

LEVEL 5 ASSESSMENT SPECIFICATION

Student name:	Paul Miller	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	12.2
Achieved 50% for Task 1. Achieved 60% for Task 2. Achieved 65% for Task 3. Achieved 85% for Task 4. Achieved 45% for Task 5. Knowledge is reasonably extensive and accurate in most parts. Show a very good understanding of most concepts and principles covered by this assignment but not all of the elements were covered for some of the tasks.		
3. Cognitive and Intellectual Skills	20	4.3
Achieved 35% for Task 1. Achieved 30% for Task 3. Achieved 0% for Task 5. For the tasks that asked for comments the work was almost wholly descriptive. Little or no critical evaluation and analysis of concepts and principles covered by these tasks.		
4. Practical Application Skills	50	32.1
Achieved 60% for Task 1. Achieved 65% for Task 2. Achieved 70% for Task 3. Achieved 70% for Task 4. Achieved 40% for Task 5. Achieved 80% for Task 6. A very good application of established methods in the use of Matlab, Multisim and practical work but in places not all of the required work was produced.		
5. Transferable Skills for Life and Professional Practice	10	5
Mostly coherent, organised and accurate work, in a suitable structure and is for the most part clearly expressed.		

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)

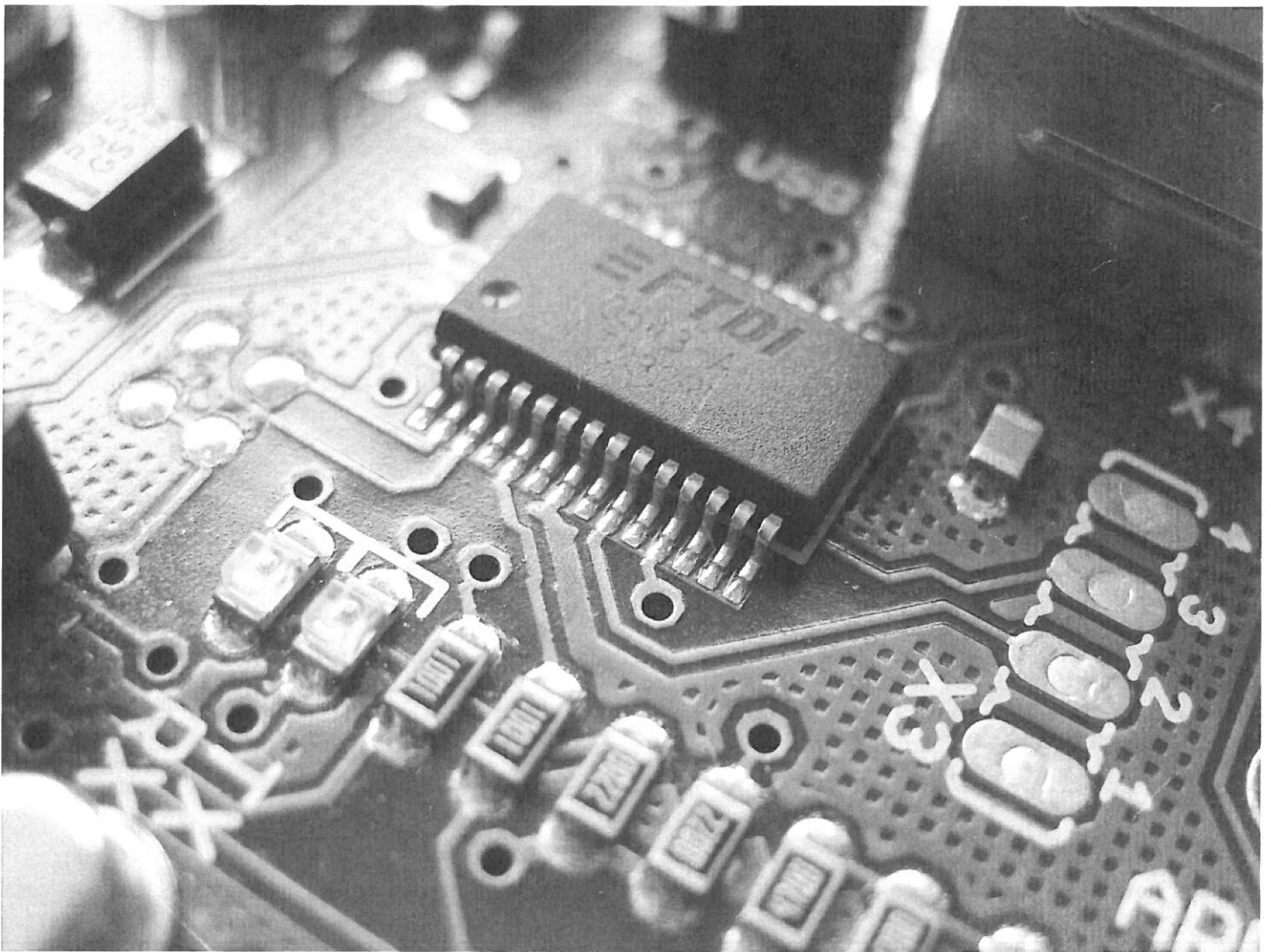
Up to 1 week late (40% Max)

Over 1 week late (0%)

54%

Electronic Principles

PDSE 5009



Paul Miller (141589)

HNC Year 2

Jason Pointer

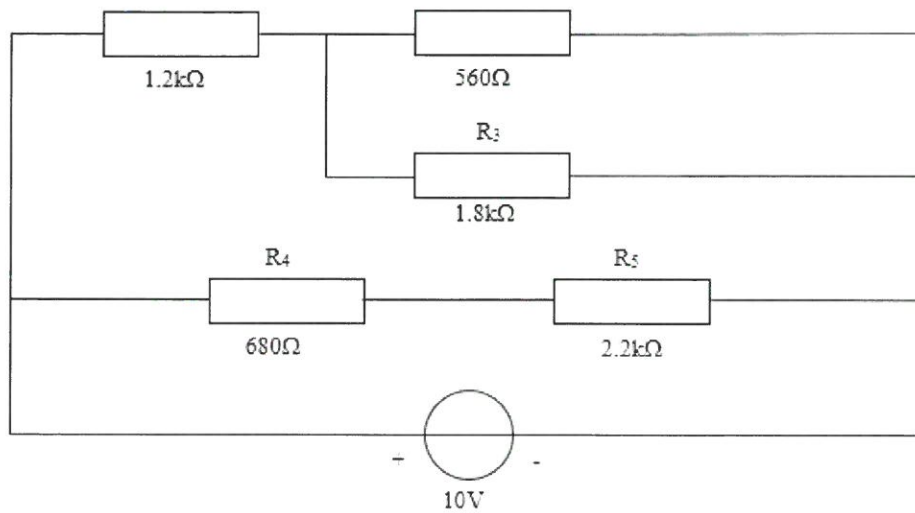
Contents:

Pages 3 – 5.....	Task 1
Pages 6 – 9.....	Task 2
Pages 10 – 13.....	Task 3
Pages 14 – 15.....	Task 4
Pages 16 – 19.....	Task 5
Pages 20 – 23.....	Task 6

