Electronic Workshop - II
Audio Amplifier

Bhuvi
Electronics and Communication
International Institute of Information Technology
Hyderabad, India
bhuvi.bhuvi@research.iiit.ac.in,
2023112015

Harsh Kapoor
Electronics and Communication
International Institute of Information Technology
Hyderabad, India
harsh.kapoor@research.iiit.ac.in,
2023112004

***Abstract*—This paper presents the design details of an Audio Amplifier, created as part of the Electronics Workshop-II course at IIIT-H. The objective of the project was to design and construct an audio amplifier capable of enhancing a given audio signal and delivering the amplified sound through a speaker, given certain constraints. The project involved circuit design, component selection, and testing to ensure effective signal amplification.**

I. Introduction

Our objective was to make an audio amplifier, which takes an input from a mic, and has a 4 stage implementation, with the stages being -

1) Pre-Amplifier
2) Gain Amplifier
3) Active Band-Pass Filter
4) Power Amplifier

The circuit should have a unified power supply at $\pm$12V, but pre-amplifier, gain amplifier and power amplifier should get $\pm$5V, and the op-amp in the filter should get $\pm$12V. The input voltage would be in the range of $5mV_{pp}$ to $10mV_{pp}$. The voltage gain should be greater than 400 times, and the circuit has to drive a 8Ω load of a speaker. Frequency range of the circuit has to 20Hz to 20kHz, with the constraints that the filter must not attenuate the signal and the power amplifier should not provide voltage gain.
With these constraints added, we made a 7 stage circuit, with the stages being -

1) Mic Circuit,
2) Pre-Amplifier,
3) Gain Amplifier,
4) Voltage Regulator,
5) Buffer Circuit,
6) Active Band-Pass Filter,
7) Power Amplifier,
8) Speaker

The stages, with the relevant details, calculations and references are given in the following sections.

II. Mic Circuit

The circuit was more or less given to us in the labs itself. It was also found by us on the web, the link for which is attached in the reference section below. The mic provided to us was a 2 pin mic as shown in the figure -

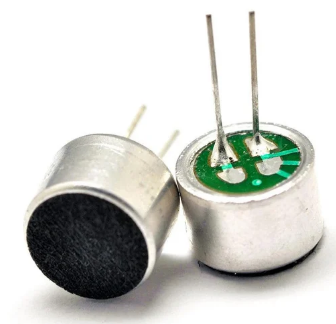

Fig. 1. Image of the mic provided

And the circuit provided to us in labs is as follows with R = 10kΩ and C = 2.2μF -

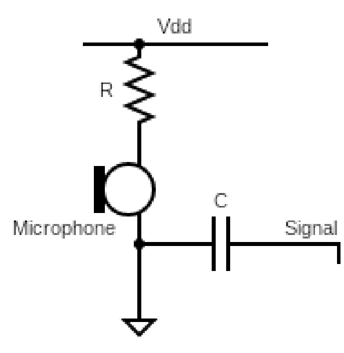

Fig. 2. Mic Circuit

This is more or less what we made, the only changes would be in the values due to the natural errors in the Resistor and Capacitor values. The output signal of this serves as our input signal for the rest of the circuit.

III. Pre-Amplifier Circuit

The first stage of our circuit is the pre amplifier stage. The preamplifier stage is a crucial component in an audio amplifier, responsible for the initial amplification of weak input signals before they proceed to the main gain stage. This stage ensures that the signal strength is sufficient for further processing while maintaining minimal noise interference.

In this project, a common-emitter differential amplifier configuration is used as the preamplifier due to its high input impedance and low output impedance, which makes it ideal for handling signals from low-power sources such as microphones. A low input resistance would result in excessive current draw, which the microphone cannot supply, leading to inefficient operation. The differential amplifier also provides good noise performance, preventing weak signals (ranging from 10mV to 40mV) from being overpowered by unwanted noise. An image for the reference circuit provided to us is as follows -

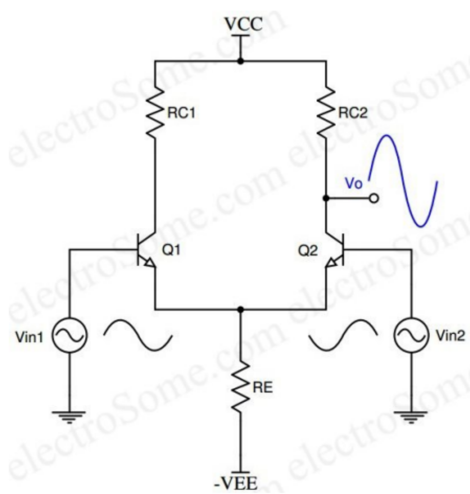

Fig. 3. Reference for the Pre-Amplifier Stage

This Differential Amplifier Topology helps us reduce noise, even if no signal is given to the inverting end of the input. This is due to the fact that the topology amplifies the difference of the inputs, so if a noise is prominent enough we can have a long twisted cable attache to the inverting end of the circuit and it will also have noise now, which will be canceled out when taking a difference. For more info please refer to the Bibliography where we have attached a link for the same. This is specially important as we were initially planning on making a signal inverter using a buffer, but it was later rejected by the TAs. Hence, our approach was using one end at input and the other end connecting to a long wire so that effectively noise can be reduced.

