

LEVEL 5 ASSESSMENT SPECIFICATION

Student name:	Sean Beynon	Student ID number:	
Programme:	HND/HNC Electrical/Electronic Engineering, HNC Instrumentation Engineering & HNC Power System Engineering		
Module:	Electrical Principles, Methods and Simulation		
Module code:	PSDE5009	Contribution to Overall Module Assessment (%):	50%
Lecturer:	Jason Pointer	Internal Verifier:	Steve Roberts
Assignment Title:	Analogue Systems	Word count (or equivalent):	2000
Submission deadline:	07/02/19	Return date of provisional marks & written feedback:	
Submission method:	All written assessments, where practical and possible, must be submitted via Turnitin unless otherwise instructed by the Lecturer. (Please DO NOT put this assessment specification into Turnitin or it will match many similarities with other students' submissions.) Alternative submission method (if applicable): Late submission of the assessment will result in a late penalty mark. Penalties for late submission: Up to one week late, maximum mark of 40%. Over one week late, 0%. Only the Extenuating Circumstances Panel may approve a change to submission dates.		
Academic honesty / referencing:	Academic honesty is required. In the main body of your submission you must give credit to authors on whose research and ideas your work is based. Append to your submission a reference list that indicates the books, articles, etc. that you have used, cited or quoted in order to complete this assessment.		

Module Learning Outcomes (from module syllabus)

1. Analyse and **evaluate electrical circuits, electronic systems** and control systems.
2. Simulate standard **electrical circuits, electronic systems** and control systems.

STUDENT FEEDBACK FORM

This section details the *extent* to which the assessment criteria are demonstrated by you, which in turn determines your mark. The marks available for each category of skill are shown. Lecturers will use the space provided to comment on the achievement of the task(s), including those areas in which you have performed well and areas that would benefit from development/improvement.

Generic Assessment Criteria	Marks available	Marks awarded
1. Engagement with Literature Skills	0	0
2. Knowledge and Understanding Skills	20	14.9
Achieved 80% for Task 1. Achieved 65% for Task 2. Achieved 75% for Task 3. Achieved 82% for Task 4. Achieved 70% for Task 5. Excellent, detailed knowledge and showed an excellent understanding of the well-established concepts and principles of the covered by this assignment.		
3. Cognitive and Intellectual Skills	20	9.2
Achieved 50% for Task 1. Achieved 44% for Task 3. Achieved 44% for Task 5. For the tasks that asked for comments there was limited attempt at critical evaluation and analysis of concepts and principles, tending towards description.		
4. Practical Application Skills	50	39.4
Achieved 85% for Task 1. Achieved 65% for Task 2. Achieved 80% for Task 3. Achieved 85% for Task 4. Achieved 80% for Task 5. Achieved 78% for Task 6. Excellent application of established methods in the use of Matlab, Multisim and practical work. Required minimum support throughout the assessment.		
5. Transferable Skills for Life and Professional Practice	10	7.5
Work is coherent, very fluent and is presented professionally.		

Assessment Mark (Assessment marks are subject to ratification at the Exam Board. These comments and marks are to give feedback on module work and are for guidance only until they are confirmed.)

Late Submission Penalties (tick if appropriate)

Up to 1 week late (40% Max)

Over 1 week late (0%)

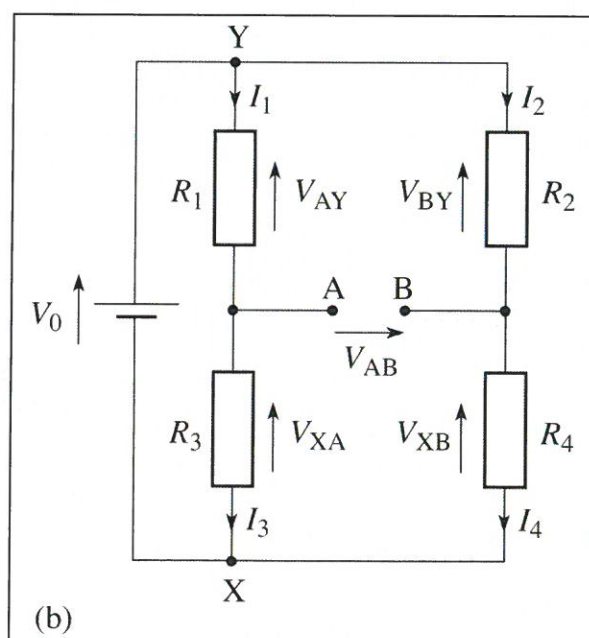
71%

HNC POWER SYSTEM ENGINEERING

Module: Electrical Principles, Methods and Simulations

Assignment: Analogue Systems

Sean Beynon
National Grid
2018



Summary

This Assignment provides a variety of techniques to scrutinize the performance of DC, AC and electronic circuits. The assignment offered the ability to cross reference values, this demonstrating the ability to perform tasks using practicable, mathematical, simulation and computerised techniques. These techniques together with the analysis of the circuits increases the understanding of circuit theory and principles.

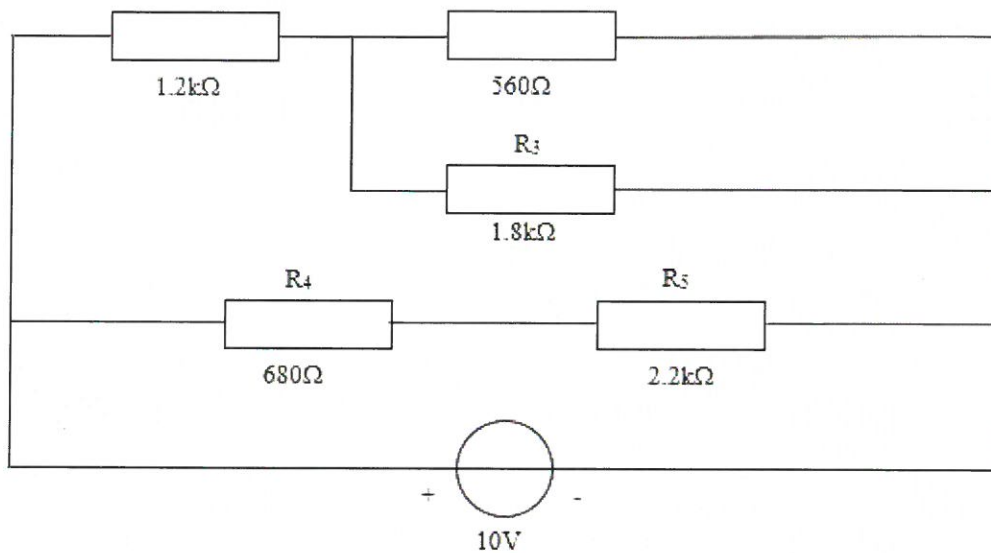
Table of contents

Table of contents	2
Task 1- Combinational Circuit.....	3
Task 2- Circuit Theorem.....	8
Task 3- Series Ac Circuit.....	Error! Bookmark not defined.
Task 4- Complex AC Circuit	Error! Bookmark not defined.
Task 5- Non-Inverting Amplifier Circuit	Error! Bookmark not defined.
Task 6- Integrator Circuit.....	Error! Bookmark not defined.

Task 1- Combinational Circuit

a)

For this section on Task 1, there is a requirement to build the DC circuit below using breadboard, supply voltage unit and resistors. On completion of the circuit build, the voltage was measure across each resistor.



In the table below the results obtained from the practical task can be seen. In the table the values for R_2 and R_3 are identical; this is due to the resistors being in parallel.