DC Circuits

Task 1 – Combinational Circuit

The circuit below shows a combinational dc circuit:



- Build the circuit using breadboard and measure the voltage across each resistor.
- Calculate the voltage across each resistance, show all workings.
- Verify the calculated voltages using Multisim and comment upon the results.
- Design a Matlab program that will determine the voltages for the circuit for different values of resistance. Test the program with the original resistor values and a new set of values, confirm the results using Multisim.

a) **Measured** voltages from each resistor:

$$\mathbf{R1\ Voltage = 7.34v}$$

$$\mathbf{R2/3\ Voltage = 2.68v}$$

$$\mathbf{R4\ Voltage = 2.32v}$$

$$\mathbf{R5\ Voltage = 7.69v}$$

b) **Calculated** voltages from each resistor:

$$\mathbf{R1 = V1 = I2 \times R1 = 6.14 \times 1200 = 7.37v}$$

$$\mathbf{R2/3 = Rt = R2 + R3 = 560 + 1800 = 427\Omega}$$

$$\mathbf{I2 = V1 / R1 + Rt = 10 / 1200 + 427.12 = 6.14mA}$$

$$\mathbf{V1 = I2 \times R1 = 6.14 \times 1200 = 7.37v}$$

$$\mathbf{V2/3 = I2 \times R2/3 = 6.14 \times 427 = 2.62v}$$

$$\mathbf{R4/5 = I1 = V / R4 + R5 = 10 / 680 + 2200 = 3.47mA}$$

$$\mathbf{R4\ Voltage = 3.47 \times 680 = 2.36v}$$

$$\mathbf{R5\ Voltage = 3.47 \times 2200 = 7.63}$$

c) Verify the calculated voltages using multisim

Figure 1 shows the circuit once it has been produced in multisim, the values are very close to those of the calculated value, and the measured values are:

$$R1 = 7.375v$$

$$R2 = 2.625v$$

$$R3 = 2.625v$$

$$R4 = 2.361v$$

$$R5 = 7.639v$$

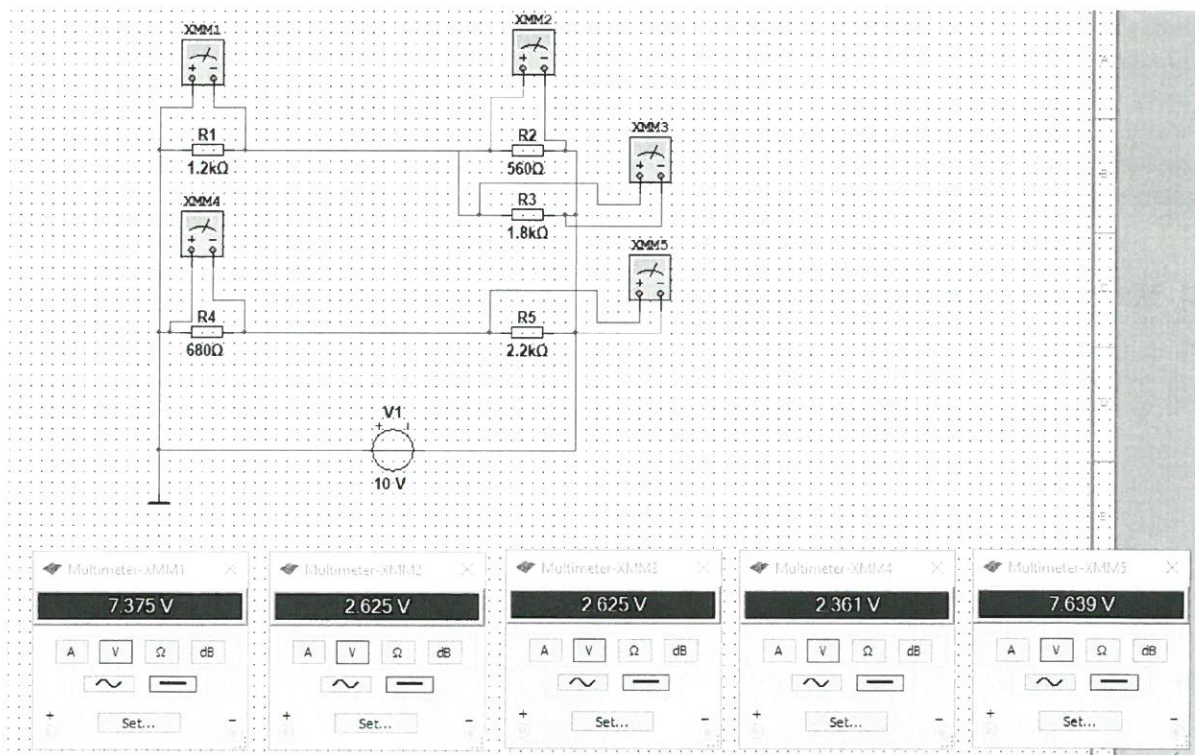


Figure 1

These small discrepancies between the calculated and simulated values can be put down to human error when rounding numbers, multisim may of rounded to a higher decimal place making it slightly more accurate than those of the calculated results.

Any other reasons?

d) Matlab Program:

Figure 2 illustrates the matlab program which was created to find the voltages of the resistors in the circuit given in task 1, **Figure 3** shows the program running, it asks the user to enter the values of the voltage and the resistors, it then puts the numbers into the equations which were programmed

```
%Task 1 - DC Circuits
%By Paul Miller (11/10/18)

V=input('Enter The Supply Voltage');
R1=input('Enter Resistor Value for R1:');
R2=input('Enter Resistor Value for R2:');
R3=input('Enter Resistor Value for R3:');
R4=input('Enter Resistor Value for R4:');
R5=input('Enter Resistor Value for R5:');

%Calculate the V4 and V5

I1=V/(R4+R5);
V4=I1*R4;
V5=I1*R5;

%Calculate the V1, V2 and V3

R23=R2*R3/(R2+R3);
I2=V/(R1+R23);
V1=I2*R1;
V2=V-V1

%Display the Values

disp('The Voltage across R1 Equals:')
disp(V1)
disp('The Voltage across R2/R3 Equals:')
disp(V2)
disp('The Voltage across R4 Equals:')
disp(V4)
disp('The Voltage across R5 Equals:')
disp(V5)
```

Enter The Supply Voltage:10
Enter Resistor Value for R1:1200
Enter Resistor Value for R2:560
Enter Resistor Value for R3:1800
Enter Resistor Value for R4:680
Enter Resistor Value for R5:2200

V2 =
2.6250

The Voltage across R1 Equals:
7.3750

The Voltage across R2/R3 Equals:
2.6250

The Voltage across R4 Equals:
2.3611

The Voltage across R5 Equals:
7.6389

Figure 3

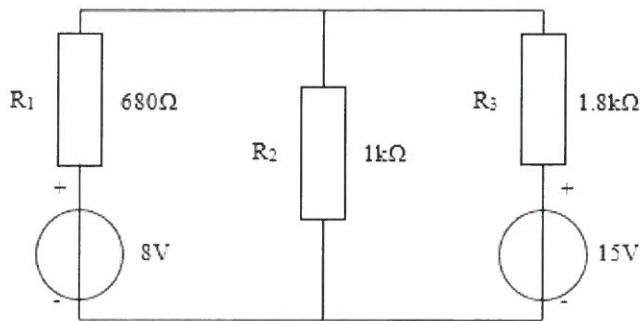
Figure 2

The answers given by the matlab program match those of the simulated values on multisim perfectly, therefore proving the calculations which were completed are correct.

