

Assignment 1

by Steffan Davies

FILE	15323_STEFFAN_DAVIES_ASSIGNMENT_1_178835_1295597715.DOCX (1.03M)		
TIME SUBMITTED	31-JAN-2021 07:31PM (UTC+0000)	WORD COUNT	2902
SUBMISSION ID	143856340	CHARACTER COUNT	15611

Course: HNC Electrical and Electronic Engineering

Module: Electronics I

Assignment: 1. Analogue Circuit Design

Name: Steffan Wyn Davies

University number: 1605160

Date submitted: 31/01/21

Word count: 2508

✓ 75-80

- ① GOOD RESEARCH AND REFERENCING.
- ② DESIGN METHOD WAS WELL EXPLAINED.
- ③ YOU GOT YOURSELF CONFUSED WITH REFINING THE DESIGN AND COMBINED GAIN/BANDWIDTH
- ④ RESULTS & ANALYSIS WERE GOOD.
- ⑤ VALID CONCLUSION & RECOMMENDATIONS MADE.

Contents

Introduction	3
Research.....	3
Amplifier design	6
Designing an Oscillator.....	13
Conclusions and recommendations.....	15
Bibliography	16
Figure 1-Class A Amplifier (Aspencore, 2020)	3
Figure 2-Transformer coupled Class B Amplifier	4
Figure 3- Cross-over distortion	Error! Bookmark not defined.
Figure 4-Class AB amplifier (Anon., 2020)	5
Figure 5-Non-sinusoidal waveform (Omegatron, 2020)	6
Figure 6-DC conditions.....	8
Figure 7-AC conditions	9
Figure 8-Input impedance calculation	10
Figure 9-bandwidth.....	10
Figure 10-Gain.....	11
Figure 11-final design for amplifier.....	12
Figure 12-Centre frequency	12
Figure 13-Bandwidth.....	12
Figure 14-Oscillator design	13
Figure 15-Oscillator output waveform.....	13
Figure 16-Oscillator output final circuit	14
Figure 17- 2N3904 Data sheet (STmicroelectronics, 2003)	17

Executive summary

Could be lower and cover all of the report

This project involved the design, simulation, testing and evaluation of electronic circuits. The primary purpose of this assignment was to design an amplifier to the specification required. Research was carried out into the different types of amplifiers and data sheets were used to analyse electronic circuits. Initially, the DC conditions were set in order to bias the transistor. After this, AC conditions were set. However, this resulted in a bandwidth far out of specifications, therefore negative feedback was introduced to reduce the bandwidth to as low as possible. Overall, the design was successful, however the final output after adding the oscillator was not perfectly sinusoidal. Due to the amount of counteractive components, it was very difficult to reach the specification

Introduction

This assignment is an investigation into semi-conductor components, their characteristics and their application in electronic circuits. Data sheets and research will be used to design, simulate, test and evaluate electronic circuits

Research

Amplifiers are used to increase the magnitude of the input signals, meaning that if you put a small input signal in then you will get a larger output. These are split into classes depending on their construction and operating characteristics (Aspencore, 2020). There are several things that can differentiate these amplifiers, such as linearity, power output, signal gain and efficiency. A few of the most common types can be seen below:

Class A amplifier

According to (Aspencore, 2020), this is one of the most common types of amplifiers as they use only one output switching transistor within the amplifier design. There are several advantages to the class A amplifier, including it's excellent linearity, high gain and low signal distortion making it very practical for audio quality applications. The Class A amplifier conducts current over the full 360 degree cycle meaning the output device is ON at all times. This is one of the main problems for the class A amplifier as it's constantly carrying current, which results in a continuous loss of power and overheating problem. See figure 1 below for a typical class A amplifier design along with the input and output signals:

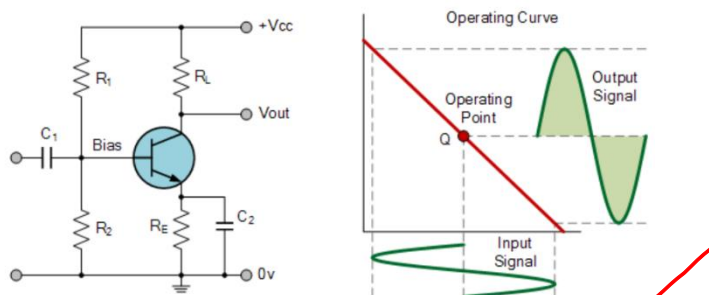
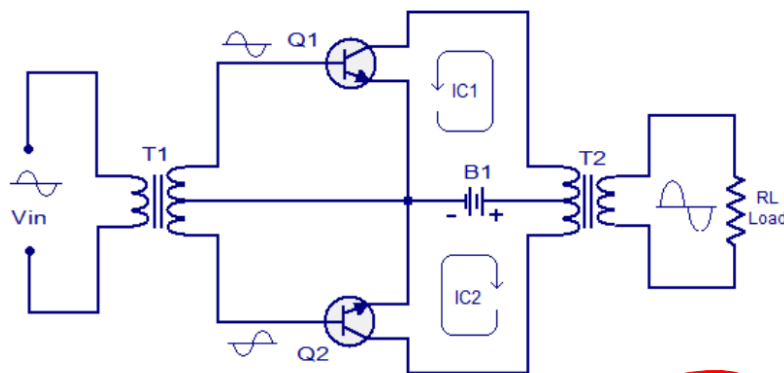


Figure 1-Class A Amplifier

(Aspencore, 2020)

Class B Amplifier

The class B amplifier was designed to overcome the class A's efficiency problem. According to (jojo, 2020) in order to achieve a practical Class B amplifier, 2 transistors need to be arranged in a push pull configuration with each transistor having the exact same input signal, but in opposite directions. This means that one of the transistors will amplify for the first half of the cycle (0-180 degrees) and the other transistor for the second half of the cycle (180-360 degrees). However, to do this there must be a way to split the input wave in order to ensure both transistors receive the same amount of input signal. The answer to this problem is a transformer coupled Class B amplifier, which can be seen in figure 2 below:



Transformer coupled Class B amplifier

www.circuitstoday.com

USE PROPER
INTEXT CITATIONS.

