulation shows there was a magnitude of 3.01dB and an approximate cutoff frequency of 29400 rad/s. These values are reasonably accurate and show the active high-pass filter is allowing the correct frequencies to pass through the filter. The phase shift of a high pass filter is 45° at the cutoff frequency as the output voltage is leading at this point, however, the inverting amplifier causes the value to be inverted, adding 180° meaning the final phase shift is 225° and this value can also be seen on the simulation.

Data on the simulation graph shows that on the roll-off of the graph over the course of a decade, the magnitude changes by approximately 20dB and this proves that the filter attenuates the undesirable low frequencies effectively and ensures the transition over the passband will be swift.



Task 9 – First Order Control System Analysis

For this task, I will be analysing the performance of an open loop control system model and a closed loop control system model through Simulink models and their responses, argand diagrams and root locus plots. For the closed loop system model, I will vary the value of the gain to show how this effects the stability of the system.

Open Loop Control System Model

Shown below is the open loop control system model, created on Simulink consisting of a step input, a transfer function and a scope. The transfer function for this model is $\frac{4.2}{s-0.68}$.

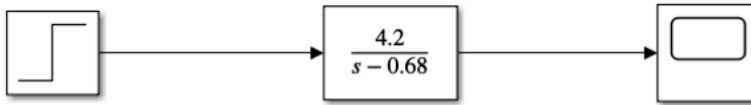


FIGURE 43 - OPEN LOOP CONTROL SYSTEM FOR TASK 9

Step Response

The graph below shows that over time the output of this system grows exponentially as the system is unstable.

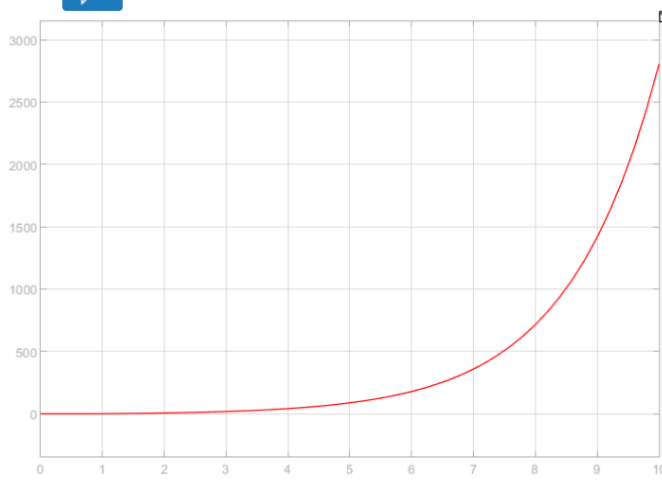


FIGURE 44 - STEP RESPONSE FOR THE OPEN LOOP SYSTEM IN TASK 9

Matlab Model

Matlab Code

Below you can see the code I inputted into Matlab:

```
s = tf('s');  
system = tf(4.2/(s - 0.68));  
pzp = pzplot(system);  
grid on
```

FIGURE 45 - MATLAB CODE FOR THE ARGAND DIAGRAM FOR THE OPEN LOOP SYSTEM IN TASK 9

Argand Diagram

33

Shown below is the Argand diagram or Pole-Zero Map showing the pole is at +0.68 meaning the pole is in the right hand plane which causes the system to be unstable.

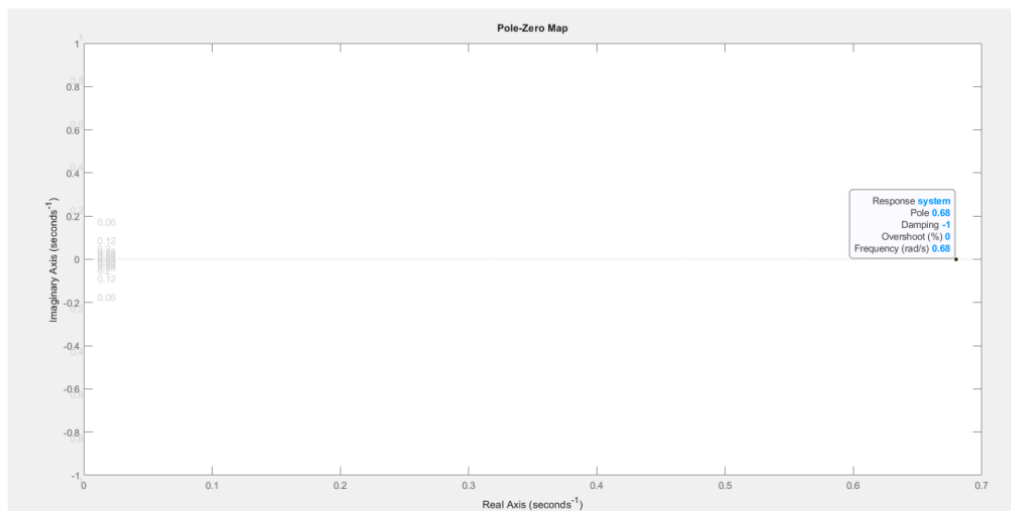


FIGURE 46 - ARGAND DIAGRAM FOR THE OPEN LOOP SYSTEM IN TASK 9

Closed Loop Control System Model

Shown below is the closed loop control system model, created on Simulink consisting of a step input, summing junction, a transfer function, a gain and a scope.

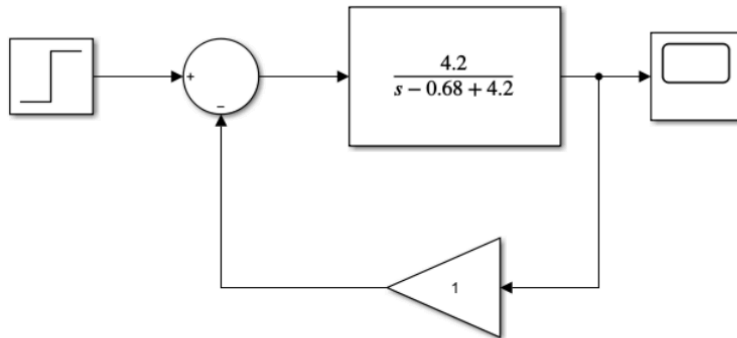


FIGURE 47 - CLOSED LOOP CONTROL SYSTEM MODEL FOR TASK 9

The closed loop transfer function is shown below:

$$\frac{\frac{4.2}{(s - 0.68)}}{1 + \frac{4.2k}{(s - 0.68)}}$$



This can then be simplified down by multiplying the top and bottom by $(s - 0.68)$:

$$\frac{4.2}{s - 0.68 + 4.2k}$$

This can then be simplified again:

$$\frac{4.2}{s + 3.52k}$$



As $k=1$, this means that the pole for this system will be located at -3.52 in the left hand plane, making the system stable.

Step Response

The graph below shows that over time the output of this system levels out as the system is stable. The system levels out around a magnitude of 0.55.

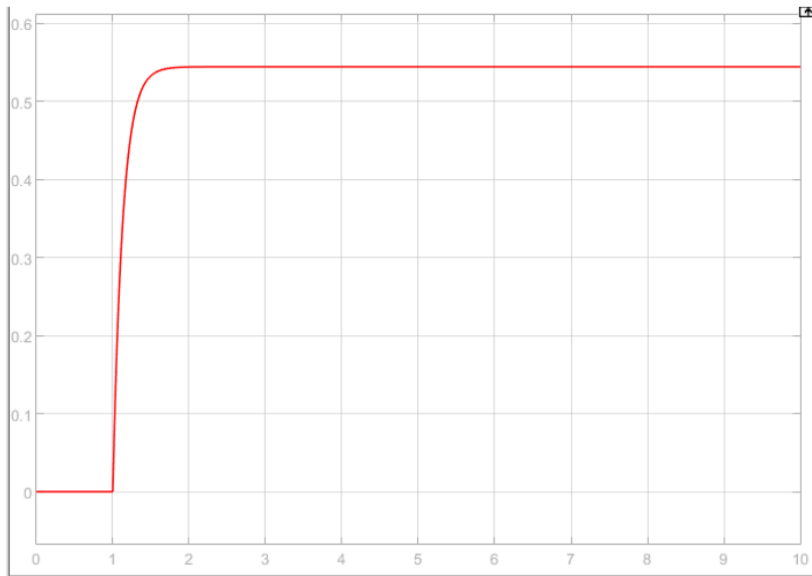


FIGURE 48 - STEP RESPONSE FOR THE CLOSED LOOP SYSTEM IN TASK 9

Matlab Model

Matlab Code

Below you can see the code I inputted into Matlab:

```
s = tf('s');  
system = tf(4.2/(s - 0.68 +4.2j));  
pzp = pzplot(system);  
grid on
```

FIGURE 49 - MATLAB CODE FOR THE ARGAND DIAGRAM FOR
THE CLOSED LOOP SYSTEM IN TASK 9

Argand Diagram

Shown below is the Argand diagram or Pole-Zero Map showing the pole is at -3.52 meaning the pole is in the left hand plane which causes the system to become stable.

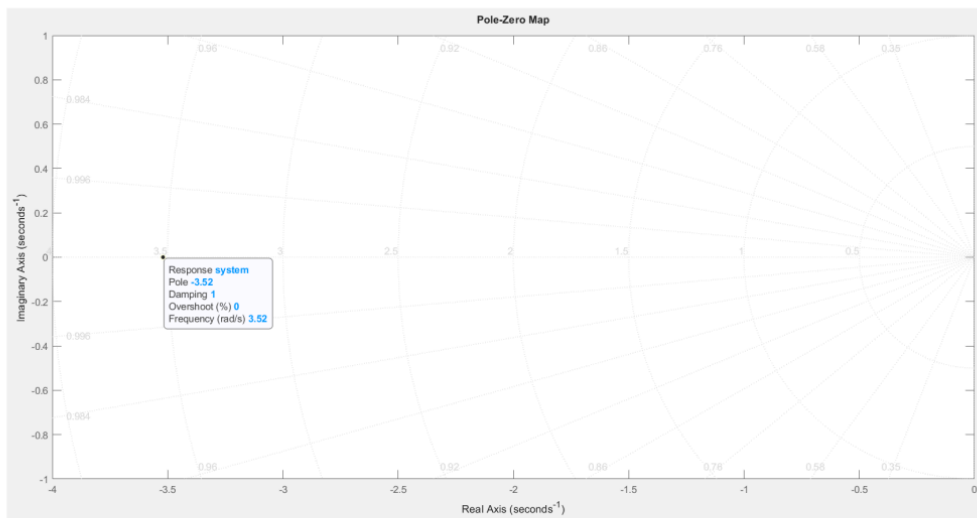


FIGURE 50 -ARGAND DIAGRAM FOR THE CLOSED LOOP SYSTEM IN TASK 9

Minimum Value of K Required for Stability

The minimum value of K needed in the feedback for system stability needs to be determined and this can then be confirmed using Matlab to create a root locus plot.