A. Calculations

We are, as mentioned earlier using a Differential Amplifier for which the small signal model will be as follows -

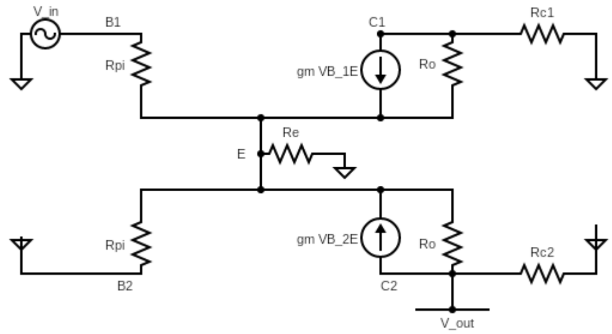

Fig. 4. Pre-Amplifier Small Signal model

From the data-sheets (attached in the References), the values of V_T, V_E and I_C are $10mV$, $-0.7V$ and $25mA$ respectively. The following Equations can be used to find the values of gain and the resistances -

$$I_E = 2I_C = \frac{V_E - V_{SS}}{R_E} => R_E = \frac{V_E - V_{SS}}{2I_C} = \frac{-0.7-(-5)}{2 \times 10 \times 10^{-3}}$$

$$R_E = 2150\Omega$$

$$g_m = \frac{I_C}{V_T} = \frac{10 \times 10^{-3}}{25 \times 10^{-3}} = 0.04\Omega^{-1}$$

$$CMRR = \frac{A_\phi}{A_C} = \frac{g_m R_C}{\frac{R_C}{R_E}} = g_m R_E = 0.04 \times 2150 = 86$$

From the small signal model (assuming R_o to be large)-

$$\frac{V_{in} - V_E}{R_\pi} + g_m(V_{in} - V_E) + \frac{0 - V_E}{R_\pi} + g_m(0 - V_E) = \frac{V_E}{R_E}$$

$$V_E = \frac{V_{in}(1 + g_m R_\pi)}{2 + 2g_m R_\pi + \frac{R_\pi}{R_E}} \tag{1}$$

$$0 - V_{out} = g_m(0 - V_E)R_C \tag{2}$$

using equations 1 and 2, we get -

$$V_{out} = \frac{V_{in}(1 + g_m R_\pi)}{2 + 2g_m R_\pi + \frac{R_\pi}{R_E}}(g_m R_C)$$

which simplifies to -

$$gain = \frac{V_{out}}{V_{in}} = g_m R_C$$

taking gain to be at 10, $10 = 0.04R_C$, $R_C = 250\Omega$

B. LT Spice Simulation

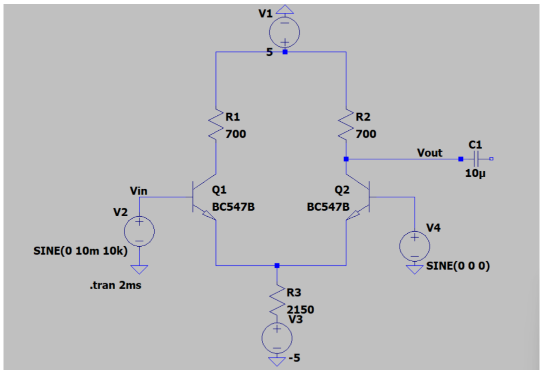

Fig. 5. Spice Circuit of Pre Amplifier

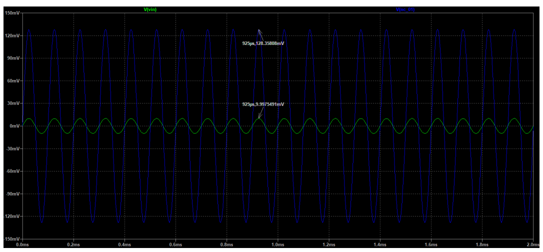

Fig. 6. Waveform in Simulation

When the calculated values of R_{C1}, R_{C2} and R_E were substituted to the circuit with V_{in2} connected to ground (as we cannot model noise in spice simulations), the following graph was obtained which approximately matches our gain value of 10, which we calculated previously. There was a DC offset observed at the output, which was corrected by placing a Capacitor temporarily for taking a reading. This DC offset along with others will be compensated for at the filter part.

C. Physical Implementation

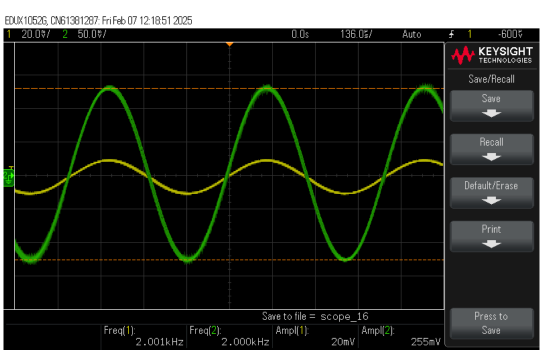

Fig. 7. Oscilloscope Waveforms

The image attached is of the oscilloscope when the input was connected to a Wave-gen and the Yellow probe, while the output was connected to the Green Probe with the circuit being supplied $\pm 5V$ at VCC and VSS. It can be noted from the measurements taken at the bottom that the gain is again approximately 10 times the input value.