Figure 2-Transformer coupled Class B Amplifier

Each transistor only begins to conduct when the input signal is around 0.7V, meaning that if there is no input signal then no current would flow through the load due to the fact that the transistors are not biased. Although this dramatically increases the class B's efficiency compared to the Class A amplifier, coupling transformers increase the cost and size and finding ideal transformers is difficult. Furthermore, the biggest problem with the class B amplifier is its crossover distortion. According to (Aspencore, 2020) the crossover distortion is when the part of the output waveform below 0.7V isn't reproduced accurately at the time when both one transistor is switching over to the other. This can be seen in Figure 3 below, demonstrating the effect of cross over distortion:

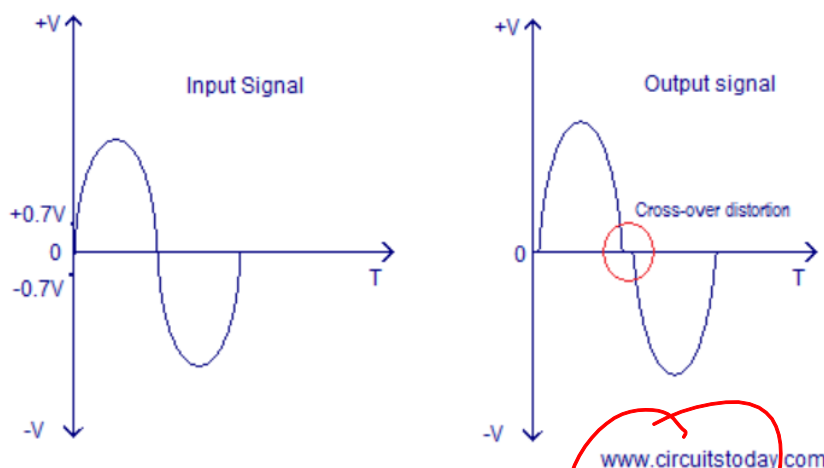


Figure 3-Crossover distortion

Class AB Amplifier

The class AB amplifier as the name suggests is a combination of Class A and Class B amplifiers. A combination of the high efficiency output of the class B amplifier and the low distortion of the class A is what's required for this amplifier. According to (Aspencore, 2020), this is achieved by pre-biasing the transistors used for the output stage. This biasing is done to ensure that both the transistors conduct at the same time during a very small part of the input waveform (more than 50% but less than 100%). This will mean that the transistor will be on for more than one half of the cycle, but much less than the full cycle of the input signal (Anon., 2020) As a result of this, the cross over distortion is greatly reduced, making this class of amplifier suitable for applications such as audio power due to it's high linearity and low crossover distortion. See figure 4 below demonstrating the construction of the amplifier and the input/output signals of one of the transistors:

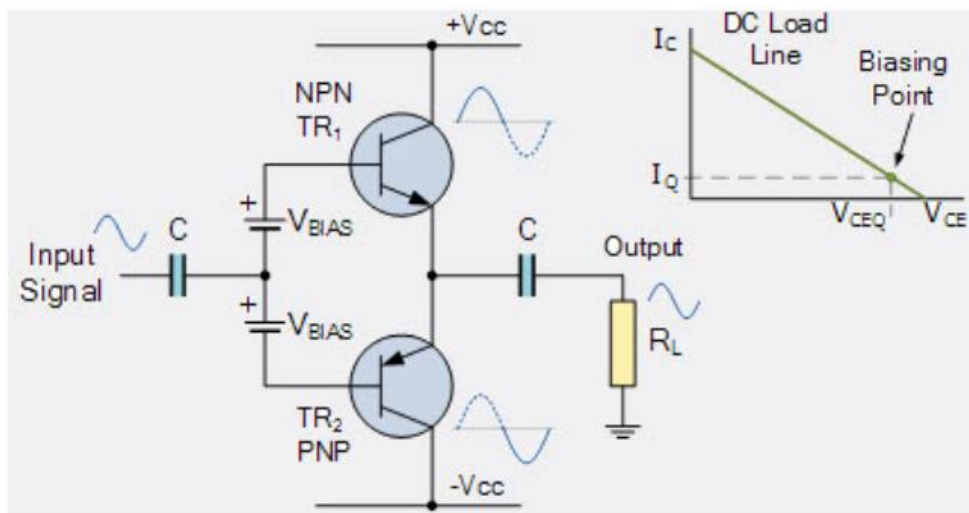


Figure 4-Class AB amplifier (Anon., 2020)

Oscillators

An oscillator produces a continuous alternating waveform without any input. It works by converting unidirectional current flow from a DC source into an alternating waveform of the desired frequency, which is decided by its circuit components. (Anon., 2020) states that oscillators are split into 2 main groups:

Sinusoidal/Harmonic oscillators

These type of oscillators generate a sine waveform and can be split into these 4 types of categories:

1. **Tuned circuit Oscillators**- Use a tuned circuit consisting of inductors and capacitors to generate high frequency signals
2. **RC Oscillators**- this type of oscillator uses resistors and capacitors to generate low/audio-frequency signals
3. **Crystal oscillators**- this type of oscillator use quartz crystals and are used to generate frequencies up to 10MHz
4. **Negative resistance Oscillators**- This type of oscillator uses negative-resistance characteristic of the devices such as tunnel diodes

Non-sinusoidal oscillators

This type of oscillators can produce frequencies from zero to 20MHz. The output of these oscillators produce a square, rectangular/triangular or saw tooth waveform, as can be seen in figure 5 below:

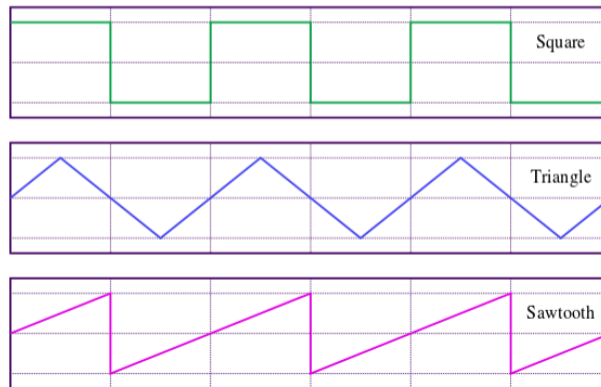


Figure 5-Non-sinusoidal waveform (Omegatron, 2020)

Amplifier design

The specification for the amplifier can be seen below:

- Centre frequency: 200KHz
- Bandwidth: 60KHz
- Gain: 50
- Input signal: 50mV

DC conditions

It's very important to set the DC conditions as the first step of designing an amplifier. Making sure that the operating points are set appropriately play an important role in biasing the transistor in order to ensure it performs as a linear amplifier. The transistor that was chosen for this task was the 2N3904 transistor. This is a silicon NPN Bipolar Junction Transistor. The reason why this specific transistor was selected is because it's good at dealing with small signals and small frequencies. By looking at the data sheet for this transistor (figure 6), we can see that the h_{FE} was between 100 and 300, therefore it was decided to pick 200 as the DC current gain parameter. A value of 30V was decided for the common collector voltage for this circuit.

