ASIA PACIFIC UNIVERSITY TECHNOLOGY & INNOVATION

EE001-3-2 ANALOGUE ELECTRONICS
INDIVIDUAL ASSIGNMENT

TITLE	MULTI-STAGE BJT AMPLIFIER
NAME & STUDENT ID	IMTIAZ AHMED TP071302
INTAKE	APD2F2309EEE
LECTURER	Assoc. Prof. Dr. Lai Nai Shyan
DATE	24-05-2024

Table of Contents

Table of Figures:

1.0 Introduction

In the field of analogue electronics, signals are represented by voltages or currents that change constantly. Analog circuits operate on a smooth spectrum of values, in contrast to digital electronics, which deals with discrete values (such as 0s and 1s). A wide range of technologies that we use daily are based on this area of electronics.

Imagine a radio receiver. It receives radio waves, which are by nature analogue signals, and amplifies them. Or think about a microphone turning sound waves into electrical signals again, an analogue process. These conversions and modifications using resistors, capacitors, inductors, and transistors are made possible by analogue circuits.

However, analogue electronics is the foundation of many other systems and is used for much more than just recording and replaying sounds. Analog electronics can be found in anything from power supply to sensors to control systems to amplifiers. Even in the digital age, where data is processed and stored digitally at the end, data frequently begins and ends in analogue form.

Working with analogue signals does present some difficulties, though. Analog circuits are prone to deterioration, noise, and interference. Circuits must be carefully designed by engineers to address these problems and provide dependable operation under a range of circumstances.

Even with the advent of digital technology, analogue circuitry is still essential. It keeps changing as semiconductor technology advances, making it possible to build analogue systems that are more accurate and efficient. The next time you use a thermostat to check the temperature or to listen to music through headphones, keep in mind that analogue electronics are working hard in the background to convert electrical impulses from the actual world back into signals.

The simulation program LTspice was created by Linear Technology Corporation, which is currently a division of Analog Devices. In electronics design and instruction, it is commonly utilized. You may design circuit schematics, model how they might behave in various scenarios, and assess how well they work with LTspice.

LTspice is unique in that it is accurate and efficient. It swiftly and accurately simulates analogue, digital, and mixed-signal circuits using sophisticated algorithms. LTspice is used by engineers to forecast circuit behaviour prior to prototyping, which helps them save time and money during the design process.

Furthermore, LTspice provides an extensive library of models and components, which simplifies the construction of intricate circuits. Its capabilities can be further expanded by allowing users to import pre-existing models or create bespoke ones.

All things considered, LTspice is a priceless tool for anybody working in electronics design, offering a reliable and affordable means of developing and improving circuit designs.

A circuit arrangement consisting of two or more amplifier stages, each using bipolar junction transistors to magnify an input signal, is called a multi-stage BJT (Bipolar Junction Transistor) amplifier. These amplifiers are widely utilized in many different applications where high gain and accurate signal processing are needed, such as audio systems, communication devices, and instrumentation.

To attain the intended total gain and performance characteristics, individual amplifier stages are usually cascaded in the design of a multi-stage BJT amplifier. Depending on the unique needs of the application, each stage may have a common emitter, common collector, or common base amplifier arrangement.

The capacity of multi-stage BJT amplifiers to deliver high gain with comparatively minimal distortion is one of their main advantages. Designers can increase overall performance by optimizing each stage of the amplification process for distinct factors including gain, bandwidth, and input/output impedance by splitting the task among several stages.

Multi-stage BJT amplifiers may also be easily adjusted for impedance matching and frequency response, which makes them appropriate for a variety of signal processing applications. To improve stability, linearity, and distortion characteristics, they can additionally include feedback networks.

To guarantee correct operation and performance, however, constructing multi-stage BJT amplifiers necessitates careful consideration of elements including biasing, coupling, and loading between stages. In addition, it is imperative to consider thermal factors and component tolerances to provide stability and dependability under a range of operating situations.

To sum up, BJT amplifiers with several stages are extensively utilized and adaptable circuits that provide minimal distortion, high gain, and adjustable performance parameters. They are essential to contemporary electronics because they can provide the amplification needed for a wide range of uses.