The following table summarizes the different values at different stages of our Circuit -

Value	Hand Calculations	LT Spice Simulation	Physical Implementation
R_C	250Ω	700Ω	650Ω
R_E	2150Ω	2150Ω	2200Ω
gain	10	12.8	12.2

IV. Gain-Amplifier Circuit

In audio signal processing, amplification is a crucial step to ensure that weak signals are strengthened before driving a speaker or further processing. In this project, we are using a Common Emitter (CE) amplifier as the second stage in an audio amplifier, following the preamplifier stage. This stage is responsible for providing higher voltage and current gain, ensuring the audio signal is strong enough for further amplification or output.The given reference circuit was as follows -

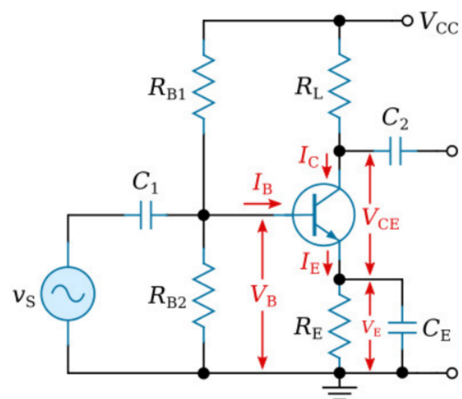

Fig. 8. Reference for Gain Amplifier Stage

The CE amplifier stage offers a combination of high gain, moderate input impedance, and low output impedance, making it well-suited for audio applications. The input capacitor (C1) blocks DC components while allowing AC signals to pass, and the output capacitor (C2) ensures proper AC coupling to the next stage. The bias resistors (RB1, RB2) establish a stable operating point for the transistor, while the emitter resistor (RE) helps with stability and linearity.

Operating within the 20 Hz – 20 kHz frequency range, the amplifier is designed to preserve the full spectrum of audible frequencies while preventing unwanted distortion or attenuation. Careful selection of capacitor and resistor values is essential to maintain signal integrity and prevent unwanted frequency roll-off.

A. Calculations

Here we are using a BJT in the Common Emitter mode and for this the small signal model is as follows -

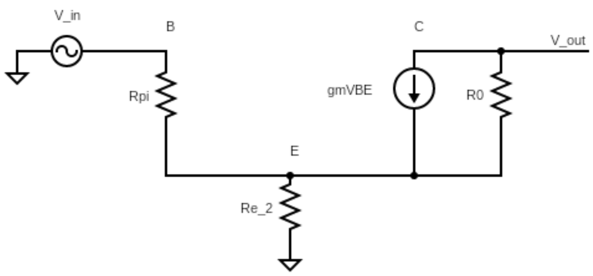

Fig. 9. Gain Amplifier Small Signal Model

The Parameters used from the Data-sheet are as follows -

- $I_C = 1.4mA$
- $\beta = 294$
- $I_E = 1mA$
- $i_c = 1.2mA$

Max swing in voltage here is +5V to -5V at collector, it has to biased at 1V

$$R_L = \frac{5-0}{1.4 \times 10^{-3}} = 3.75k\Omega$$

we know that, $V_B > V_C$, and since we wont get an exact +5V to -5V swing, $V_B = -4.1V$,

$$V_{RB2} = \frac{10R_{B2}}{R_{B1} + R_{B2}} = 0.9$$

$$\frac{R_{B1}}{R_{B2}} = \frac{91}{9}$$

we also know that,

$$I_E = \frac{\beta}{\beta+1} I_C$$

$$\frac{10}{10(R_{B1} + R_{B2})} > \frac{I_C}{\beta}$$

$$R_{B1} + R_{B2} < \frac{\beta}{I_C} = 210000$$

using these results we take $R_{B1} = 182k\Omega$ and $R_{B2} = 18k\Omega$. Now using this we can say that $V_E = -4.1 - 0.7 = -4.8V$.

$$R_E = \frac{-4.8 - (-5)}{I_E} = 157.14\Omega$$

To get a voltage swing of +4V to -4V -

$$I_C - i_c = \frac{5-4}{3.75 \times 10^3} = 0.28 \times 10^{-3} A$$

$$I_C + i_C = 2.6mA = I_c \exp\left(\frac{V_{BE}}{V_T}\right)$$

solving this will give us $V_{BE} = 15.075mV$

$$100mV - 15.075mV = 85.924mV$$

$$R_{E2} = \frac{85.924}{1.4} = 96.48\Omega, R_{E1} = 60.66\Omega$$

Reactance of the Capacitors should be less than 1/10th of Resistors. Therefore C at input is chosen to be $5\mu F$ and the one at emitter to be $1mF$.