Resistor	Resistor Value (Ω)	Measured Voltage across resistor(V)
R1	1.2 K Ω	7.33 V
R2	560 Ω	2.67 V
R3	1.8 K Ω	2.67 V
R4	680 Ω	2.35 V
R5	2.2 K Ω	7.65 V

b)

Using Ohms Law and the Potential Divider rule, the DC circuit in Task 1 a) was calculated to provide mathematical values for the voltages across each resistor.

$$R4 + R5 = 680 + 2.2 \times 10^3 = 2.88 \times 10^3 = 2.88 \text{ k}\Omega$$

$$I1 = \frac{V}{R45} = \frac{10}{2.88 \times 10^3} = 3.472 \times 10^{-3} = 3.472 \text{ mA}$$

$$V4 = I1 \times R4 = 3.472 \times 10^{-3} \times 680 = 2.36 \text{ V}$$

$$V5 = I1 \times R5 = 3.472 \times 10^{-3} \times 2.2 \times 10^3 = 7.64 \text{ V}$$

$$R23 = \frac{R2 \times R3}{R2 + R3} = \frac{560 \times 1.8 \times 10^3}{560 + 1.8 \times 10^3} = \frac{1.008 \times 10^6}{2.36 \times 10^3} = 427.12 \Omega$$

$$R123 = R1 + R23 = 1.2 \times 10^3 + 427.12 = 1.627 \times 10^3 = 1.627 \text{ k}\Omega$$

$$I2 = \frac{V}{R123} = \frac{10}{1.627 \times 10^3} = 6.14 \times 10^{-3} = 6.14 \text{ mA}$$

$$V1 = I2 \times R1 = 6.14 \times 10^{-3} \times 1.2 \times 10^3 = 7.37 \text{ V}$$

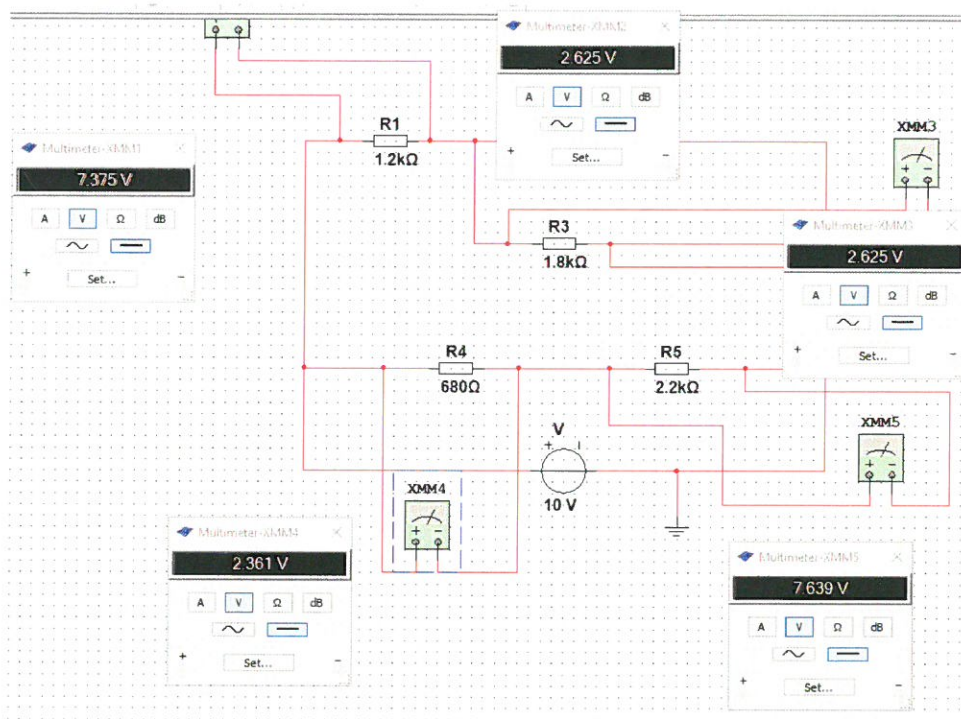
$$V2 = V3 = V - V1 = 10 - 7.37 = 2.63 \text{ V}$$

The Table below illustrates the values obtained in the calculations.

Resistor Identification	Voltage across the Resistor(v)
<i>R1</i>	7.37V
<i>R2</i>	2.63V
<i>R3</i>	2.63V
<i>R4</i>	2.36V
<i>R5</i>	7.64V

c)

The DC circuit shown in Task 1 a) was simulated using Multisim computer software. Below the results can be seen using the multi-meter function in the program.



The table below clarifies the simulated results gathered from Multisim.

Resistor Identification	Voltage across the Resistors using Multisim (v)	Calculated Voltage across the Resistors(v)	Measured Voltage across the Resistors (v)
R1	7.37V	7.37V	7.33 V
R2	2.63V	2.63V	2.67 V
R3	2.63V	2.63V	2.67 V
R4	2.36V	2.36V	2.35 V
R5	7.64V	7.64V	7.65 V

d)

Using the circuit values provided a Matlab program was developed to determine the voltage across each resistor.

A copy of the designed program is illustrated below, providing an explanation of each stage of the program design.

```
1  % Task 1 DC Cicuits by Sean Beynon 11/10/18
2
3  - V=input('Enter the Voltage: ');           %Supply Voltage
4  - R1=input('Enter R1: ');                   %Resistor value for R1
5  - R2=input('Enter R2: ');                   %Resistor value for R2
6  - R3=input('Enter R3: ');                   %Resistor value for R3
7  - R4=input('Enter R4: ');                   %Resistor value for R4
8  - R5=input('Enter R5: ');                   %Resistor value for R5
9
10 % Calculate the V4 and V5
11 - I1 = V/(R4+R5);                           %Calculates current through R4 and R5
12 - V4= I1*R4;                                %Calculates the Voltage across R4
13 - V5=I1*R5;                                %Calculates the Voltage across R5
14
15 % Calculate V1, V2 and V3
16 - R23=R2*R3/(R2+R3);                       %Calculates resistance of R2 and R3 in parallel
17 - I2=V/(R1+R23);                           %Calculates current through R1 and R23
18 - V1=I2*R1;                                %Calculates voltage across R1
19 - V2=V-V1;                                  %Calculates Voltage across R23
20 - V3=V2;
21
22 % Display the Values
23 - disp('the voltage across R1 equals:')
24 - display(V1)
25 - disp('the voltage across R2 equals:')
26 - display(V2)
27 - disp('the voltage across R3 equals:')
28 - display(V3)
29 - disp('the voltage across R4 equals:')
30 - display(V4)
31 - disp('the voltage across R5 equals:')
32 - display(V5)
```

The original values provided in the task description were entered as below:

Enter the Voltage: 10
Enter R1: 1200
Enter R2: 560
Enter R3: 1800
Enter R4: 680
Enter R5: 2200

On completion of entering the specified data, the program outputted a set of results as shown below.

The voltage across R1 equals: $V1 = 7.375$

The voltage across R2 equals: $V2 = 2.625$

The voltage across R3 equals: $V3 = 2.625$

The voltage across R4 equals: $V4 = 2.361$

The voltage across R5 equals: $V5 = 7.638$

The results from the Matlab program provides the same or very similar results to those collated from Multisim, calculated and measured results which are illustrated in part c) of this task.

The only small discrepancies are in the measured set of results, which could be due to either resistor tolerances or the calibration of the test equipment.

Part b) of Task 1 also required a new set of values to be inputted into the Matlab program to ensure the program is operating correctly.

The new values that were selected are as shown below:

Enter the Voltage: 12

Enter R1: 980

Enter R2: 1300

Enter R3: 680

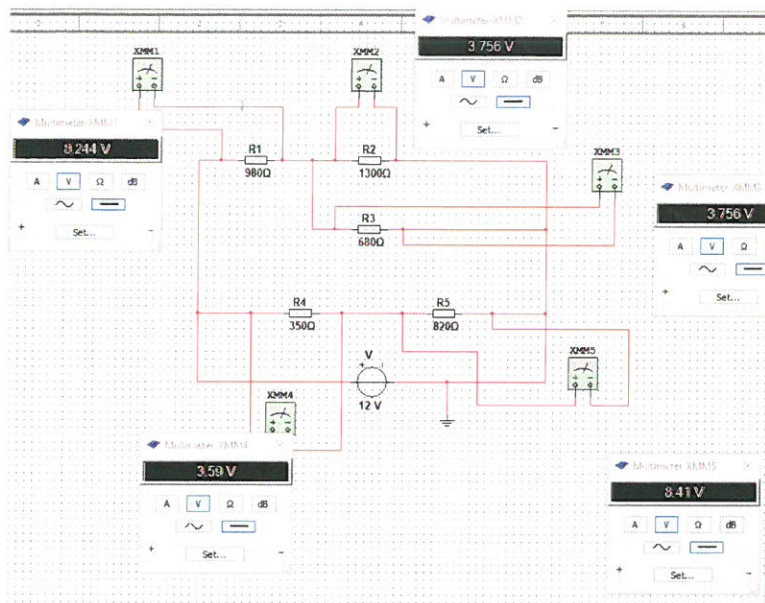
Enter R4: 350

Enter R5: 820

The results for both the Matlab and Multisim are collated in the following table. Multisim was used to confirm that the program is functioning correctly. The table demonstrates that this is the case.