To calculate the minimum value of k:

$$s - 0.68 + 4.2k = 0$$

Move s to the left:

$$s = 0.68 - 4.2k$$

Set so to equal 0:

$$0 = 0.68 + 4.2k$$

Take 0.68 away from both sides:

$$-0.68 = -4.2k$$

Divide both sides by -4.2:

$$k = \frac{-0.68}{-4.2} = 0.161904$$



Matlab Model

Matlab Code

Below you can see the code I inputted into Matlab:

```
s = tf('s');  
system = tf(4.2/(s - 0.68));  
rlocus(system);  
grid on
```

FIGURE 51 - MATLAB CODE FOR THE ROOT LOCUS PLOT
FOR THE CLOSED LOOP SYSTEM IN TASK 9

Root Locus Plot

Shown below is the Root Locus Plot showing the minimum value of K for a stable circuit is 0.16 as shown through the calculations above. There is another data point at the pole at +0.68 which shows the gain is equal to zero, while there is an additional data point where the gain is 0.229 with a pole of -0.575. These data points prove that the higher the gain, the more stable the system is.

37

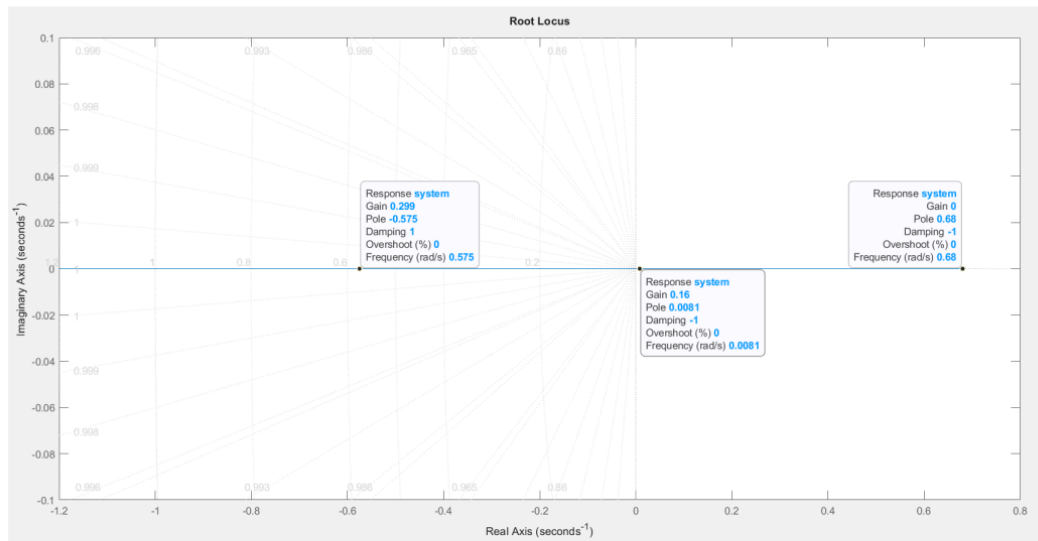


FIGURE 52 - ROOT LOCUS PLOT SHOWING THE MINIMUM VALUE OF K REQUIRED FOR STABLE SYSTEM IN TASK 9

System Response to Different Values of K

The value of k will be given as the minimum value needed for system stability (0.161904), a value greater than the minimum value (0.299) and finally a value lesser than the minimum value required for system stability (0).

Response for the Minimum Value of K for a Stable System

Model

Shown below is the closed loop control system model, created on Simulink consisting of a step input, summing junction, a transfer function, a gain with a value of 0.161904 and a scope.

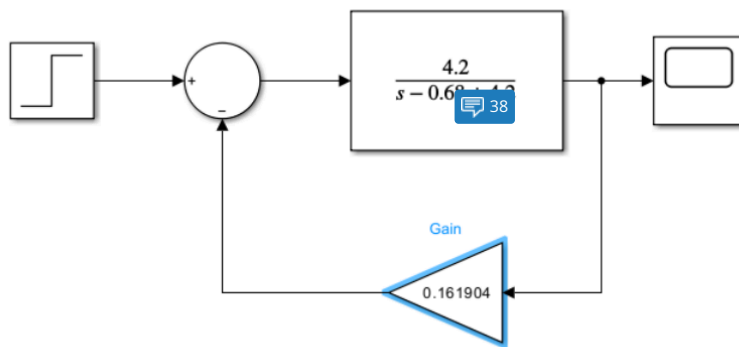


FIGURE 53 - CLOSED LOOP SYSTEM CONTROL MODEL WITH THE MINIMUM K VALUE REQUIRED FOR A STABLE SYSTEM IN TASK 9

Step Response

The graph below shows that over time the output of this system levels out as the system is stable. The system levels out at a magnitude of 1.

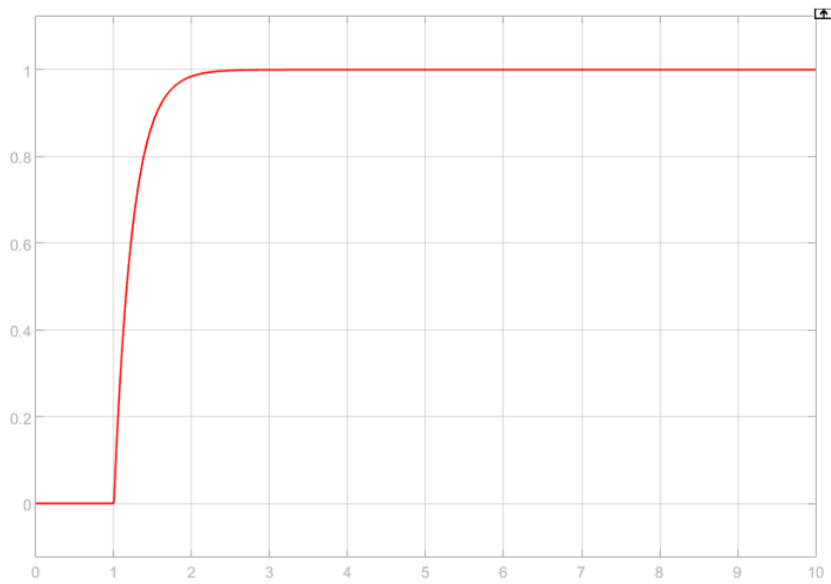


FIGURE 54 - STEP RESPONSE FOR THE MINIMUM VALUE OF K REQUIRED FOR A STABLE SYSTEM IN TASK 9

Response for a Value Greater than the Minimum Value of K for a Stable System

Model

Shown below is the closed loop control system model, created on Simulink consisting of a step input, summing junction, a transfer function, a gain with a value of 0.299 and a scope.

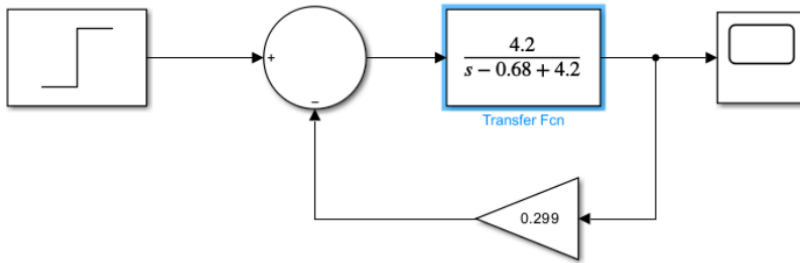


FIGURE 55 - CLOSED LOOP SYSTEM CONTROL MODEL WITH A GAIN GREATER THAN THE MINIMUM K VALUE REQUIRED FOR A STABLE SYSTEM IN TASK 9

Step Response

The graph below shows that over time the output of this system levels out as the system is stable. The system levels out at a magnitude of around 0.875.

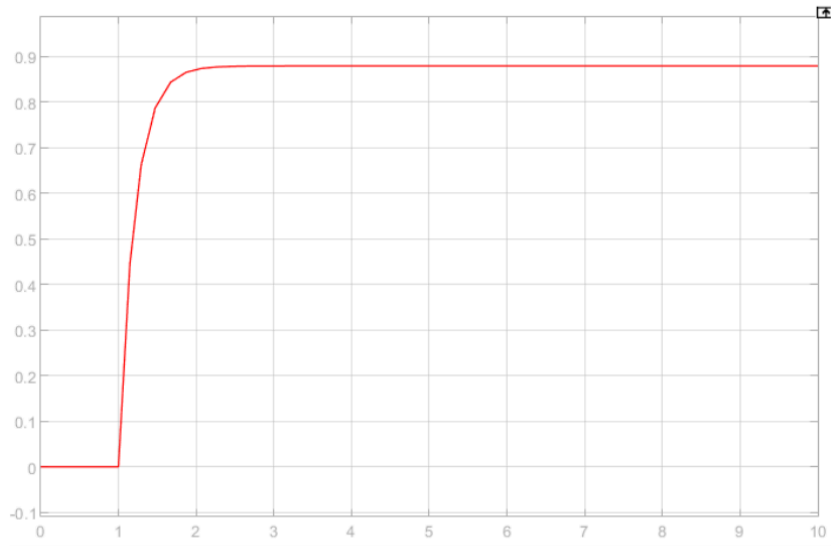


FIGURE 56 - STEP RESPONSE FOR A SYSTEM WITH A GAIN GREATER THAN THE MINIMUM VALUE OF K REQUIRED FOR A STABLE SYSTEM IN TASK 9

Response for a Value Lesser than the Minimum Value of K for a Stable System

Model

Shown below is the closed loop control system model, created on Simulink consisting of a step input, summing junction, a transfer function, a gain with a value of 0 and a scope.

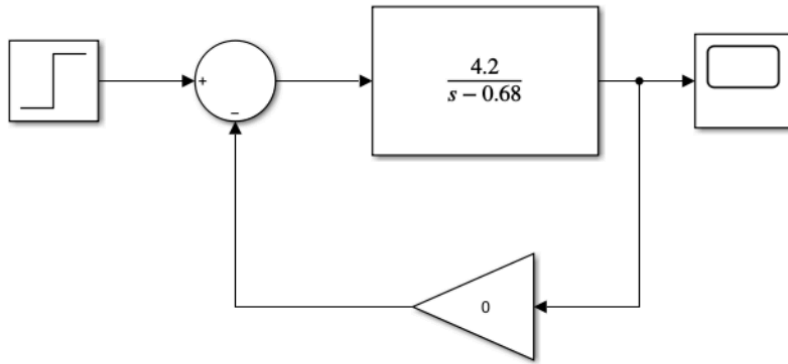


FIGURE 57 - CLOSED LOOP SYSTEM MODEL WITH A GAIN OF 0 IN TASK 9

Step Response

The graph below shows that over time the output of this system grows exponentially as the system is unstable.

