

1702304

by Benjamin Morgan

FILE	19736_BENJAMIN_MORGAN_1702304_178835_122532745.DOCX (5.27M)		
TIME SUBMITTED	31-JAN-2021 04:15PM (UTC+0000)	WORD COUNT	3982
SUBMISSION ID	143842118	CHARACTER COUNT	20182

Course: HNC Electrical and Electronic Engineering

Module: SGEE4003 - Electronics I

Assignment Title: Analogue Circuit Design

Name: Benjamin James Morgan

University number: 1702304

Date Submitted: 31/01/2021

Word Count: 3571 words

80 - 85

1000 OVER LIMIT

- ① RESEARCH + REFERENCE LIST ARE VERY GOOD, YOU SHOULD USE INTEXT CITATIONS WITHIN THE REPORT RATHER THAN THE FULL CITATION.
- ② METHOD WAS EXPLAINED LOGICALLY AND ALL RELEVANT POINTS IN CLARER
- ③ TESTING WAS COMPREHENSIVE.
- ④ VERY GOOD ANALYSIS OF RESULTS AND VALID RECOMMENDATIONS MADE.

Table of Contents

1. Executive Summary.....	3
2. Research into amplifier design techniques.....	4
3. Practical task, design method, choices, and calculations	9
4. Testing, results, and analysis.....	14
5. Conclusions and recommendations.....	17
6. Reference list and bibliography	18
7. Annexe for diagrams, data sheets, results tables etc	19

1. Executive Summary

This report describes the process of designing, building, and testing an amplifier circuit.

Firstly, research was conducted into what amplifiers are, how they are classed and what types they are. Then Oscillators were researched, and the theory of negative feedback was introduced and explored.

Next the design process of the amplifier was described, all of the equations used were given and manipulated, with all of the corresponding calculations shown. Once sufficient calculations had been completed, the initial amplifier could be constructed. This circuit was tested and was found to need refinement, this refinement process was then described along with the reasons for the improvements. Now that the amplifier circuit was complete, an oscillator had to be modified to replace the signal source used in the initial circuit. After tweaking the component values within the oscillator so that it fit the criteria, it was connected to the amplifier circuit. At this point final tests were conducted on the circuit to ensure that it fit the design brief.

The report is then concluded, in the conclusion the circuit values are compared to the desired values. Then recommendations were made as to how the circuit values could be improved in the future.

VERY GOOD, YOU WOULD SAY
WHAT THE RECOMMENDATIONS ARE

2. Research into amplifier design techniques

Amplifiers are electronic components that have the ability to amplify an input signal. When a signal is amplified the power is increased. An amplifier is classed as an active device, an active device is a circuit component capable of electrically controlling the electric charge flow. (Kuphaldt, 2021) The function of an active device is to control the flow of electric charge through them, they can either use voltage or another current for this control. When amplifying the input signal, the only characteristic of the input to change is the amplitude, all other characteristics of the wave stay constant. Amplifiers can do this because they are powered by an external power source. This is how they can increase power without breaking the law of conservation of energy.

There are 3 different types of amplifiers: Voltage, Current and Power. Voltage amplifiers are the most common type of amplifier and they increase the amplitude of the voltage; current amplifiers increase the current amplitude; and the power amplifiers increase the power. Power amplifiers increase the power by altering the voltage and current, as power is equal to the product of voltage and current. This means that although the power will be amplified the output voltage or current could be lower than the input voltage or current. Transistors are active devices that are often used as amplifiers, they can be grouped into 2 sections: bipolar transistors and field effect transistors. Bipolar junction transistors (BJT's) are electronic devices that come in 2 forms: NPN and PNP. Both of these refer to the type of bipolar junctions within the device. In an NPN transistor the charge carriers in the n-type emitter are electrons whereas in an PNP transistor the charge carriers in the p-type emitter are holes.

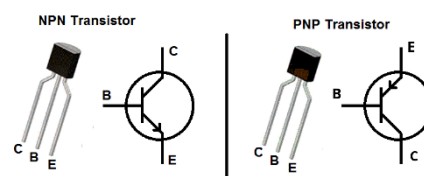


Figure 1- diagram of different junctions
<https://shoptransmitter.com/blog/what-is-the-difference-between-pnp-and-npn/>

USE AN INTERNET
CITATION.

BJT's can have 3 different configurations as they have 3 different legs, these are: Common base, Common emitter and Common collector. The common base

configuration provides a voltage gain but no current gain. The common collector configuration is the opposite of this, current gain but no voltage gain. The common emitter configuration provides both current and voltage gain. The term “common” in these descriptions refers to what leg on the transistor the input and output are connected. (Bird, 2014) There are also Field Effect Transistors (FET’s) and there are 2 variations: junction gate and insulated gate. All FET’s are unipolar, meaning that the charge flow through them is either electrons through the N-type semiconductor or holes through the P-type semiconductor.

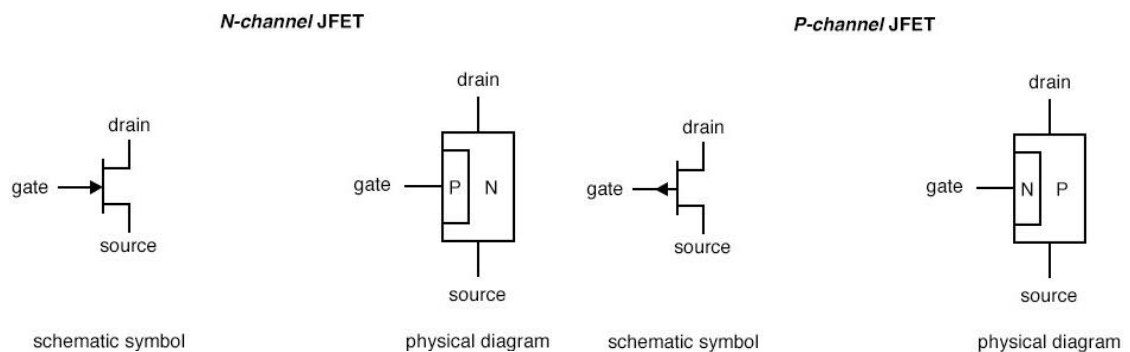


Figure 2 -
<https://www.allaboutcircuits.com/textbook/semiconductors/chpt-5/junction-field-effect-transistors-jfet/>

Figure 3 -
<https://www.allaboutcircuits.com/textbook/semiconductors/chpt-5/junction-field-effect-transistors-jfet/>

The N-type FET’s are more common than the P-type, the FET’s allow the current being controlled to flow from source to drain or visa-versa, the controlling voltage is applied between the gate and source, this acts as a “pinch” that limits the current flow.