What about checking the program using other values?

Task 2 – Circuit Theorem

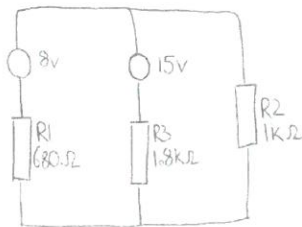
The circuit below can be classed as a complex dc circuit:



- a) Using Thevenin's or Norton's theorem determine the current flowing through R_2 and check the value using Multisim.

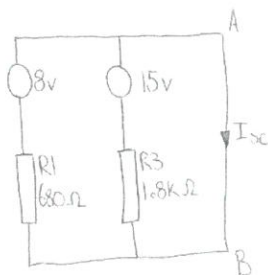
- i) Rearrange The formula:

which method.

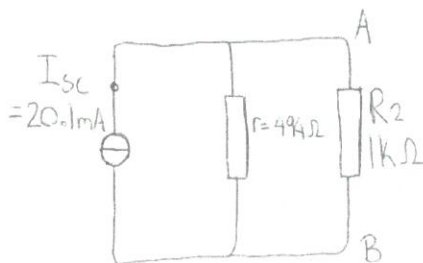


- ii) Short out R_2 branch and find I_{sc} :

$$I_{sc} = I_1 + I_2 = 8 / 680 + 15 / 1800 = 20.1\text{mA}$$



- iii) $r = 680 \times 1800 / 680 + 1800 = 494\Omega$



- iv) $I = (494 / 494 + 1000) \times 20.1 \times 10^{-3} = 6.65\text{mA}$

Figure 4 illustrates the simulated value which was obtained by recreating the circuit on multisim, the calculated value and the measured value are 0.008 difference:

Simulated value: 6.642mA

Calculated Value: 6.65mA

This small difference can be put down to human error when rounding numbers up.

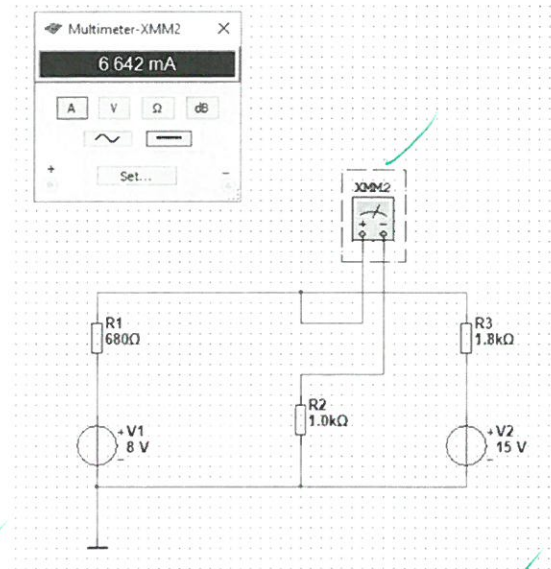


Figure 4

- b) Design a Matlab program that will find the value of the current through R_2 using Thevenin's and Norton's theorem. Modify the program to find the current flowing through each resistor and the voltage across them.

Matlab Program:

```
%By Paul Miller

%Enter Resistor Values and Voltages

R1=input( 'Enter value of R1:' );
R2=input( 'Enter value of R2:' );
LR=input( 'Enter value of Load Resistor LR:' );

E1=input( 'Enter value of E1:' );
E2=input( 'Enter value of E2:' );

%Find the load current with Thevenins
I1=(E1-E2) / (R1+R2);

%Find the open circuit voltage
E=E1-I1*R1;

%Find the total resistance at the open circuit terminals
r=(R1*R2) / (R1+R2);

%Find the current through the load
It=E/ (r+LR);
disp( 'The load current by using Thevinins equals: ' )
disp(It)

%Find current using Nortons

%Short circuit current
Isc= (E1/R1)+(E2/R2);

%Determine the load current
In= (r*Isc) / (LR+r);
disp( 'The load current using Nortons equals:' )
disp(In)
```

Results for Matlab program:

```
Enter value of R1:680
Enter value of R2:1800
Enter value of Load Resistor LR:1000
Enter value of E1:8
Enter value of E2:15
The load current by using Thevinins equals:
    0.0066
The load current using Nortons equals:
    0.0066
>> |
```

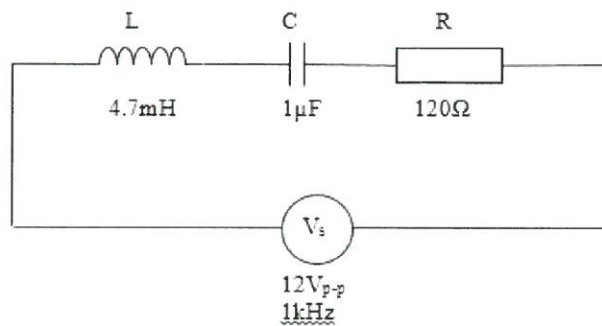
The results which are displayed by the Matlab program are very similar to those of the calculated and simulated values, slight discrepancies can be put down due to Matlab displaying the value as **0.0066** rather than **6.6mA**.

Have not found the current in the other resistors.

AC Circuits

Task 3 – Series AC Circuit

The circuit below shows a series ac circuit:



- Calculate the phase angle at 1kHz and resonant frequency.
- Using Multisim determine the phase angle and resonant frequency.
- Design a Matlab program that will determine the phase angle, resonant frequency and also produce a plot of the waveforms showing the phase difference.
- Construct the circuit on breadboard and compare the practical and simulation results.

a) Calculate the phase angle at 1kHz:

$$X_L = 2 \times \pi \times F \times L = 2 \times \pi \times 1000 \times 4.7 \times 10^{-3} = 29.53\Omega$$

$$X_C = 1/(2 \times \pi \times F \times C) = 1/(2 \times \pi \times 1000 \times 1 \times 10^{-6}) = 159.15\Omega$$

$$\text{PhaseAng} = \tan^{-1} (X_L - X_C)/R = \tan^{-1} (29.53 - 159.15/120) = -47.207^\circ$$

$$R_F = 1/(2 \times \pi \times \sqrt{L \times C}) = 1/(2 \times \pi \times \sqrt{4.7 \times 10^{-3} \times 1 \times 10^{-6}}) = 2321.5134\text{Hz}$$

b) Using multisim determine the phase angle and frequency response:

Figure 5 illustrates the circuit which was built in multisim to determine the phase angle and frequency response, to find the phase angle, the time difference between the phase had to be found, this was done by using an oscilloscope, from this time difference the sum:

TD x Frequency x 360 = Phase angle

$$131.827 \times 10^{-6} \times 1000 \times 360 = 47.457^\circ$$

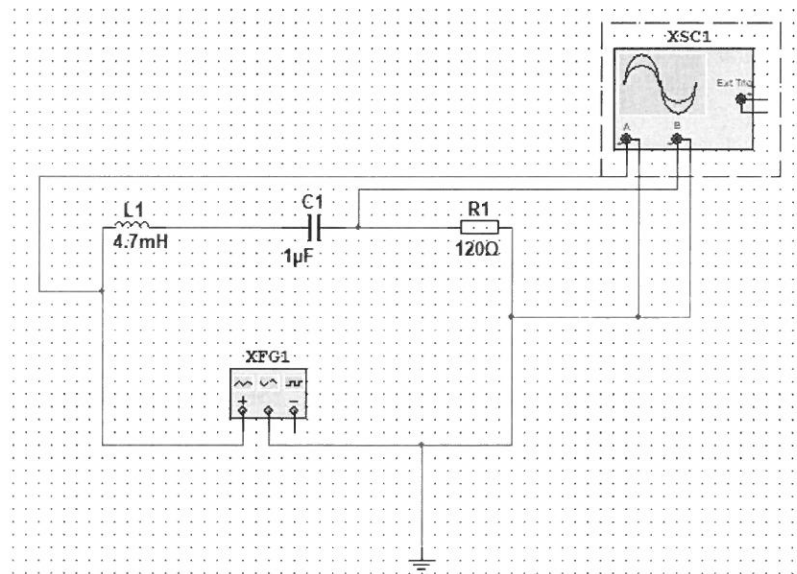


Figure 5

Figure 6 shows the graph which has the two phases displayed on it, it also shows the phase time difference which is 131.827×10^{-6} s

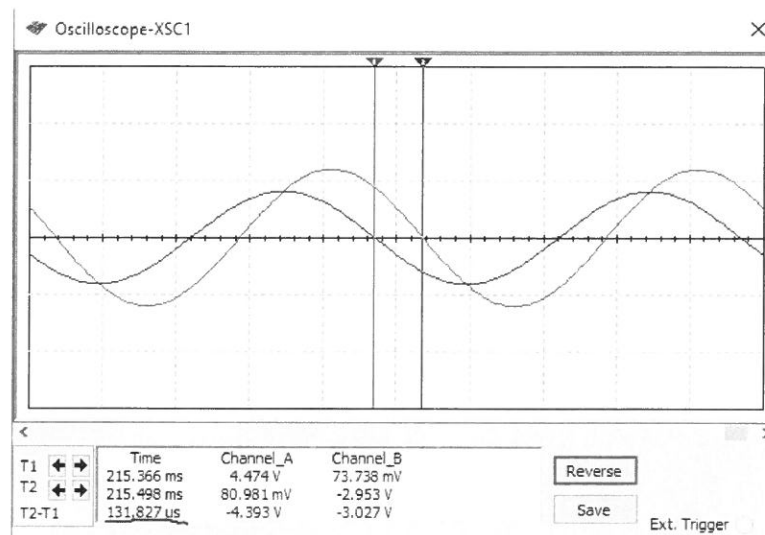


Figure 6

Figure 7 shows the resonant frequency which was obtained using a bode plotter, It shows the peak value which is 2.136 kHz, which is within tolerance of the calculated values and values which were produced using the matlab program.

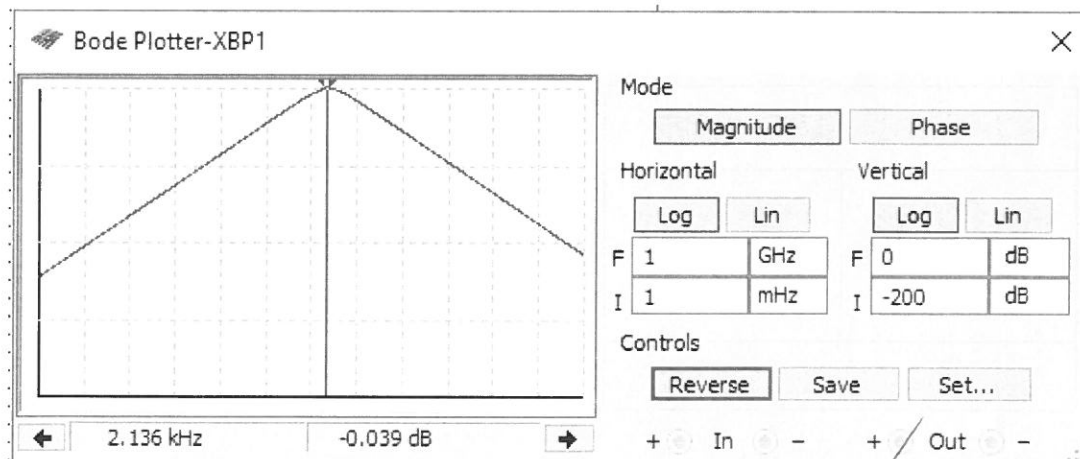


Figure 7

c) Design a Matlab program:

Figure 8 illustrates the matlab program created to find the phase angle for the series AC circuit in task 3, it illustrates the answers which are the phase angle, and the resonant frequency:

Phase angle = -47.2079°

Resonant Frequency = 2321.5134Hz

```
%Paul Miller Assignment 1 task 3c

%INPUTS
L=input ('Enter the inductor value ');
C=input ('Enter the capacitor value ');
V=input ('Enter the supply voltage ');
R=input ('Enter the resistor value ');
F=input ('Enter the frequency of supply voltage ');

%REACTANCE CALCULATION
Xl=2*pi*F*L;
Xc=1/(2*pi*F*C);

%PHASE ANGLE
PhaseAng=(180/pi)*atan((Xl-Xc)/R);

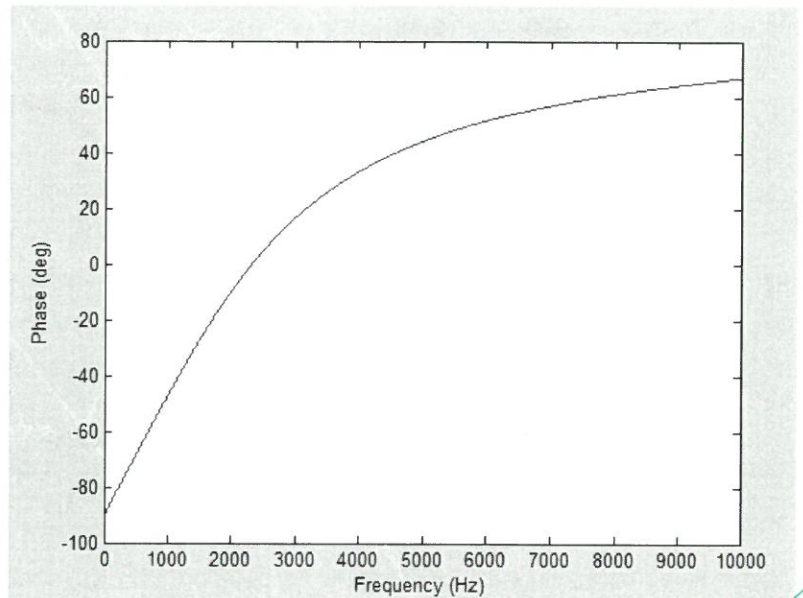
%RESONANT FREQUENCY
RF=1/(2*pi*sqrt(L*C));

%FOR LOOP FOR PHASE ANGLE
for pa=1:10000
    Freq(pa)=pa;          %Variable Freq
    Xl=2*pi*Freq(pa)*L;
    Xc=1/(2*pi*Freq(pa)*C);
    PA(pa)=(180/pi)*atan((Xl-Xc)/R);
end

X1=['The Phase Angle Equals ',num2str(PhaseAng),' degrees'];
disp(X1)

X2=['The Resonant Frequency Equals ',num2str(RF),' Hz'];
disp(X2)

plot(Freq,PA,'b-');
xlabel('Frequency (Hz)');
ylabel('Phase (deg)');
```



```
Enter the inductor value .0047
Enter the capacitor value .000001
Enter the supply voltage 12
Enter the resistor value 120
Enter the frequency of supply voltage 1000
The Phase Angle Equals -47.2079 degrees
The Resonant Frequency Equals 2321.5134Hz
```