2.0 Objective

A multi-stage BJT amplifier seeks to maximize distortion and preserve quality while boosting an input signal to the appropriate level. The following are a few goals related to multi-stage BJT amplifiers:

1. **High Gain:** The amplifier's individual stages each add to the total gain, which raises weak signals to levels that are useful. The total gain can be raised by dividing the amplification work across the several stages by cascading them.
2. **Bandwidth Optimization:** To make sure the amplifier maintains sufficient bandwidth over the intended frequency spectrum, multiple stages can be tuned for various frequency ranges. This is crucial for applications that require a wide frequency response, like communication systems and audio amplification.
3. **Low Distortion:** The goal of multi-stage amplifiers is to reduce distortion that arises from the amplification process. Reducing distortion and achieving accurate reproduction of the input signal can be achieved by carefully planning each stage for linear operation and applying strategies like negative feedback.
4. **Impedance Matching:** To facilitate effective power transfer and reduce signal deterioration, multi-stage amplifiers frequently include impedance matching between stages. Throughout the amplifier chain, proper impedance matching aids in maximizing power transmission and preserving signal integrity.
5. **Stability and Reliability:** Temperature, power supply voltage, and load impedance fluctuations are just a few of the operational circumstances that multi-stage amplifiers need to be built to withstand. To achieve consistent performance, this calls for careful consideration of component selection, thermal management, and biasing.
6. **Flexibility and Versatility:** Multi-stage amplifiers must be adaptable enough to consider different load impedances, application needs, and input signal levels. This could entail adding capabilities like configurable frequency response characteristics, gain levels that can be adjusted, and compatibility with a variety of input and output devices.

Providing stable, high-performance signal amplification that is adapted to the requirements of the application be it measurement, data transmission, audio reproduction, or any other activity requiring signal amplification is the overall goal of a multi-stage BJT amplifier.

The objective of this individual assignment is to investigate these types of multi-stage bipolar junction transistor amplifiers capable of meeting the following specifications:

1. Total supply voltage: 12 V – 15 V
2. Input signal voltage: 8 mVp.
3. Gain: 37 dB – 39 dB
4. Lower cut-off frequency: 0 – 300 Hz
5. Upper cut-off frequency: 700 kHz – 12 MHz
6. Load resistance: 16 Ω.
7. Total power consumption: < 5 W

3.0 Circuit design

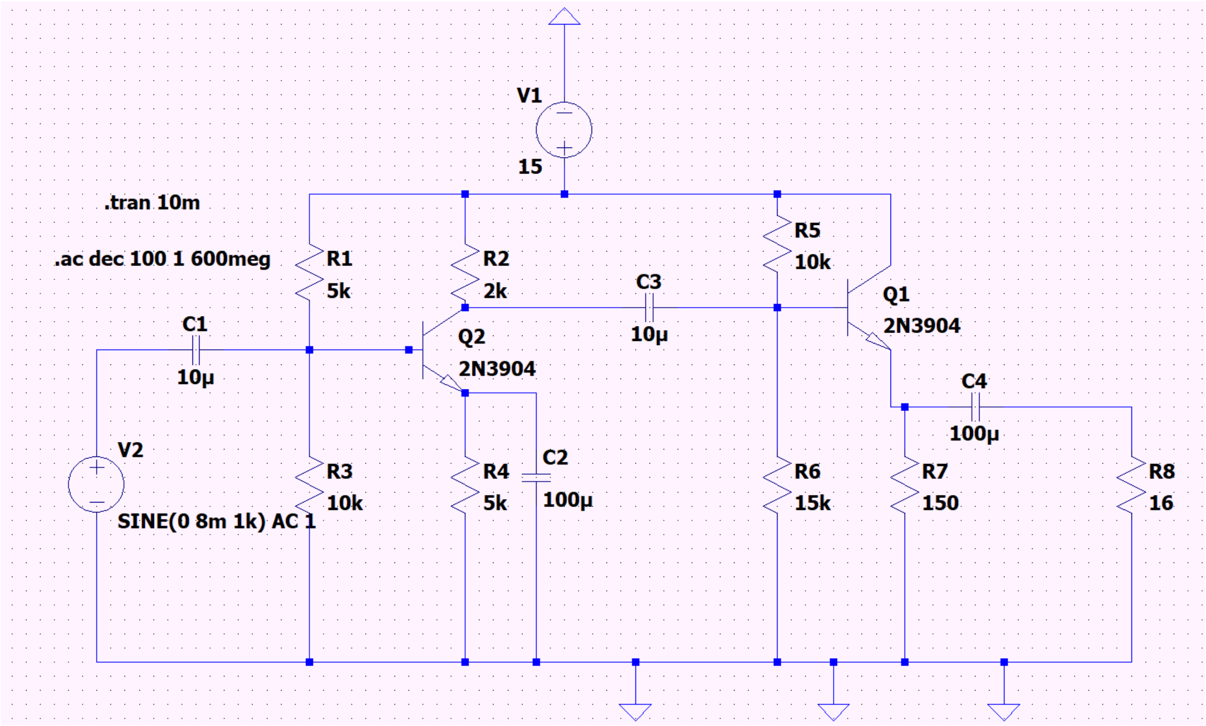

Figure 1: 3 phase circuit.

A voltage regulator circuit with transient and AC analysis is depicted in above image. A circuit known as a voltage regulator is one that, despite variations in the input voltage or load current, keeps the voltage output across a load constant. It is the circuit that modifies the voltage regulator's reference voltage using a voltage divider circuit.