Another type of amplifier is the operational amplifier (op-amp), these are complex integrated circuits that can have many functions. These are commonly used to preform mathematical operations and act as a high gain DC and AC voltage amplifier. Op-amps have 3 main properties: high open-loop voltage gain, which decreases with increasing frequency; very high input impedance, this ensures that most of input voltage is passed through the op-amp; and they have a very low output impedance, this ensures that the output voltage is transferred efficiently. (Bird, 2014) The circuit symbol for an op-amp is as below.

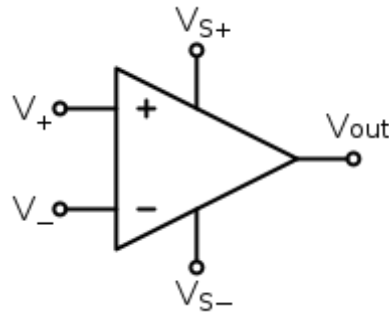


Figure 4 -
https://en.wikipedia.org/wiki/Operational_amplifier

The op-amp has 1 output (V_{out}), and 2 inputs, these are the inverting input (V_-) and the non-inverting input (V_+). V_{S+} and V_{S-} are where an external power supply would be connected, the 0v point from the power supply is usually common to both the input and output side of the circuit, this provides the voltage reference level. In general, the op-amp can be viewed as a differential voltage amplifier, this means that it amplifies the difference between the 2 input voltages. This means we can have 3 different situations:

1. $V_2 > V_1$, therefore V_{out} is positive
2. $V_1 > V_2$, therefore V_{out} is negative
3. $V_1 = V_2$, therefore V_{out} is zero

A general equation for op-amps is $V_{out} = A_0 (V_2 - V_1)$, where A_0 is the open loop voltage gain. (Bird, 2014)

Amplifiers can be classed by their duty cycle, the most common classes are: A, AB, B, and C. These classes identify what the amplifier affects and its uses, the class of an amplifier indicates how much of the input waveform is used. Class A amplifiers are the most common and use 100% of the input and therefore are conducting all the time, this results in low distortion and low efficiency as lots of energy is lost due to heat. The low-level distortion means these are ideal for audio amplifiers. Class B amps use 2 transistors, 1 BJT and 1 FET, they both amplify half of the input signal each, this means that the transistors are turning off for half of the wavelength, this results in a low distortion but better efficiency, roughly twice as efficient than class A. Class AB work with the same principle as class B, these amplifiers have very low distortion, around 0.1% and the efficiency is around 50% or lower. These amplifiers are

commonly used in Hi-Fi systems. Class C amps are very high distortion and very high efficiency, and these are used for radio frequency amplification. (Elprocus, 2021)

Oscillators are commonly used with amplifiers and they have different types. An oscillator produces a continuous, repeating, changing waveform, they are used to convert a DC source into a desired waveform. Oscillators can be sectioned into 2 groups: harmonic and relaxation. Harmonic oscillators pass the energy from the active components to the passive. Whereas relaxation oscillators exchange the energy between the active and passive components. The frequency of oscillation for the harmonic is determined by the feedback path and for the relaxation the charging and discharging time constants of the exchange determines the frequency. Oscillators can be also be grouped depending on the type of feedback mechanism, these 2 groups are, positive feedback and negative feedback. Some common types of oscillators are, Wein Bridge and Colpitts. (Electrical 4 U, 2020)

Negative feedback is the technique of sampling a part of the output signal and comparing that to the input signal. This makes the input partly reliant on the output, and by doing this it is possible to gain very precise control of the system.

Negative feedback is commonly used in electronics and there are 4 types: Series-parallel, series-series, parallel-parallel, and parallel-series. These refer to how the input/output of the amplifier is connected to the input/output of the feedback circuit. These different configurations will alter the input/output impedances, and therefore the configuration will alter the use case.

Type of feedback	Input Impedance (Z_{in})	Output impedance (Z_{out})	Ideal use case
Series-Parallel	High	Low	Voltage amplifier
Series-Series	High	High	Voltage to current transducer
Parallel-Parallel	Low	Low	Current to voltage transducer
Parallel-Series	Low	High	Current amplifier

Connecting circuits in series yields a high impedance, whereas connecting in parallel gives a low impedance. These configurations provide different models for the circuit,

the model refers to how the circuit behaves, a Series-Parallel configuration will provide a voltage-controlled voltage source, this is perfect for a voltage amplifier. A Parallel-Series configuration yields a current-controlled current source, ideal for a current amplifier. Series-Series and Parallel-Parallel do not provide a gain, instead they convert an input voltage to an output current (series-series) or an input current to an output voltage (parallel-parallel). (Fiore, 2021)

RESEARCH IS VERY GOOD, BUT YOU COULD
HAVE REMOVED SOME UNNECESSARY SENTENCES
TO KEEP YOUR WORD COUNT DOWN.

3. Practical task, design method, choices, and calculations

The first step of the design process was to select DC components, firstly a power supply for the transistor had to be chosen, the decided value was 30v (V_{CC}). Next a transistor was selected, the amplifier had to be small signal and low frequency, it must also be on Multisim. Therefore, a 2N3904 transistor was chosen. The data sheet then had to be analysed to determine what quiescent collector current (I_Q) would be suitable. The sections of the data sheet that were used are in the annexe. The quiescent current is the collector current when no signal is applied. The value for I_Q should be below 10% of the $I_{C_{MAX}}$ value, the $I_{C_{MAX}}$ was 200mA, and the chosen I_Q was 5mA. Now the value for the load resistor (R_L) can be calculated, this is done using the equation:

$$R_L = \frac{V_C}{I_Q}, \text{ where, } V_C = \frac{1}{2} V_{CC} .$$

$$\text{Hence, } R_L = \frac{15}{5 \times 10^{-3}} = 3\text{k}\Omega$$

Next the value for the emitter resistor (R_E) must be calculated, the equation used for this was:

$$R_E = \frac{V_E}{I_Q}, \text{ where, } V_E = 10\% V_{CC}$$

$$\text{Therefore, } R_E = \frac{3}{5 \times 10^{-3}} = 600\Omega$$

Before the next value can be calculated the data sheet for the transistor must be consulted to find the value for h_{fe} . This is known as the transistors current gain; this value will be different for each transistor even though they are the same type. Therefore, the base current (I_B) that will be calculated next is an estimate. To calculate I_B the following equation is used:

$$I_B = \frac{I_Q}{h_{fe}}$$

$$I_B = \frac{5 \times 10^{-3}}{200} = 25\mu A$$

Now the base voltage (V_B) can be calculated, the base voltage should be roughly 0.7V higher than the emitter voltage (V_E), V_E was found to be 3V, therefore V_B is 3.7V. The 0.7V increase is to make sure that the input signal is biased on the linear part of the transistor characteristic. Next the current flowing in the DC bias network, which consists of R_1 and R_2 , can be calculated.