Figure 8

d) Compare practical and simulation results:

Figure 9 shows a table of results which was obtained from the practical results, it shows the intervals at which the frequency was increased and it also shows the voltage out. With this table of values, a graph was produced which shows the frequency response curve. **Figure 10** illustrates the graph which was produced using the practical recorded results.

Freq	V-Out
100	1.39
500	4.63
1000	6.45
1500	7.13
2000	7.41
2500	7.41
3000	7.29
3500	7.09
4000	6.85
4500	6.51
5000	6.3

Figure 9

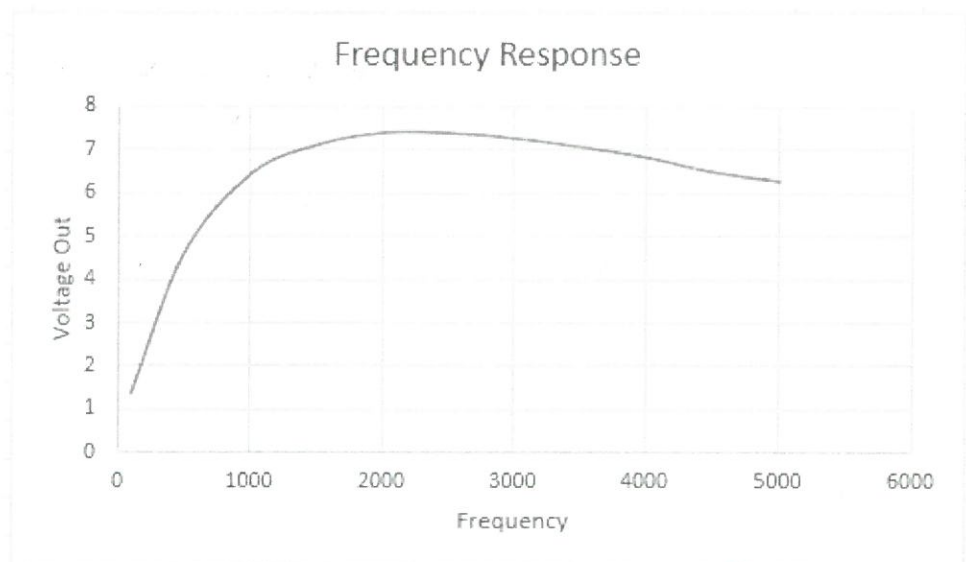


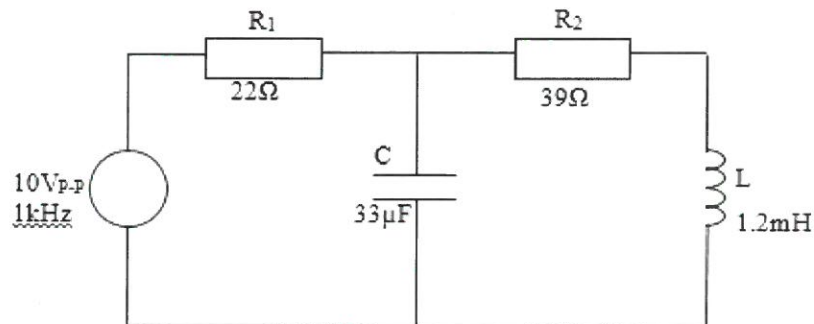
Figure 10

Comparing these results to those of the calculated value, they both match up identically, as the calculated value was **2321.513Hz** for a peak voltage, the measured peak voltage was measured to be between **2000Hz** and **2500Hz**, which means the calculated value is within tolerance which confirms the results.

What about the phase angle?

Task 4 – Complex AC Circuits

The circuit below show a complex AC circuit.



- Using Mesh theorem calculate the current through C.
- Using Multisim to confirm the value of the current.

a) Calculate the current through C, using mesh theorem:

a.

$$X_L = 2\pi fL = 2\pi \times 1000 \times 1.2 \times 10^{-3} = j7.53\Omega$$

$$X_C = -j4.82\Omega$$

Loop 1

$$V = R_1 I_1 + X_C (I_1 - I_2)$$

$$10 = 22I_1 + (-j4.82)(I_1 - I_2)$$

$$10 = (22 - j4.82)I_1 + j4.82I_2 \quad (1)$$

Loop 2

$$0 = R_2 I_2 + X_L I_2 + X_C (I_2 - I_1)$$

$$0 = 39I_2 + j7.53I_2 + (-j4.82)(I_2 - I_1)$$

$$j7.53 - j4.82 = j2.71$$

$$0 = j4.82I_1 + (39 + j2.71)I_2 \quad (2)$$

Rearrange eq (2)

$$j4.82I_1 = - (39 + j1.46) I_2$$

$$I_1 = - (39 + j2.7198) I_2 / j4.82 \quad (3)$$

Substituting the value of I1 into eq (1).

$$10 = j0.207 (22 - j4.82) (39 + j2.7198) I_2 + j4.82 I_2$$

$$10 = j0.207 (8.71 - j128.36) I_2 + j4.82 I_2$$

$$10 = (26.52 + j180.31) I_2 + j4.82 I_2$$

$$I_2 = 10 / 2.65 + j185.14 = 0.0076 - j0.0529$$

Substituting the value of I2 into eq (3).

$$I_1 = -(39 + j2.7198) (0.0076 - j0.0529) / j4.82$$

$$I_1 = 0.4237 + j0.0913$$

$$I = I_1 - I_2 = 0.4237 + j0.0913 - 0.0076 + j0.0529$$

$$I = 0.4161 + j0.1442$$

$$0.44037 \angle 19.1138 \text{ Peak to Peak}$$

$$V_{rms} = 0.44037 / \sqrt{2} = 0.1557 \angle 19.1138 \text{ A} = 155.7 \text{ mA}$$

b) Simulated values using multisim:

Figure 11 shows the simulated value which was produced on multisim, the current flowing through C1 on the simulation is:

$$C1 = 155.197\text{mA}$$

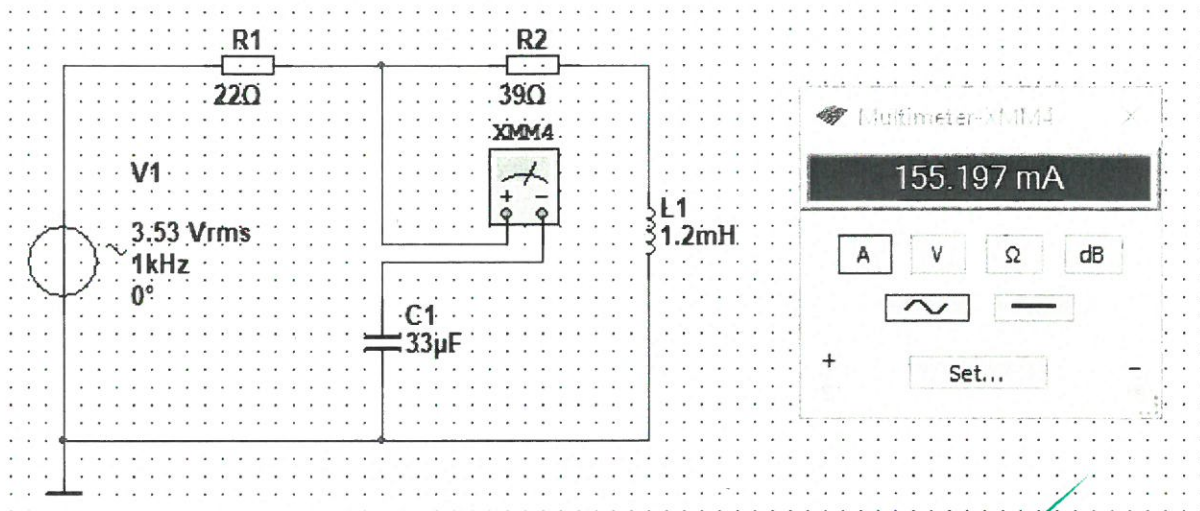


Figure 11

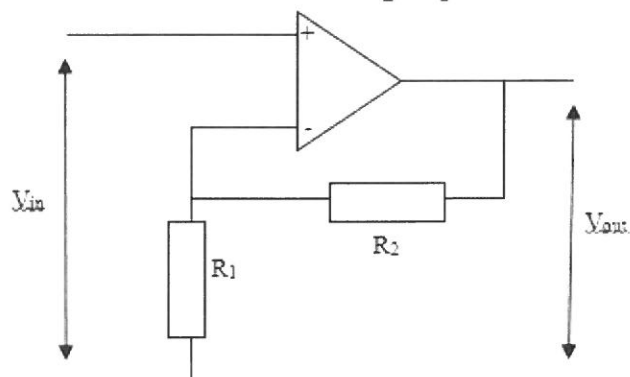
These discrepancies between calculated and simulated values can be put down to human error when rounding, as the equations are complex for the mesh circuit.

Well done.

Electronic Circuits

Task 5 – Non-Inverting Amplifier Circuits

The circuit below shows an inverting amplifier.



- Determine the values for R_{in} and R_f to produce a gain of 200.
- Using the resistor value that were determined in a) and a 741 op amp construct the circuit on breadboard and determine the gain and bandwidth for the circuit. Confirm the results using Multisim and comment upon the results.

a) Determine the values for R_{in} and R_f :

$$V_{out}/V_{in} = A_v = R_f / R_{in} + 1$$

$$A_v - 1 = R_f/R_{in}$$

$$R_f = (A_v - 1) \times R_{in} = (200-1) \times R_{in}$$

$$\text{Let } R_{in} = 4.7\text{k}\Omega$$

$$R_f = 199 \times 4700 = 935.3\text{k}\Omega$$

$$R_f = \text{Nearest Value} = 1\text{M}\Omega$$

b) Confirm results using multisim:

Figure 12 illustrates the circuit which was created to check and compare the calculated values which were calculated in part a

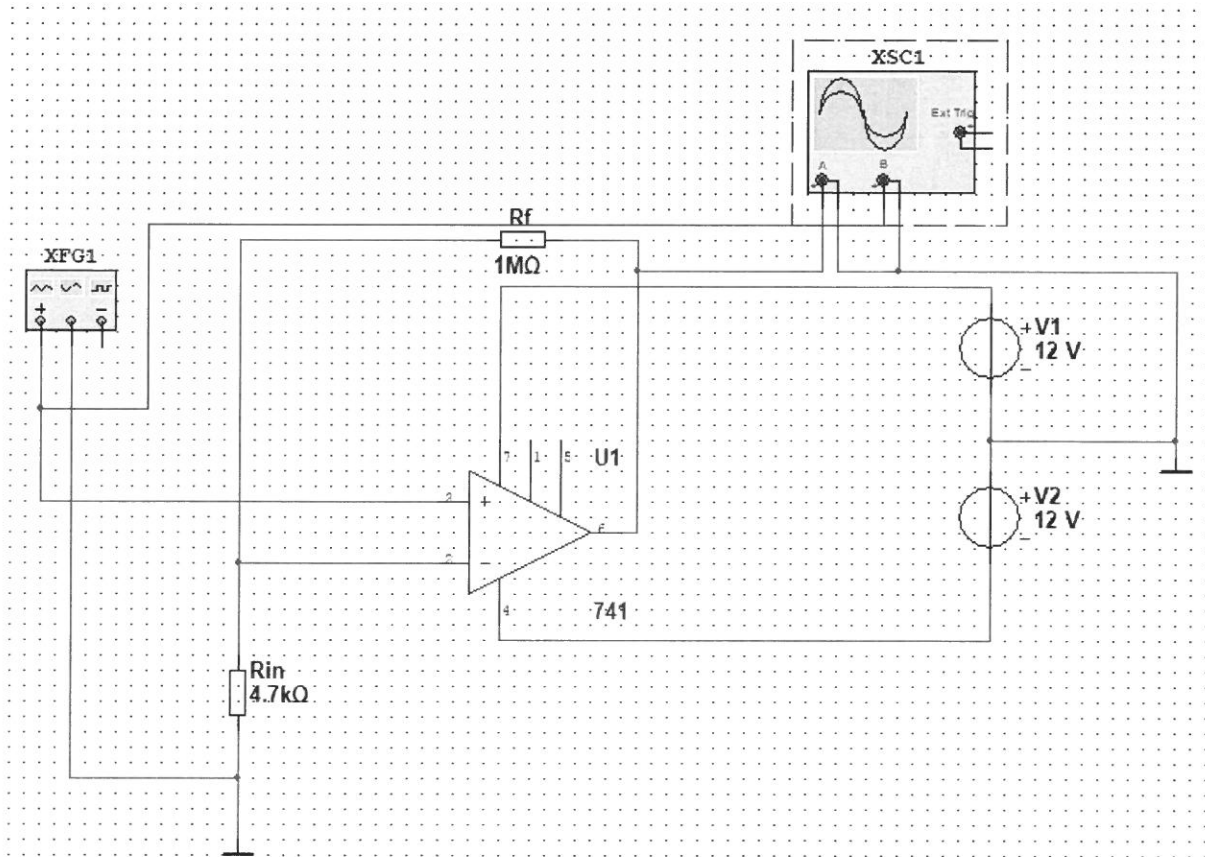


Figure 12

Figure 13 shows the results which were obtained when simulating the circuit which is **Figure 12**, from these results the gain can be produced by using both the voltages from each channel:

Channel **A** = **10.848V**

Channel **B** = **49.995mV**

So Gain = $V_{out}/V_{in} = 10.848 / 49.995 \times 10^{-3} = 217$

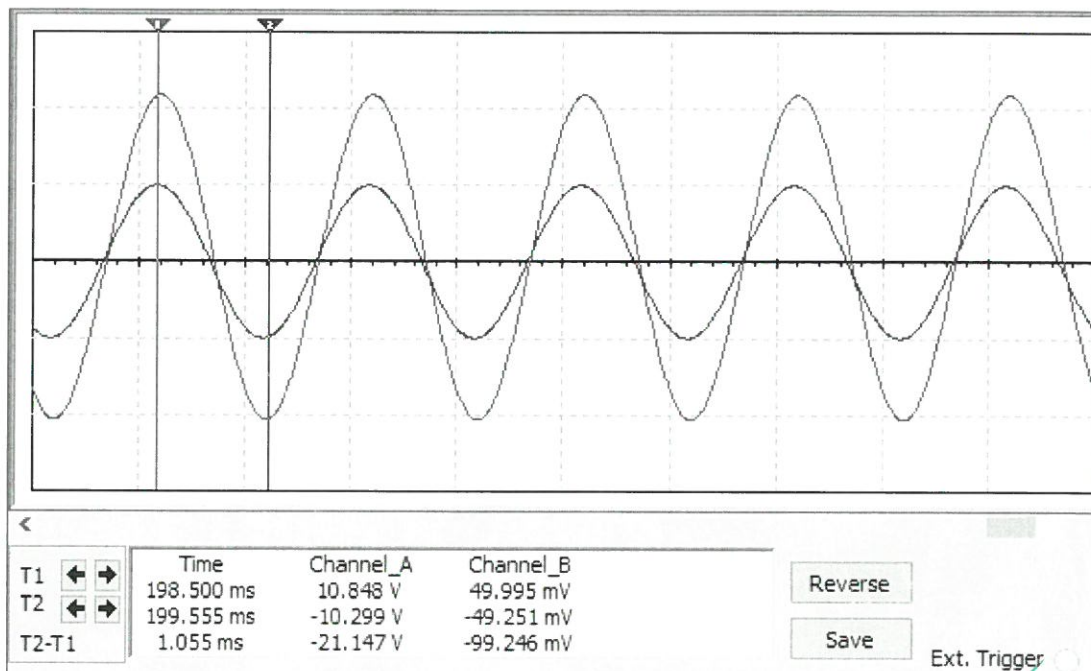


Figure 13

From the simulations carried out on the circuit the bandwidth can also be found, this was done by removing the **oscilloscope** and adding in a **bode plotter**, from the bode plotter, a graph is produced, this graph displays the frequency response, the max reading on the graph is **46.587db**, to find the bandwidth then, **3db** is then taken away from the max value, this means the bandwidth is near to **43.445db**, **Figure 14** illustrates this.

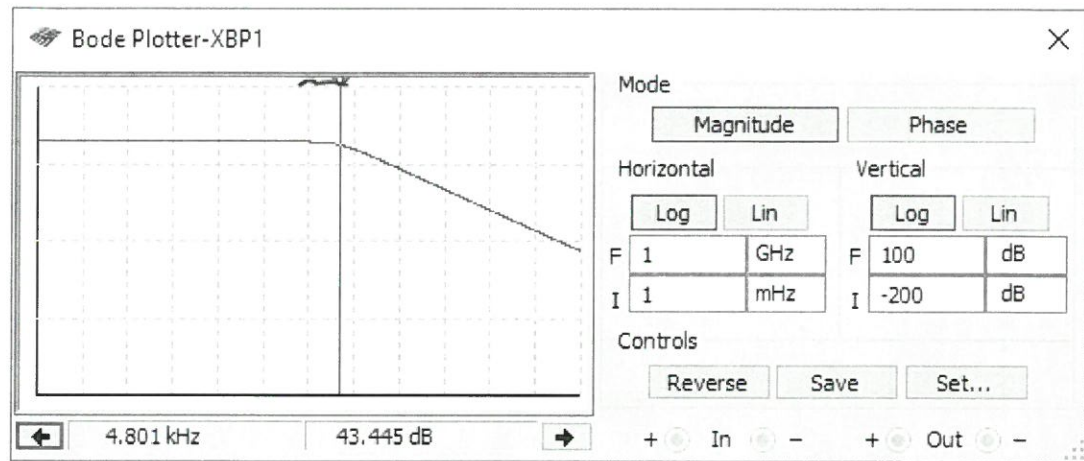
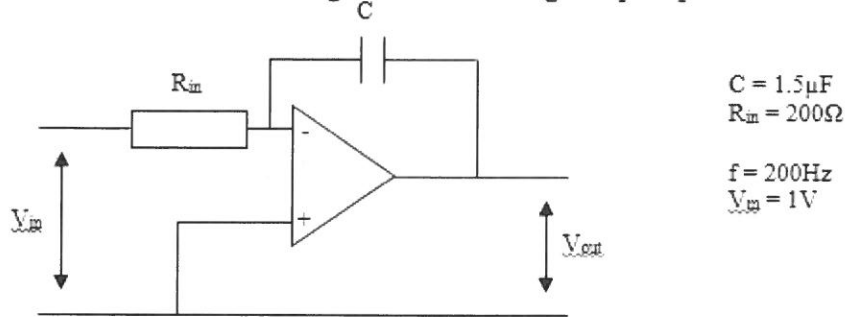


Figure 14

No practical results.

Task 6 – Integrator Circuits

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



- Using Matlab write a program to display the output waveform when a square wave is applied.
- Check the results using Multisim.

a) Matlab Program for Task 6:

Figure 15 shows the results which were obtained from the matlab program, Figure 16 is another part of the program which was produced to work alongside figure 17 which will ensure the program runs correctly. Figure 17 illustrates the matlab program which was produced for task 6.