Voltage across Resistor	Matlab results in volts	Multisim results in volts
R1	8.24	8.24
R2	3.76	3.76
R3	3.76	3.76
R4	3.59	3.59
R5	8.41	8.41

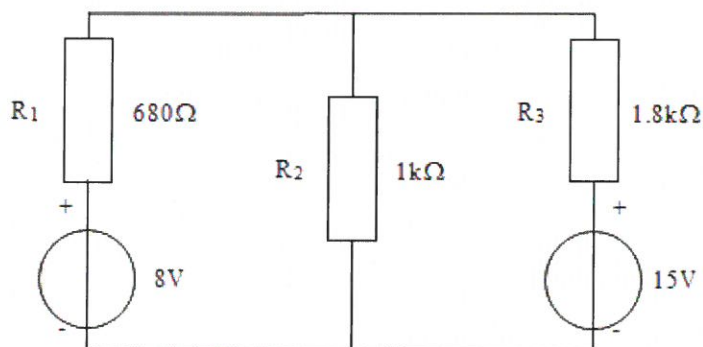
The Multisim circuit below provides evidence of the results achieved.



Task 2- Circuit Theorem

a)

The circuit shown below is classed as a complex DC circuit. Task 2 requires different methods to determine the current flowing through the resistor R2.



The first method applied to determine R2 was mathematical calculation using 'Norton's theorem'.

$$I = \left(\frac{r}{R + r} \right) I_{sc}$$

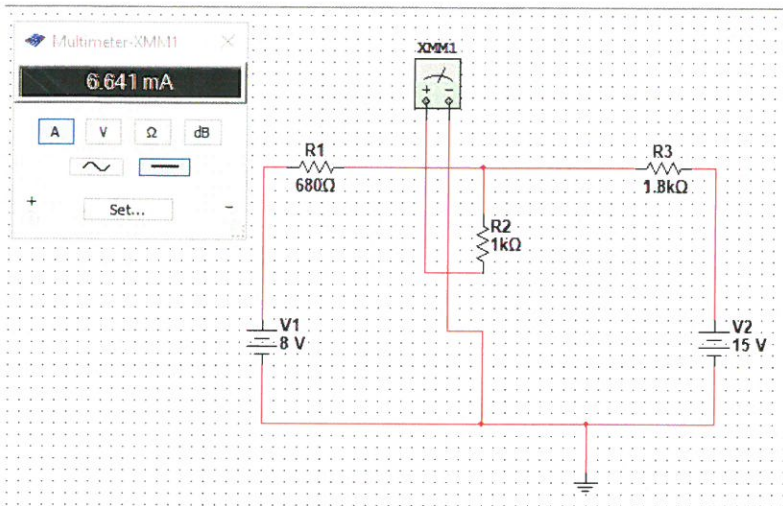
The calculation below illustrates the working out.

$$I_{sc} = I_1 + I_2 = \frac{8}{680} + \frac{15}{1800} = 0.02 \text{ A}$$

$$R = \frac{680 \times 1800}{680 + 1800} = 493.55 \Omega$$

$$\left(\frac{493.55}{493.55 + 1000} \right) \times 0.02 \text{ A} = 6.6 \text{ mA}$$

The results was then confirmed by using Multisim, this is shown in the following circuit diagram. Both theories confirm that the current flowing through **R2 = 6.6 Ma.**



b)

Attached below is the designed Matlab program for calculating the current flowing through R2 using both Thevenin's and Norton's Theorem, providing explanation of each stage of the Calculations.

% Task 2 Thevenin's and Norton's by Sean Beynon 11/10/18

```
V1=input('Enter the 1st Supply Voltage: '); %1st Supply Voltage
V2=input('Enter the 2nd Supply Voltage: '); %2nd Supply Voltage
R1=input('Enter resistance for R1: '); %Resistor value for R1
R2=input('Enter resistance for R2: '); %Resistor value for R2
R3=input('Enter resistance for R3: '); %Resistor value for R3
```

%Thevinens

% Calculate the current I1

```
I1 = (V2-V1)/(R1+R3);
```

%Calculates current I1

% Calculate voltages

```
E1= V1+(I1*R1);
```

%Calculates volatage 1

```
E2= V2-(I1*R3);
```

%Calculates voltage 2

%Calculate equivalent resistance

```
r=(R1*R3)/(R1+R3);
```

%Calculates resistance

%Calculate equivalent current

```
I=E1/(R2+r);
```

%Calculates equivalent current

%Power Dissapated

```
P=(I^2)*r ;
```

%Nortons

```
In1=V1/R1;
```

%Calculate current 1

```
In2=V2/R3;
```

%Calculates current 2

```
Isc=In1+In2; %Calculates short circuit current

rn=((R1*R3)/(R1+R3)); %Calculates resistance

%Norton equivalent
In=(rn/(rn+R2))*Isc; %Calculates equivalent current

% Display the Values
disp('Thevenins, current I1:')
display(I1)
disp('Thevenins, equivalent resistance r:')
display(r)
disp('Thevenins, equivalent current I:')
display(I)
disp('Thevenins, power dissipated P:')
display(P)
disp('Nortons, current 1 In1:')
display(In1)
disp('Nortons, current 2 In2:')
display(In2)
disp('Nortons, short circuit current Isc:')
display(Isc)
disp('Nortons, equivalent resistance rn:')
display(rn)
disp('Nortons, equivalent current In:')
display(In)
```

The values in the complex DC circuit diagram were inputted into the Matlab program.

These values are re-iterated below:

Enter the 1st Supply Voltage: 8

Enter the 2nd Supply Voltage: 15

Enter resistance for R1: 680

Enter resistance for R2: 1000

Enter resistance for R3: 1800

The results for Thevenin's and Norton's theorems are highlighted below in red. The other values have been displayed to allow cross reference to the calculated values at each stage of the calculation. This also provides evidence that the Matlab program operates correctly.

Thevenin's, current I1 = 0.0028

Thevenin's, equivalent resistance r = 493.5484

Thevenin's, equivalent current I = 0.0066

Thevenin's, power dissipated P = 0.0218

Norton's, current 1 $I_{n1} = 0.0118$

Norton's, current 2 $I_{n2} = 0.0083$

Norton's, short circuit current $I_{sc} = 0.0201$

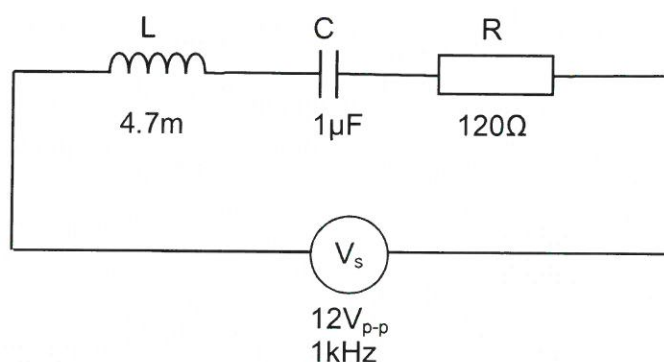
Norton's, equivalent resistance $r_n = 493.5484$

Norton's, equivalent current $I_n = 0.0066$

Have not found the current in each resistor

Task 3- Series AC circuit

Task 3 involves investigating different methods to determine the phase angle and resonant frequency of the series AC circuit shown below.



a)

The first method requires the mathematical calculation of the phase angle and resonant frequency.

$$X_L = 2\pi fL = 2\pi * 1000 * 4.7 \times 10^{-3} = 29.531 \Omega$$

$$X_C = \frac{1}{2\pi fC} = \frac{1}{2\pi * 1000 * 1 \times 10^{-6}} = 159.155 \Omega$$