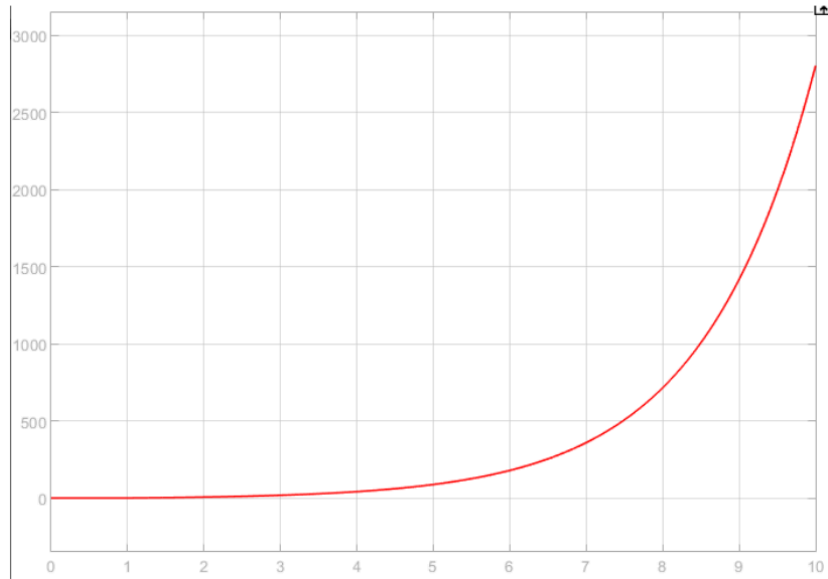


FIGURE 58 - STEP RESPONSE FOR A SYSTEM WITH A GAIN OF 0 IN TASK 9

Comparison of Results

I can now compare the different responses I achieved when altering the value of K in the Simulink simulation. I will compare the values of k and the poles and also compare the responses I achieved.

K	Pole	Response
1	-3.52	Stable
0.299	-0.575	Stable
0.161904	0.0081	Marginally Stable
0	0.68	Stable

TABLE 9 - COMPARISON OF THE VALUES OF THE POLES AND THE RESPONSES ACHIEVED WHEN ALTERING THE VALUE OF K

Analysis

During this task, the open loop and closed loop systems displayed the effect of feedback through their responses and how varying the value of gain in the feedback loop can also affect the response of the system. The responses proved that the gain that was utilised in the closed loop system increased stability.

It is crucial that systems can become stable in order to obtain greater control and the table shown above in the comparison of results section indicates that the gain value is crucial in helping the system become stable, as the higher the gain the more stable the systems seemed to become.

The data labels in the Root locus plot, include the damping value of the system and proves that when the pole is in the left-hand plane it is negative and the system is stable with a damping value of 1, and when the pole is in the right-hand plane the damping is negative 1 and the system becomes unstable.



Task 10 – DC Motor Model Analysis

For this task a DC Motor Model will be designed through Simulink in block diagram form. The electrical and mechanical transfer functions will be used to investigate how the motor behaves in respect to time. Appropriate conclusions will then be made using responses formed from when the motor does and doesn't have load and when the torque is and isn't ignored.



Parameters

For this DC Motor Model all the specific parameters are set out in the brief and will be kept constant throughout this task. All appropriate parameters are shown below:

Armature Voltage (R_a) = 1.5Ω

Armature Inductance (L_a) = $0.2H$

Rotor Inertia (J) = 0.00035 kg/m^2

Friction Coefficient (b) = $3.68 \times 10^{-6} \text{ Nm/rad/s}$

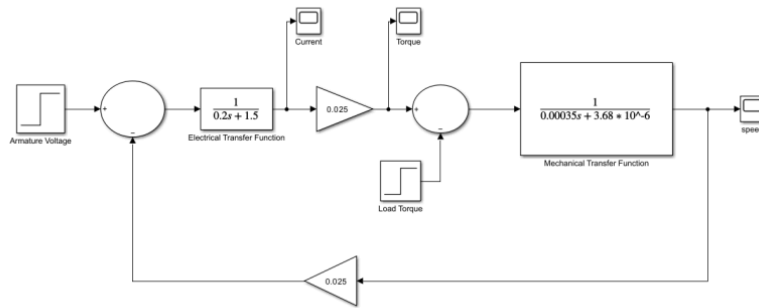


Motor Voltage Constant (k_e) = 0.025 V/rad/s

Motor Torque Constant (k_t) = 0.025 Nm/A

Maximum Armature Voltage ($V_a(\text{max})$) = $24V$

DC Motor Model



43

FIGURE 59 - DC MOTOR MODEL BUILT IN SIMULINK

This DC Motor Model includes a mechanical and an electrical transfer function. Both transfer functions are vital as they predict how the motor will react in terms of mechanical and electrical principles. The electrical transfer function, which is $\frac{1}{0.2s + 1.5}$, examines the change in current against voltage while the mechanical transfer function, which is $\frac{1}{0.00035s + 3.68 \times 10^{-6}}$, examines the rotor speed against the applicable torque.

There are two unit step inputs into the system, one is the armature voltage which will be applied to the motor while the other is a load torque which is simulating a disturbance applied to the system to see how the motor will react. The system utilises negative feedback to stabilise the motor and helps to limit excessive current and torque thus increasing accuracy in response from the motor. The gains in the DC Motor Model, both with a value of 0.025, represent the torque present in the system and the voltage present in the system. Additionally, there are 3 scopes that are reading parameters from the system; they will each measure either current, torque or speed. The graphical responses from these scopes will allow conclusions to be drawn for how the motor reacts and for trends to be established.

Also, there are three different varieties of torque that are present in the DC Motor Model which are load torque (T_L) which is the force the motor is fighting against to move the mechanical system, friction torque (b_w) which attempts to prevent parts from causing friction and the torque required to start the motor ($J \frac{d\omega}{dt}$) which is greater than the running torque as the motor is static for this torque. Combined these torques equate to the complete mechanical torque T_m of the system.

Calculations

In this DC Motor Model there are different definitions of speed that can be calculated and ultimately compared against values collated from the Simulink responses of the system. The parameters specified that the maximum armature voltage is 24V, however, 14V will be used as this is specified in the brief as the armature voltage input. All necessary calculations are shown below:

$$\text{Natural Frequency (wn)} - \omega_n = \sqrt{\frac{Ra \times b}{La \times J} + \frac{Kt \times Ke}{La \times J}} = \sqrt{\frac{1.5 \times 3.68 \times 10^{-6}}{0.2 \times 0.0035} + \frac{0.025 \times 0.025}{0.2 \times 0.0035}}$$

$$= 3.01 \text{ rad/s}$$

$$\text{Damping Factor } (\zeta) - \zeta = \frac{b \times Ra}{J \times La} = \frac{3.68 \times 10^{-6} \times 1.5}{0.0035 \times 0.2} = 1.252$$

$$\text{Motor Torque (stall)} - T_{M(STALL)} = \frac{K_T}{R_a} \times V_a = \frac{0.025}{1.5} \times 14$$

$$T_{M(STALL)} = 0.233 \text{ Nm}$$

$$\text{No Load Speed (without friction)} - \Omega_{NL} = \frac{V_a}{k_E} = \frac{14}{0.025}$$

$$\Omega_{NL} = 560 \text{ rad/s}$$

$$\text{No Load Speed (with friction)} - \Omega = \frac{K_T \times V_a}{(b \times R_a) + (K_T \times K_E)} = \frac{0.025 \times 14}{(3.68 \times 10^{-6} \times 1.5) + (0.025 \times 0.025)}$$

$$\Omega = 555.09 \text{ rad/s}$$

$$\text{Under Load Speed} - \Omega_L = \frac{V_a - (R_a \times T_L)}{\left(\frac{R_a \times b}{R_T}\right) + k_E} = \frac{14 - \left(\frac{1.5 \times 0.05}{0.025}\right)}{\left(\frac{1.5 \times 3.68 \times 10^{-6}}{0.025}\right) + 0.025}$$

$$\Omega_L = 436.14 \text{ rad/s}$$



Stall torque is the maximum starting torque the motor can produce when stationary, if this is exceeded by the load torque the motor will not run correctly.

The other three calculations account for the speeds of the motor depending on if a load is applied and if there is or isn't friction.

The responses can now be gathered from the Simulink design in order to collect values to compare against these calculated values. One graph will be when load is applied and the other will be when load is not applied and will help to demonstrate the effect of adding a load has on the DC Motor Model.

Simulink Responses

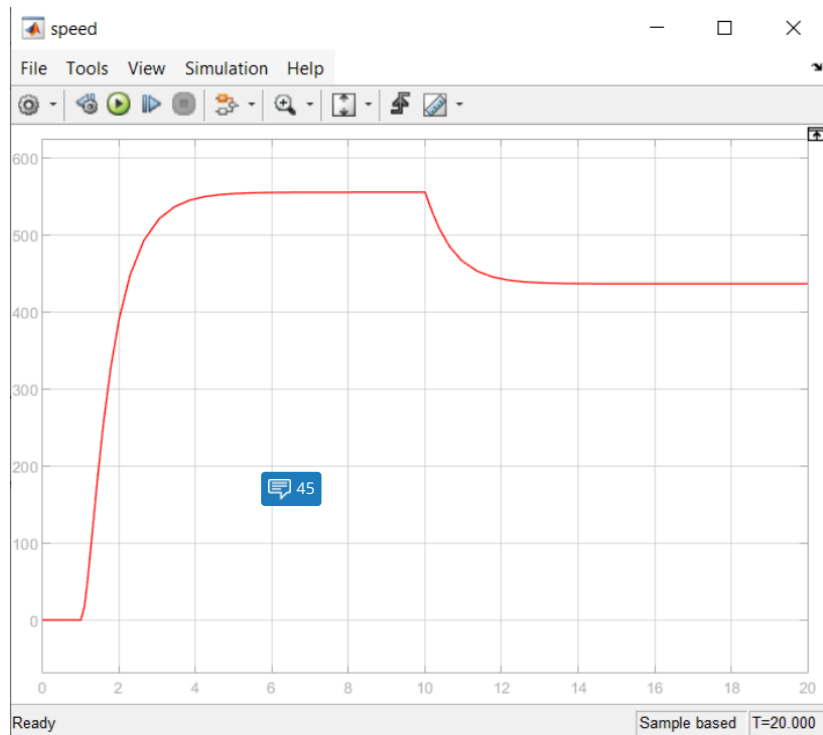


FIGURE 60 - SIMULINK GRAPHICAL RESPONSE FOR SPEED WITH LOAD

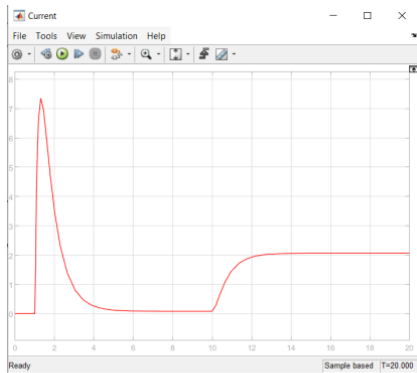


FIGURE 61 - SIMULINK GRAPHICAL RESPONSE FOR CURRENT WITH LOAD

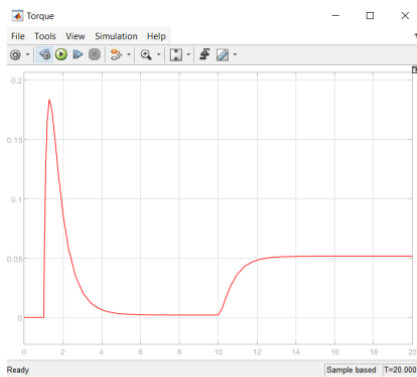


FIGURE 62 - SIMULINK GRAPHICAL RESPONSE FOR TORQUE WITH LOAD

Figures 60, 61 and 62 above shows the response of the DC motor model with a load being applied after 10 seconds. The speed has dropped to approximately the same value as was calculated for underload speed at 436 rad/s, and as the load is applied the current and the torque have increased to attempt to drive the motor with the additional load applied.