B. LT Spice Simulations

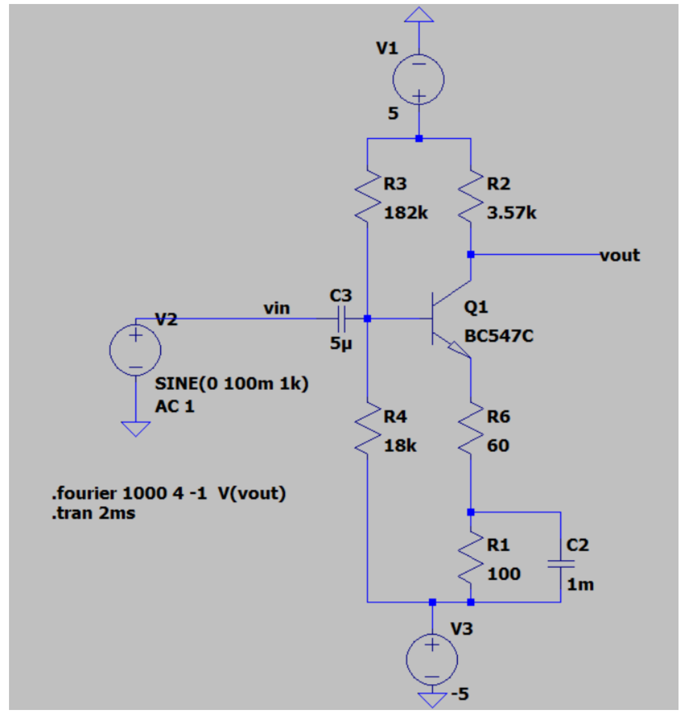

Fig. 10. Spice Circuit of Gain Amplifier

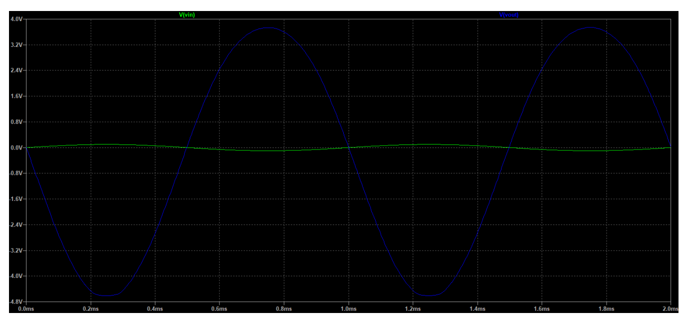

Fig. 11. Waveform in Simulation

When the calculated values of Resistances and Capacitances were substituted to the circuit, the following graph was obtained which approximately matches our gain value of 40, which we calculated previously. There was a DC offset observed at the output, which was corrected by placing a Capacitor temporarily for taking a reading. This DC offset along with others will be compensated for at the filter part.

C. Physical Implementation

The image attached is of the oscilloscope when the input was connected to a Wave-gen and the Yellow probe, while the output was connected to the Green Probe with the circuit being supplied $\pm 5V$ at VCC and VSS. It can be noted from the measurements taken at the bottom that the gain is again approximately 50 times the input value. Please note the phase inversion in this stage of the circuit.

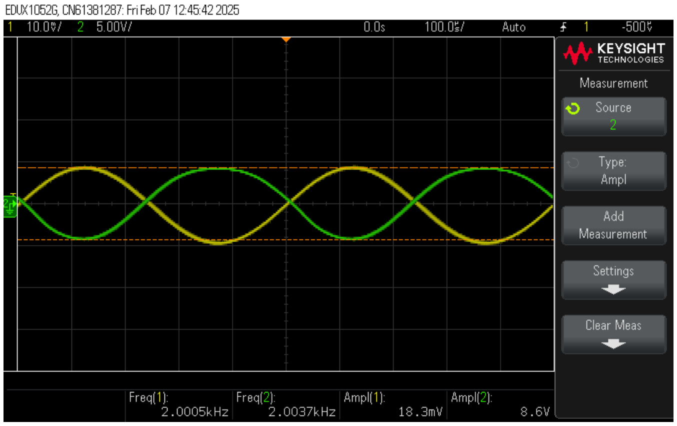

Fig. 12. Oscilloscope Waveforms

The following table summarizes the different values at different stages of our Circuit -

Value	Hand Calculations	LT Spice Simulation	Physical Implementation
R_{B1}	$182k\Omega$	$182k\Omega$	$180k\Omega$
R_{B2}	$18k\Omega$	$18k\Omega$	$20k\Omega$
R_C	$3.75k\Omega$	$3.75k\Omega$	$4k\Omega$
R_{E1}	60.66Ω	60Ω	60Ω
R_{E2}	86.48Ω	100Ω	100Ω
C_E	$5\mu F$	$5\mu F$	$10\mu F$
C_B	$1mF$	$1mF$	$10\mu F$
gain	50	44	46.9

V. Voltage Regulator Circuit

One of the many last minute changes made in this project. The Voltage regulators are used as we were not allowed to use two different Power Supplies for the $\pm5V$ and $\pm12V$ which were used by the transistors and the op-amps respectively. To overcome this problem the op-amps were given a direct $\pm12V$ supply with a branch of it being fed to the regulator ICs, which then supplied $\pm5V$ for the rest of the circuit. The ICs were LM-7805 and LM-7905, the datasheet of which are again in the reference section. The circuits were also commonly available on the internet with the following topology being used by us -

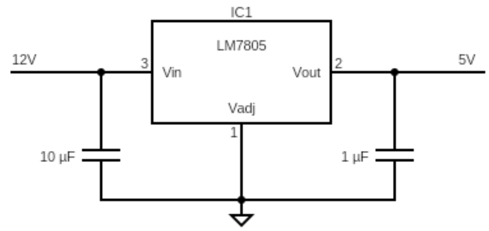

Fig. 13. Circuit to get +5V from +12V

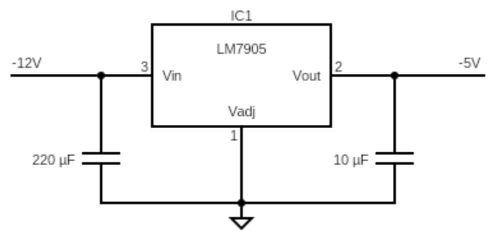

Fig. 14. Circuit to get -5V from -12V

VI. Buffer Circuit

The buffer stage in this audio amplifier serves as an intermediary between amplifier stages and the filter stage, ensuring impedance matching and signal integrity. It provides high input impedance and low output impedance, minimizing signal loss and distortion while preventing excessive loading on the preceding stage. Buffers help improve the overall efficiency of the amplifier by maintaining signal strength and preventing unwanted interactions between stages. The configuration used by us is a Unity Gain Amplifier on Op-Amp. The following image shows the topology used by us, using the chip UA-741, the data-sheet for which is given in the reference section of the report.