In order to decide on an appropriate value for the quiescent current (I_q), ideally it needed to be between 1-5mA, so the value that was decided was 3mA. From this, the collector voltage can be determined as it is half of V_{CC} , therefore:

$$V_C = \frac{30}{2}$$

$$V_C = 15V$$

Using the following equation, the resistance of the load resistor can be calculated below:

$$R_L = \frac{V_{CC} - V_c}{I_q}$$

$$R_L = \frac{30 - 15}{3\text{mA}}$$

$$R_L = 5\text{K}\Omega$$

The emitter resistance can be calculated after calculating a value for V_e , this is typically 10% of $V_{CC}(30\text{V})$. Therefore assuming that V_e is 3, then:

$$R_E = \frac{V_E}{I_Q}$$

$$R_E = \frac{3\text{V}}{3\text{mA}}$$

$$R_E = 1000\Omega$$

As already mentioned, the h_{FE} was determined at 200, therefore allowing us to work out the base current (I_B) using the following equation:

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

$$I_B = \frac{3\text{mA}}{200}$$

$$I_B = 15\mu\text{A}$$

The value of V_B could then be calculated by using the following equation

$$V_B = V_E + 0.7\text{V}$$

$$V_B = 3\text{V} + 0.7\text{V}$$

$$V_B = 3.7\text{V}$$

The next step was to calculate the DC bias network's current. The current flowing through R_1 and R_2 needs to be ten times greater than the base current, This is to ensure good DC stability. The current can be calculated using the following equation:

$$I(R_1 + R_2) = I_B \times 10$$

$$I(R_1 + R_2) = 15\mu\text{A} \times 10$$

$$I(R_1 + R_2) = 150\mu\text{A}$$

The calculation for resistance values for R_1 and R_2 can be seen below:

$$R_1 = \frac{V_{CC} - V_B}{I(R_1 + R_2)} \quad R_2 = \frac{V_B}{I(R_1 + R_2)}$$

$$R_1 = \frac{30 - 3.7}{150\mu\text{A}} \quad R_2 = \frac{3.7}{150\mu\text{A}}$$

$$R_1 = 175.3\text{k}\Omega \quad R_2 = 24.6\text{k}\Omega$$

In figure 7 below, the DC conditions for this circuit can be seen, showing the resistor values and voltage values for VC, VB and VE:

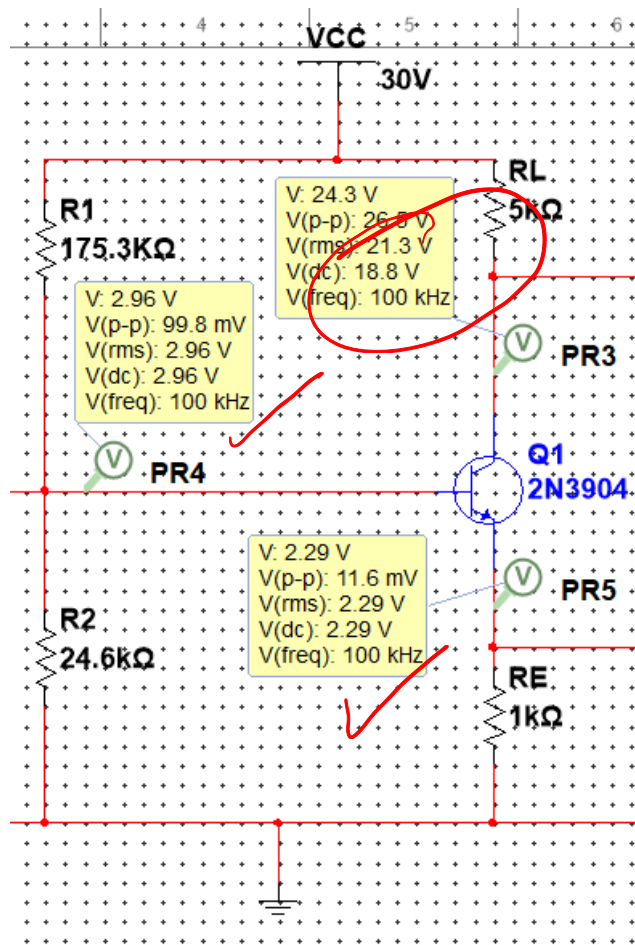


Figure 6-DC conditions

AC conditions

Capacitors C1 and C2 are then added to the circuit in order to act as a coupling capacitor. According to (EETechmedia, 2020), the purpose of coupling capacitor are to be connected in series to the signal path and are used to filter out DC signal. The purpose of coupling capacitors is to ensure that the incoming AC signal doesn't interfere with the bias voltage applied to the transistor. When selecting a capacitor, the value must be chosen appropriately to ensure that it allows signal at all the desired frequencies to pass. The following equation was used to calculate a value for the capacitor C1 and C2

$$X_c = \frac{1}{2\pi fC}$$

Re-arranged to

$$C = \frac{1}{2\pi fX_c}$$

$$C = \frac{1}{2\pi \times 200\text{KHz} \times 1\Omega}$$

$$C = 800\text{nF}$$

Another capacitor was then added across the emitter resistor RE. The purpose of this was to prevent any AC voltages appearing across RE. If AC voltages would be present across RE the amplifiers gain would be reduced. In order to calculate the value for this capacitor, the following equation was manipulated and used:

$$X_c = \frac{1}{2\pi FC}$$

Re-arranged to

$$C = \frac{1}{2\pi FX_c}$$

$$C = \frac{1}{2\pi \times 200KHz \times 1\Omega}$$

$$C = 800nF$$

After adding these capacitors into the circuit, it looked like figure 7, seen below:

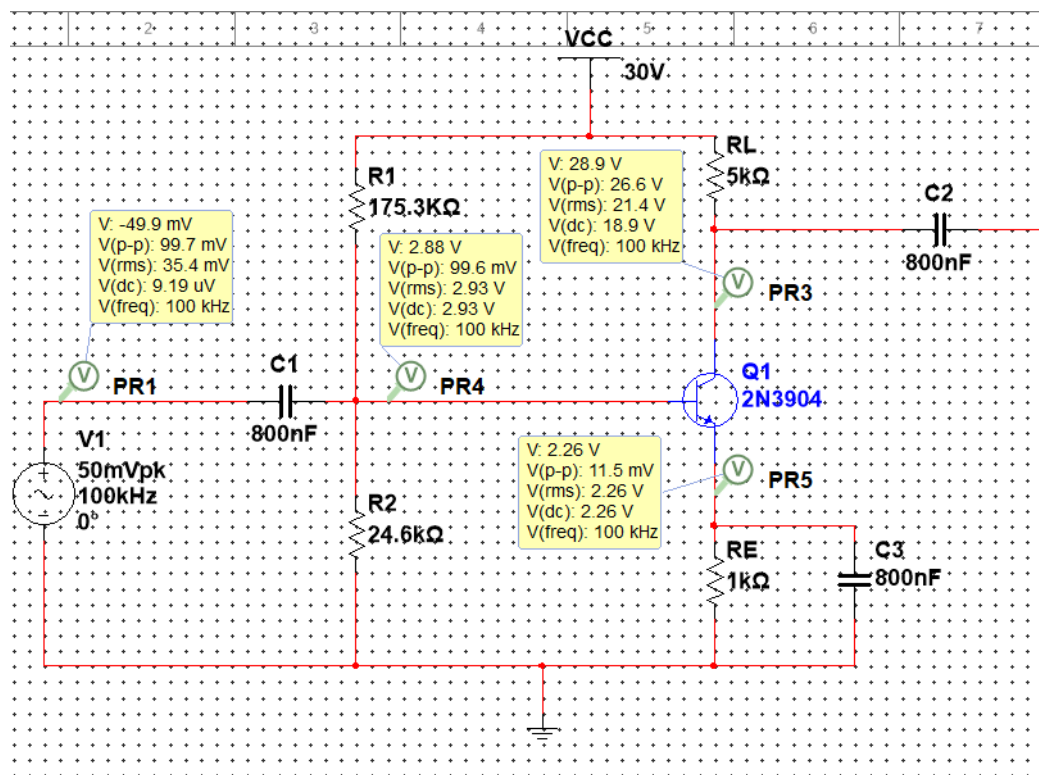


Figure 7-AC conditions

From figure 7, the gain of the circuit can be calculated using the following formula:

$$Gain = \frac{V_{out}}{V_{in}}$$

$$Gain = \frac{26.6V}{99.7mV}$$

$$Gain = 266.8$$

As can be seen from figure 7, the output peak to peak voltage was 26.6V. The circuit impedance was then calculated using a 1K variable resistor set at 0Ω. At that point, the voltage source is equal to the input voltage. After calculating the output voltage at 26.6V, then the variable resistor was adjusted for half of the output voltage. This can be seen in figure 8 below:

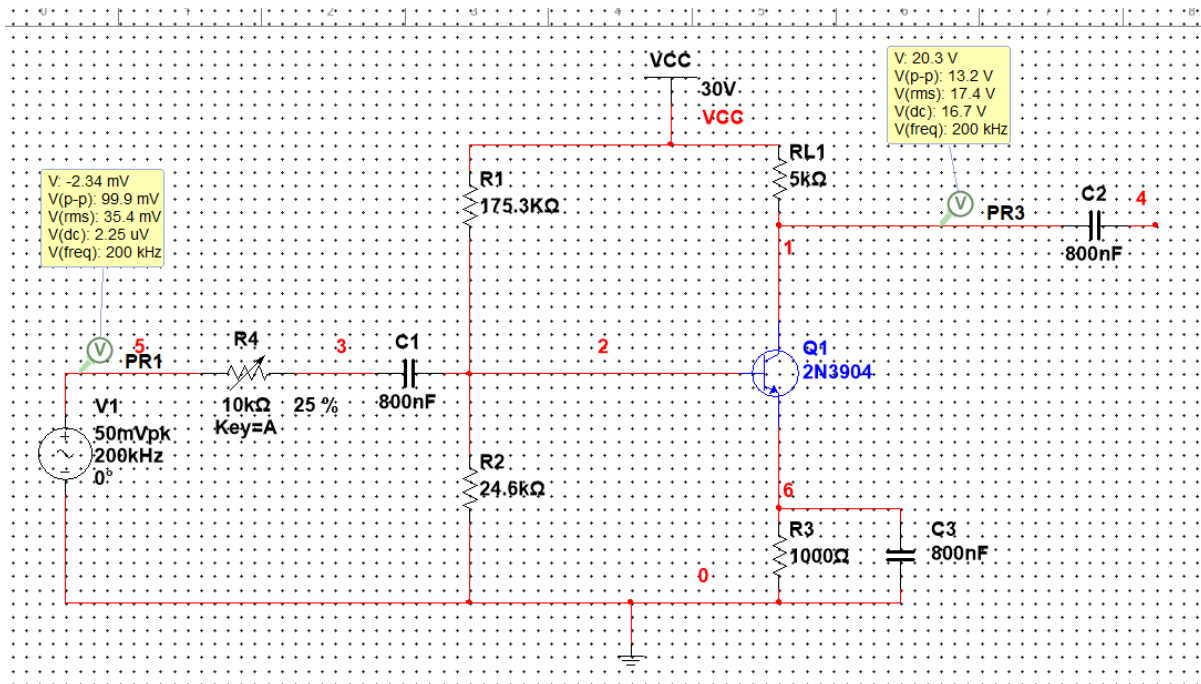


Figure 8-Input impedance calculation

As can be seen from figure 8, the 10KΩ variable resistor was set to 25% when the output voltage was half the original value, therefore the input impedance of this circuit is 2500Ω. Below is a screenshot of the AC sweep before adjusting C1 and C4. As can be seen from figure 9 below, the bandwidth is 12.3778M. This is far from the required specification and therefore negative feedback was required to reduce the bandwidth:

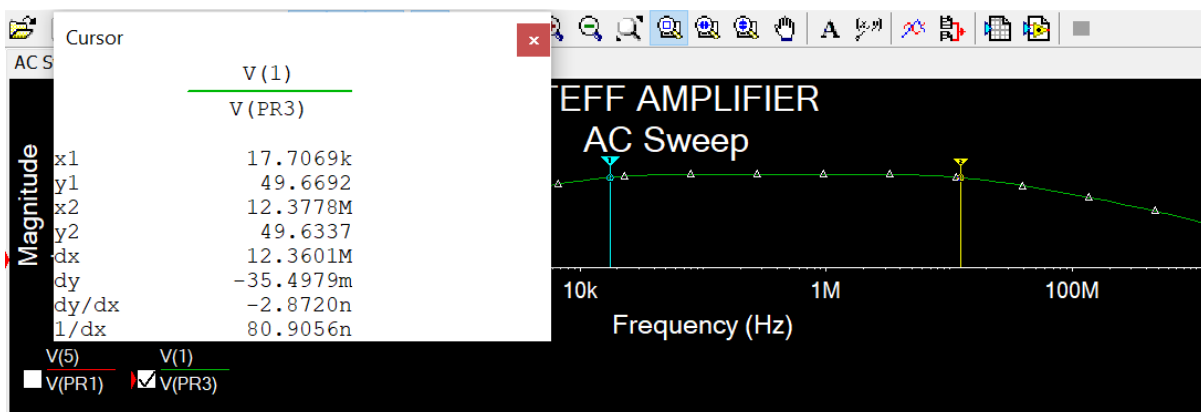


Figure 9-bandwidth

USE A WHITE
BACKGROUND

Negative feedback

AND GAIN

The purpose of introducing negative feedback to a circuit is to reduce distortion and noise.