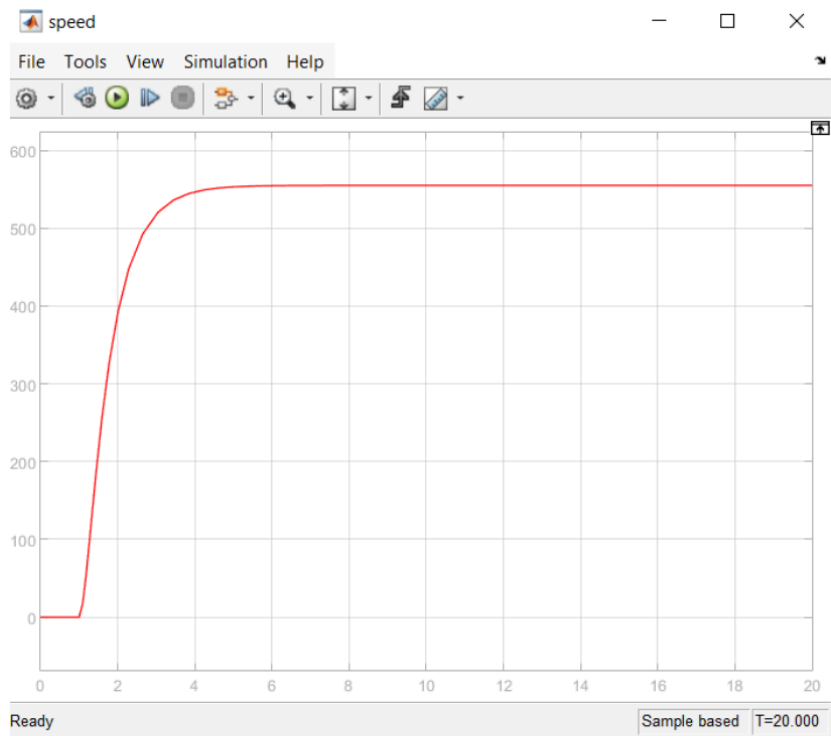


FIGURE 63 - SIMULINK GRAPHICAL RESPONSE FOR SPEED WITH NO LOAD

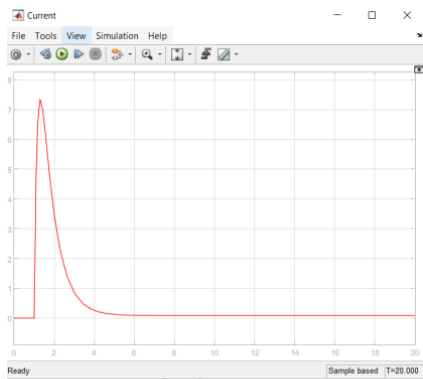


FIGURE 64 - SIMULINK GRAPHICAL RESPONSE FOR CURRENT WITH NO LOAD

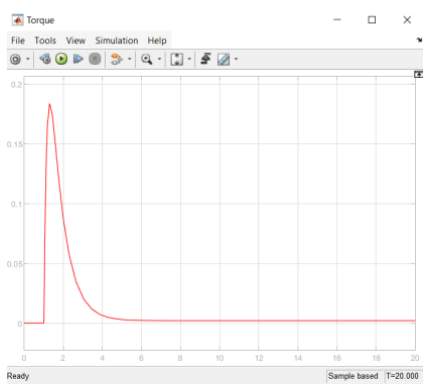


FIGURE 65 - SIMULINK GRAPHICAL RESPONSE FOR TORQUE WITH NO LOAD

Figures 63, 64 and 65 above shows the response of the DC motor model without a load being applied after 10 seconds. You can see from the responses for the current and the torque that both graphs peak really early in the simulation to get the motor up to speed and then both drop off as they are no longer required as the motor keeps a constant speed at approximately 555 rad/s.

Comparison of Results

	Calculated (rad/s)	Simulated (rad/s)	Error (rad/s)
No Load Speed (Ω)	555.09	555.1	0.01
Under Load Speed (Ω)	436.14	436.2	0.06

TABLE 10 - COMPARISON OF CALCULATED AND SIMULATED VALUES OF SPEED

Analysis



From the results provided in Table 10 above, conclusions can be drawn to support that applying a load onto the DC motor model will have a drastic negative impact on the motors responses. As the load is applied, the speed of the motor dropped accordingly, this then caused a reduced back electromagnetic field which meant additional current was needed to develop more torque to drive the motor at its underload speed. This is supported by the responses shown during the simulation.

Whether a load was applied to the DC motor model or not, there was a large inrush current spike at the start of each simulation. As the motor is stationary at this point, large currents are required to produce enough torque to overcome the initial inertia and create the back electromagnetic field. The results provided above support these claims with tiny errors between the calculated values and the simulated values.

Task 11 – Proportional Speed Control System

In this task, the motor model from the previous task will be taken and converted into a subsystem to make the system easier to comprehend. A proportional controller will also be introduced, with a 140 rad/s setpoint and a disturbance of 0.03 Nm after 10 seconds. The proportional controller will be set at a K_p of 3, before the system is then duplicated twice, with the second system having a K_p of 1.5 and the third system having a K_p of 6. The percentage of overshoot, rise time, settle time and steady state error both with and without the disturbance will be recorded from the response of each subsystem to analyse the effects of changing the value of K_p in the proportional controller.

Shown below in Figure 66, is the auto tune feature in Simulink which displays that the optimum value of K_p is 0.19321 which would provide the system with the best response.

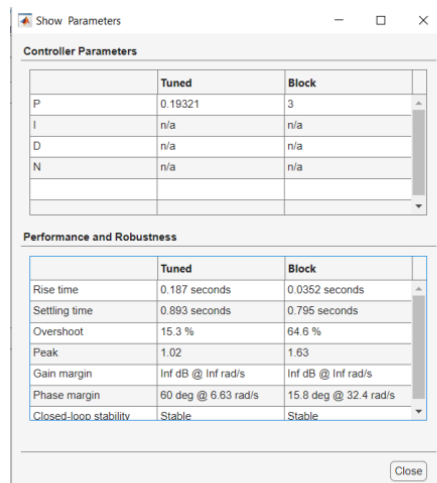


FIGURE 66 - AUTO TUNE FEATURE IN SIMULINK SHOWING THE OPTIMUM K_p VALUE

Motor Subsystem Model

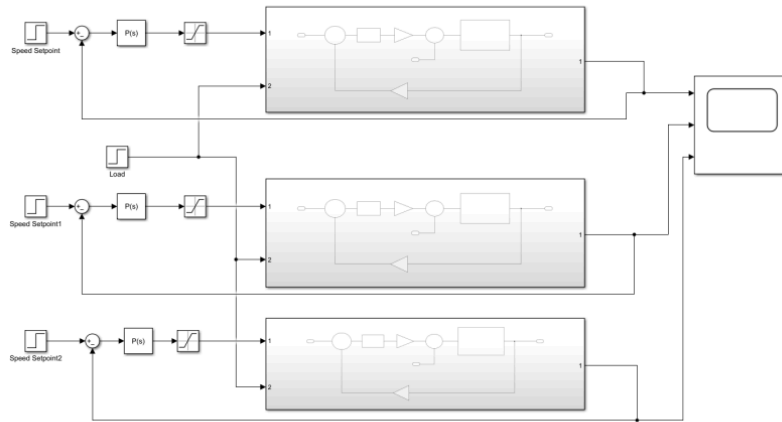


FIGURE 67 -SUBSYSTEMS WITH P CONTROLLER BUILT IN SIMULINK

Shown above in Figure 67 is the three subsystems. Each system has a speed setpoint set at 140 rad/s, a proportional controller, a voltage limiter set at +/-24V and a summing junction where the error will be fed back to, with this error then being supplied to the controller. Each subsystem then feeds into the 3 input scope so that all three responses can be displayed together on the same graph and there is also a load disturbance of 0.03Nm after 10 seconds applied to each system. As previously discussed, system 1 has a Kp value of 3, system 2 has a Kp value of 1.5 and system 3 has a Kp value of 6.