To make sure that the bias is stable the current should be 10x more than I_B . I_B was calculated as $25\mu A$, so $I_{R1/R2}$ is simply $250\mu A$. Now the value for R_1 can be calculated, this is done using the equation:

$$R_1 = \frac{V_{CC} - V_B}{I_{R1/R2}}$$

$$R_1 = \frac{30 - 3.7}{250 \times 10^{-6}}$$

$$R_1 = 105.2k\Omega$$

The final DC component to be selected is R_2 , to calculate this V_B will be divided by $I_{R1/R2}$. Doing this gives us a value of $14.8k\Omega$.

Now the DC components for the amplifier have been chosen, the AC components must now be decided upon. 4 capacitors must be added to our selection, C_1 , C_2 and C_3 will be the first to be chosen, C_1 and C_2 are known as coupling capacitors, these will be placed at the input and output side. Their job is to allow AC signals to pass through them while blocking any DC signals. This is to stop the bias conditions for the amplifier changing when it is connected to other circuits. C_3 is connected between the emitter resistor and the 0V, it is called the decoupling capacitor. Its job is to remove any AC signals acting across R_E , any AC acting on the emitter leg of the transistor would be in phase with the base leg signal, this will mean that the signals will fluctuate together and the input signal and therefore the gain will be reduced. To calculate the capacitor values the following equation must be used:

$$C = \frac{1}{2\pi f X_c}$$

The frequency of the circuit is $200kHz$ and X_c will set as 1 as the capacitive reactance must be a low value when compared to the input impedance of the amplifier. Therefore,

$$C = \frac{1}{2\pi \times 200 \times 10^3}$$

$$C = 800nF$$

Enough of the components have now been selected to enable the circuit to be built, the values gathered are not the final values and they will have to be altered to make the circuit act according to the brief. The circuit can be seen in the annexe Fig. 5. With the circuit built

the gain and the bandwidth can be calculated, the gain calculation is shown on Fig 5 and comes out as 264.5, to calculate the bandwidth an AC sweep is conducted on Multisim, the results for this test is below.

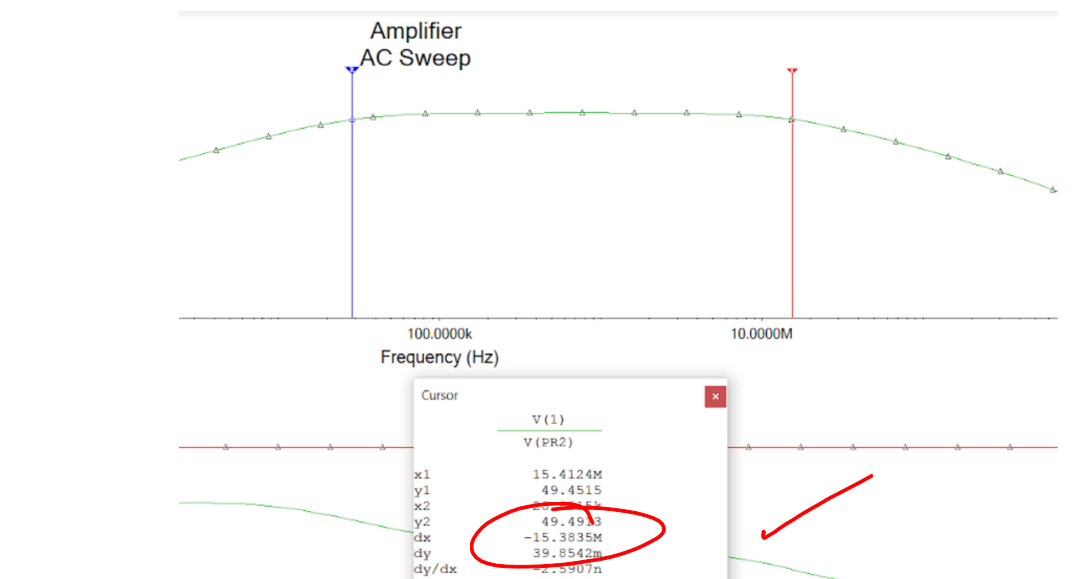


Figure 6 - AC Sweep of Initial Circuit

To calculate the bandwidth, the cursors must first be moved to the max value on the Y axis, this will give the reading of the centre frequency. Then 3db must taken away from this value and each cursor must be moved to this point on the Y axis, this value is exhibited twice on the graph, one on each side of the centre frequency. At this point the value for dx can be read from the AC sweep, this is the bandwidth. The bandwidth of the initial circuit was 15.383MHz. To refine the circuit, the value for C_1 must be tweaked and a 4th capacitor must

~~NEW HEADING - INPUT IMPEDANCE~~
~~and output impedances must be calculated. To calculate the input impedance the circuit must be modified, a variable resistor (VR_1) must be placed before C_1 , this will be set to 0Ω initially, under these conditions $V_{source} = V_{input}$. Now the V_{output} is measured, next the resistance of VR_1 is adjusted until the voltage across it is equal to $\frac{1}{2} V_{output}$. When these conditions are met, the input impedance is equal to the value of VR_1 . With VR_1 integrated into the circuit and set to 0Ω , V_{output} was found to be 9.77mV, when the voltage across VR_1 dropped to 4.84mV the resistance was 1.7k Ω , therefore the input impedance of the circuit is also 1.7k Ω . The output impedance of the circuit is simply the value of the R_L , 3k Ω . Now that these values have been found it is possible to refine the value for C_1 and add C_4 . To find the values of C_1 and C_4 the following equations are used:~~

$$C_1 = \frac{1}{2\pi \times f_L \times Z_{in}} \text{ and } C_4 = \frac{1}{2\pi \times f_H \times Z_{out}}$$