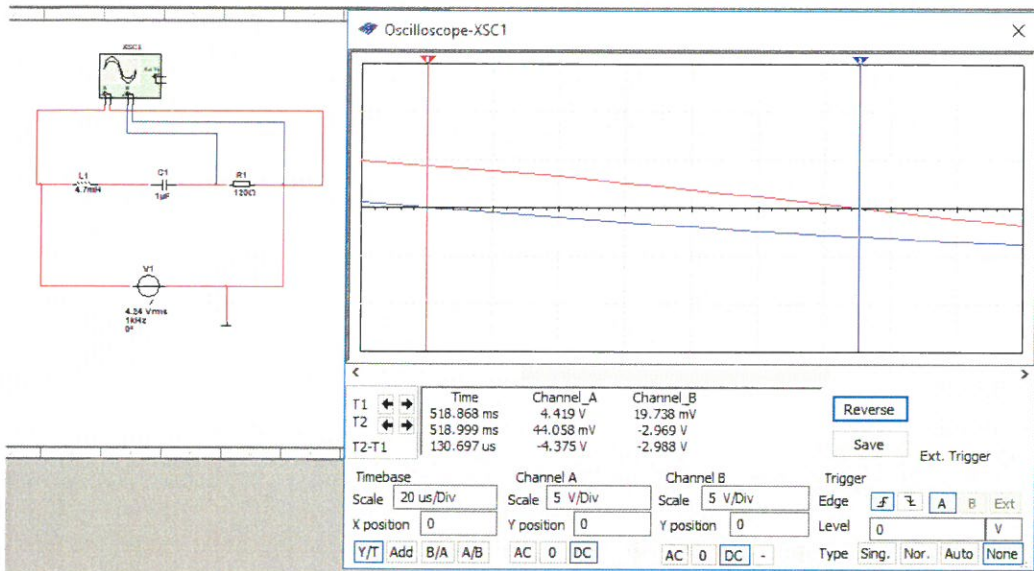
$$\theta = \tan^{-1} \left(\frac{X_L - X_C}{R} \right) = \tan^{-1} \left(\frac{29.531 - 159.155}{120} \right) = 47.21^\circ$$

$$f_0 = \frac{1}{2\pi\sqrt{LC}} = \frac{1}{2\pi\sqrt{4.7 \times 10^{-3} * 1 \times 10^{-6}}} = 2.32 \text{ kHz}$$

The series AC circuit produced a phase angle of 47.21° and a resonant frequency of 2.32kHz.

b)

The series AC circuit was constructed within Multisim, using the oscilloscope measuring facility to determine the time (t) between each phase as it crosses the origin.



Time (t) =

130.697 µs

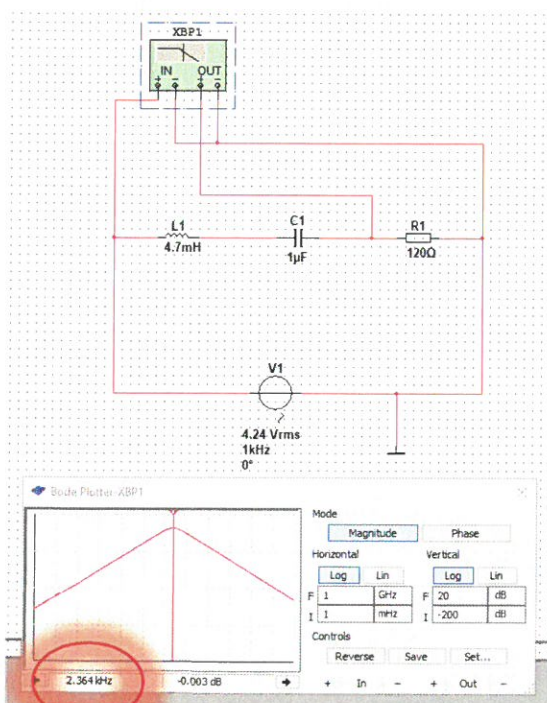
Frequency (f)

= 1 kHz

Once the time is confirmed, a short calculation can be completed to determine the phase angle.

$$\phi = f * t * 360^\circ = 1000 * 130.697 * 10^{-6} * 360 = 47.05^\circ$$

A phase angle of 47.05° was achieved using Multisim. The phase angle slightly differs from the 47.21° that was achieved from the calculated method, this could be due the scale of the oscilloscope when the points were placed.



Within the Multisim package there is a facility to determine the resonant frequency by measuring the peak of the waveform.

The image of the Bode plotter provides evidence that the resonant frequency of the circuit measured in Multisim is 2.364 kHz

Again this measured is slightly different to the calculated value of 2.32 kHz. This could be effected by the scale of the plotter when the point is located at the peak and/ or the rounding off of numbers when carrying out the calculations.

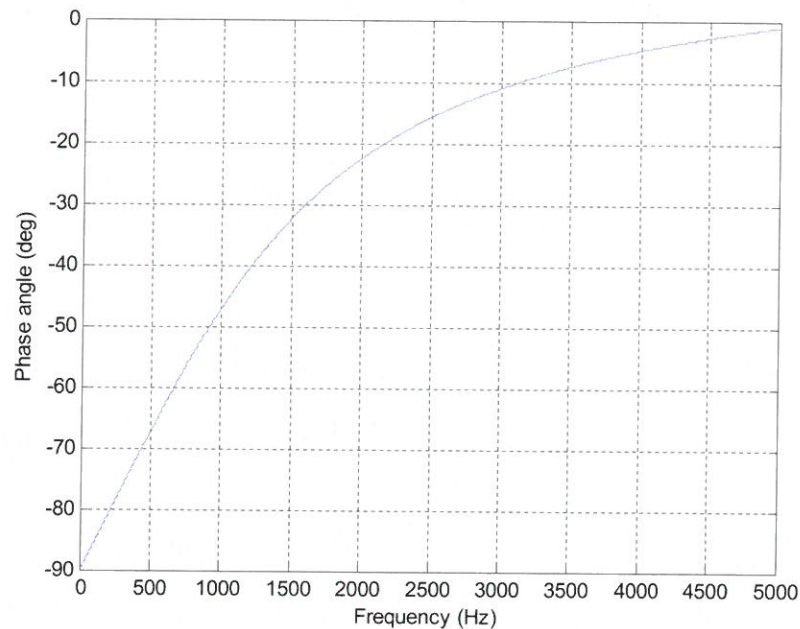
c)

In this part of the task a Matlab program was designed to determine the phase angle and resonant frequency, producing a waveform plot to illustrate the phase difference. Below is the program that was created to deliver the necessary values.

```
1  % Task 3 Series AC Circuit by Sean Beynon 29/11/18
2
3  %Inputs
4
5  L=input('Enter the value of the Inductor:');
6  C=input('Enter the value of the Capacitor:');
7  R=input('Enter the value of the Resistor:');
8  V=input('Enter the supply voltage:');
9  f=input('Enter the frequency of the supply voltage:');
10 fprintf('\n') %Blank line
11
12 %Reactance Calculation
13
14 Xl=2*pi*f*L;
15 Xc= 1/(2*pi*f*C);
16
17 %Phase angle calculation
18
19 Angle=(180/pi)*atan((Xl-Xc)/R);
20
21 %Resonant Frequency
22
23 RF=1/(2*pi*sqrt(L*C));
24
25 %For loop to calculate the phase angle
26
27 for A=1:5000
28     freq(A)=A;
29     Xl=2*pi*freq(A)*L;
30     Xc= 1/(2*pi*freq(A)*C);
31     Phzang(A)=(180/pi)*atan((Xl-Xc)/R);
32 end
33
34 %Display the results
35
36 X1=['The phase angle is: ',num2str(Angle),'degrees'];
37 disp(X1)
38
39 X2=['The resonant frequency is: ',num2str(RF),'Hz'];
40 disp(X2)
41
42 plot(freq,Phzang,'b-');
43 xlabel('Frequency (Hz)');
44 ylabel('Phase angle (deg)');
45 grid;
46
```

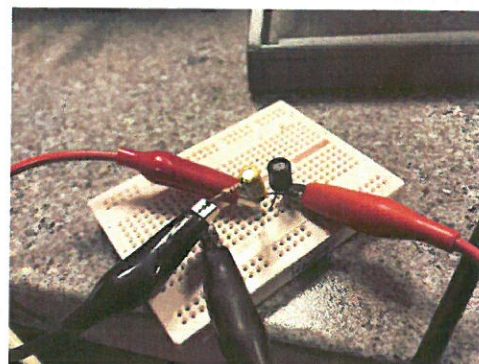
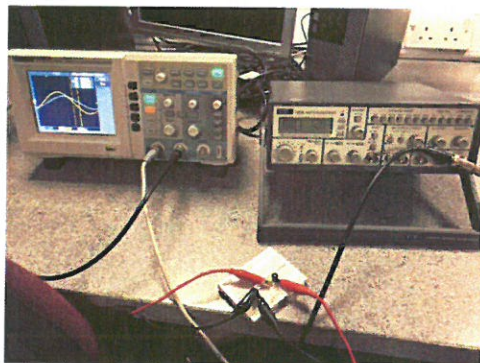
The values from the series AC circuit were entered into the Matlab program, providing results of 47.21° phase angle and a resonant frequency of 2.32 kHz.