Simulink Responses

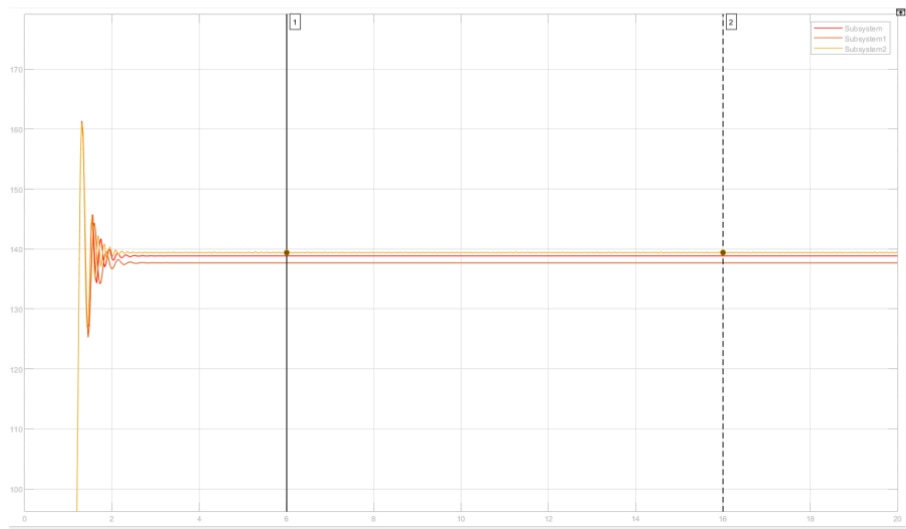


FIGURE 68 - SIMULINK RESPONSES FOR THE THREE SYSTEMS WITH NO LOAD

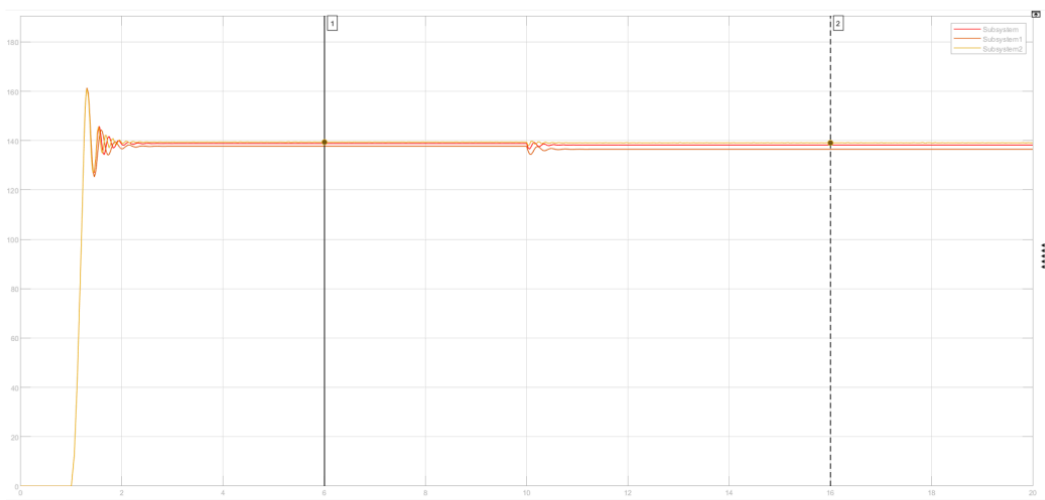


FIGURE 69 - SIMULINK RESPONSES FOR THE THREE SYSTEMS WITH LOAD

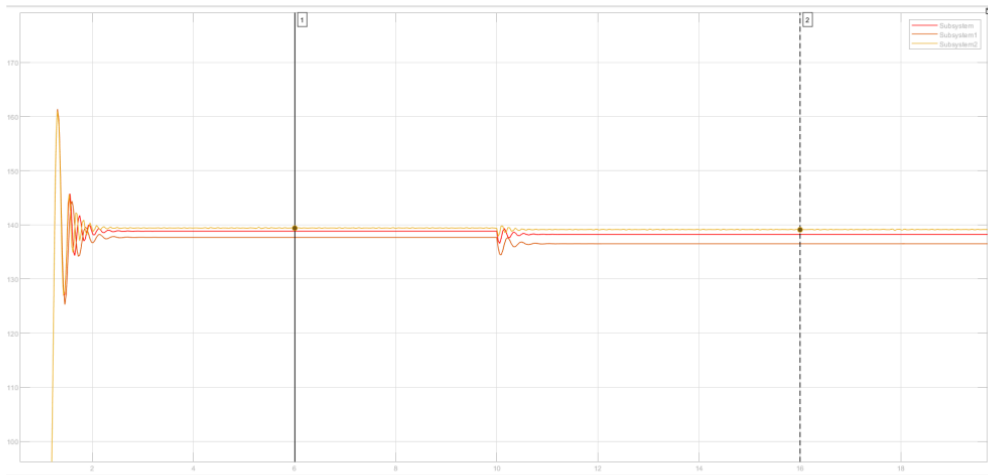


FIGURE 70 - ZOOMED IN SIMULINK RESPONSES FOR THE THREE SYSTEMS WITH LOAD

It is shown above in Figures 68, 69 and 70, that increasing or decreasing the value of K_p in the proportional controller does have a drastic effect on the response for each system. It is displayed that when the K_p value is increased this consequently increases the oscillations and the percentage of overshoot. System 1 was shown to be quite aggressive with a high overshoot percentage of 64.6% and a lot of oscillations which resulted in a small steady state error. System 3, which had an even greater K_p value, was more aggressive resulting in overshoot of 73.5% and a small steady state error of 0.9 rad/s. System 2 had the lowest K_p value which meant less oscillations and a smaller overshoot, resulting in a far greater steady state error of 3.5. The three responses additionally displayed that the higher the K_p value, the faster the rise time with system 3 with a K_p value of 6 having very quick rise time of 0.0241. However, there wasn't as much correlation with the settle times; system 2 having the longest settle time with the smallest K_p value, then system 3 with the greatest K_p value being the second fastest and system 1 with a K_p value of 3 having the quickest settle time at 0.795 seconds.



Comparison of Results

I can now compare the different responses I achieved when altering the value of K_p in the Simulink simulation. I will compare the values of Rise Time(s), Settle Time(s), Steady State Error(rad/s) with and without load and percentage of Overshoot depending on the value of K_p .

	Kp	Overshoot(%)	Rise Time(s)	Settle Time(s)	Steady State Error(rad/s) with Load	Steady State Error(rad/s) without Load
Subsystem 1	3	64.6	0.0352	0.795	1.8	1.2
Subsystem 2	4.5	53.4	0.0511	0.845	3.5	2.3
Subsystem 3	6	73.5	0.0241	0.826	0.9	0.6

TABLE 11 - COMPARISON OF THE VARIABLES WHEN ALTERING THE VALUE OF K_p IN EACH SUBSYSTEM

Analysis

From the tabulated results, you can see that the responses showed that the higher the value of K_p for the Proportional controller, the smaller the steady state error became as the system was more aggressive. It was also proven through these responses that the higher the Proportional value, the higher the percentage of overshoot became, and as the K_p value was increased the rise time also became lesser. This is because the poles shift further to the left in the s-plane when the proportional value is 6 which simultaneously increases the natural frequency. The higher the value of the natural frequency, the quicker the response of the system is. Additionally, the higher the value of K_p , generally it took longer for the system to settle as the oscillations increased. This system is largely accurate with the largest steady state error being 3.5 rad/s with a load applied, however the introduction of Integral and Derivative control in future tasks will theoretically eliminate the steady state error.

Task 12 – Proportional-Integral Speed Control System

In this task, the Proportional Speed Control System from the previous task will be taken and each controller has been converted into a Proportional-Integral controller. The setpoint will continue as 140 rad/s with a disturbance of 0.03 Nm after 10 seconds. The proportional value will be constant with a value of 3, while the integral value will be varied with system 1 having K_i of 0.3, system 2 having a K_i value of 0.6 and system 3 having a K_i value of 0.9. The percentage of overshoot, rise time, settle time and steady state error both with and without the disturbance will be recorded from the response of each subsystem to analyse the effects of changing the value of K_i in the Proportional-Integral controller.

Shown below in Figure 71, is the auto tune feature in Simulink which displays that the optimum value of K_p is 0.037976 and the optimum K_i value is 0.077202 which would provide the system with the best response.

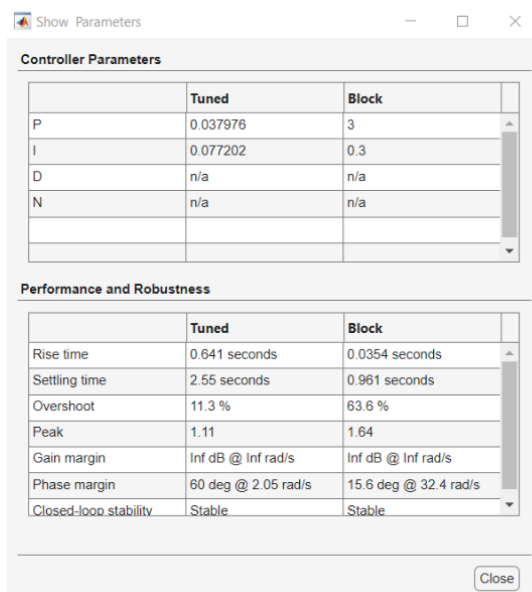


FIGURE 71 - AUTO TUNE FEATURE IN SIMULINK SHOWING THE OPTIMUM K_p AND K_i VALUES

Motor Subsystem Model

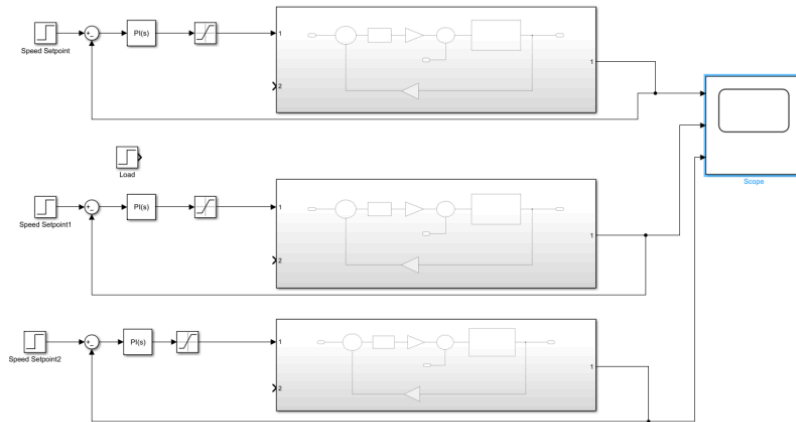


FIGURE 72 - SUBSYSTEMS WITH PI CONTROLLERS BUILT IN SIMULINK

Shown above in Figure 72 is the three subsystems. Each system has a speed setpoint set at 140 rad/s, a proportional-integral controller, a voltage limiter set at $\pm 24V$ and a summing junction where the error will be fed back to, with this error then being supplied to the controller. Each subsystem then feeds into the 3 input scope so that all three responses can be displayed together on the same graph and there is also a load disturbance of 0.03Nm after 10 seconds applied to each system. As previously discussed, the proportional value is kept constant at 3 for each system while, system 1 has a K_i value of 0.3, system 2 has a K_i value of 0.6 and system 3 has a K_i value of 0.9.

