Section 1: General review of sound amplification needs and goals

Objectives:

The aim of this project is to design the electro-acoustic transducers. There are two parts of circuits constituting the electro-acoustic transducers. The first part is audio amplifier and the second part is power supply circuit. For the whole system, the output power is set to 2W and the load impedance is 8Ω. For the audio amplifier, the integrated circuit TDA2050 is used to design the amplifier to amplify the audio signal. The input voltage to amplifier is 140mV and our frequency bandwidth at -3dB is set to 40Hz~16KHz. For the power supply circuit, the integrated circuit LM117 is used to design the linear linear voltage regulator capable of rectifying a sinusoidal input voltage by transformer(24V rms,50Hz) and produce a stable and regulated 15V DC output voltage to the audio amplifier as power.

Principles of electro-acoustic transducers:

Audio amplifier and power supply circuit constitute the electro-acoustic transducers. The function of the audio amplifier is to amplify the audio signal both current and voltage which means power to meet the requirements and set the bandwidth of output signal which is 40Hz~16KHz. The function of the power supply circuit is to provide a ripple free and stiff 15V DC voltage to the amplifier by rectify and regulate the 24V rms, 50Hz AC input signal which is provided by a transformer.

Types of amplifier:

TDA2050 is a monolithic integrated circuit in Pentawatt package for audio class AB audio amplifier with high power capability.

LM117 is packaged in standard transistor packages which are easily mounted and handled. In addition, the LM117 is easy to use and require only two external resistors to set the output voltage.

Section 2: General description of our electro-acoustic transducers

Audio amplifier:

The figure 2.1 is on the datasheet of TDA2050. Based on this figure, the R1 ,R2 and R3 are our biasing resistors. R4 and R5 are the feed back resistors. The function of R6 and C6 is to ensure the frequency stability of the output voltage to prevent the danger of oscillations. C1 is non-inverting input DC decoupling capacitor.C1 and R3 also constitute a high pass filter C4 is inverting input DC decoupling capacitor. C2 is supply voltage rejection capacitor. C3 and C5 are supply voltage bypass capacitors. C7 is output DC decoupling capacitor . RL is output load resistor. RL combined with C7 constitutes the high pass filter. Based our design, there is also a capacitor C8 parallel with R5 as the low pass filter.

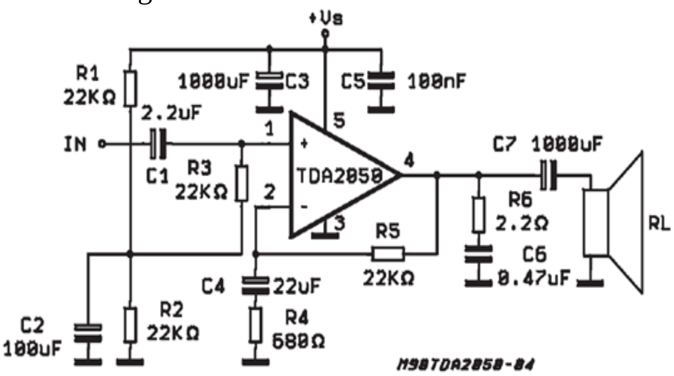

Figure 2.1: The general description of the audio amplifier

Power supply circuit:

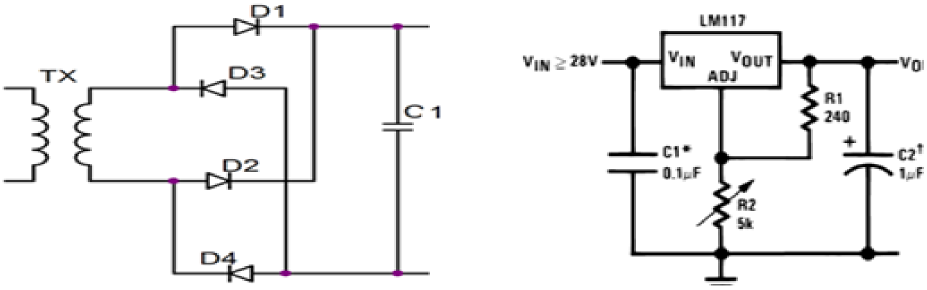

Figure 2.2: The general description of power supply circuit.

The transformer is used as our main supply and it provides 24Vrms AC. The function of four didoes are to rectify the AC voltage from the transformer. Capacitor C1 is to smooth the rectified voltage. LM117 is to regulated the voltage. R2 is our variable resistor. R1 the our output resistor. Capacitor C2 is to smooth the regulated output voltage.