The task criteria required a waveform to be plotted. The plot below illustrates how the phase angle differs at different frequencies.

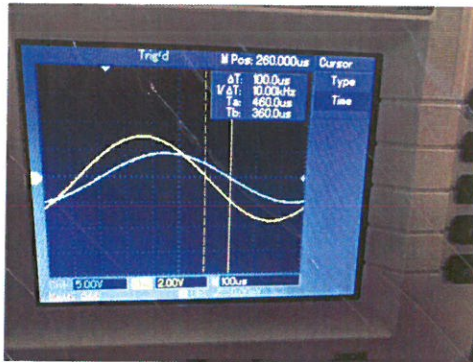


d)

The final method for determining the phase angle and resonant frequency was to complete a practical assessment. The series AC circuit was constructed using breadboard, function generator, oscilloscope and components specified in the task description.



The function generator was set at a voltage at 12V and a frequency of 1 kHz. The image of the function below provides evidence of the values set. As seen in the image below.



The markers on the oscilloscope were set on each waveform where they cross the origin. This provides a time interval between the waveforms, allowing a phase angle to be calculated using the $\Theta^t = f \cdot t \cdot 360$.

$$\Theta^t = f \cdot t \cdot 360$$

$$\Theta^t = 1\text{kHz} \cdot 100 \mu\text{s} \cdot 360^\circ$$

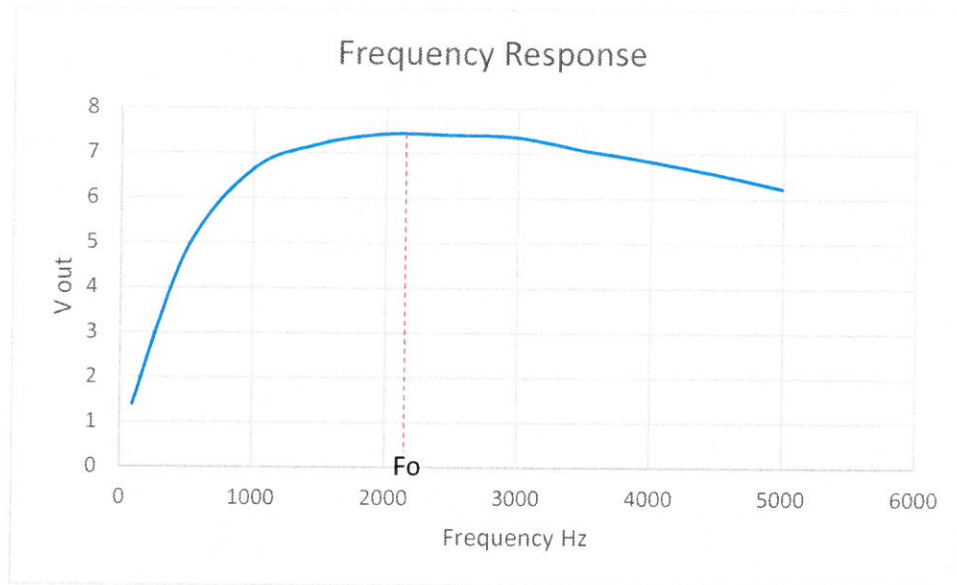
$$\Theta^t = 36^\circ$$

The practical method presents a quite substantial difference of phase angle at 36° compared to the simulated and calculated methods. This could be due to the calibration of equipment, such as the function generator and the oscilloscope and/ or component tolerance values.

A frequency response test was carried out to characterize the dynamics of the system. This is achieved by changing the frequency to different values and recording the voltage output. The table below shows the results from this investigation.

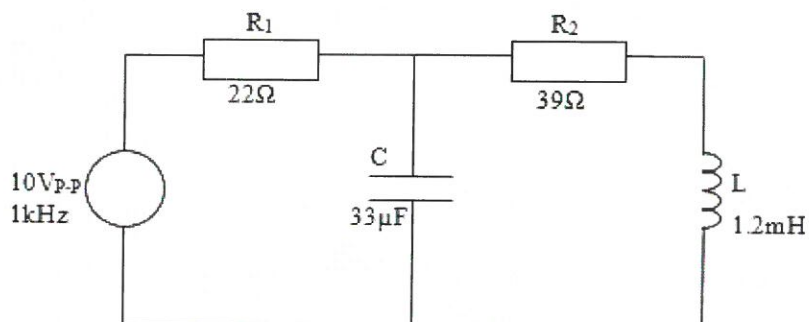
FREQUENCY (Hz)	Voltage Out (V)
100	1.39
500	4.8
1000	6.62
1500	7.19
2000	7.42
2500	7.4
3000	7.35
3500	7.07
4000	6.83
4500	6.55
5000	6.23

After completing the table of results a graph was plotted to illustrate the dynamics of the system. The red dotted line demonstrates where on the plotted graph the calculated resonant frequency is located.



Task 4- Complex AC circuits

The values in the complex AC circuit below were used to calculate the current flowing through the $33\mu\text{F}$ capacitor using Mesh theorem.



$$X_L = 2\pi fL = 2\pi \times 1000 \times 1.2 \times 10^{-3} = j7.539 \quad \checkmark$$

$$X_C = \frac{1}{2\pi fC} = \frac{1}{2\pi \times 1000 \times 33 \times 10^{-6}} = -j4.82 \quad \checkmark$$

Loop 1

$$\left. \begin{aligned} V &= R_1 I_1 + X_C(I_1 - I_2) \\ 10 &= 22 I_1 + (-j4.82)(I_1 - I_2) \\ 10 &= (22 - j4.82)I_1 + j4.82 I_2 \end{aligned} \right\} \text{Equation (1)} \quad \checkmark$$

Loop 2

$$\left. \begin{aligned} 0 &= R_2 I_2 + X_L I_2 + X_C(I_2 - I_1) \\ 0 &= 39 I_2 + j7.539 I_2 + (-j4.82)(I_2 - I_1) \\ 0 &= j4.82 I_1 + (39 + j2.72) I_2 \end{aligned} \right\} \text{equation (2)} \quad \checkmark$$

Rearrange equation 2

$$\left. \begin{aligned} j4.82 I_1 &= -(39 + j2.72) I_2 \\ I_1 &= \frac{-(39 + j2.72) I_2}{j4.82} \end{aligned} \right\} \text{equation (3)} \quad \checkmark$$

Sub I_1 equation (3) into equation (1)

$$10 = \frac{-(22 - j4.82)(39 + j2.72) I_2}{j4.82} + j4.82 I_2 \quad \checkmark$$

$$10 = j0.207 (22 - j4.82)(39 + j2.72) I_2 + j4.82 I_2$$

$$10 = j0.207 (871.11 - j128.14) I_2 + j4.82 I_2 \quad \checkmark$$

$$10 = (26.525 + j180.32)I_2 + j4.82 I_2$$

$$I_2 = \frac{10}{26.525 + j185.14}$$

$$I_2 = 0.0076 - j0.0529$$

Substitute I_2 into equation (3)

$$I_1 = \frac{- (39 + j2.72) \times (0.0076 - j0.0529)}{j4.82}$$
$$= 0.4237 + j0.0913$$

$$I = I_1 - I_2 = (0.4237 + j0.0913) - (0.0076 - j0.0529)$$
$$= 0.4161 + j0.1442$$
$$= 0.44 \angle 19.11^\circ \text{ A}$$