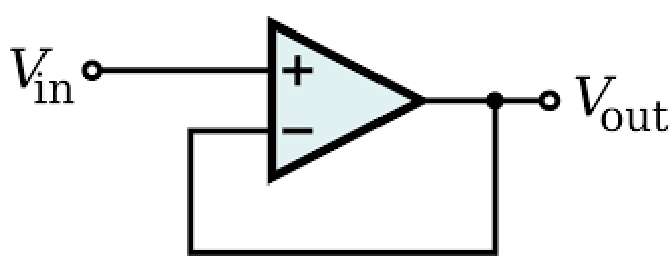

Fig. 15. Unity Gain Buffer using Op-Amp

Since we are giving the input to the positive end of the Op-Amp the output is non inverted.

VII. Active Band Pass Filter Circuit

The active band-pass filter in an audio amplifier selectively attenuates signals outside a specific frequency range while keeping frequencies inside this range same. Unlike passive filters, active band-pass filters use operational amplifiers (op-amps) to achieve greater gain and sharper frequency response. This stage is crucial for enhancing desired audio frequencies while minimizing unwanted noise or interference. By precisely tuning the cutoff frequencies, the filter helps shape the amplifier's overall sound quality and performance. The range for non-attenuation is set as 20Hz and 20kHz as it is the audible range, but for safety so that even the roll offs don't affect the signal, the cutoff frequencies are chosen outside this region.

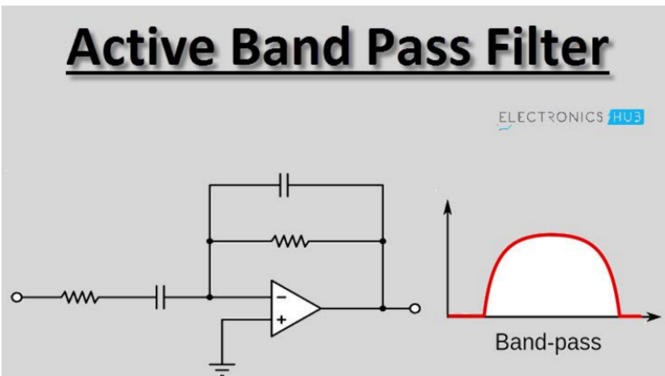

Fig. 16. Refernce circuit provided

A. *Calculations*

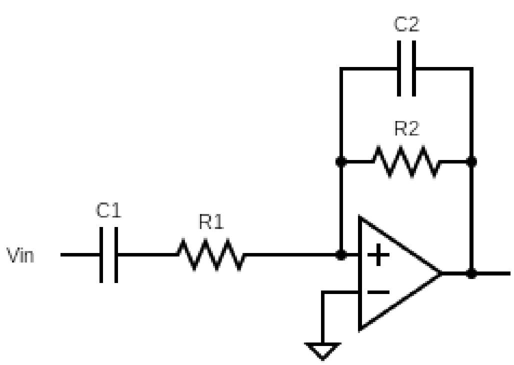

Fig. 17. Circuit of the Active bandpass filtr

Assuming the condition of virtual short of the op-amp input terminals, we get a ground at the positive terminal as well.

$$\frac{V_{in}-0}{Z_1}=\frac{0-V_{out}}{Z_2}$$

where Z_1 is R_1 and C_1 in series and Z_2 is R_2 and C_2 in parallel.

$$Z_1 = R_1 + \frac{1}{sC_1}$$

$$Z_2 = \frac{R_2\frac{1}{sC_2}}{R_2+\frac{1}{sC_2}}$$

from these equations, we get -

$$\frac{V_{Out}}{V_{in}}=\frac{-Z_2}{Z_1}=-\frac{\frac{R_2}{1-sC_2R_2}}{\frac{1+sC_1R_1}{sC_1}}$$

$$\frac{V_{out}}{V_{in}}=\frac{-sC_1R_2}{(C_1R_1C_2R_2)s^2+(C_1R_1+C_2R_2)s+1}$$

solving for the poles of the equation we get -

$$s=j\omega=\frac{-(C_1R_1+C_2R_2)\pm\sqrt{(C_1R_1+C_2R_2)^2-4(C_1C_2R_1R_2)}}{2C_1C_2R_1R_2}$$

$$\omega=\frac{1}{C_1R_1}or\frac{1}{C_2R_2}$$

therefore -

$$f_l = 20Hz = \frac{1}{2\pi R_1C_1}$$

$$f_h = 20kHz = \frac{1}{2\pi R_2C_2}$$

and for the magnitude of $\frac{V_{out}}{V_{in}}=-\frac{R_2}{R_1}$, therefore for no gain we get $R_2 = R_1 = 100k\Omega$, and we get $C_1 = 0.1\mu F$ and $C_2 = 0.04nF$.