Where f_L and f_H are the ideal lower and higher frequency limit of the bandwidth, because the bandwidth is meant to be 60kHz and the centre frequency is 200kHz, this means f_L and f_H will be 170kHz and 230kHz respectively. Using these values:

$$C_1 = \frac{1}{2\pi \times 170 \times 10^3 \times 1.7 \times 10^3} \text{ and } C_4 = \frac{1}{2\pi \times 230 \times 10^3 \times 3 \times 10^3}$$

$$C_1 = 550.71pF \text{ and } C_4 = 230.66pF$$

The final step for designing the amplifier is to introduce negative feedback, by introducing negative feedback the gain of the system will be reduced. The gain is reduced, and it is made much more reliable, this is because the gain was initially determined by the ratio of R_L and R_E . The decoupling capacitor C_3 makes this ratio almost infinite and therefore the gain is now determined by h_{fe} . The introduction of negative feedback changes this and now makes it rely on the fixed component values. Negative feedback will also reduce distortion and it will help to even the gain over the whole bandwidth. To introduce negative feedback into the system R_E must be split into 2, with C_3 connected in parallel with the bottom resistor.

To reduce the gain to the desired value of 50, trial and error was used. By altering the value of R_E and R_3 the gain can be adjusted. Initially the value of R_E is split in two, it was 600Ω, so 2 300Ω resistors were used. This produced a very small gain, to adjust this the value of R_E is reduced and R_E is increased, ensuring the total resistance was around 600 ohms. After a few attempts a gain of 50 was achieved using the values; $R_3 = 30\Omega$ and $R_E = 560\Omega$.

Now the amplifier circuit is in its final configuration and the component values can be tested and compared to calculated/desired values, the circuit used for testing can be seen in Fig.7, in the annexe.

Once the amplifier circuit design was complete, a Colpitts oscillator was tuned, this was used to replace the signal generator. The basic version of the oscillator was provided but it did not meet the requirements for the circuit. The provided oscillator can be seen in fig.8. The frequency had to be modified, to do this the values for C_1 , C_2 and L_1 must be changed, although the values for C_1/C_2 must be the same. To produce the required frequency trial and error was used, interactive testing was done and the frequency values from the data points were read. The oscillator produced a signal frequency of 200kHz when $C_1/C_2 = 98pF$

and L1 was 13mH. The oscillator contains 2 op-amps, U1 is a buffer amplifier and U2 is an inverting amplifier. U2 reduces the output signal according to the equation:

$$Gain = \frac{R_4}{R_3}$$

The required output of U2 is 100mV (p-p), the incoming voltage is 4.43V, therefore the gain required is 0.0226. To produce this the following resistor values were used, R3=900Ω and R4=20Ω, these changes resulted in the output being 98-101mV (p-p). The final oscillator used can be seen in Fig.9.

To complete the construction of the circuit the oscillator and the amplifier must be connected. To do this the signal generator is removed from the circuit and the output of U2 is connected to the base of the amplifier, the final configuration of the circuit can be seen in Fig.10. The circuit will be tested in this configuration to ensure it fits the criteria.

YOUR EXPLANATION IS VERY DETAILED, BUT
YOU COULD REDUCE THE WORD COUNT BY
WRITING MORE SUCCINCTLY.

4. Testing, results, and analysis

Testing was conducted throughout the design process; the initial testing was used to confirm that the calculated values were correct and were providing the expected results. Testing conducted later into the design process was used to refine the circuit to produce the desired values. The testing done within Multisim was either interactive testing, where values could be read at desired points in real time; or AC sweeps were conducted to obtain readings for the bandwidth and centre frequency. The first set of tests were conducted to check the DC voltages on each leg of the transistor, this was done with interactive testing, the results are below.

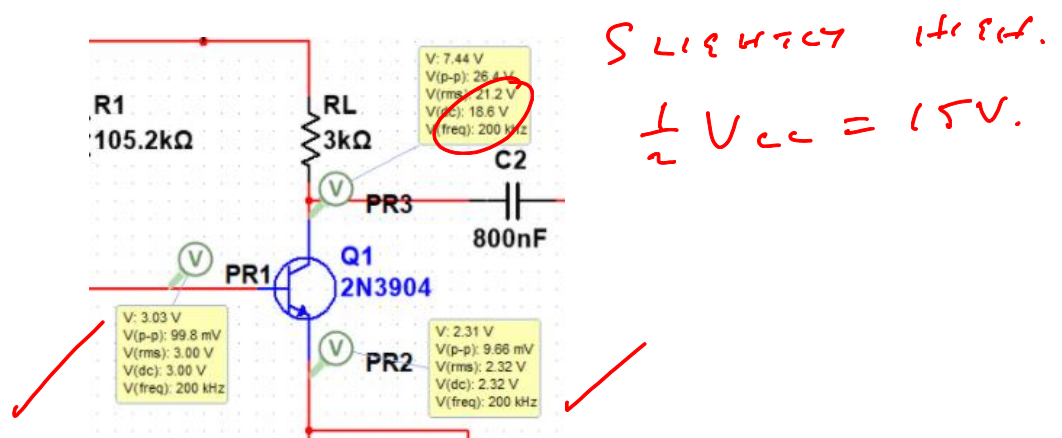


Figure 11- DC Voltages on Each Leg of Transistor

The values here are not the same as the values that were gained through calculations, they are all slightly lower than expected. Importantly though the difference between the emitter and base voltages remained as 0.7V. Then the initial circuit testing for bandwidth and gain were conducted, these results were described in the previous section and shown in Fig.6.

The next group of testing was carried out once the values for C_1 and C_4 were tweaked and negative feedback was introduced through the splitting of the emitter resistor. The circuit used for these tests is shown in Fig.7. The results from this test showed that the gain was exactly 50 and the bandwidth was 348.4627kHz. When the AC sweep was conducted it was

noted that the centre frequency had shifted to 125.89kHz, this was not correct and therefore the value for C_1 had to be altered.