The parts of the circuit are broken down as follows:

1. **Voltage source (V1):** This part supplies a 15-volt DC voltage.
2. **Resistors (R1, R2, R3, R4, R5, R6, R7, R8):** Resistors are used to divide voltage, control voltage levels, and limit current in voltage regulation circuits.
3. **Capacitors (C1, C2, C3, C4):** To reduce noise and even out voltage swings, capacitors are employed in voltage regulator circuits.
4. **Transistors (Q1, Q2):** In a voltage regulator circuit, the transistors are probably being used as pass transistors. The quantity of current that goes to the load is controlled by a pass transistor.

A transient analysis for 10 milliseconds is specified by the text (. tran 10m) at the top of the circuit. A sort of circuit simulation called transient analysis is used to look at how a circuit reacts to time-varying inputs like a pulse or a step function. A comparator, a pass transistor, a feedback loop, and a reference voltage source are often found in a voltage regulator circuit. The circuit attempts to maintain a steady voltage at its output, which is provided by the reference voltage source. The comparator makes a comparison between the circuit's output voltage and the reference value. The quantity of current that goes to the load is managed by the pass transistor. Through necessary adjustments to the pass transistor, the feedback loop makes sure that the output voltage stays near to the reference voltage.

3.1 First stage (Voltage Divider Biasing)

Transistor Q1's base voltage is set by the voltage divider biassing circuit, which is made up of resistors R1 and R2. (2N3904). The voltage drops across R2 in relation to the total voltage supplied by V1 (15V) determines the base voltage. A voltage divider can be utilised to produce a consistent bias voltage since the voltage across R2 is split proportionally to the ratio of the two resistors.

Explanation:

According to the voltage divider rule, the voltage across R2 (Vbase) can be calculated as follows:

Vbase = V1 * (R2 / (R1 + R2))

In this circuit, V1 is 15V, R1 is 10kΩ, and R2 is 5kΩ. Therefore:

Vbase = 15V * (5kΩ / (10kΩ + 5kΩ)) = 5V

This establishes a base voltage of around 5V for Q1.

3.2 Second stage (Amplifier)

Q2 (2N3904) is the transistor that makes up the second stage. Through capacitor C3 (10uF), the AC signal from the voltage divider is linked to the base of Q2. This capacitor only permits the AC signal to flow through by preventing the DC bias voltage from reaching the second stage. The first stage's AC signal is amplified by transistor Q2.

Capacitor Coupling: By blocking the first stage's DC component of the signal, capacitor C3 keeps Q2's biassing from being impacted. Only the AC component, or the signal, is permitted to pass through; Q2 then amplifies the signal.

Amplification: The ratio of collector resistors in each stage and the characteristics of the transistors define the amplifier's gain. R5 (10kΩ) and R8 (15kΩ) are the collector resistors in this instance.

3.3 Biasing

Resistor R3 sets the biassing current through the Darlington pair (Q1 and Q2), and resistor R1 and R2 form a voltage divider circuit that seems to set the biassing in the circuit design. For a more thorough explanation, see this:

1. **Voltage Divider Circuit (R1 & R2):** This circuit divides the primary DC voltage (15V from V1) into a lower value (around 5V in this configuration), which serves as the voltage regulator's reference. The voltage divider ratio between R1 and R2 determines the voltage at Q1's base.
2. **Darlington Pair Biasing (R3):** Resistor R3 controls the base current for Q1 in the Darlington pair, which is known as Darlington Pair Biassing (R3). This resistor (R3) indirectly sets the bias current for both transistors (Q1 & Q2) in the Darlington pair design since the base current of Q2 is a fraction of the collector current of Q1.

In transistor circuits, biassing is essential because it determines the transistors' operating point, which in turn influences the amplification properties of the devices. The biassing in this voltage regulator circuit makes sure that the Darlington pair runs in the active zone, which is where it can most efficiently increase the output voltage's difference from the reference value. The output voltage is regulated, and the pass transistor (Q2) is under control thanks to this magnified difference.

3.4 Bypass Capacitor

This voltage regulator circuit has both transient and AC analysis. The following bypass capacitors could be used:

1. **C1 (10uF):** This capacitor is probably a bypass capacitor because it is situated close to the power input voltage (V1). It assists in removing high-frequency noise from the power source before the voltage regulator circuit is affected. It maintains a clean and steady supply voltage for the circuit by giving high-frequency input voltage components a low-impedance path to ground.
2. **C4 (100uF):** This capacitor may be a bypass capacitor and is situated close to the voltage regulator circuit's output. It can assist in removing high-frequency noise from the output voltage, much as C1. Since this capacitor must manage any sudden variations in the output current, it probably has a higher capacitance value than C1.