Simulink Responses

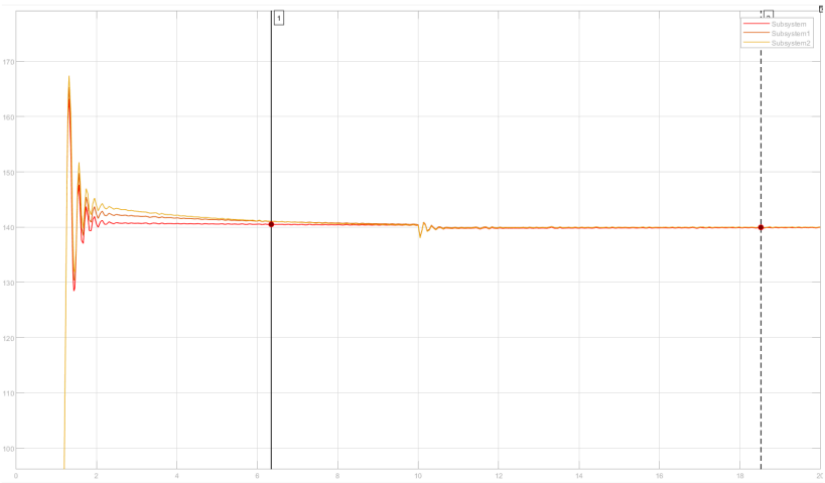


FIGURE 73 - SIMULINK RESPONSES FOR THE THREE SYSTEMS WITH LOAD

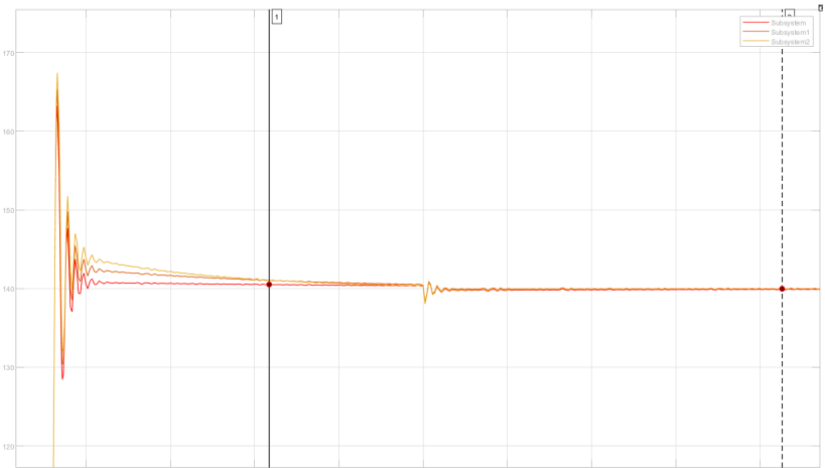


FIGURE 74 - ZOOMED IN SIMULINK RESPONSES FOR THE THREE SYSTEMS WITH LOAD

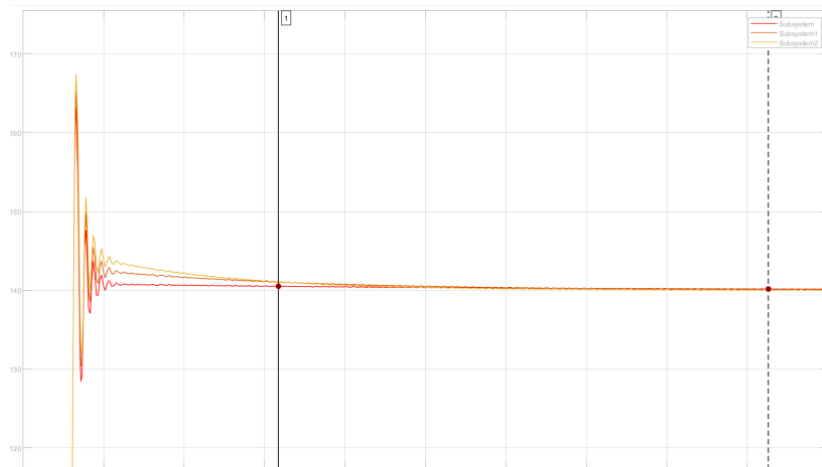


FIGURE 75 - SIMULINK RESPONSES FOR THE THREE SYSTEMS WITH NO LOAD

It is shown above in Figures 73, 74 and 75 that converting the controllers in each system to Proportional-Integral controllers has had a drastic effect in the responses by eliminating as much steady state error as possible. It can be seen that compared to the Proportional controllers from the previous task, where the steady state error was varied depending on the value of K_p , the systems are more stable with negligible errors. System 1 has a slight error 0.1 rad/s when there is a disturbance, while there is no error at all when there is no disturbance. System 2 and 3, which have the slightly more aggressive integral values have no steady state error where the disturbance is applied to the systems or not. For all 3 systems, however, there are still very slight oscillations after the load is applied and rectifying this could be a possible improvement of the systems. It can also be seen from the responses that the overshoot is slightly greater when the K_i is greater with system 3 having an overshoot value of 64.5%. The addition of the Integral to the controller has made the rise times of all three systems more consistent with each system having a time of approximately 0.0353 seconds. The settle times of each system has also decreased slightly with system 1, with the less aggressive K_i value, having a settle time of 0.961 seconds. Systems 2 and 3 both have a settle time of 0.804 seconds. These responses display that the addition of Integral to each controller has caused each system to become considerably more stable, with the slightly more aggressive K_i values making their respective systems moderately more efficient.

Comparison of Results

I can now compare the different responses I achieved when altering the value of K_i in the Simulink simulation. I will compare the values of Rise Time(s), Settle Time(s), Steady State Error(rad/s) with and without load and percentage of Overshoot depending on the value of K_p .

	Ki	Overshoot(%)	Rise Time(s)	Settle Time(s)	Steady State Error(rad/s) with Load	Steady State Error(rad/s) without Load
Subsystem 1	0.3	63.6	0.0354	0.961	0.1	0
Subsystem 2	0.6	64.1	0.0353	0.804	0	0
Subsystem 3	0.9	64.5	0.0353	0.804	0	0

TABLE 12 - COMPARISON OF THE VARIABLES WHEN ALTERING THE VALUE OF K_i IN EACH SYSTEM

Analysis

In this task, the introduction of Integral control to the feedback loop was shown to be very effective in attempting to eliminate the steady state error and make the systems far more efficient. In the previous task, the Proportional gain used in the controller could only bring the steady state error down to 0.6 rad/s at best so introducing Integral control was crucial in order to reach the 140 rad/s setpoint with almost no error. Similar to the previous task, the systems were largely more efficient when the K_i value was more aggressive, with steady state error only occurring with lowest K_i value of 0.3 when load was applied to the system. All other scenarios showed that all steady state error had been eliminated. The introduction of Integral control was also crucial in eliminating the error after load was applied to the system as the proportional control on its own could not achieve this in the previous task. The only drawbacks with the responses shown for the systems above is that the overshoot is relatively high for each system as they are more aggressive, and each system is taking longer to settle from the oscillations. The introduction of Derivative control in the next task will hopefully make each system even more efficient and reduce these values.



Task 13 – Proportional-Integral-Derivative Speed Control System

In this task, the Proportional-Integral Speed Control System from the previous task will be taken and each controller has been converted into a Proportional-Integral-Derivative controller. The setpoint will continue as 140 rad/s with a disturbance of 0.03 Nm after 10 seconds. The proportional value will be constant with a value of 0.50679, while the integral value will also be constant with a value of 1.0117. However, the values of Kd will be varied with system 1 having a Kd of 0.025, system 2 having a Kd value of 0.058002 and system 3 having a Kd value of 0.1. The percentage of overshoot, rise time, settle time and steady state error both with and without the disturbance will be recorded from the response of each subsystem to analyse the effects of changing the value of Kd in the Proportional-Integral-Derivative controller.

Shown below in Figure 76, is the auto tune feature in Simulink which displays that the optimum value of Kp is 0.042874, the optimum Ki value is 0.075335 and the optimum value of Kd is 0.0027025 which would provide the system with the best response.

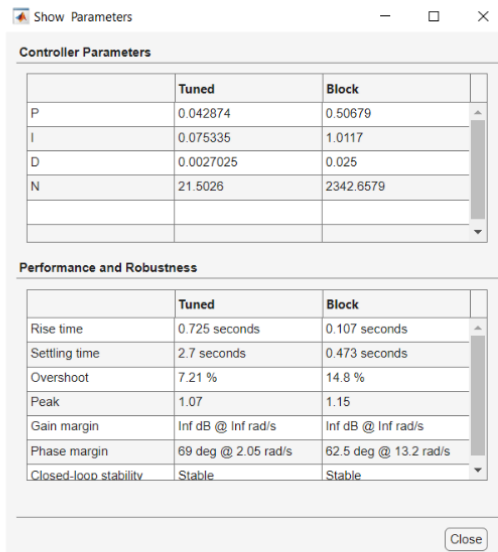


FIGURE 76 - AUTOTUNE FEATURE IN SIMULINK SHOWING THE OPTIMUM VALUES OF Kp, Ki AND Kd

Motor Subsystem Model

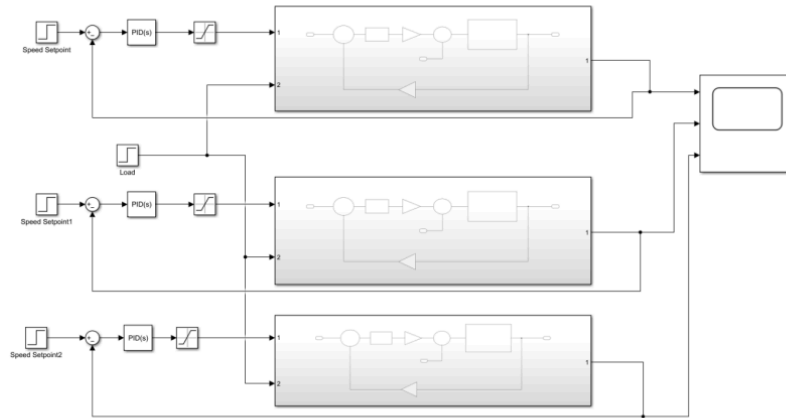


FIGURE 77 - SUBSYSTEMS WITH PID CONTROLLERS BUILT IN SIMULINK

Shown above in Figure 77 is the three subsystems. Each system has a speed setpoint set at 140 rad/s, a proportional-integral-derivative controller, a voltage limiter set at $\pm 24V$ and a summing junction where the error will be fed back to, with this error then being supplied to the controller. Each subsystem then feeds into the 3 input scope so that all three responses can be displayed together on the same graph and there is also a load disturbance of 0.03Nm after 10 seconds applied to each system. As previously discussed, the proportional value is kept constant at 0.50679 and the integral value at 1.0117 for each system while, system 1 has a Kd value of 0.025, system 2 has a Kd value of 0.058002 and system 3 has a Kd value of 0.1.