Section 3: Design with computer assistance (simulations)

Design circuit diagram of the audio amplifier:

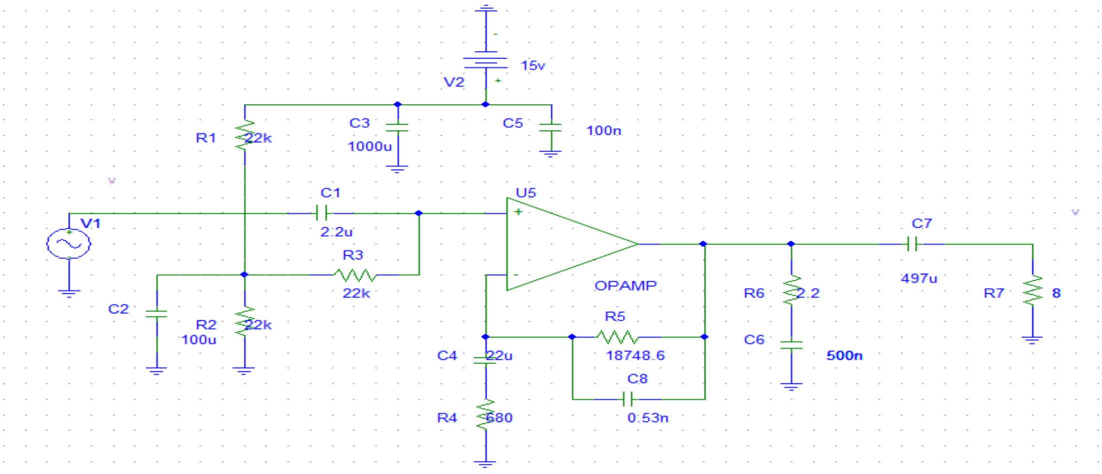

Figure 3.1: The diagram of design circuit.

This is our schematic of the design circuit, Following are the detailed explanations of major functions aspects:

Biased voltage of the amplifier:

The values of R1 and R2 are 22Ω in addition, they are in series and equal, so that the voltage across R3 is half of the DC supply(15V) i.e. 7.5V. In that case,our biased voltage for the amplifier is set to 7.5V.

Gain:

R4 and R5 are the feed back resistors which set the gain of the amplifier. In order to obtain 2W output power , according to our load impedance 8Ω, so the output voltage should be $Pout = \frac{Vout^2}{Zload}$, $Vout = \sqrt{Pout \times Zload} = \sqrt{8 \times 2} = 4V$ according to the AC input voltage is

140mV, so $Gain = \frac{Vout}{Vin} = \frac{4V}{140mV} = 28.57$ The gain of the amplifier in the figure 3.1 is $Gain = 1 + \frac{R5}{R4}$,R4 is used 680 Ω . Hence, $R5 = Gain \times R4 - R4 = 28.57 \times 680 - 680 = 18748.57\Omega$

Bandwidth and cut off frequency:

Our designed bandwidth is 40Hz~16KHz.

C7 and the load impedance R7 constitutes the high pass filter of the output signal. In that case,

$f = \frac{1}{2\pi RC}$ $C7 = \frac{1}{2\pi R_7 f} = \frac{1}{2\pi \times 8 \times 40Hz} = 497uF$. This will ensure our lower cut off frequency of the band pass filter is 40Hz

C8 and feed back resistor R5 constitutes the low pass filter of the output signal. In that case, $f = \frac{1}{2\pi RC}$ $C8 = \frac{1}{2\pi R_5 f} = \frac{1}{2\pi \times 18748.6 \times 16KHz} = 0.53nF$ This will ensure our upper cut off frequency of the band pass filter is 40Hz

C1 and R3 constitutes a high pass filter of the input signal. In that case, out cut off frequency is $f = \frac{1}{2\pi R_3 C_1} = \frac{1}{2\pi \times 22k \times 2.2uF} = 3.29Hz$

Other detail explanations of the circuit.

C1 is to block the DC signal to the AC source and to pass the DC signal to the non-inverting input Pin 1 of the amplifier. C2 influences the transient response of the amplifier. C4 aims to block the DC signal to the ground and to pass the DC signal to the inverting input Pin 2 of the amplifier. The function of C3 and C5 is to remove the noises from the DC supply. R6 and C6 are to protect the danger of oscillations of the amplifier. C7 is to block the DC signal to the load for protection because the load is a speaker and it has inductance.