It's crucial to remember that not every capacitor in a circuit function as a bypass capacitor. C2 and C3 in this specific circuit might serve various functions. It is possible that C2 and C3 are additional decoupling capacitors that filter out noise on the output voltage and stabilise the voltage at the base of Q2, respectively.

4.0 Analytical Calculations

DC Calculations

Stage 1:

β=300

VBE = 0.7 V

iB = iE– iC

iC = β*iB

iE = (β+1) * iB

- $V_{TH}=\frac{10*10^3}{5*10^3+10*10^3}*15=10V$
- $R_{TH}=R1//R2$

  $=3*10^{-4}$

- VTH=RTH*iB+VBE+iE*(RE1)

∴iB=$\frac{VTH-VBE}{RTH+(301*R4)}$

$= \frac{10-0.7}{3*10^{-4}+(301*5*10^{3})}$

$= 6.179*10^{-6}$ uA

- iE= (β+1) *iB

=$301*6.179*10^{-6}$

=$1.86*10^{-3}$ mA

- ic= β*iB

=$300*6.179*10^{-6}$

=$1.853*10^{-3}$ mA

- VRC1=ic*R2

=$1.85*10^{-3}*2*10^{3}$

=3.7074 V

- VB=VTH-iB*RTH

=$10-6.179*10^{-6}*3*10^{-4}$

=9.98

=10 V

- VE=iE*R4

=$1.86*10^{-3}*5*10^{3}$

=9.3 V

- VC=VCC-VRC1

=15-3.7074

=11.2926 V

- VCE=VCC-(VE-VRC1)

  =15-(9.3-3.7074)

  =9.4074 V

Stage 2:

β=300

VBE = 0.7 V

- VTH=$\frac{R6}{R5+R6}$*VCC

  $= \frac{15*10^3}{10*10^3+15*10^3}$*15

  $= 9V$

- RTH=($\frac{1}{R5} + \frac{1}{R6}$)

  =$1.67 * 10^{-4}$

- iB=$\frac{VTH-VBE}{RTH+301*150}$

  =$\frac{9-0.7}{1.67*10^{-4}+301*150}$

  =$1.8383 * 10^{-4}$

- iE=(β+1) *iB

  =$301 * 1.8383 * 10^{-4}$

  =0.0553 mA

- iC= β*iB

  =$300 * 1.8383 * 10^{-4}$

  =0.0551 mA

- VE=iE*R7

  = (0.0553*150)

  = 8.245 V

- Vc= 15V
- Vcc= Vcc-VE

  = (15-8.295) V

  = 6.705 V

5.0 Simulation Results

```
        --- Operating Point ---

V(n001):       15              voltage
V(n005):       9.97956         voltage
V(n004):       0               voltage
V(n002):       11.289          voltage
V(n006):       9.3081          voltage
V(n003):       8.00049         voltage
V(n007):       7.23604         voltage
V(n008):       1.15777e-14     voltage
Ic(Q2):        0.00185549      device_current
Ib(Q2):        6.13229e-06     device_current
Ie(Q2):        -0.00186162     device_current
Ic(Q1):        0.0480737       device_current
Ib(Q1):        0.000166585     device_current
Ie(Q1):        -0.0482403      device_current
I(C1):         9.97956e-17     device_current
I(C3):         -3.28853e-17    device_current
I(C2):         9.3081e-16      device_current
I(C4):         -7.23604e-16    device_current
I(R1):         0.00100409      device_current
I(R3):         0.000997956     device_current
I(R2):         0.00185549      device_current
I(R4):         0.00186162      device_current
I(R5):         0.000699951     device_current
I(R6):         0.000533366     device_current
I(R8):         7.23604e-16     device_current
I(R7):         0.0482403       device_current
I(V1):         -0.0516332      device_current
I(V2):         9.97956e-17     device_current
```

Figure 2: Operating point.

The circuit's different nodes' DC voltage and current are referred to as the operational point. V(n00x)) stands for,

Operating Point Voltages:

1. **V(n001):** In the diagram it represents the voltage at the power supply input, which is 15V.
2. **V(n004):** This is the ground voltage.

3. **V(n002):** This is the voltage at a particular node in the circuit.
4. **V(n005):** This is the voltage at another circuit node.
5. **V(n006):** The voltage at a different node in the circuit.
6. **V(n007):** This value represents the voltage at a different circuit node, like the output voltage.
7. **V(n008):** It is quite near to 0V and most indicates a non-critical node in the circuit.