Simulink Responses

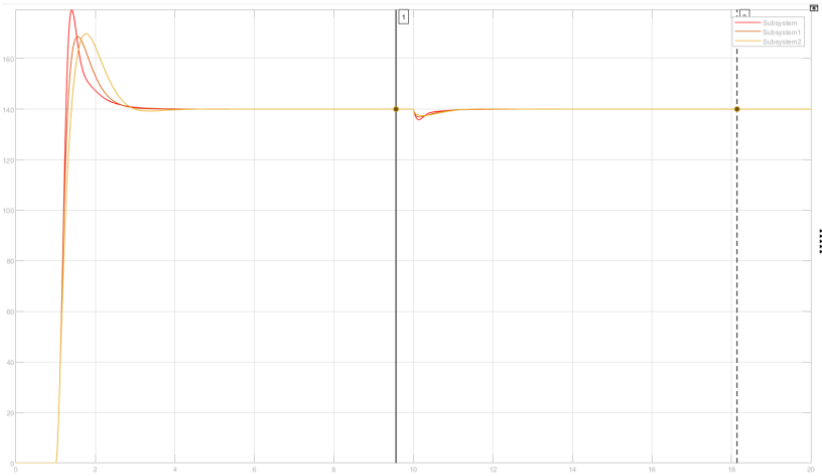


FIGURE 78 - SIMULINK RESPONSE FOR THREE SYSTEMS WITH LOAD

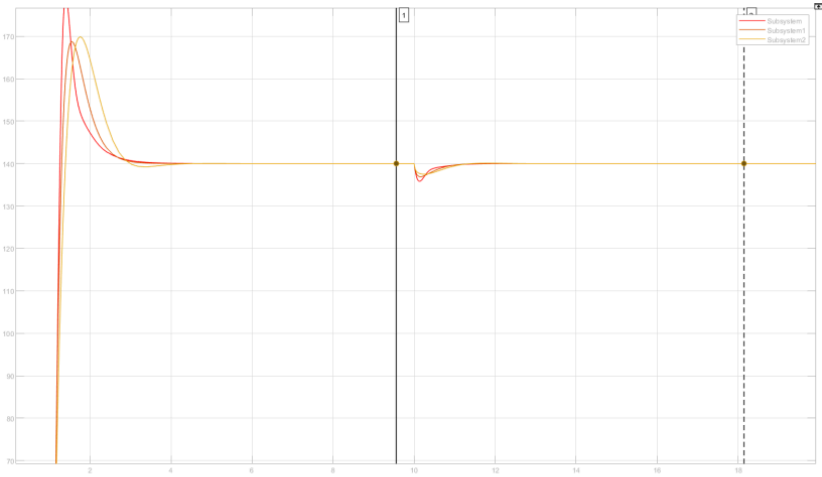


FIGURE 79 - ZOOMED IN SIMULINK RESPONSE OF THE THREE SYSTEMS WITH LOAD

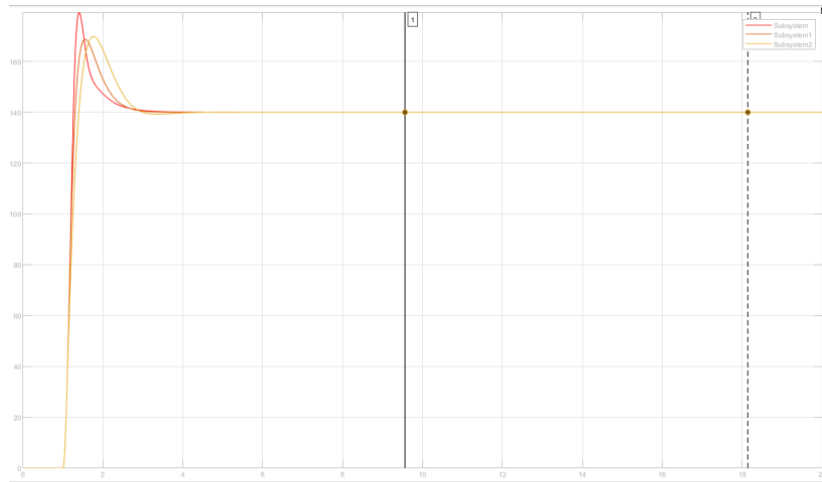


FIGURE 80 - SIMULINK RESPONSE FOR THE THREE SYSTEMS WITH NO LOAD

It is shown above in Figures 78, 79 and 80 that converting the controllers in each system to Proportional-Integral-Derivative controllers has had a drastic effect in the responses with the steady state error being completely eliminated for each subsystem wherever there was a load applied to the system or not. The introduction of derivative control also helped to minimise the percentage of overshoot and the settle time of each subsystem compared to the previous two tasks. The rise time has increased slightly, but this just means the system is slightly more aggressive in order to eliminate the error and stabilise the system as quickly as possible. The responses shown and the data extracted from these responses prove that Proportional-integral-derivative control is the most efficient way to stabilise each subsystem and eliminate all the steady state error.

Comparison of Results

I can now compare the different responses I achieved when altering the value of K_d in the Simulink simulation. I will compare the values of Rise Time(s), Settle Time(s), Steady State Error(rad/s) with and without load and percentage of Overshoot depending on the value of K_p .

	Kd	Overshoot(%)	Rise Time(s)	Settle Time(s)	Steady State Error(rad/s) with Load	Steady State Error(rad/s) without Load
Subsystem 1	0.025	14.8	0.107	0.473	0	0
Subsystem 2	0.058002	3.32	0.103	0.736	0	0
Subsystem 3	0.1	2.15	0.0901	0.897	0	0

TABLE 13 - COMPARISON OF THE VARIABLES WHEN ALTERING THE VALUE OF K_d IN EACH SYSTEM

Analysis

In this task, The PID controller was completed with the introduction of Derivative control. These systems are very efficient with no steady state error being present during either simulation and the highest overshoot value recorded being 14.8%. Each part of the PID controller seems to be working correctly in each simulation with Proportional gain correcting the errors, Integral gain accumulating the previous errors in an attempt of eliminating the steady state error, and finally derivative control seems to be predicting future errors to keep the systems stable and keep the speed at the setpoint. It is evident that the introduction of derivative control has also assisted in decreasing the settling times of each system and a correlation can be made that overshoot was lower when the K_d value was greater. A general conclusion can be drawn that the introduction of derivative control to each system has made each system as accurate and as efficient as possible.

Task 14 – Controller Comparisons

In this task, the Proportional-Integral Speed Control System from the previous task will be taken and one controller has been converted into a Proportional controller, another has been converted into a Proportional-Integral controller and the final subsystem will be continue as a Proportional-Integral-Derivative controller. This has been done in order to examine the effects of using Proportional control, Proportional-Integral control and Proportional-Integral-Derivative control. The setpoint will continue as 140 rad/s with a disturbance of 0.03 Nm after 10 seconds. The proportional value will be constant with a value of 0.50679 for all three systems, while the integral value will also be constant with a value of 1.0117 but only used in systems 2 and 3. System 3 will then also have derivative control with a K_d value of 0.058002. The percentage of overshoot, rise time, settle time and steady state error both with and without the disturbance will be recorded from the response of each subsystem to analyse the effects of using Proportional control, Proportional-Integral control and Proportional-Integral-Derivative control.

Motor Subsystem Model

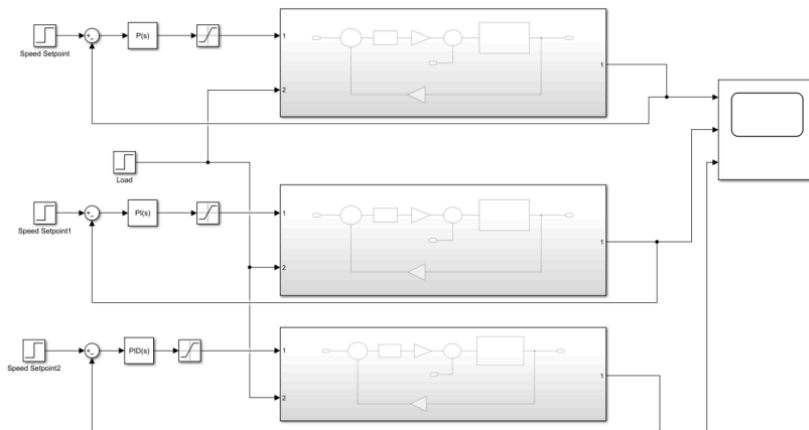


FIGURE 81 - SUBSYSTEMS WITH P, PI AND PID CONTROLLERS BUILT IN SIMULINK

Shown above in Figure 81 is the three subsystems. Each system has a speed setpoint set at 140 rad/s, a voltage limiter set at $\pm 24V$ and a summing junction where the error will be fed back to, with this error then being supplied to the controller. Subsystem 1 has a Proportional controller, subsystem 2 has a Proportional-Integral controller and subsystem 3 has a Proportional-integral-derivative controller. Each subsystem then feeds into the 3 input scope so that all three responses can be displayed together on the same graph and there is

also a load disturbance of 0.03Nm after 10 seconds applied to each system. As previously discussed, the proportional value is kept constant at 0.50679 for each system. Subsystem 2 and 3 have K_i values of 1.0117 and subsystem 3 has K_d value of 0.58002.

Simulink Responses

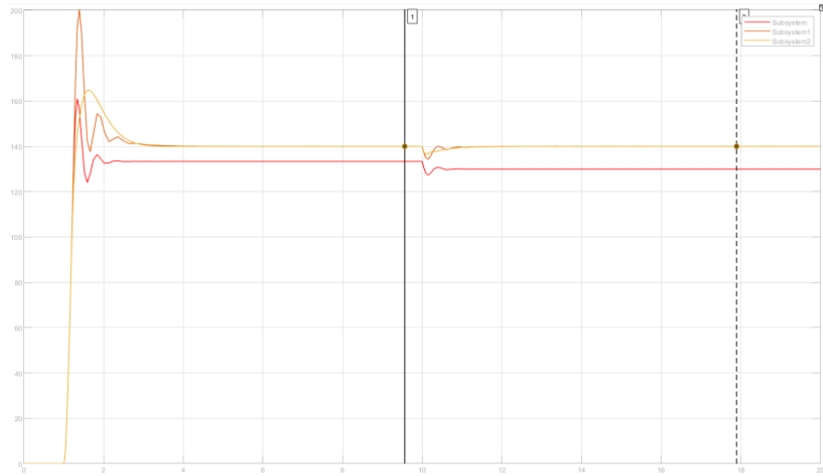


FIGURE 82 - SIMULINK RESPONSE FOR THE THREE SUBSYSTEMS WITH LOAD

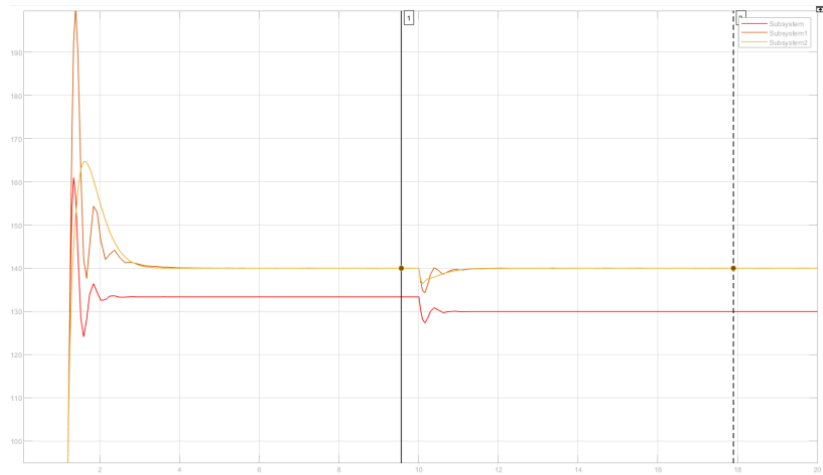


FIGURE 83 - ZOOMED IN SIMULINK RESPONSE FOR THE THREE SYSTEMS WITH LOAD

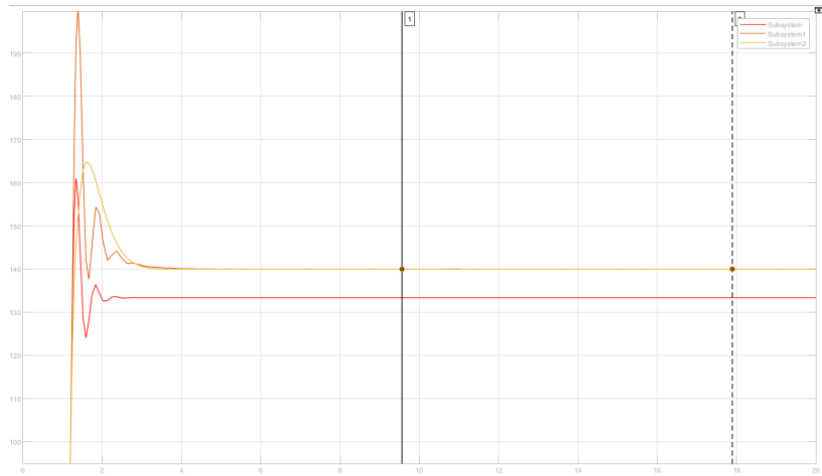


FIGURE 84 - SIMULINK RESPONSE FOR THE THREE SYSTEMS WITHOUT LOAD

It is shown above in Figures 82, 83 and 84 how the introduction of proportional, Proportional-Integral and proportional-Integral-Derivative controllers can affect the response of each system and stabilise the systems. System 1 with only proportional control had a speed error of 10 rad/s with load applied to the system and 7 rad/s without load applied to the system. System 2 then introduced integral control and the steady state error was completely eliminated, system 3 additionally had no steady state error when derivative control was introduced. System 1 had the lowest overshoot but this can be down to this system not being as aggressive as the two systems with the integral and derivative control added. In system 3 where derivative control is introduced the rise time and settle time are lowered as much as possible so that the setpoint is reached as efficiently as possible.