B. *LT Spice Simulations*

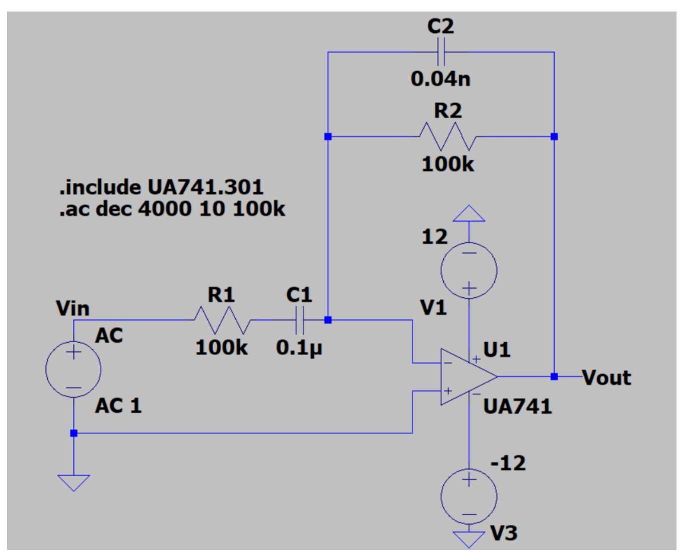

Fig. 18. Spice simulation of the circuit

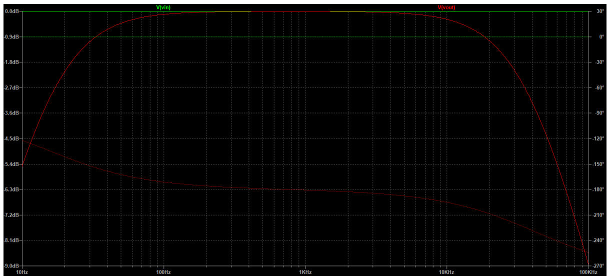

Fig. 19. Spice Simulation Frequency Response

The above images show the results for the circuit when they were simulated in LT SPICE, please no the -3dB cutoffs were well beyond 20Hz and 20kHz so that no attenuation takes place.

C. *Physical Implementation*

The following image is of the frequency response of the filter, the red one being the phase response and the blue one being the amplitude response. Please note that the 4dB gain is due to the fact that the signal was passed through the pre-amplifier in the implementation while testing.

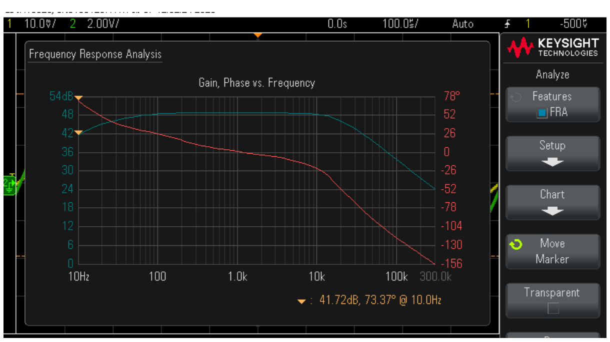

Fig. 20. Oscilloscope Frequency Response

The following table summarizes the different values used in the hand calculations, simulations and the physical implementation.

Value	Hand Calculations	LT Spice Simulation	Physical Implementation
R_1	$100k\Omega$	$100k\Omega$	$100k\Omega$
R_2	$100k\Omega$	$100k\Omega$	$100k\Omega$
C_1	$0.1\mu F$	$0.1\mu F$	$0.5\mu F$
C_2	$0.04nF$	$0.04nF$	$0.1nF$

VIII. Power Amplifier

In an audio amplifier system, the power amplifier plays a crucial role in boosting the low-power audio signal to a level sufficient to drive a loudspeaker. It is the final stage of amplification, ensuring that the amplified signal retains its fidelity while delivering the necessary power output.

For our audio amplifier, we implemented a Class AB power amplifier, which combines the efficiency of Class B with the low distortion of Class A. This design minimizes crossover distortion while maintaining a good balance between linearity and efficiency. Unlike Class A, which operates with continuous conduction and high power dissipation, and Class B, which suffers from non-linearity at the crossover point, Class AB operates with both transistors slightly conducting even at zero input. This small bias current ensures smooth transitions between the positive and negative halves of the waveform, reducing distortion and improving sound quality.

Our Class AB power amplifier stage effectively drives the speaker while preserving the clarity and accuracy of the audio signal, making it a suitable choice for high-fidelity sound reproduction in our system.

The following is an image provided to us as a reference for a class B power amplifier.