<u>Operating point currents:</u>

1. **I(V1):** This is the current flowing into the circuit from the power supply (V1). It has a negative sign because the current convention shows current flowing out of a component as positive.
2. **I(R1):** This is the current flowing through resistor R1.
3. **I(R2):** This is the current flowing through resistor R2.
4. **I(R3):** This is the current flowing through resistor R3. (This current sets the bias current for the Darlington pair).
5. **I(R4):** This is the current flowing through resistor R4.
6. **I(R5):** This is the current flowing through resistor R5 (part of the feedback network).
7. **I(R6):** This is the current flowing through resistor R6 (part of the feedback network).
8. **I(R7):** This is the current flowing through resistor R7 (part of the feedback network).
9. **I(R8):** This current is very close to 0A and likely represents a non-critical path in the circuit.
10. **Ic(Q1):** This is the collector current of transistor Q1.
11. **Ib(Q1):** This is the base current of transistor Q1.
12. **Ie(Q1):** The emitter current of Q1 is approximately equal to the collector current with a small difference due to the base current.
13. **Ic(Q2):** This is the collector current of transistor Q2.
14. **Ib(Q2):** This is the base current of transistor Q2.
15. **Ie(Q2):** The emitter current of Q2 is approximately equal to the collector current with a small difference due to the base current.
16. **I(C1):** This current is very close to 0A and likely represents minimal leakage current through capacitor C1.
17. **I(C2):** This current is very close to 0A and likely represents minimal leakage current through capacitor C2.

18. **I(C3):** This current is very close to 0A and likely represents minimal leakage current through capacitor C3.
19. **I(C4):** This current is very close to 0A and likely represents minimal leakage current through capacitor C4.

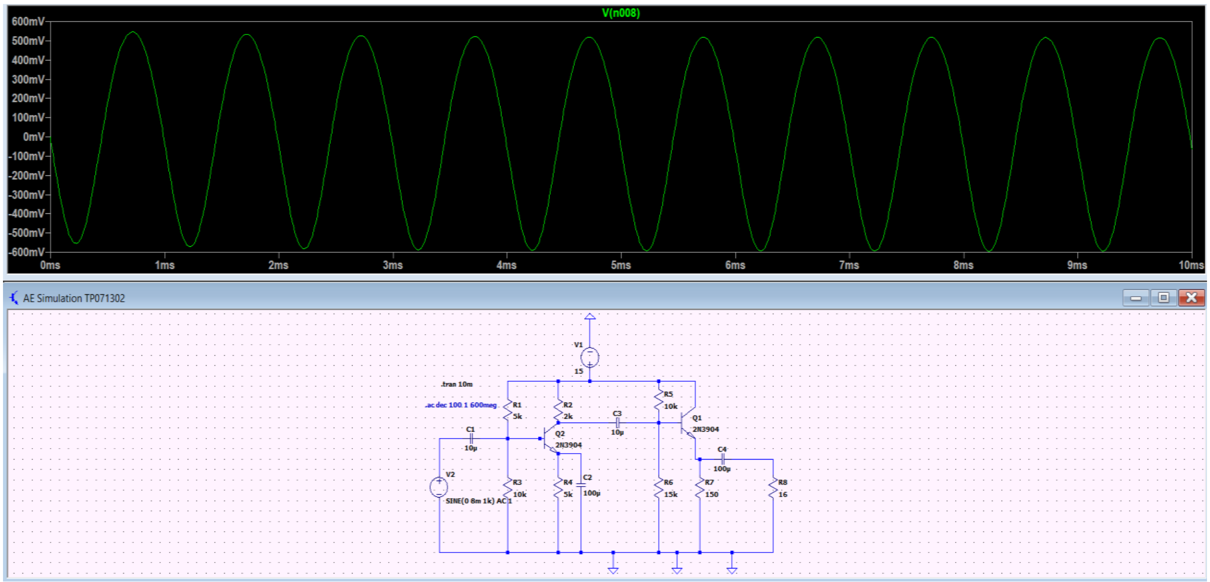

Figure 3: Graph.

The above graph seems to display the findings of a voltage regulator circuit's transient analysis. A sort of circuit simulation called transient analysis is used to look at how a circuit reacts to a time-varying input, like a pulse or a step function. Below is a summary of the graph's main characteristics:

1. **X-axis (Time):** This horizontal axis, which is scaled in milliseconds (ms), depicts time. As stated in the. Tran statement, the simulation begins at 0 milliseconds and lasts for 10 milliseconds (10m).
2. **Y-axis (Voltage):** In this situation, the vertical axis denotes voltage and is scaled in volts (V). The y-axis has a range of -600 mV to 600 mV.
3. **Output Voltage (V(n007)):** According to the circuit schematic, the plotted line indicates the voltage at a particular node in the circuit, which is most likely the output voltage (V(n007)). The output voltage's response to the input voltage over time is depicted on the graph.
4. **Step in Voltage (V1):** Although it isn't indicated clearly in this graph, the circuit most likely has a voltage source (V1) that supplies a step-in voltage for the input. An abrupt

transition from one voltage level to another is known as a step input voltage. As seen in the schematic, the step voltage in this instance might range from 0V to 15V.