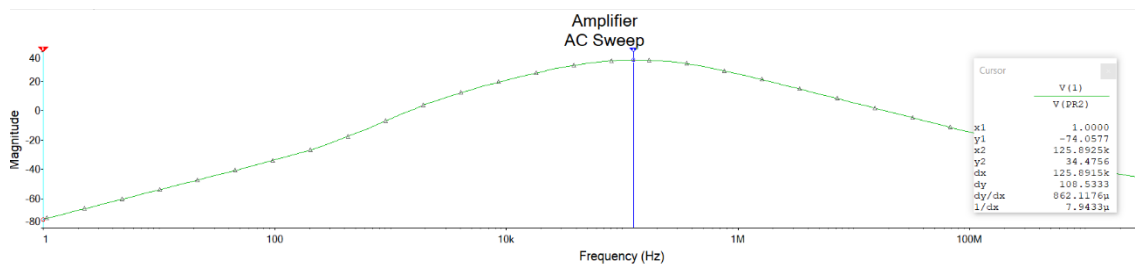


Figure 12 - AC Sweep Showing Incorrect Centre Frequency

Through repeated testing the new value for C_1 became 200pF, when the AC sweep was tried again the centre frequency had become 199.53kHz, this fit the criteria for the circuit. Now the amplifier circuit had been fully tested and was complete.

Next the oscillator was built and tested. The value for frequency was found through interactive testing, the data points showed the frequency. The values for C_1/C_2 and L_1 were altered until these data points showed that the frequency was 200kHz. The gain of the op-amps were measured in the same way, the resistor values were determined through calculations so that they provided the correct ratio. Once the resistors were changed to the desired values, the p-p voltage readings from the data points were read, and the desired output voltage of 100mV was confirmed. The last test done on the oscillator was conducted by connecting an oscilloscope to the output, the test was to check that the output signal was relatively sinusoidal. The oscilloscope wave is below.

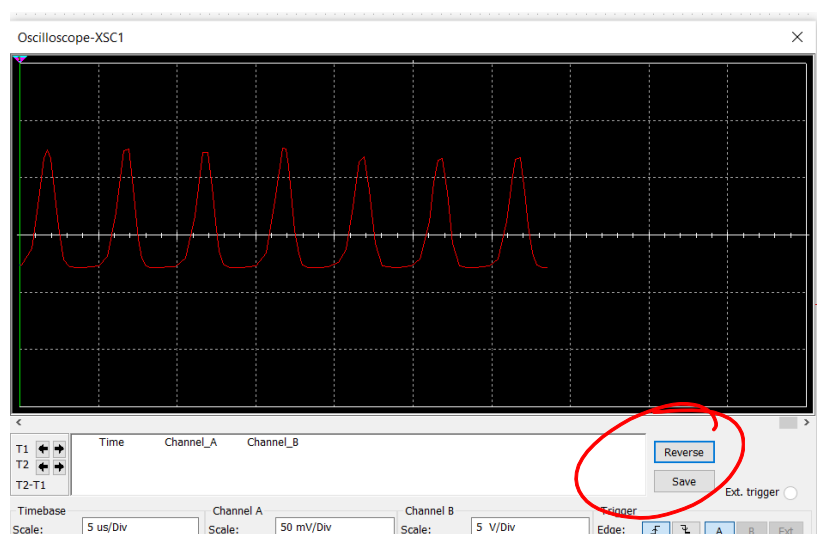


Figure 13 - Oscilloscope Results Showing Oscillator Waveform

USE THIS TO GIVE A WHITE BACKGROUND

This waveform is acceptable for this application.

Now the 2 circuits could be combined to produce the final, desired circuit, this is shown in Fig.10. With the circuit in this configuration the final round of tests can be completed. The final circuit provided a gain of 33.4, this was found by dividing the output voltage by the input.

$$Gain = \frac{3.34}{0.1} = 33.4$$

The data point at the output of the circuit showed the frequency was 200kHz. Finally, the waveform was checked to see if it was sinusoidal, to do this an oscilloscope was connected between the output and the 0V rail. Here is the resulting waveform:

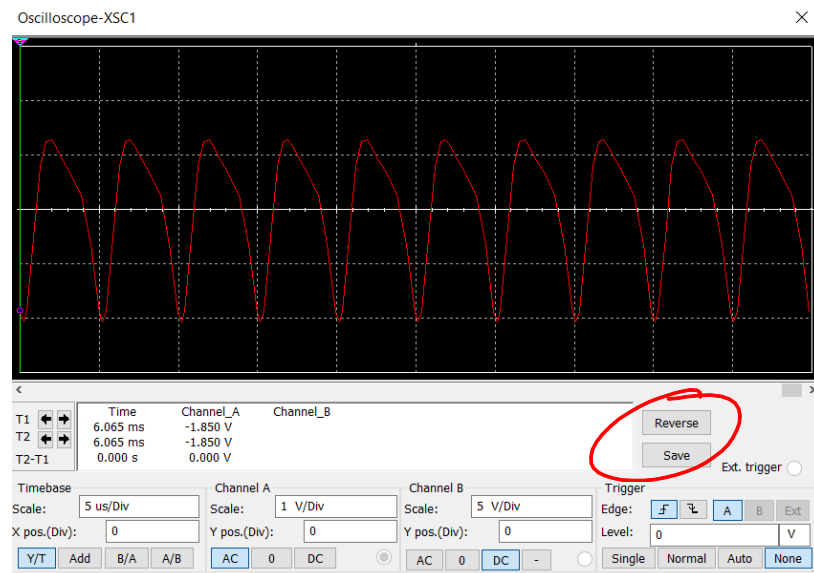


Figure 14 - Oscilloscope Reading of Output Signal

This waveform is the correct shape, this concluded the testing for the circuit.

5. Conclusions and recommendations

In conclusion the amplifier built was only partially able to achieve the criteria set. The characteristics of the built amplifier have been tabulated to be compared to the brief.

Criteria	Specification	Built Circuit
Gain	50	33.4
Centre Frequency	200 kHz	199.53 kHz
Bandwidth	60 kHz	348.4627 kHz

These results show that the combined circuit only met 1 of 3 criteria. The constructed amplifier had a gain of 50 before being connected to the oscillator, it is unknown why the gain experienced this drop. The values shown above were achieved through repetitive testing and adjustments. The bandwidth was reduced from 15.383MHz to 348.4627kHz, and the gain reduced from 264 to 50. This was done by introducing negative feedback to the amplifier. The negative feedback was connected in the 'series-parallel' configuration described in the research. The task required the voltage to be amplified, the research showed that it was best to connect the circuit in series-parallel for this application.