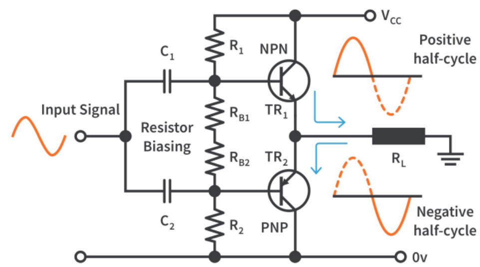

Fig. 21. Reference Circuit Provided

A. *Calculations*

Using the small signal model -

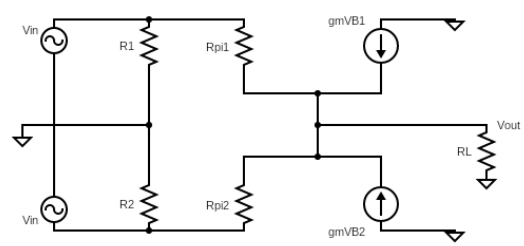

Fig. 22. Power Amplifier Small Signal Model

$$V_{out} = I_C R_L$$

$$V_{in} - V_{BE} - V_{out} = 0$$

$$V_{out} = V_{in} - \frac{I_C}{g_m}$$

$$\frac{1}{g_m} = r_e$$

$$V_{in} = I_C(R_L + r_e)$$

$$\frac{V_{out}}{V_{in}} = \frac{R_L}{R_L + r_e} \approx 1$$

considering r_e to be small in comparison to R_L, we get gain = 1

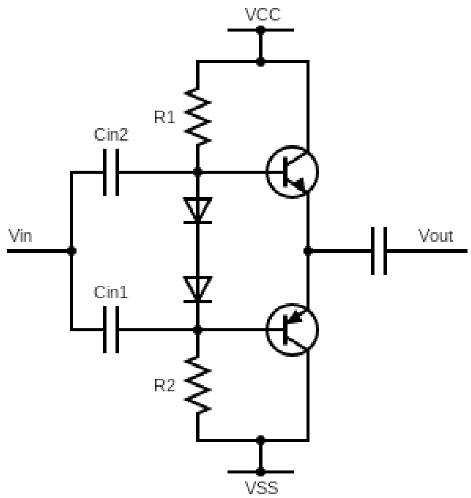

Fig. 23. Class AB Power Amplifier

B. LT Spice Simulations

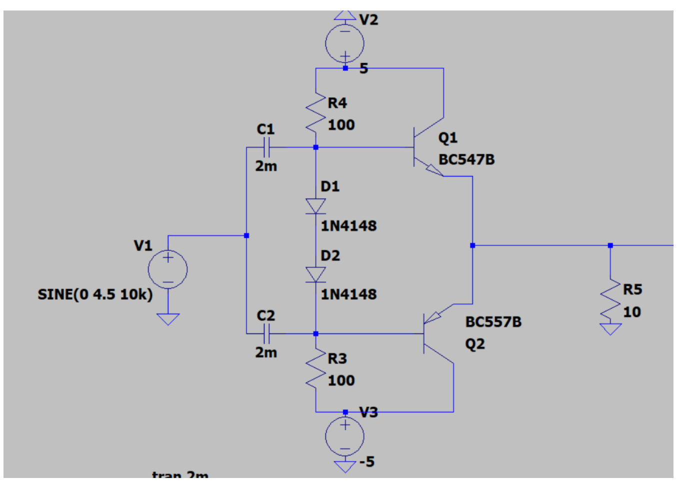

Fig. 24. Spice Circuit

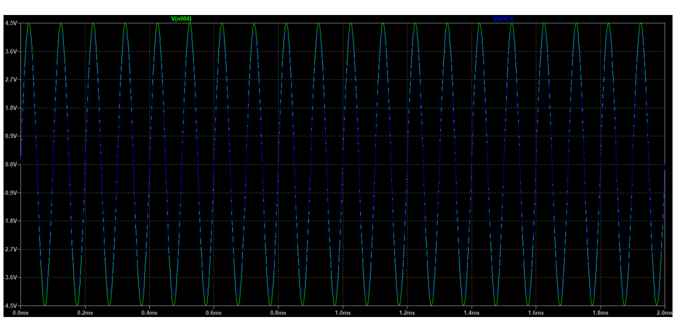

Fig. 25. Simulation results- Output Waveform

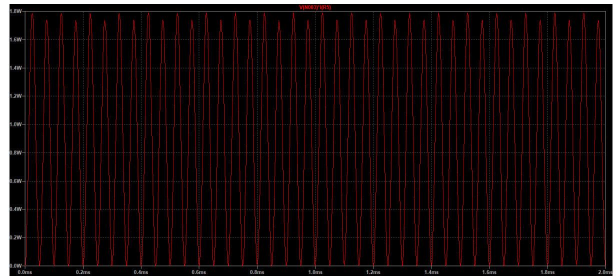

Fig. 26. Simulation results- Output Power

The above images show the circuit when simulated in LT Spice, please note that there is no attenuation in the signal as a whole and the power peaks at 1.8 Watts when driving a 10Ω load.

C. Physical Implementation

The following image is of the oscilloscope taken from the completed circuit. The gain and the values of voltages are the input and output voltages of the completed circuit after the power amplifier.

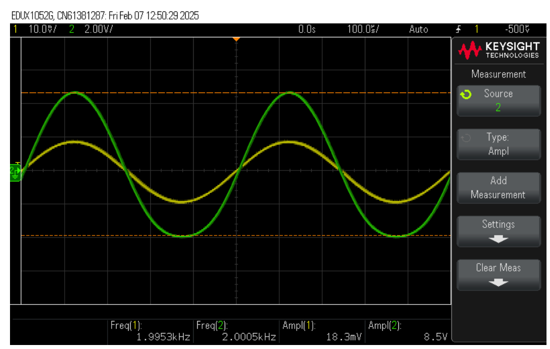

Fig. 27. Oscilloscope Waveforms

Please note that a different value was used for R_{up} and R_{down} as in practice NPN and PNP transistors have a slightly different threshold voltage. Also note that the capacitance C_out is used only to block the DC offset of the output signal and had to be set so that it doesn't affect the signal as a low-pass filter with the resistances. The same goes with C_in.

IX. Speaker

The following speaker was provided to us which had a resistance of 8Ω and a power rating of 5 Watt. It is modelled as a resistance in the entire simulation.

Fig. 28. Speaker Used

X. Complete Circuit

The following images are of the completed circuit and the outputs of it.