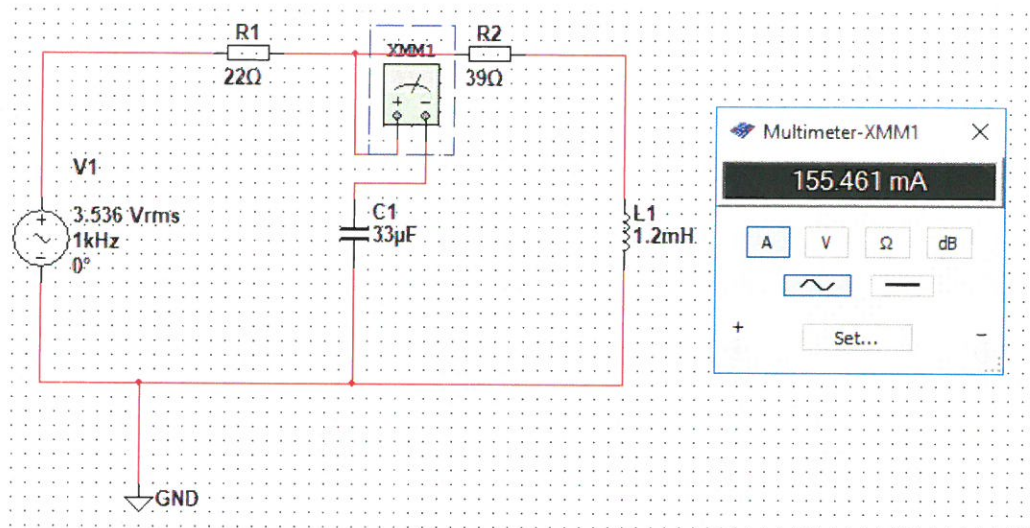
Change from Peak to Peak. to Irms

$$\frac{0.44}{2 \times \sqrt{2}} = 0.156 \angle 19.11^\circ \text{ A}$$

Good work but you might want to type this up.

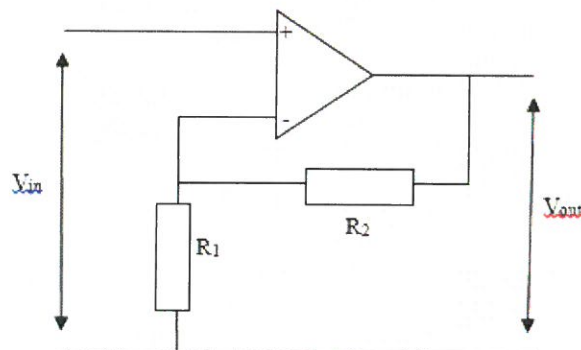
Multisim was used to confirm that the calculated results are correct and similar values are obtained.

The circuit produced in Multisim shown below illustrates that the simulated current of 155.45mA is very similar to the calculated value of 0.157A. This very slight difference is probably due to rounding off of numbers when carrying out the calculations.



Task 5- Non-inverting amplifier circuits

The following circuit shows a non-inverting amplifier circuit. To be able complete a practical investigation, firstly the values for R_{in} and R_f need to be determined.



a)

The task description defines that a gain of 200 must be achieved. $AV = 200$

$$A_v = 1 + \frac{R_f}{R_{in}}$$

$$\frac{R_f}{R_{in}} = AV - 1$$

$$R_f = (AV - 1) * R_{in}$$

$$R_f = (200 - 1) * R_{in}$$

For the purpose of this task I chose a 3.3KΩ resistor for R_{in} .

$$R_f = 199 * 3.3 \text{ k}\Omega$$

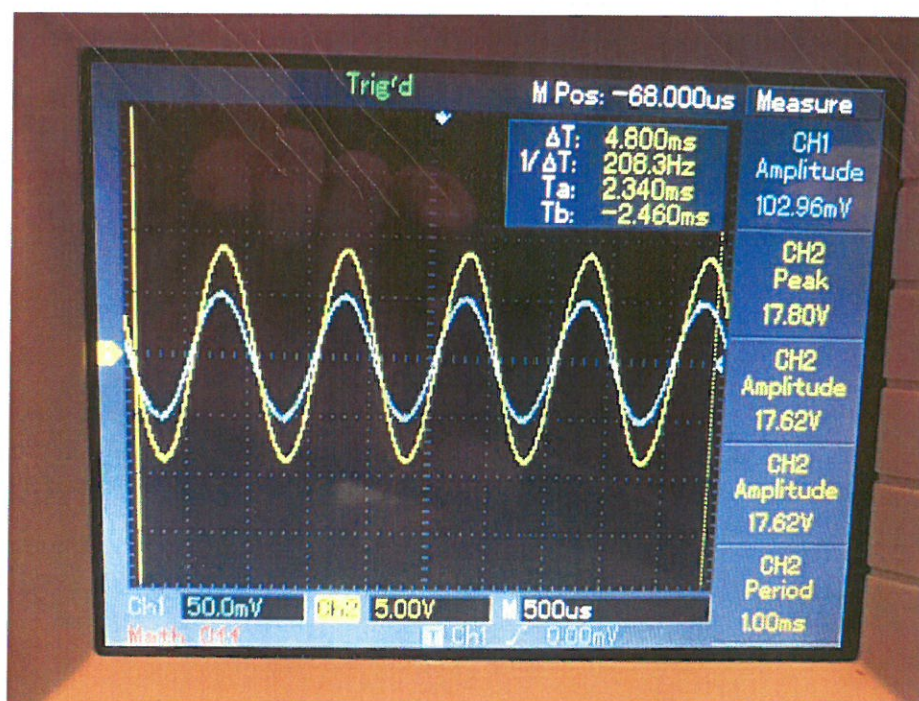
$$R_f = 656.7 \text{ k}\Omega$$

b)

The circuit was constructed using breadboard, a 741 op amp, and two resistors.

For the practical investigation a 680 k Ω resistor was used, due to it being the closest available resistor to the calculated results, and also a 3.3 k Ω resistor.

An oscilloscope was used to determine the voltage gain by using the amplitude for each waveform.



The image of the oscilloscope above provides evidence of the amplitudes achieved from each waveform.

Output voltage= 17.62 V

Input voltage= 102.96 mV

$$\text{Voltage gain} = \frac{\text{Output voltage}}{\text{input voltage}}$$

$$\text{Voltage gain} = \frac{17.62}{102.96 \times 10^{-3}}$$

Voltage gain = 171.13

To determine the bandwidth for the circuit, voltages were recorded at various different frequencies. The gain for each voltage could be calculated using the formula:

$$\text{Voltage gain} = \frac{\text{Output voltage}}{\text{input voltage}}$$

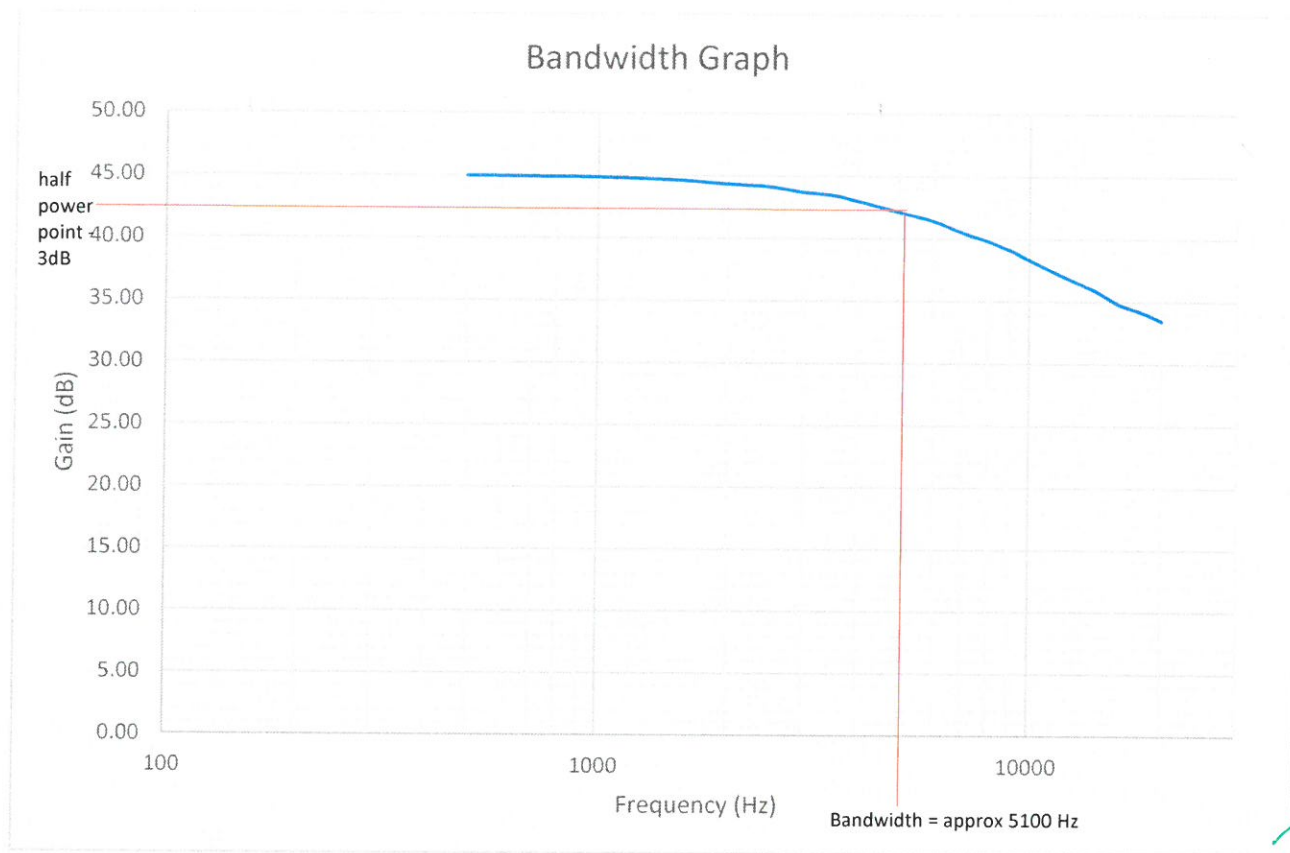
On completion of the calculation for the Gain, the Gain (dB) could be calculated using the formula:

$$\text{Gain (dB)} = 20 \cdot \log(\text{Gain})$$

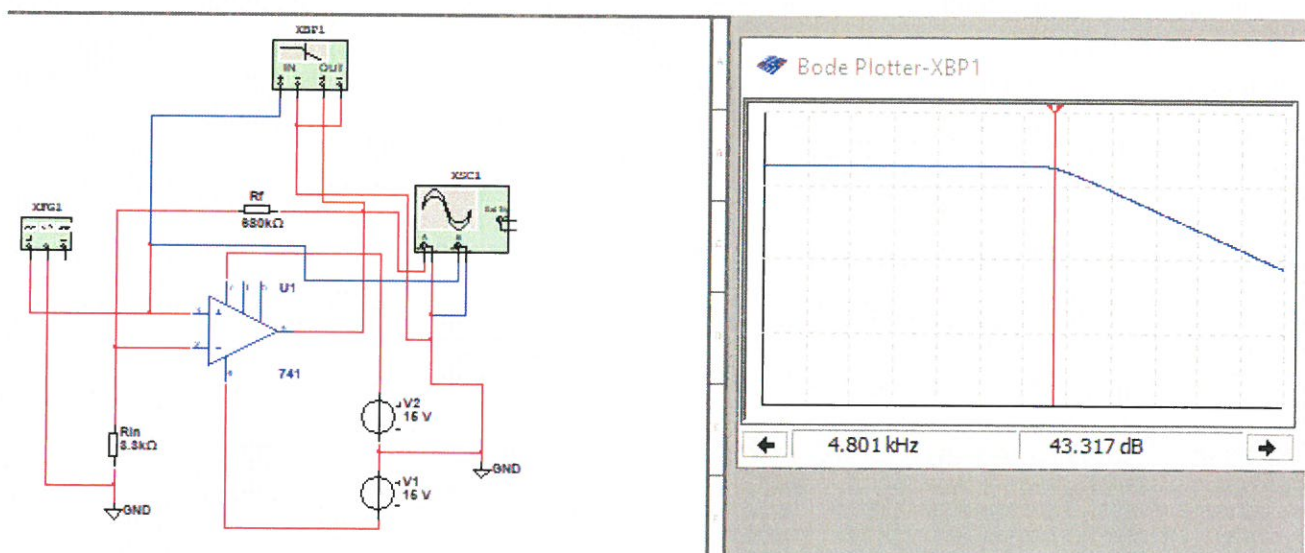
The results can be seen in the table below.

Frequency (Hz)	Output voltage (V)	Gain	Gain (dB)
500	18	176.47	44.93
1000	17.8	174.51	44.84
1500	17.4	170.59	44.64
2000	16.8	164.71	44.33
2500	16.4	160.78	44.12
3000	15.6	152.94	43.69
3500	15.2	149.02	43.46
4000	14.4	141.18	43.00
5000	13	127.45	42.11
6000	12	117.65	41.41
7000	10.8	105.88	40.50
8000	10	98.04	39.83
9000	9.2	90.20	39.10
10000	8.4	82.35	38.31
12000	7.2	70.59	36.97
14000	6.4	62.75	35.95
16000	5.6	54.90	34.79
18000	5.2	50.98	34.15
20000	4.8	47.06	33.45

A graph of the Gain (dB) against frequency was plotted from the results in the above table to determine the bandwidth. The half power point, which is a point of the Gain that is 3dB lower than the peak, is marked on the graph to determine the bandwidth. The following graph shows that the bandwidth is approximately 5.1 kHz.



The results were confirmed using a Multisim simulation of the circuit.



The 'bode plotter' function was used to establish the peak frequency. Then the half power point could be marked on the plot to confirm the bandwidth frequency.

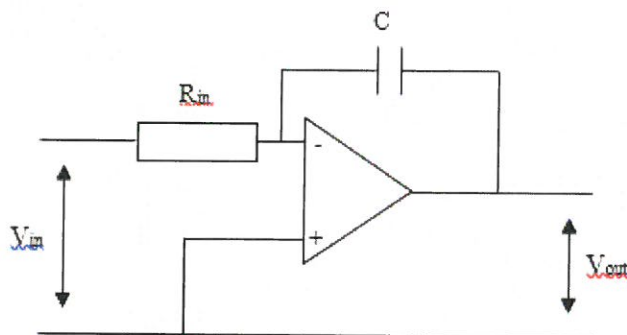
$$46.3\text{dB} - 3\text{dB} = 43.3\text{dB}$$

Therefore the bandwidth frequency = **4.8 kHz**

The conclusion of this investigation shows a small difference of 0.3 kHz. This discrepancy could be a result of a number of affects, such as accuracy of reading graphs, accuracy of test equipment and reading the results, possible resistor tolerances of 5% and also the accuracy of the function generator. Although there are when contributing factors, the result are considered fairly accurate.

Task 6- Integrator circuits

The circuit below shows an integrator circuit using an op amp.



$C = 1.5\mu\text{F}$
 $R_{in} = 200\Omega$
 $f = 200\text{Hz}$
 $V_{in} = 1\text{V}$

a)

A Matlab program was developed to display the output waveform when a square waveform is applied.