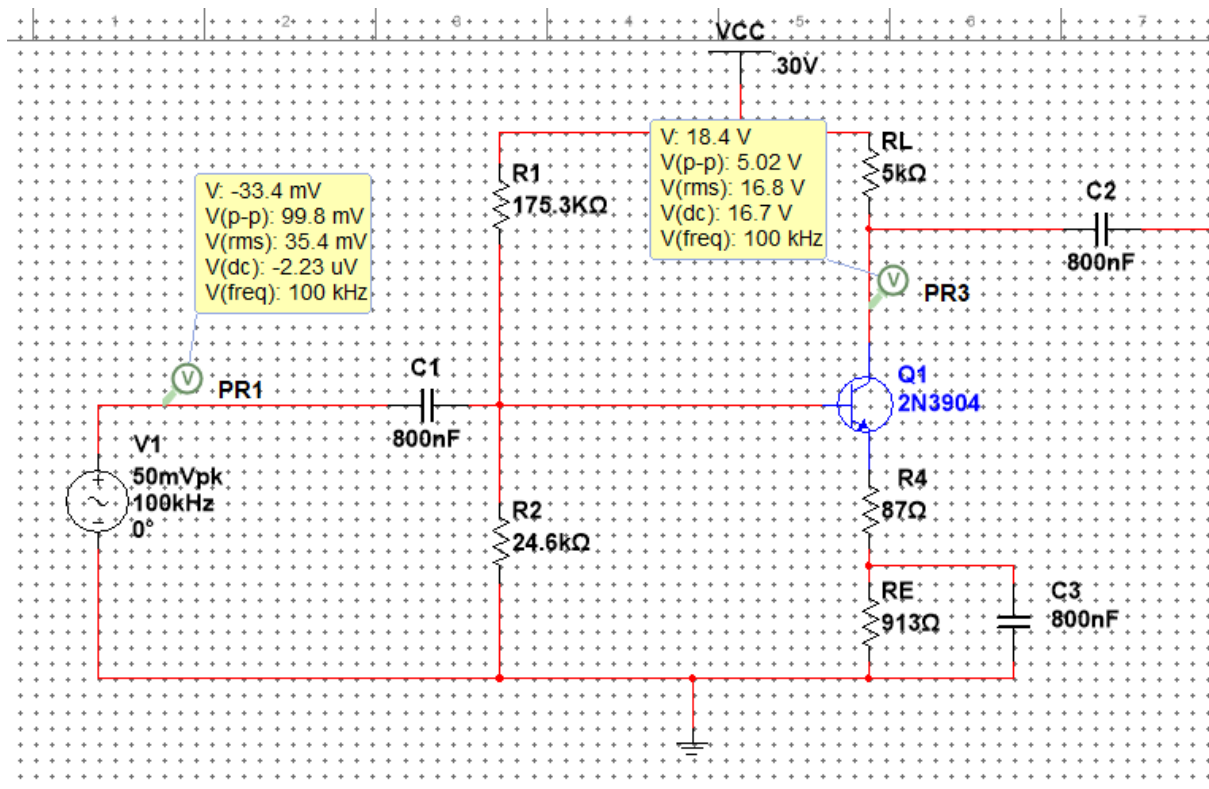


Figure 10-Gain

After achieving the gain required by the requirements of the circuit, the value of C1 was changed and another capacitor(C4) was added to the circuit in order to reduce the bandwidth to around 60KHz. The capacitor values were determined using the following equations:

$$C1 = \frac{1}{2\pi \times Fl \times Xc}$$

$$C1 = \frac{1}{2\pi \times 170KHz \times 2500\Omega}$$

$$C1 = 374.5pF$$

$$C4 = \frac{1}{2\pi \times Fh \times Xc}$$

$$C4 = \frac{1}{2\pi \times 230KHz \times 5k\Omega}$$

$$C4 = 138.4pF$$

THIS IS BANDWIDTH
NOT GAIN.

The decoupling effect of C3 was reduced by splitting the emitter resistor into two different values adding up to the value of RE. These 2 resistors were adjusted via a trial and error method, until a gain of 50 was achieved. The gain was calculated using the following equation, and can be seen in figure 9 below:

$$Gain = \frac{5.02V}{99.8mV}$$

$$Gain = 50.3$$

YOU HAVE COMBINED
ADJUSTING GAIN + B/W
THEY ARE SEPARATE
PROCESSES.

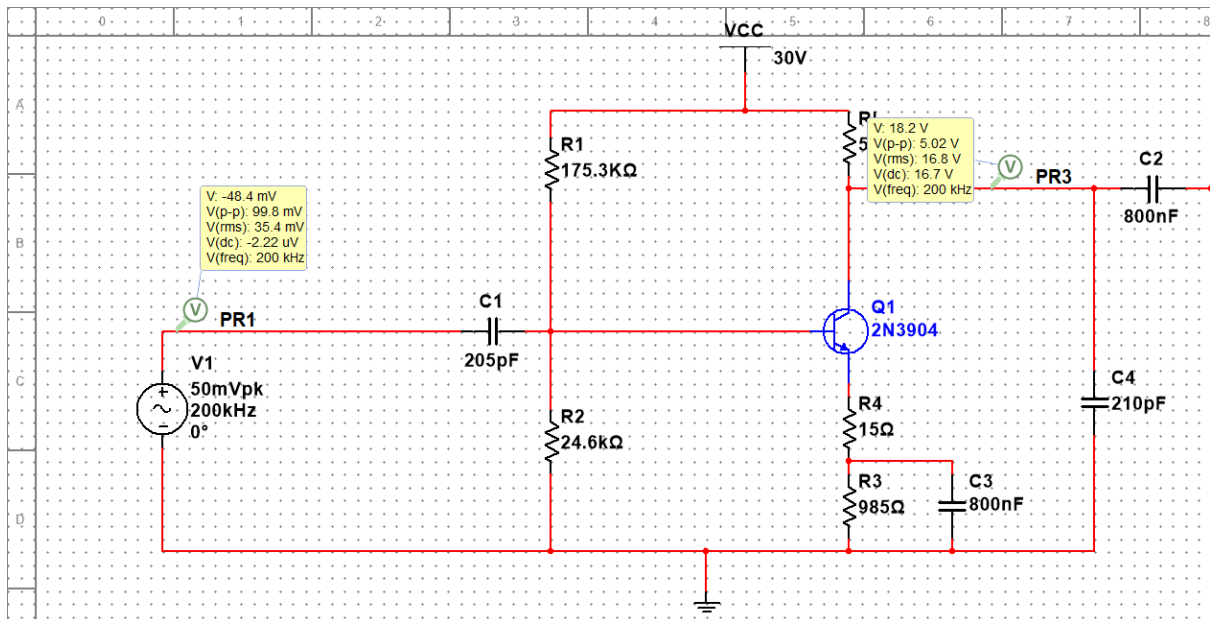


Figure 11-final design for amplifier

Figure 10 above is the final design for the amplifier. This is the closest possible design to the specifications due to the fact that by adjusting the bandwidth, the gain would change and vice versa. Figure 10 and 11 below show the centre frequency and bandwidth of the circuit:

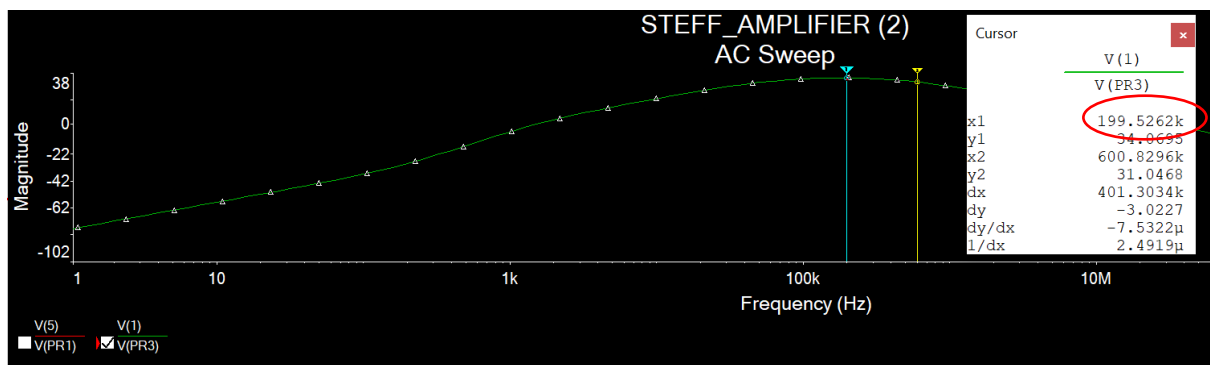


Figure 12-Centre frequency

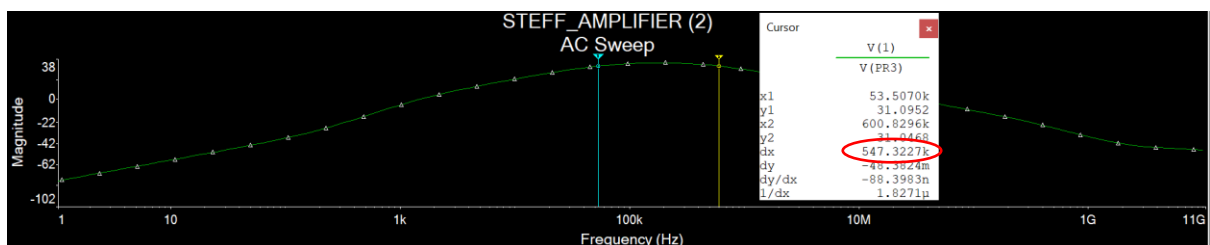


Figure 13-Bandwidth

SHOW PLOTS WITH
A WHITE BACKGROUND

Designing an Oscillator

This task required designing a sine-wave oscillator to operate at 200KHz with an output of 100mV peak to peak value. Figure 10 below shows the oscillator after being built to the required specification:

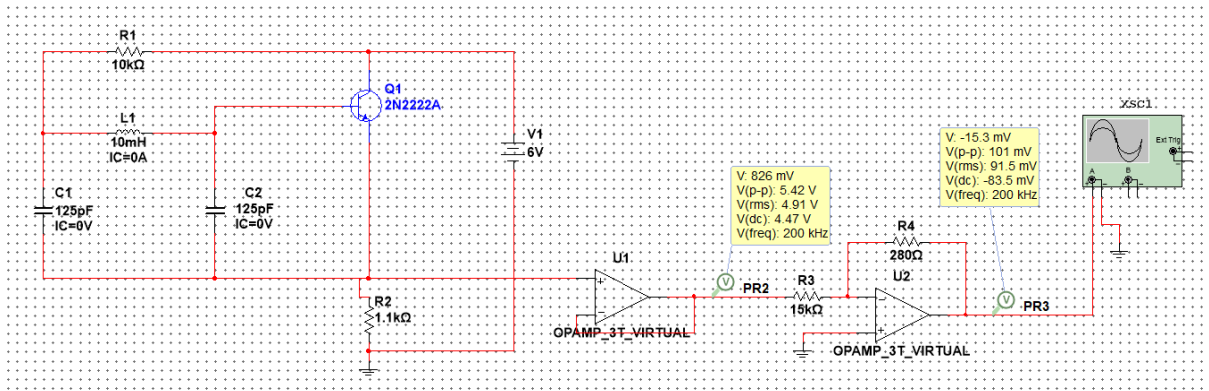


Figure 14-Oscillator design

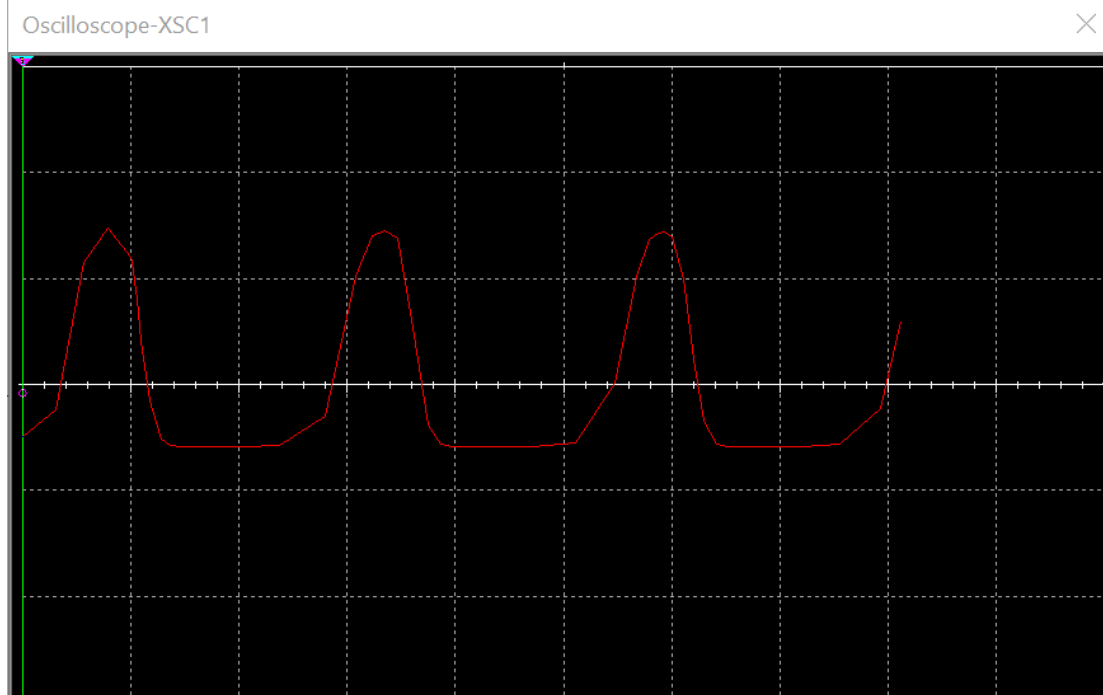
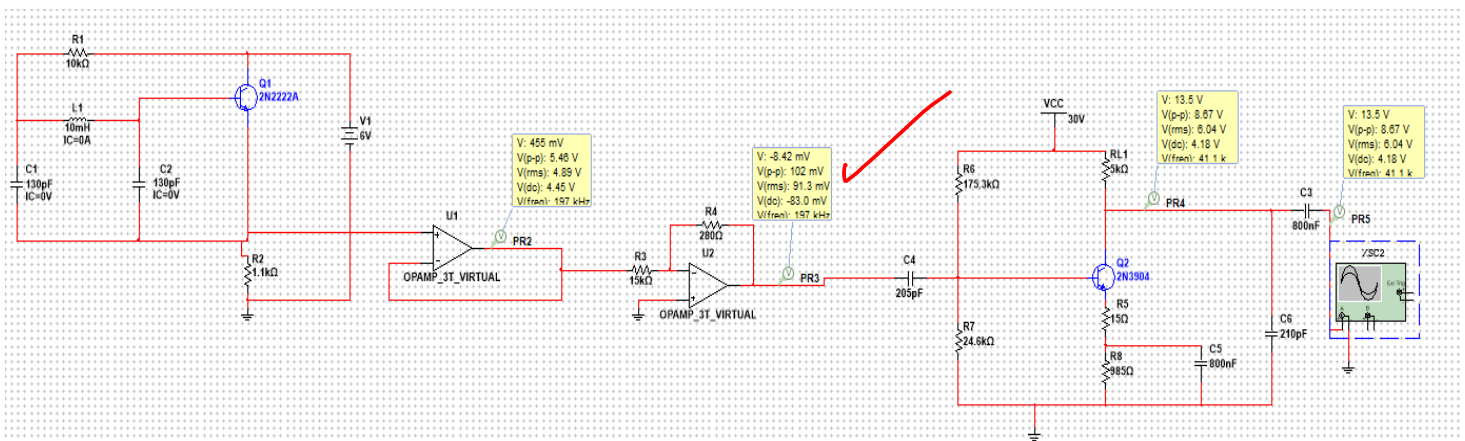