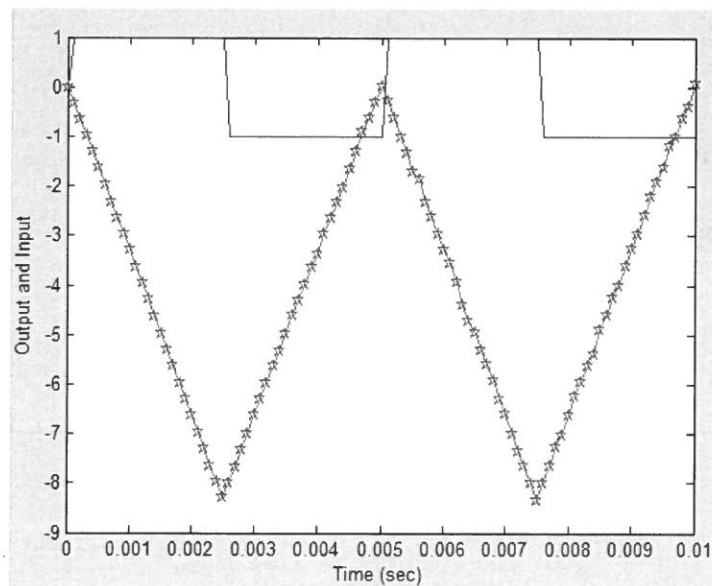


Figure 15

```
%Paul Miller
%Task 6 Program
```

```
% Parameters
R1 =200; C=0.0000015;
```

```
% Time vector
t = 0:0.0001:0.01;
```

```
% Output Waveform
```

```
for P = 1:length(t)
    Vo(P) = -1/(R1*C)*quad('Vin',0,t(P));
end;
```

```
% Input Signal
Vi = 1*sign(sin(2*pi*200*t));
```

```
% Display the Results
plot(t,Vi,t, Vo,'p-');
xlabel('Time (sec)'); ylabel('Output and Input');
```

Figure 17

```
%Calculate a function value at point x
function[V] = Vin(x)

%The function expression
V = 1*sign(sin(2*pi*200*x));
```

Figure 16

b) Check results using multisim:

Figure 18 illustrates the circuit which was produced to compare the results with the matlab program, there is a function generator connected along with an oscilloscope which is what is used to measure the waveforms.

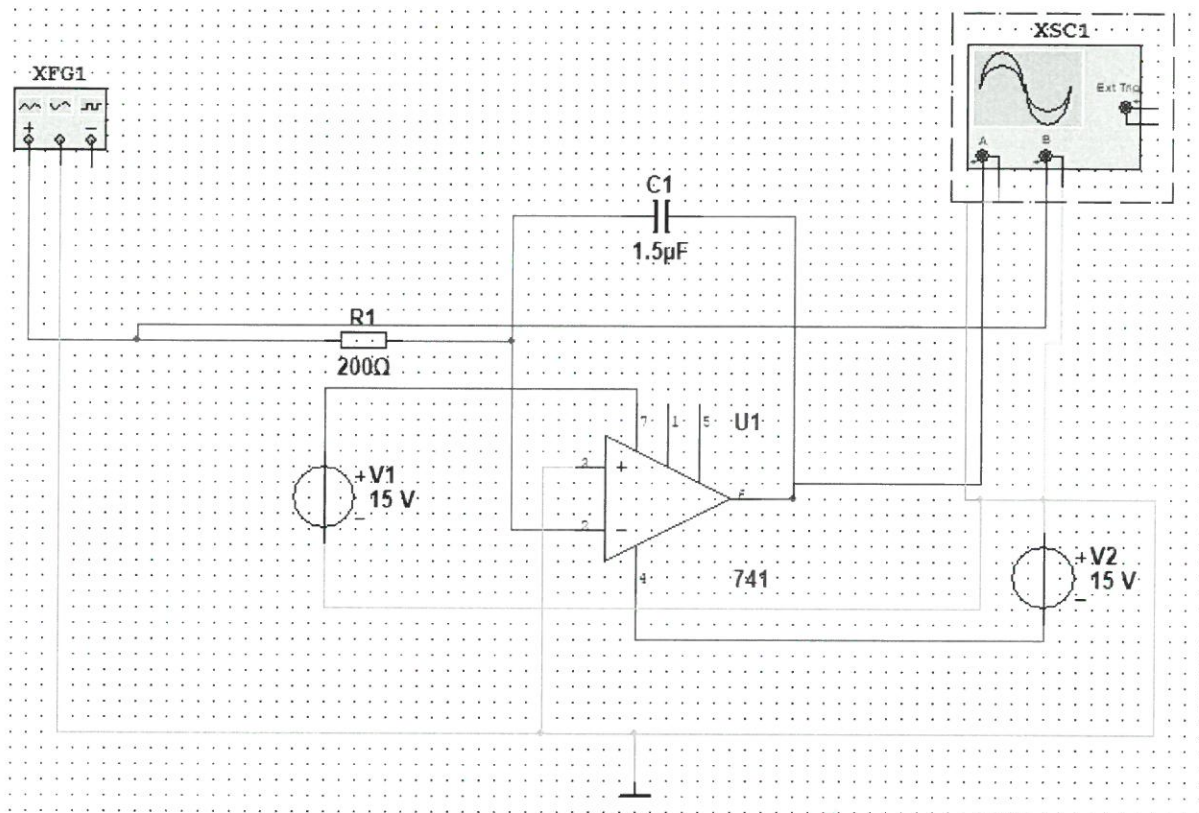


Figure 18

Figure 19 shows the wave which was produced by the circuit, channel A is the blue waveform plotted on the graph, and the red waveform is the is channel B, channel B is what is being recorded coming out of the function generator, and then channel A is what is being recorded coming out of the 741 op amp.

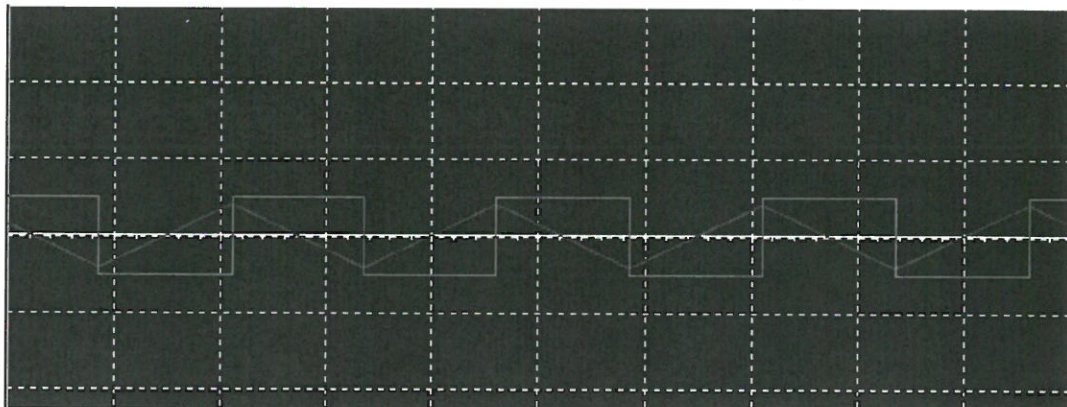


Figure 19

Not all the requirements were covered in this assignment and your comment were weak or non-existent. This has caused you to lose a number of marks.