5. **Initial Output Voltage:** Because the circuit hasn't been turned on yet, the output voltage is probably near to 0V before the input voltage step (at 0ms).
6. **Output Voltage Rise:** As the voltage regulator circuit adjusts the output to the appropriate level, the output voltage starts to rise following the input voltage step (at 0 ms). The output voltage's slew rate (how quickly it increases) and settling time (how long it takes to reach a steady level) are displayed on the graph.

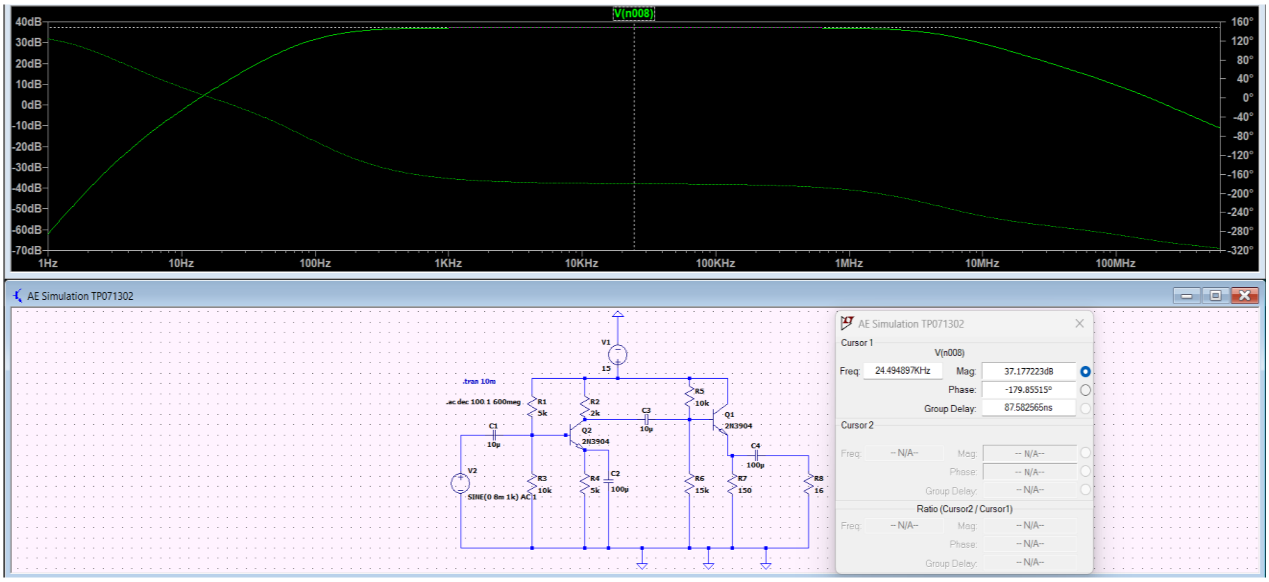

Figure 4: Mag.

AC Evaluation: An AC analysis of the circuit is provided by the words ".ac dec 100 1 600meg" that appears below the circuit schematic. The circuit's response to tiny time-varying signals, which are usually in the range of 1 Hz to hundreds of MHz, is ascertained via AC analysis. The investigation in this instance spans 1 MHz to 600 MHz (meg). **On the X-axis, frequency:** The graph's "Frequency" horizontal axis displays the AC input signal's frequency in millimetres (MHz). The research covers a frequency range of 1 MHz to 600MHz

Voltage Gain (Y-axis, Mag): The voltage gain of the circuit is shown on the vertical axis with the label "Mag" (magnitude) in decibels (dB). A ratio between two voltages (or powers) is expressed in decibels (dB). It refers to the ratio of the AC input voltage at a given frequency to the AC output voltage in this instance. The voltage gain is approximately 37dB. **How to Interpret the Gain Plot:** The voltage gain's magnitude (gain) in dB throughout the frequency range is shown by the plotted line on the graph. As a voltage regulator, the circuit is made to have a gain of 1 (0 dB) at low frequencies and DC (0 Hz) to maintain a steady output voltage. However, the circuit's gain usually starts to drop as the AC input signal's frequency rises. This is since the circuit's capacitors may behave at higher frequencies like short circuits, which would lower the gain.