The bandwidth could have been improved more using a tuned circuit. A tuned circuit is an L-C circuit, this would have to be connected to the output of the amplifier. The L-C circuit can tune the output signal by setting the component values to a desired resonant frequency. The equation to calculate this is:

$$f_r = \frac{1}{2\pi\sqrt{LC}}$$

The L-C circuit will be constructed to the required frequency and when connected to the output it will filter any frequencies that aren't close to the 200kHz resonant frequency. The range of the filtered frequencies is determined by the quality factor. The equation for Q factor is:

$$Q = \frac{2\pi f_r L}{R}$$

The higher the value for the Q factor, the better the selectivity of the circuit will be.

6. Reference list and bibliography

Bird, J., 2014. *Electrical Circuit Theory and Technology*. 5th ed. London: Routledge.

Electrical 4 U, 2020. *Oscillators: What Are They? (Definition, Types, & Applications)*. [Online]
Available at: <https://www.electrical4u.com/what-is-an-oscillator/#:~:text=An%20oscillator%20is%20a%20circuit,decided%20by%20its%20circuit%20components.>

[Accessed 6 January 2021].

Elprocus, 2021. *The Classes and Classification of Amplifiers With Their Applications*. [Online]
Available at: <https://www.elprocus.com/classification-of-amplifiers-classes-its-applications/>
[Accessed 6 January 2021].

Fiore, J.M., 2020. The Four Variants of Negative Feedback. Available at:
<https://chem.libretexts.org/@go/page/3561> [Accessed 6 January, 2021].

Kuphaldt, T. R., . A. a. C., 2021. *Lessons in Electric Circuits*. [Online]
Available at: <https://www.allaboutcircuits.com/textbook/semiconductors/chpt-1/active-versus-passive-devices/>
[Accessed 4 January 2021].

GOOD, RELIABLE SOURCES.