Figure 15-Oscillator output waveform

After achieving the correct specifications, the oscillator was ready to be connected to the amplifier. Firstly the existing signal generator was removed from the circuit and then the output of the oscillator was connected directly to the input of the other this circuit. The final circuit can be seen below:



Oscilloscope-XSC2

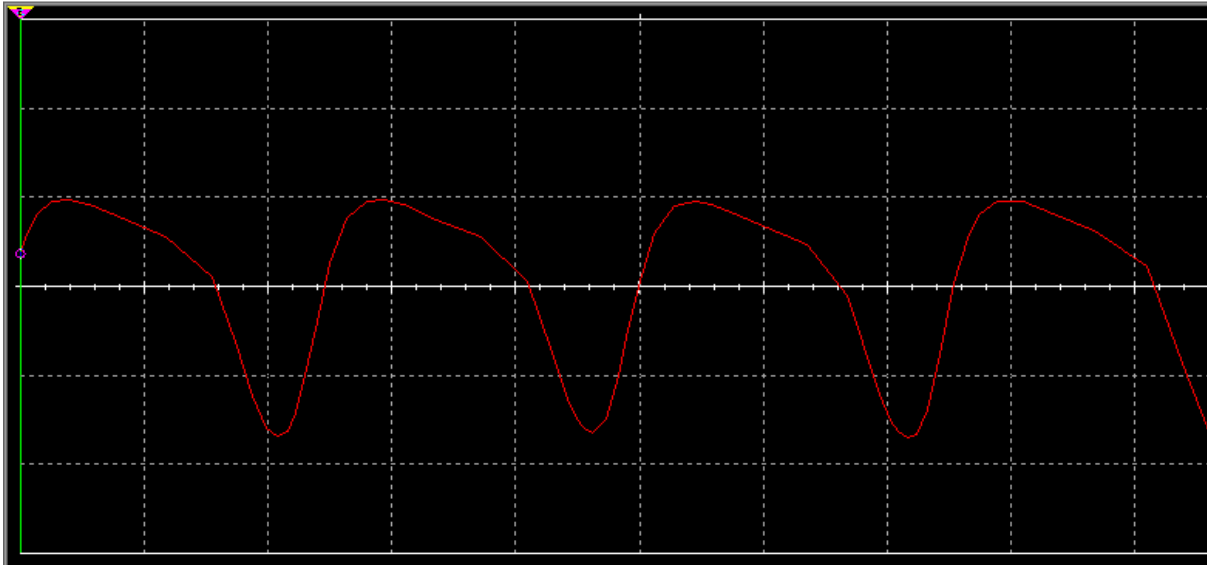


Figure 16-Oscillator output final circuit

The above figure is an image of an output of the final circuit after adding an oscilloscope. It was not possible to do an AC sweep at this stage therefore the only option to analyse it was to add an oscilloscope to the output of the circuit. As can be seen, the output is not a perfect sine wave as would be expected if the circuit was designed perfectly

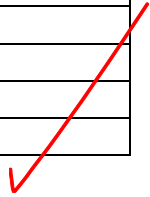
Conclusions and recommendations

The purpose of this task was to design a small signal class A voltage amplifier and design an oscillator with an output of at least 100mVpk and operating at a frequency of 200KHz. A buffer amplifier was required in order to help put together the both circuits. According to (dsculley, 2020), the buffer amplifier helps solve many impedance issues, as it's input impedance is very high and output impedance is very low. The buffer amplifier also helps keep both the circuits separate and therefore cant interfere with each other.

After calculating the DC conditions, negative feedback was introduced to the circuit. Feedback also reduces the overall gain of a system according to (Aspencore, 2020). When dealing with DC conditions only, the gain was dependant on the ratio between the load and the emitter resistance. The voltage drop across the emitter resistor was almost zero due to the low reactance of C3. Before introducing negative feedback, the gain of the circuit was totally dependant on the Hfe of the transistor. According to (Coates, 2021), the principle of negative feedback is that a portion of the output signal is fed back to the input and combined with the input signal in order to reduce gain. As a result of this, it would be possible to make many different versions of this design with a very similar gain.

In conclusion, this task has outlined that it's very difficult to make a perfect amplifier. One reason for this is that there are several counteractive components. For example, when altering the gain it would distort the bandwidth, and altering the bandwidth would change the gain. This made designing an amplifier to specification required a difficult task. Below is a table comparing the specification to the final design:

	Specification	Actual value
Centre Frequency	200KHz	199.5262K
Bandwidth	60KHz	547.3227Khz
Gain	50	50.3
Input signal	50mV	49.9mV



By looking at the table above, we can see that the actual value is quite similar to the specification with a relatively low error. However, the bandwidth is far out of spec with an error of 487KHz. A possible suggestion for the future would be to use a tuned amplifier in order to improve the results in the above table. Furthermore, a class B push pull amplifier could be used to improve the efficiency of this circuit. However, the class A amplifier has the most linear output, therefore making it the most suitable for this task

By looking at the output of the final circuit after adding in the oscillator, we can see that figure 16 doesn't show a perfect sine wave form. A possible reason for this could be that the transistor wasn't biased properly. This would ensure that the transistor operates as a linear amplifier and ensures that any AC signal is amplified correctly by the transistor. Another transistor could also be used in future to improve the design of the amplifier. Another option to improve the design could be to change the values of C2 and C3. Although C1 and C4 are the critical capacitors in this design, altering C2 and C3 using a similar method to how C1 and C4 were altered could have a positive effect on the circuit. However, designing an amplifier to the specs in the time we had was very difficult. Designing an amplifier takes allot of time and includes many equations, simulators and workshop time.

Bibliography

Anon., 2020. *DAEnotes*. [Online]

Available at: <https://www.daenotes.com/electronics/digital-electronics/oscillators>

[Accessed 28 12 2020].

Anon., 2020. *Electrical4U*. [Online]

Available at: <https://www.electrical4u.com>

[Accessed 28 12 2020].