A. *LT SPICE Simulations*

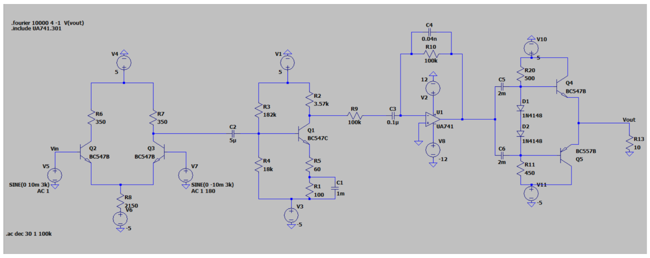

Fig. 29. Complete Circuit in Spice Simulation

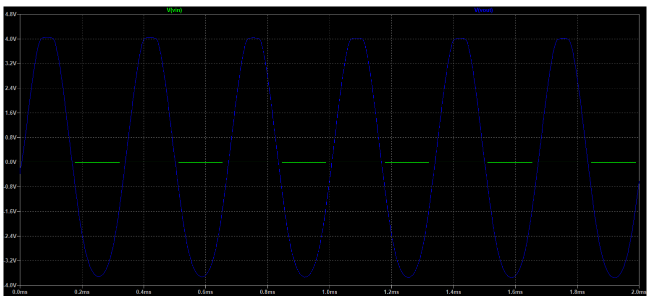

Fig. 30. Output Waveforms in Simulation

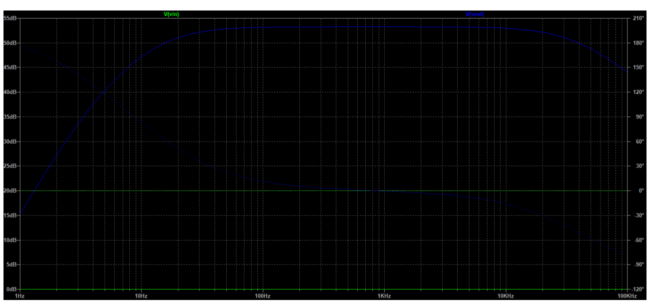

Fig. 31. Frequency Response in Spice Simulation

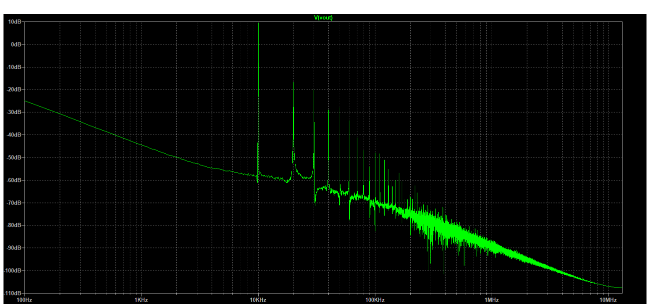

Fig. 32. FFT Simulation

```
Direct Newton iteration for .op point succeeded.
Changing Tseed to 9.76562e-10
N-Period=100.00
Fourier components of V(vout)
DC component:-0.120673

Harmonic    Frequency    Fourier     Normalized    Phase      Normalized
 Number       [Hz]       Component   Component    [degree]    Phase [deg]
    1       1.000e+4     4.155e+0    1.000e+0     107.10°        0.00°
    2       2.000e+4     2.053e-1    4.941e-2    -141.95°     -249.05°
    3       3.000e+4     1.388e-1    3.340e-2     134.77°       27.67°
    4       4.000e+4     5.025e-2    1.210e-2    -116.44°     -223.54°
Partial Harmonic Distortion: 6.085373%
Total Harmonic Distortion:   6.872589%


Date: Thu Mar 20 21:42:20 2025
Total elapsed time: 0.931 seconds.
```

Fig. 33. Total Harmonic Distortion Calculation

B. *Physical Implementation*

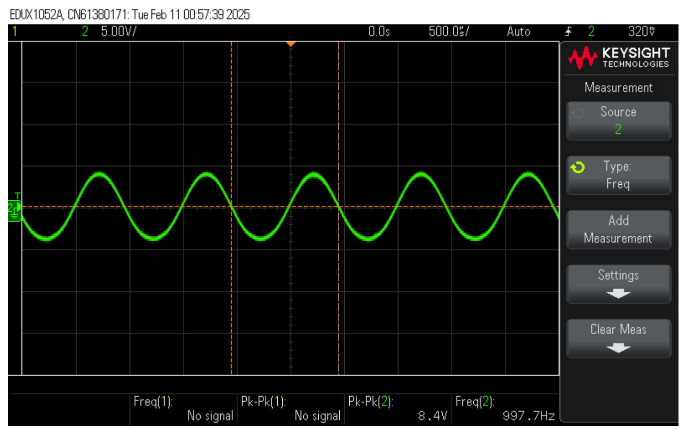

Fig. 34. Oscilloscope Outputs

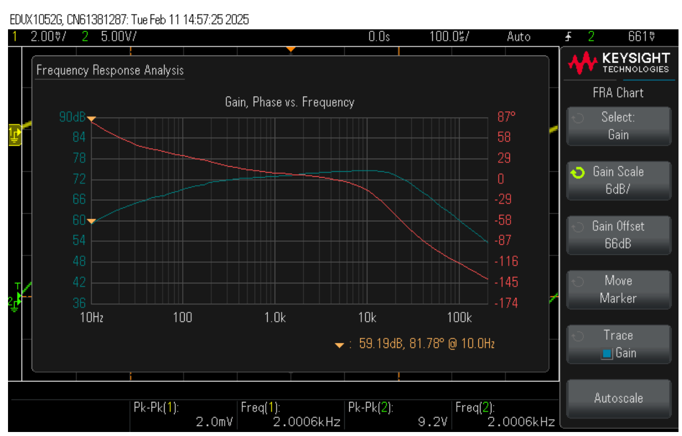

Fig. 35. Oscilloscope Frequency Response

And with this our circuit and our report comes to an end. An effort of 2.5 months is laid before your eyes in 10 short pages. If you are reading this, hats off to you. With this the report ends.

Acknowledgment

We would like to express our sincere gratitude to all those who supported and guided us throughout the completion of this project.

First and foremost, we extend our heartfelt thanks to our Professors, for providing us with the opportunity to work on this project and for their valuable insights, continuous encouragement, and constructive feedback. Their guidance has been instrumental in shaping our understanding of the subject.

We would also like to acknowledge the efforts of our TAs for providing us with the necessary resources and a conducive learning environment. Additionally, we are grateful to our peers for their collaboration, discussions, and suggestions, which helped us refine our work. This experience has been both enriching and rewarding for both of us.

Bibliography

- Mic Circuit
- Differential Amplifier
- BJT Data-sheet
- LM-7805 Data-sheet
- LM-7905 Data-sheet
- UA-741 Data-sheet
- Circuit Creator used for all the circuit images in the report
- Source for Class AB Power Amplifier