7. Annexe for diagrams, data sheets, results tables etc

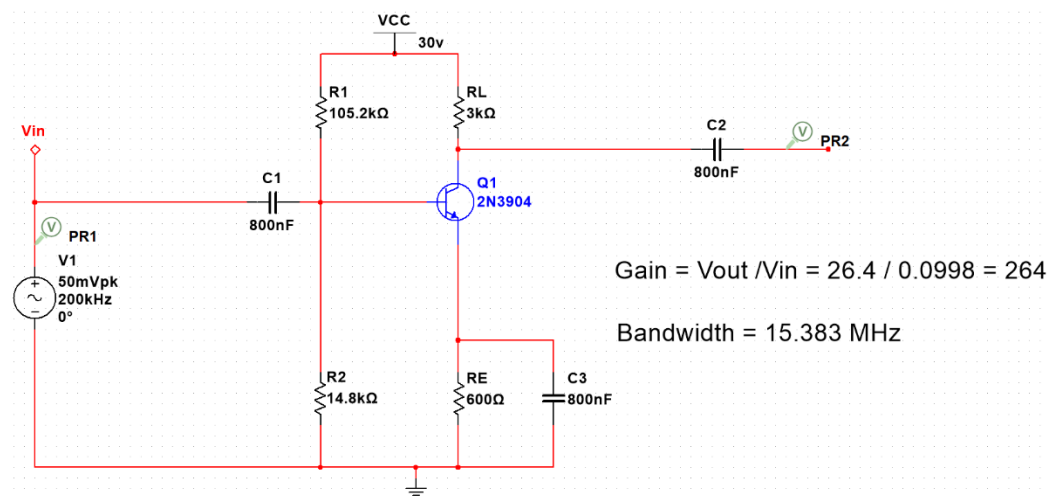


Figure 5 - Initial Circuit Design

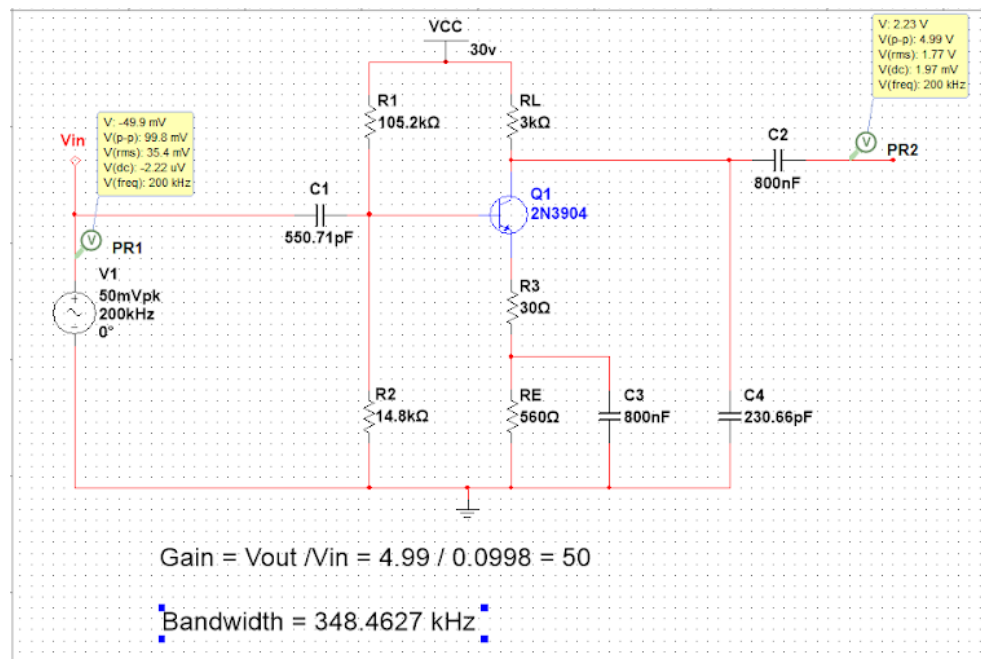


Figure 7 - Complete Amplifier Circuit

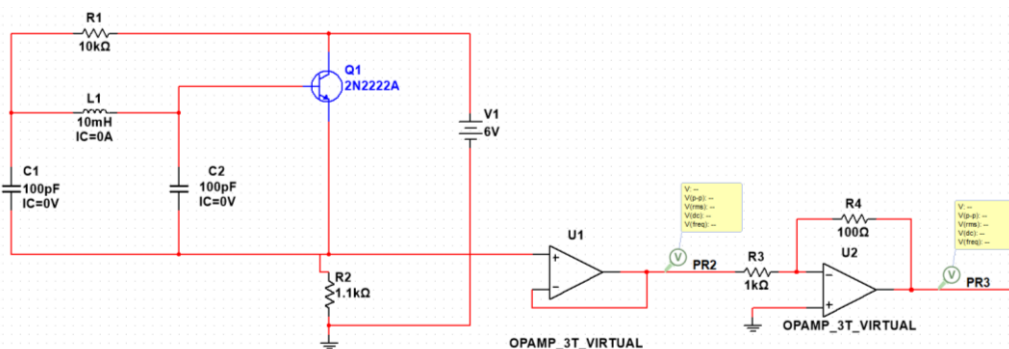
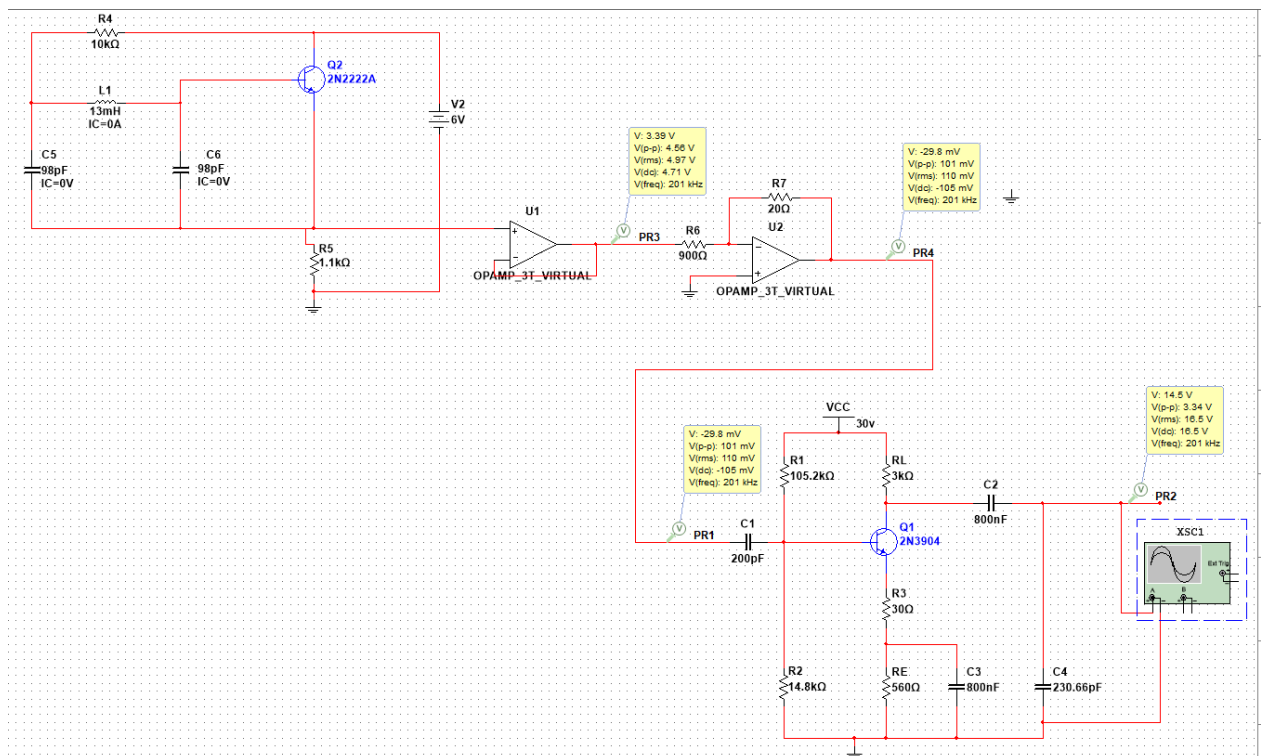
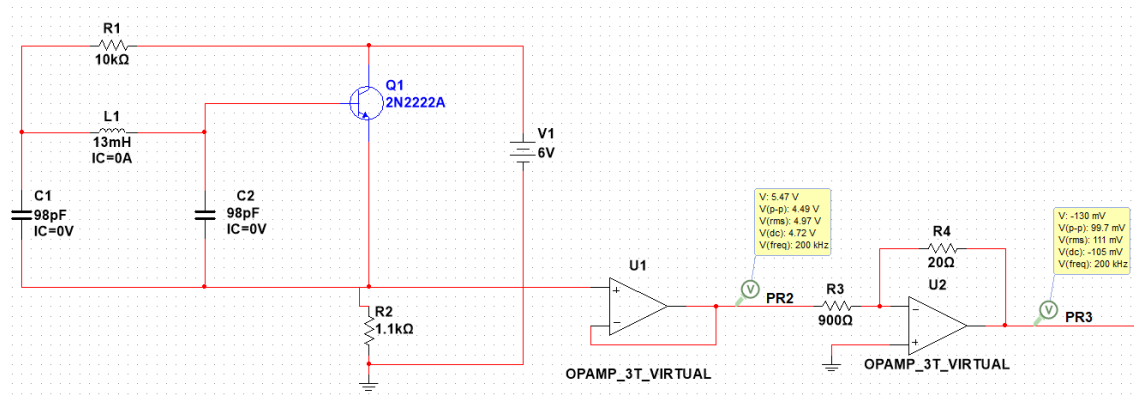


Figure 8 - Provided Colpitts Oscillator



2N3903, 2N3904

General Purpose Transistors

NPN Silicon

Features

- Pb-Free Packages are Available*

MAXIMUM RATINGS

Rating	Symbol	Value	Unit
Collector – Emitter Voltage	V_{CEO}	40	Vdc
Collector – Base Voltage	V_{CBO}	60	Vdc
Emitter – Base Voltage	V_{EBO}	6.0	Vdc
Collector Current – Continuous	I_C	200	mAdc
Total Device Dissipation @ $T_A = 25^\circ\text{C}$ Derate above 25°C	P_D	625 5.0	mW mW/ $^\circ\text{C}$
Total Device Dissipation @ $T_C = 25^\circ\text{C}$ Derate above 25°C	P_D	1.5 12	W mW/ $^\circ\text{C}$
Operating and Storage Junction Temperature Range	T_J, T_{stg}	-55 to +150	$^\circ\text{C}$

THERMAL CHARACTERISTICS (Note 1)

Characteristic	Symbol	Max	Unit
Thermal Resistance, Junction-to-Ambient	$R_{\theta JA}$	200	$^\circ\text{C/W}$
Thermal Resistance, Junction-to-Case	$R_{\theta JC}$	83.3	$^\circ\text{C/W}$

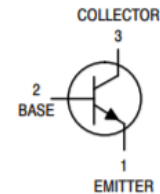
Stresses exceeding Maximum Ratings may damage the device. Maximum Ratings are stress ratings only. Functional operation above the Recommended Operating Conditions is not implied. Extended exposure to stresses above the Recommended Operating Conditions may affect device reliability.