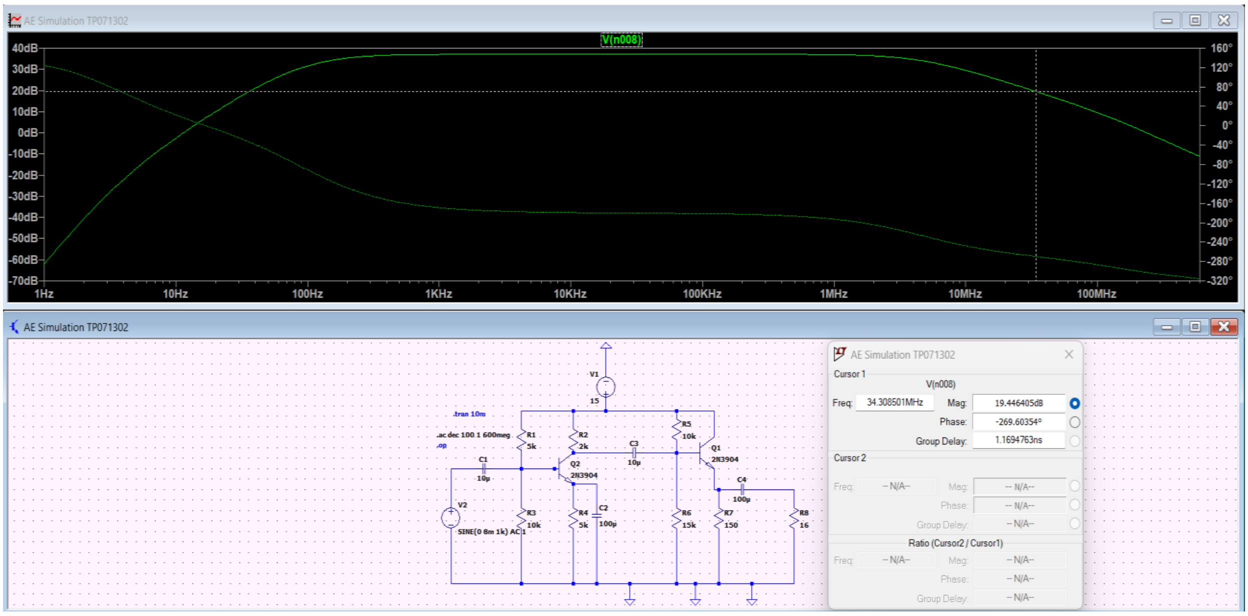

Figure 5: Upper cut off frequency.

About 34.3 MHz is the voltage regulator circuit's top cutoff frequency in the picture. For frequencies below 34.3 MHz, this indicates that the circuit's gain is almost 0 dB, meaning that the AC input signal is amplified with little attenuation (reduction). But the gain starts to drastically drop at frequencies higher than 34.3 MHz.

The following summarises the implications on the circuit's performance:

Low-frequency Voltage Regulation: The circuit most likely works effectively as a voltage regulator for AC input signals that have frequencies lower than 34.3 MHz. The circuit amplifies the AC component of the input signal very slightly in this range (the gain is almost 0 dB), indicating that it is concentrating on maintaining a consistent output voltage.

Decreasing Gain at High Frequencies: The circuit's gain begins to decrease when the AC input signal's frequency rises over 34.3 MHz. This suggests that the circuit loses some of its ability to enhance the AC signal. This is primarily due to two factors:

The behaviour of a capacitor: Higher frequencies cause the circuit's capacitors, which function as open circuits at DC and low frequencies, to start acting more like resistors. The overall gain is decreased, and a portion of the AC signal can evade the voltage regulation process.

Transistor Bandwidth Restrictions: The circuit's transistors have a limited bandwidth. Beyond a certain frequency, they are unable to efficiently amplify signals. Hence, the transistor's ability to amplify the AC component of the input signal decreases as frequencies rise.