Anon., 2020. *Electronic projects focus*. [Online]

Available at: <https://www.elprocus.com/classification-of-amplifiers-classes-its-applications/>

[Accessed 28 12 2020].

Aspencore, 2020. *Electronics tutorials*. [Online]

Available at: www.electronics-tutorials.ws

[Accessed 27 11 2020].

Coates, E., 2021. *learnabout-electronics*. [Online]

Available at: <https://learnabout-electronics.org/Amplifiers/amplifiers31.php>

[Accessed 01 01 2021].

Dellasalla, G., 2020. *Audioholics*. [Online]

Available at: <https://www.audioholics.com/audio-amplifier/amplifier-classes>

[Accessed 27 11 2020].

dsculley, 2020. *eeecs*. [Online]

Available at: <https://www.eecs.tufts.edu/~dsculley/tutorial/opamps/opamps5.html>

[Accessed 03 01 2021].

EETechmedia, 2020. *EE POWER*. [Online]

Available at: <https://eepower.com/capacitor-guide/applications/coupling-and-decoupling/#>

[Accessed 01 01 2021].

jojo, 2020. *Circuits today*. [Online]

Available at: <https://www.circuitstoday.com/class-b-power-amplifiers>

[Accessed 28 12 2020].

Omegatron, 2020. *Wikipedia*. [Online]

Available at: https://en.wikipedia.org/wiki/Square_wave#/media/File:Waveforms.svg

[Accessed 31 12 2020].

STmicroelectronics, 2003. *Sparkfun*. [Online]

Available at: <https://www.sparkfun.com/datasheets/Components/2N3904.pdf>

[Accessed 01 01 2021].

Symbol	Parameter	Test Conditions	Min.	Typ.	Max.	Unit
I_{CEX}	Collector Cut-off Current ($V_{BE} = -3\text{ V}$)	$V_{CE} = 30\text{ V}$			50	nA
I_{BEX}	Base Cut-off Current ($V_{BE} = -3\text{ V}$)	$V_{CE} = 30\text{ V}$			50	nA
$V_{(BR)CEO}^*$	Collector-Emitter Breakdown Voltage ($I_B = 0$)	$I_C = 1\text{ mA}$	40			V
$V_{(BR)CBO}$	Collector-Base Breakdown Voltage ($I_E = 0$)	$I_C = 10\text{ }\mu\text{A}$	60			V
$V_{(BR)EBO}$	Emitter-Base Breakdown Voltage ($I_C = 0$)	$I_E = 10\text{ }\mu\text{A}$	6			V
$V_{CE(sat)}^*$	Collector-Emitter Saturation Voltage	$I_C = 10\text{ mA}$ $I_B = 1\text{ mA}$ $I_C = 50\text{ mA}$ $I_B = 5\text{ mA}$			0.2 0.2	V V
$V_{BE(sat)}^*$	Base-Emitter Saturation Voltage	$I_C = 10\text{ mA}$ $I_B = 1\text{ mA}$ $I_C = 50\text{ mA}$ $I_B = 5\text{ mA}$	0.65		0.85 0.95	V V
h_{FE}^*	DC Current Gain	$I_C = 0.1\text{ mA}$ $V_{CE} = 1\text{ V}$ $I_C = 1\text{ mA}$ $V_{CE} = 1\text{ V}$ $I_C = 10\text{ mA}$ $V_{CE} = 1\text{ V}$ $I_C = 50\text{ mA}$ $V_{CE} = 1\text{ V}$ $I_C = 100\text{ mA}$ $V_{CE} = 1\text{ V}$	60 80 100 60 30		300	
f_T	Transition Frequency	$I_C = 10\text{ mA}$ $V_{CE} = 20\text{ V}$ $f = 100\text{ MHz}$	250	270		MHz
C_{CBO}	Collector-Base Capacitance	$I_E = 0$ $V_{CB} = 10\text{ V}$ $f = 1\text{ MHz}$		4		pF
C_{EBO}	Emitter-Base Capacitance	$I_C = 0$ $V_{EB} = 0.5\text{ V}$ $f = 1\text{ MHz}$		18		pF
NF	Noise Figure	$V_{CE} = 5\text{ V}$ $I_C = 0.1\text{ mA}$ $f = 10\text{ Hz}$ to 15.7 KHz $R_G = 1\text{ K}\Omega$		5		dB
t_d	Delay Time	$I_C = 10\text{ mA}$ $I_B = 1\text{ mA}$			35	ns
t_r	Rise Time	$V_{CC} = 30\text{ V}$			35	ns
t_s	Storage Time	$I_C = 10\text{ mA}$ $I_{B1} = -I_{B2} = 1\text{ mA}$			200	ns
t_f	Fall Time	$V_{CC} = 30\text{ V}$			50	ns

* Pulsed: Pulse duration = 300 μs , duty cycle $\leq 2\%$

Symbol	Parameter	Value	Unit
V_{CBO}	Collector-Base Voltage ($I_E = 0$)	60	V
V_{CEO}	Collector-Emitter Voltage ($I_B = 0$)	40	V
V_{EBO}	Emitter-Base Voltage ($I_C = 0$)	6	V
I_C	Collector Current	200	mA
P_{tot}	Total Dissipation at $T_C = 25\text{ }^\circ\text{C}$	625	mW
T_{stg}	Storage Temperature	-65 to 150	$^\circ\text{C}$
T_j	Max. Operating Junction Temperature	150	$^\circ\text{C}$

Figure 17- 2N3904 Data sheet

(STmicroelectronics, 2003)

Assignment 1

GRADEMARK REPORT

FINAL GRADE

GENERAL COMMENTS

Instructor

76/100

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

STEFFAN DAVIES

0

RESEARCH-INFORMED LITERATURE



There was good evidence of further reading and research from credible and reliable sources. Your interpretation and understanding of the additional research was good and demonstrated an underlying appreciation of the subject. Your references were appropriate to the task and applied with few errors or inconsistencies. Your reference list was of an appropriate length for this assignment.

7/10

KNOWLEDGE AND UNDERSTANDING



The knowledge and understanding of the subject demonstrated by this assignment was reasonably extensive. There were very few gaps in your knowledge of the basic principles and your understanding of the higher level concepts was demonstrated to a limited degree. You have demonstrated some critical understanding of the main concepts and expanded your knowledge beyond that contained in the course notes, however your explanations need to be more detailed and written with a greater emphasis on critical evaluation rather than description. Further

22/30

work - There were a few omissions and misunderstandings throughout the task, which could be read up on for future reference.

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.

22/30

PRACTICAL APPLICATION AND DEP.



Your application of practical skills for this task was generally excellent. 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 written.

17/20

SKILLS FOR PROFESSIONAL PRAC.



In this assignment you have demonstrated skills in working individually and also collaboratively. Your written report was written in a professional and confident style, however you still need to be more efficient with your word count. The report was very well presented, with excellent diagrams in appropriate places, good sentence/paragraph structure and clear calculations.

8/10