1. Indicates Data in addition to JEDEC Requirements.

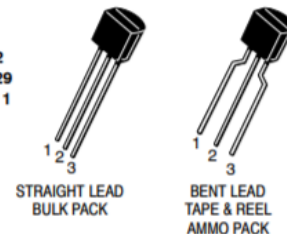


ON Semiconductor®

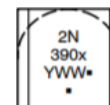
<http://onsemi.com>



TO-92
CASE 29
STYLE 1



MARKING DIAGRAMS



2N3903, 2N3904

ELECTRICAL CHARACTERISTICS ($T_A = 25^\circ\text{C}$ unless otherwise noted)

Characteristic	Symbol	Min	Max	Unit
OFF CHARACTERISTICS				
Collector – Emitter Breakdown Voltage (Note 2) ($I_C = 1.0\text{ mAdc}$, $I_B = 0$)	$V_{(BR)CEO}$	40	–	Vdc
Collector – Base Breakdown Voltage ($I_C = 10\text{ }\mu\text{Adc}$, $I_E = 0$)	$V_{(BR)CBO}$	60	–	Vdc
Emitter – Base Breakdown Voltage ($I_E = 10\text{ }\mu\text{Adc}$, $I_C = 0$)	$V_{(BR)EBO}$	6.0	–	Vdc
Base Cutoff Current ($V_{CE} = 30\text{ Vdc}$, $V_{EB} = 3.0\text{ Vdc}$)	I_{BL}	–	50	nAdc
Collector Cutoff Current ($V_{CE} = 30\text{ Vdc}$, $V_{EB} = 3.0\text{ Vdc}$)	I_{CEX}	–	50	nAdc
ON CHARACTERISTICS				
DC Current Gain (Note 2) ($I_C = 0.1\text{ mAdc}$, $V_{CE} = 1.0\text{ Vdc}$)	h_{FE}	20	–	–
($I_C = 1.0\text{ mAdc}$, $V_{CE} = 1.0\text{ Vdc}$)		40	–	–
($I_C = 10\text{ mAdc}$, $V_{CE} = 1.0\text{ Vdc}$)		35	–	–
($I_C = 10\text{ mAdc}$, $V_{CE} = 1.0\text{ Vdc}$)		70	–	–
($I_C = 50\text{ mAdc}$, $V_{CE} = 1.0\text{ Vdc}$)		50	150	–
($I_C = 100\text{ mAdc}$, $V_{CE} = 1.0\text{ Vdc}$)		100	300	–
Collector – Emitter Saturation Voltage (Note 2) ($I_C = 10\text{ mAdc}$, $I_B = 1.0\text{ mAdc}$)	$V_{CE(sat)}$	–	0.2	Vdc
($I_C = 50\text{ mAdc}$, $I_B = 5.0\text{ mAdc}$)		–	0.3	Vdc
Base – Emitter Saturation Voltage (Note 2) ($I_C = 10\text{ mAdc}$, $I_B = 1.0\text{ mAdc}$)	$V_{BE(sat)}$	0.65	0.85	Vdc
($I_C = 50\text{ mAdc}$, $I_B = 5.0\text{ mAdc}$)		–	0.95	Vdc

FINAL GRADE

GENERAL COMMENTS

Instructor

85/100

PAGE 1

PAGE 2

PAGE 3

PAGE 4

PAGE 5

PAGE 6

PAGE 7

PAGE 8

PAGE 9

PAGE 10

PAGE 11

PAGE 12

PAGE 13

PAGE 14

PAGE 15

PAGE 16

PAGE 17

PAGE 18

PAGE 19

GRADING FORM: CSG-RUBIC

BENJAMIN MORGAN**0****RESEARCH-INFORMED LITERATURE**

There was evidence of extensive further reading and research from credible and reliable sources. Your interpretation and understanding of the additional research was very good and demonstrated a deep underlying appreciation of the subject. Your references were consistently applied and appropriate to the task, however you need to cite diagrams using the short citations as well. Your reference list was of an appropriate length for this assignment, but an extra couple of sources wouldn't hurt.

8/10**KNOWLEDGE AND UNDERSTANDING**

The knowledge and understanding of the subject demonstrated by this assignment was exceptional. There were no gaps in your knowledge of the basic principles and the higher level concepts were extremely well explained. Your explanation of the design process and calculations was also excellent

27/30

and you have demonstrated an exceptionally detailed knowledge and critical understanding of the main concepts and expanded your knowledge beyond that contained in the course notes.

ANALYSIS



Your analysis of the task was excellent, both in class and as shown in this assignment. You have demonstrated highly competent analytical skills and your approach to the task was highly logical. You carried out a detailed evaluation of the results and included an excellent conclusion relating your practical results to accepted theory. You have demonstrated a capacity for independent thought and your interpretation of the quantitative and qualitative data was excellent with sound judgements based on wholly valid arguments.

27/30

PRACTICAL APPLICATION AND DEP.



Your application of practical skills for this task was generally exceptional. You used both the basic and advanced features of the resources appropriately when designing the circuit and you used appropriate advanced options for testing and evaluating the circuit performance. You managed to design an effective circuit that functioned well within the specification and achieved this by rarely asking for assistance thus demonstrating some advanced problem solving skills. Your results were comprehensive and allowed for a detailed conclusion to be attempted.

18/20

SKILLS FOR PROFESSIONAL PRAC.



In this assignment you have demonstrated skills in working individually and also collaboratively. Your written report was exceptionally well structured and written in a highly coherent and logical way. The report was presented in a professional manner using appropriate language and formatting, good diagrams in appropriate places, good sentence/paragraph structure and clear calculations. Word count exceeded by 1000 words.

5/10