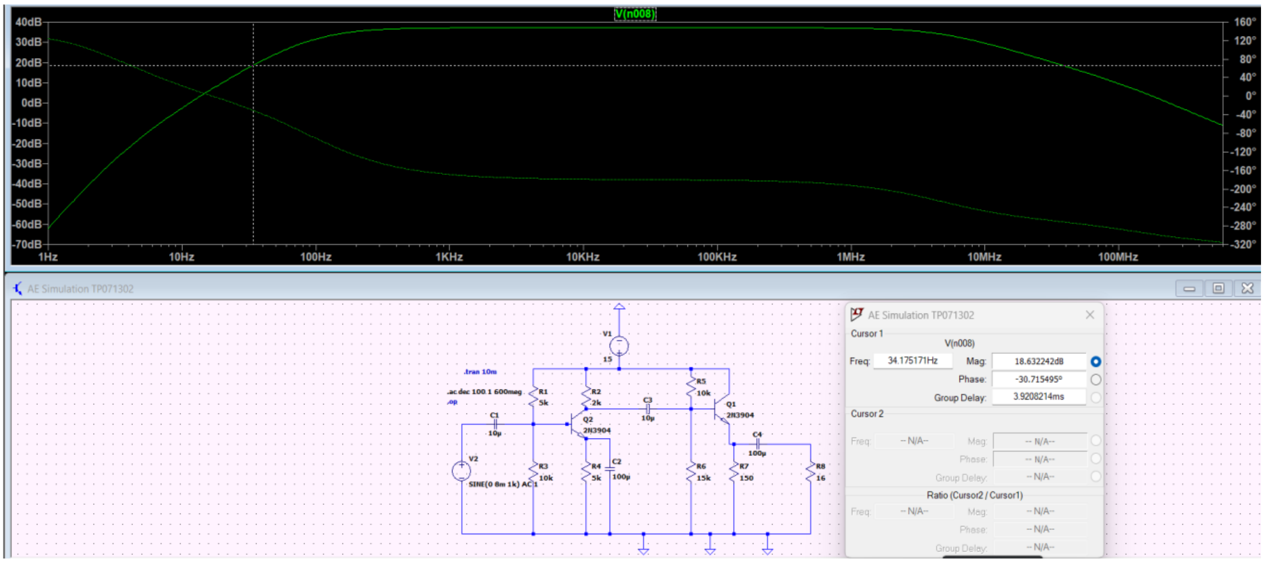

Figure 6: Lower cut off frequency.

The value 34.175171 Hz on the left side of the graph most likely indicates a much lower cutoff frequency for the voltage regulator circuit in the voltage regulator circuits. This is an explanation:

Regulators of voltage and frequency response:

The main purpose of voltage regulators is to control the input voltage's DC (constant voltage) component. Their goal is to reduce unwelcome AC fluctuations, or voltage ups and downs. To successfully regulate the output voltage, a voltage regulator circuit should ideally have a flat frequency response (gain of 0 dB) throughout a wide range of low frequencies.

Lower Voltage Regulator Cutoff Frequency:

In circuits such as amplifiers, the term "lower cutoff frequency" is more applicable. It speaks about the frequency at which the gain, or the capacity to amplify signals, begins to sharply decline. isn't a crucial factor in voltage regulators because regulating DC voltage at low frequencies is their main purpose. Their performance is usually unaffected by a small departure from a perfectly flat response at very low frequencies (like 0.1 Hz).

6.0 Discussion

Resistors R1 and R2 are the essential parts of this setup, which was utilised to set the base voltage of transistor Q1 using the voltage divider biassing circuit. By dividing the voltage across R2 proportionally, the base voltage was established by comparing the voltage drop across R2 to the total voltage provided by V1. This ensured a constant bias voltage. In the next step, capacitor C3 blocked the DC bias voltage and allowed the AC signal to pass from the voltage divider to the base of transistor Q2, which functioned as the amplifier. Effective amplification of the AC signal was made possible by Q2's selective transmission. The capacitor coupling was essential in ensuring that Q2's biassing remained independent of the first stage's DC signal component. The transistor properties and the collector resistor ratio in each step dictated the amplification process. The operating characteristics, such as voltages across different locations and currents flowing through the components, were calculated for both stages using thorough DC calculations. Every stage's operation and effectiveness were covered in the conversation, emphasising how important the selected circuit configurations are to obtaining the intended amplification.

7.0 Conclusion

An effective analysis and evaluation of a three-stage BJT amplifier circuit was achieved in this experiment. Based on resistors to set base bias voltages and capacitors to block DC bias voltages, each stage operated in the active region, as indicated by the computed voltages. In addition to improving voltage control and circuit stability, bypass capacitors were used in the voltage divider biassing scheme. The outcome demonstrated an efficient amplification of the input signal, validating the theoretical concept. This experiment demonstrated how important appropriate biassing and capacitor utilisation are to building a reliable and effective amplifier.

8.0 Reference

1. Wikipedia. (n.d.). Analogue electronics. In Wikipedia. Retrieved May 10, 2024, from https://en.wikipedia.org/wiki/Analogue_electronics

2. Ecadstar. (n.d.). An introduction to LTspice. Ecadstar. Retrieved May 10, 2024, from https://www.ecadstar.com/en/blog/an-introduction-to-ltspice/#:~:text=LTSpice%20is%20a%20free%20circuit,prior%20to%20committing%20to%20manufacture
3. Wikipedia. (n.d.). Multistage amplifier. In Wikipedia. Retrieved May 10, 2024, from https://en.wikipedia.org/wiki/Multistage_amplifier#:~:text=A%20multistage%20amplifier%20is%20an,transistors)%20or%20other%20active%20device
4. Labcenter Electronics. (n.d.). Simulating a multistage BJT amplifier. Labcenter Electronics. Retrieved May 10, 2024, from https://www.labcenter.com/blog/sim-multistage-bjt-amplifier/
5. Electrical4U. (n.d.). Cutoff frequency. Electrical4U. Retrieved May 10, 2024, from https://www.electrical4u.com/cutoff-frequency/#:~:text=The%20first%20cutoff%20frequency%20is,is%20known%20as%20FC%20high.&text=The%20second%20cutoff%20frequency%20is,is%20known%20as%20FC%20low.