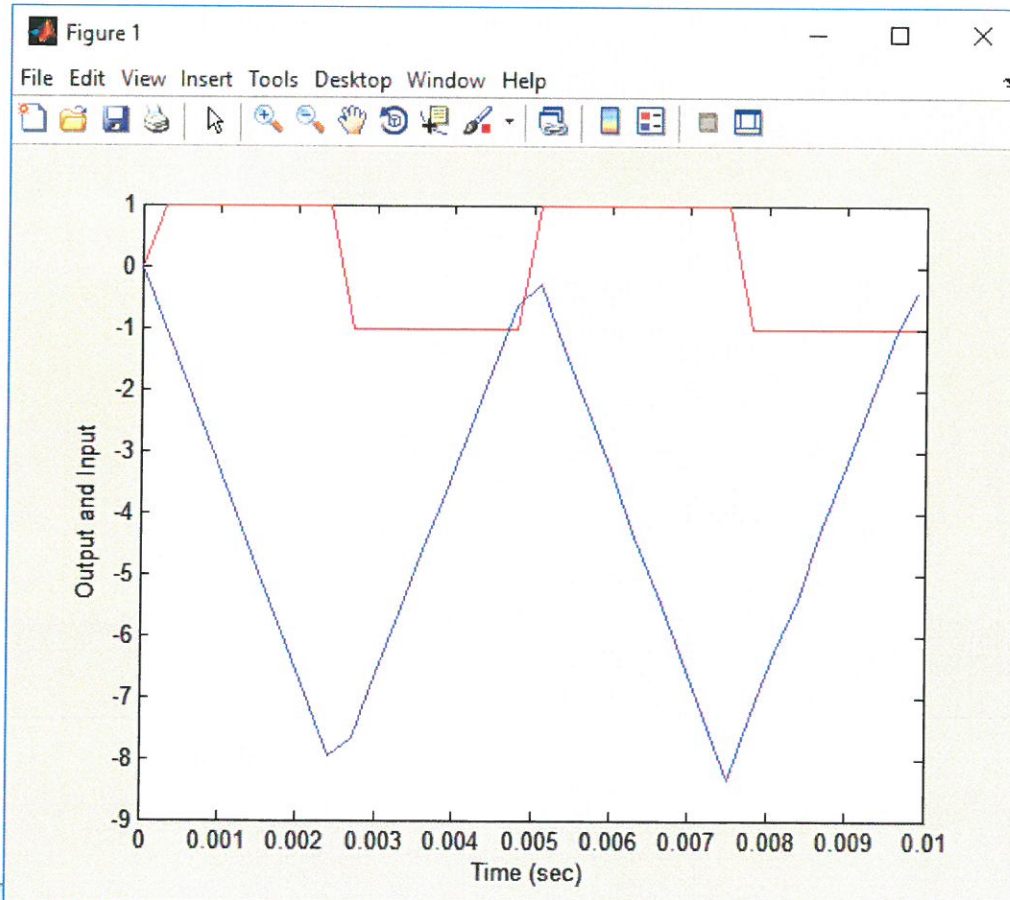
The first program was developed to create a function, Vin, that will utilized in the main program that will display the output waveform.

```

1      % Task 6    Integrator circuit (Vin) | by Sean Beynon 17/01/18
2
3      %Calculate a function value at point x
4      function[V] = Vin(x)
5
6      %The function expression
7 -    V = 1*sign(sin(2*pi*200*x));
  
```

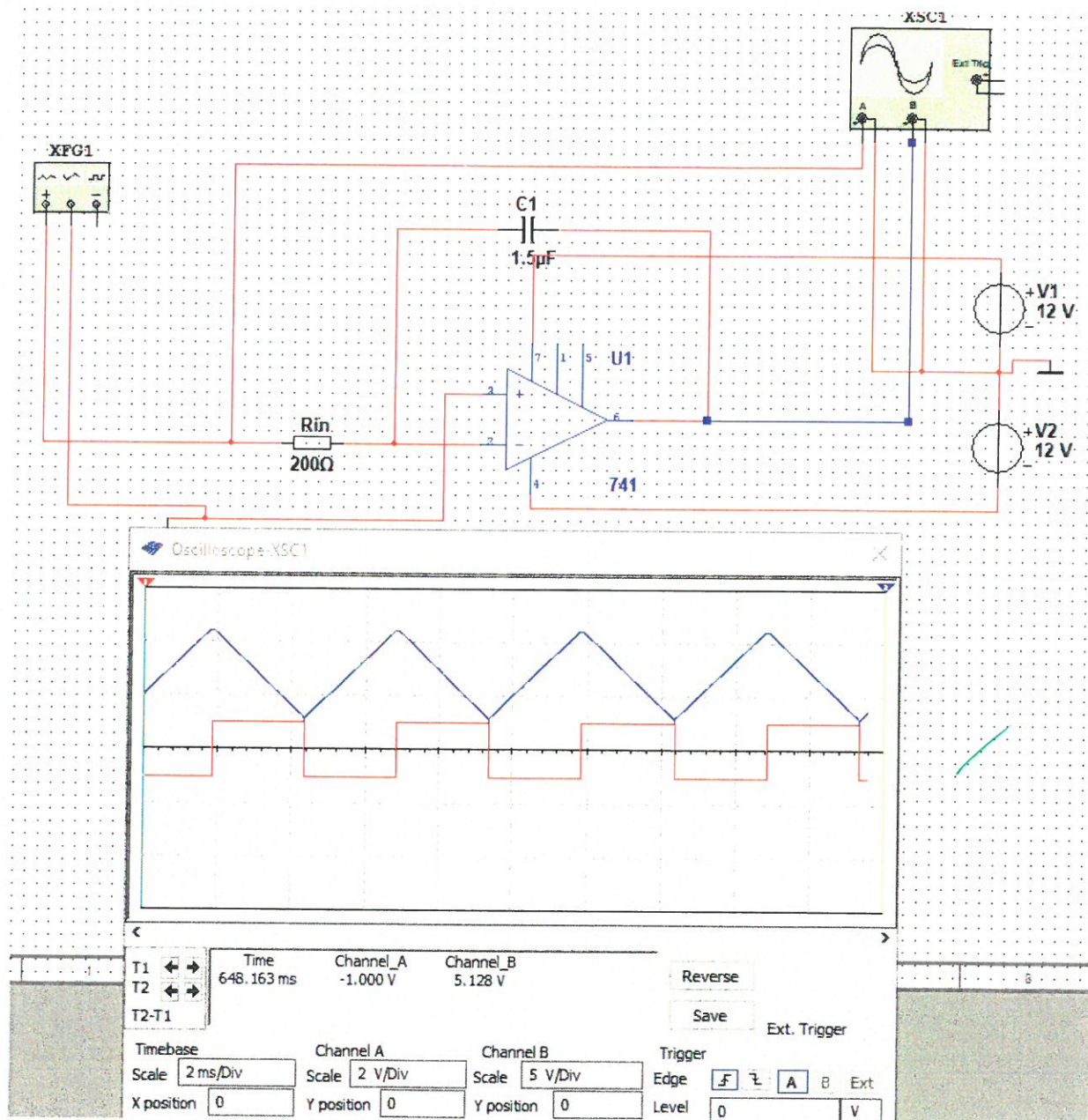
The following image shows the Matlab program created. The plot in the image illustrates the waveforms, the red showing the inputted square waveform and the blue showing the output triangular waveform.

```
1 % Task 6 Integrator circuit by Sean Beynon 17/01/18
2
3 % Parameters
4 - R1 =200; C=0.0000015; f= 200; %Input values
5
6 % Time vector
7 - t = 0:0.0003:0.01; %Time scale
8
9 % Output signal
10 - for m = 1:length(t) %Output signal calculations
11 - Vo(m) = -1/(R1*C)*quad('Vin',0,t(m));
12 - end;
13
14 % Input signal
15 - Vi = 1*sign(sin(2*pi*f*t)); %Input signal calculation
16
17 % Display results
18 - plot(t,Vi,'r-',t, Vo,'b-');
19 - xlabel('Time (sec)'); ylabel('Output and Input');
20
21
```



b)

The results from the Matlab program were checked by inputting the circuit diagram into Multisim, using the oscilloscope function to determine the waveforms.



As a conclusion it can be seen that both methods of determining the waveform, shows that the input square waveform converts to a triangular waveform output in an integrator circuit.

Although the Matlab plot of the circuit doesn't look as sharp and as square as the waveform produced from the Multisim circuit. This is due to the number of samples obtain during the Matlab program.

A very good assignment that showed
a good understanding of the requirements
of the assignment.

Very good presentation, would have
been nice if all your calculations were
done on computer.