Comparison of Results

I can now compare the different responses I achieved when altering the value of K_d in the Simulink simulation. I will compare the values of Rise Time(s), Settle Time(s), Steady State Error(rad/s) with and without load and percentage of Overshoot depending on the value of K_p .

	K_p	K_i	K_d	Overshoot(%)	Rise Time(s)	Settle Time(s)	Steady State Error(rad/s) with Load	Steady State Error(rad/s) without Load
Subsystem 1 (P)	0.50679	N/A	N/A	33.5	0.0984	0.803	10	7
Subsystem 2 (PI)	0.50679	1.0117	N/A	45.5	0.0942	1.04	0	0
Subsystem 3 (PID)	0.50679	1.0117	0.058002	45	0.0181	0.85	0	0

TABLE 14 - COMPARISON OF THE VARIABLES IN P, PI AND PID CONTROLLERS

Analysis

During this task, it has been illustrated the purposes of each of form of gain in the controller and how important each one is to create a stable, efficient system. The results shown above in Table 14, clearly display that when the controller has Proportional, Integral and Derivative control the system is stable and far more efficient when compared to the other two systems which is as expected.

In subsystem 1, where only Proportional control was utilised, there was a steady state error of 10 rad/s when load was applied so the setpoint is never met. The overshoot value for this system is 33.5% with a slow rise time and this is because the system isn't aggressive enough to reach the speed setpoint and adjust to the disturbance applied to the system. In system 2, where integral control was introduced, the steady state error is completely eliminated which is as expected from the introduction of this form of control. The overshoot value and settle time have also increased as the system is now more aggressive but isn't efficient enough to allow the system to settle fast enough. The system now also reacts positively when the disturbance is applied and still manages to reach the desired setpoint. Finally, in subsystem 3 where Derivative control was included, it can be seen that the overshoot decreases slightly with significant decreases in the rise and settle times which in effect makes this system far more efficient in reaching the setpoint of 140 rad/s even when the load is applied to the system after 10 seconds. The derivative control predicts future error and then tries to act on these possible errors before they occur to hopefully improve the stability and efficiency of the system.

Task 15 – Overall Conclusions and Recommendations

This assignment, which included the analysis, simulation and evaluation of electrical circuits and systems using knowledge of electrical principles, includes 14 tasks that can be separated into 3 topic areas. Tasks 1 through 6 were the electrical based tasks which included RL, RC and RLC circuits, tasks 7 and 8 were the electronic tasks where a low pass active filter and a high pass active filter were constructed and then task 9 through 14 were the control based tasks. During these control based tasks, a DC motor model was constructed, and this was later further developed to include a PID controller with a closed loop system.

In the electrical based tasks, a transient analysis, which is in regard to the time domain, and a frequency performance analysis, which is in regard to the frequency domain, were carried out on both a first order RL circuit and a first order RC circuit. For the transient analysis tasks, tau was calculated for the specified component values and the systems were then simulated on Matlab in order to obtain a simulated value of tau, an error was then calculated between these values. A similar process was followed for the frequency performance analysis tasks, with cut off frequency being the parameter that was compared through calculated and simulated values. These tasks were successful, with negligible errors between values. Additionally, second order series RLC circuits, which contained an additional energy storing component, were constructed with both analysis methods being conducted again. For these situations, however, the responses of the circuits were observed when values such as the resistance, damping factor and Q factor were varied in order to obtain the required damping responses.

For each task, the circuits were deeply analysed, with complicated calculations conducted and accurate circuit simulations taking place. These analysis methods increased the quality of the tasks, which required basic electrical principle knowledge to complete and this theory could be easily explained throughout via the circuit drawings, simulation responses and numerical results.

The electronic based assignment tasks were task 7 and task 8 where low pass and high pass active filters were designed, simulated and analysed. Both circuits contained op amps, but if you moved the capacitor in parallel with a resistor a low pass filter would be created and if the capacitor was in series, then the circuit would be high pass filter. During both tasks, values were calculated and simulated for the cut off frequency, the magnitude and the phase shift for each circuit. The phase shift was crucial for deciding if the circuit was lagging or leading, and the inverting amplifier effected these phase shift values during the tasks with 180° being added to the phase shift values. Negligible errors were discovered between simulated and calculated results which proved that the theory used to come up with the calculated values for each factor was correct.

The control tasks that featured between tasks 9 and 14, began with a simple open loop system before a gain was then added to create a closed loop system in order to compare and reflect upon the different responses between the two different systems. This talk highlighted how crucial feedback is in controls to keep a system and being able to withstand system disturbances. A complex DC motor model was constructed in a later task with a mechanical transfer function and an electrical transfer function being present to account for their respective factors. Several simulations then took place to see how the motor would run normally and then how it would run when a disturbance (load) was applied to the system. Speed, current and torque responses were ed for both scenarios to see how the motor would react under these conditions. The final tasks involved developing a PID controller in order to make the DC motor model system more accurate and efficient, with the roles of Proportional, Integral and Derivative being discussed. Key results such as the steady state error, rise time and the percentage of overshoot were obtained from response graphs and debated. The effect the different types of gain had on these response values was later discussed, which each variant having their own unique role in making the system as accurate yet efficient as possible.

Throughout this assignment module I was able to utilise my preexisting electrical principle knowledge to assist me in completing each task, and additionally various other examples of electrical principle theory I learnt during the module allowed me to complete the tasks to a high standard with great accuracy in my results throughout with thorough and detailed analysis. I also gained crucial experience in the simulation and testing of the circuits and systems with the software packages provided such as Matlab and Simulink, in order to discover the behaviours of the electrical systems and to meet all the criteria set out in the assignment brief.

In order to further enhance my electrical principle knowledge and my electrical skillset as a whole; building several of the circuits and systems in real life and testing these circuits and collecting the relevant data and responses so that a third set of data could have been compared during each task could have been especially beneficial. Seeing the circuits in operation in a real world scenario would allow you to validate the calculated and simulated data and responses but also, the real world performance of the systems and the component interactions could then be taken into account. Issues such as environmental factors, component tolerances and system reliability could then be uncovered and further understood. During the simulation process, however, real world parameters and problems could have been introduced such as the component tolerances and temperature changes to see how the systems would react under these non-ideal conditions. Also, the circuits could possibly have been simulated under fault conditions to unearth the weaknesses within the systems.

67 /100

PAGE 1

PAGE 2



Comment 1

Good work here.



Comment 2

Gr



Comment 3

A good summary.

PAGE 3

PAGE 4

PAGE 5

PAGE 6

PAGE 7

PAGE 8



Comment 4

Useful information here.

PAGE 9



Comment 5

Add a circuit diagram at the outset.



Comment 6

0.92ms



Comment 7

Good work here



Comment 8

Good work.



Comment 9

Source?



Comment 10

Good work in this task.



Comment 11

Source?



Comment 12

Correctly annotated.



Comment 13

There are more quantities you could have tabulated.



Comment 14

A good analysis, avoid vague terminology like swift.



Comment 15

Correctly annotated.



Comment 16

See comment for previous task.



Comment 17

WW



Comment 18

You are correct.



Comment 19

Good work here.



Comment 20

You could have avoided repetition by stating method once only.

PAGE 42



Comment 21

Fiar but largely descriptive, ty to relate your trends to theory, put into context, what does this mean in a real world system.

PAGE 43

PAGE 44

PAGE 45

PAGE 46

PAGE 47

PAGE 48

PAGE 49

PAGE 50

PAGE 51

PAGE 52

PAGE 53

PAGE 54

PAGE 55

PAGE 56

PAGE 57

PAGE 58

PAGE 59



Comment 22

Correct reults, well explained and well annotated.

PAGE 60



Comment 23

Efficient in what way? Superior to what? Use scientific terminology, avoid vague statements.



Comment 24

Where is this theory proved?



Comment 25

Confusing last paragraph.



Comment 26

Electronic not Electrical



Comment 27

What is the benefit of the op-amp filter to the passive filter?



Comment 28

Correctly annotated.



Comment 29

Ok, but largely descriptive and missing the point of adding the op-amp.



Comment 30

Correctly annotated.



Comment 31

See comment for previous analysis.



Comment 32

Quantify what this means and the reason.



Comment 33

Define a pole.



Comment 34

Relate to $G/(1+GH)$



Comment 35

Incorrectly mathematically explained.

PAGE 72

PAGE 73

PAGE 74



Comment 36

Good work here.

PAGE 75



Comment 37

Good work.

PAGE 76



Comment 38

Not apparently obvious where this TF comes from.

PAGE 77

PAGE 78

PAGE 79

PAGE 80

PAGE 81

PAGE 82



Comment 39

Mistake here....typo?



Comment 40

Reasonable, confusing last paragraph.

PAGE 83



Comment 41

WW



Comment 42

Show the circuit arrangement.

PAGE 84



Comment 43

State the TFs in terms of quantities rather than values.

PAGE 85



Comment 44

Correct, for higher grade show the derivations.

PAGE 86



Comment 45

Missed opportunity to place cursors.

PAGE 87

PAGE 88

PAGE 89

PAGE 90



Comment 46

Accurate speed results, you could have expanded this to include other results, e.g. torque, current.

PAGE 91

PAGE 92



Comment 47

Good work.

PAGE 93

PAGE 94



Comment 48

Avoid using a paragraph of text to describe results, tabulate and look for trends.

PAGE 95



Comment 49

Put in numerical order, it helps identify trends.



Comment 50

Descriptive in nature, use the mathematical describing equations for the PID gains to explain the findings, thus relating directly to theory.

PAGE 96



Comment 51

You have acried out the task correctly and logically, however, for the report you should use the mathematical describing equations for the PID gains to explain the findings, thus relating directly to theory. Avoid long confusing, difficult to read sections of text when a table or graph can do the same job. Also work becomes repetative and difficult to follow. Analysis also then lacks depth due to the amount of words used to describe results.



Comment 52

See previous comment.



Comment 53

See previous comment.



Comment 54

Same comment as previous applies, your work should not be a description of what you have done, rather an analysis, relating to theory of the findings and explanations given which relate to mathematical relationships.

1. ENGAGEMENT WITH LITERATURE NA

0

2. KNOWLEDGE AND UNDERSTANDING Excellent knowledge displayed via sound mathematical calculations, well explained methods and results. 21/30

21

3. COGNITIVE AND INTELLECTUAL Analysis for the most part is sound and accurate and some theoretical predictions are included, however, it should not be a description of what you have done/learned, rather an analysis, relating to theory compared with your findings and explanations given which relate to mathematical relationships.

20

4. PRACTICAL APPLICATION Very good application of technical methods both in the mathematics and the simulation aspects. See comments for improvement opportunities. 21/30

21

5. TRANSFERABLE SKILLS The report contains effective written communication but some aspects are repetitive,

5

see comments. 5/10