Saturation:

For the max amplitude of output should be 7.5V and the minimum amplitude of output voltage should be -7.5V. This is because our limitation of amplifier is 0-15V and our biased voltage is 7.5V.

Simulations results of the audio amplifier:

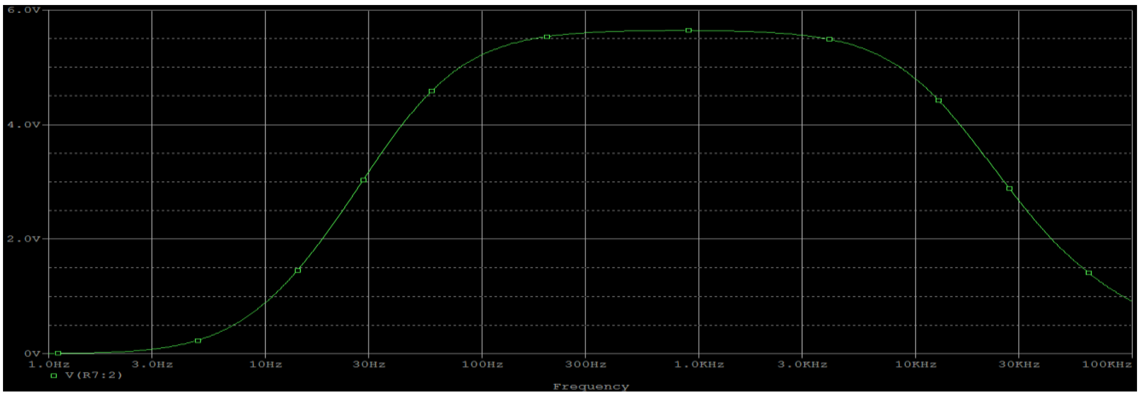

Figure 3.2: The output of design circuit of the amplifier in the frequency domain.

From the figure 3.2, our Peak voltage at the flat region is 5.64 V. The amplitude of the input is 198mV. Hence, $Gain = \frac{5.64V}{198mV} = 28.48$.This result is very close to our calculated value which is 28.57. The voltage at cut off frequency should be $5.64/\sqrt{2} = 3.98V$. Therefore, the lower cut off frequency which could be found on the graph is 42.62Hz and the upper cut off frequency is 16.06KHz. These values are slightly higher than calculated value. These errors could be explained by the lack of sufficient samples when simulating and the unconsidered impedance such as the capacitance of C4 and C8.

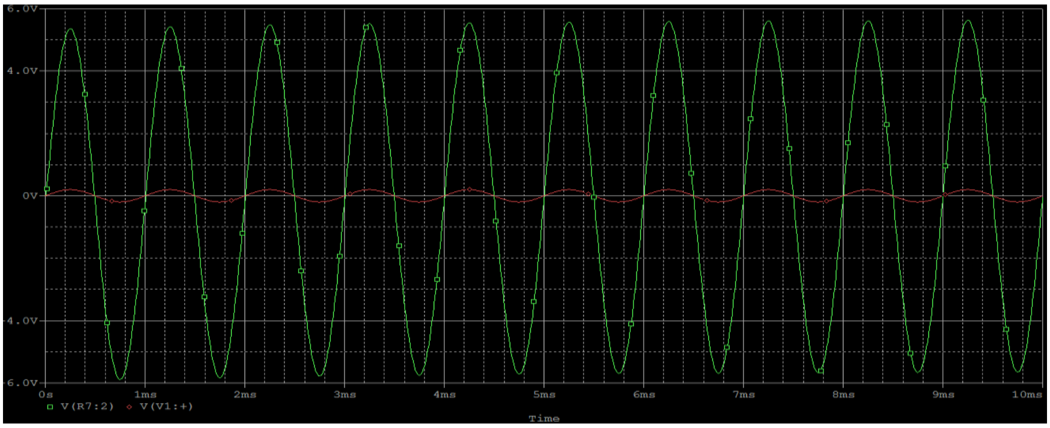

Figure 3.3: The input and output voltage of design circuit of the amplifier in the time domain at f=1KHz.

From the figure 3.3, it can be concluded that the input signal is amplified and the output signal is obtained. The peak value of the input is 198mV and the peak value of the output is 5.51V. Gain=5.51V/198mV=27.83. The peak dissipation $Ppeak = \frac{Vout^2}{RL} = \frac{5.54^2}{8} = 3.84W$

The output power is $Pout = \frac{Vout^2}{RL} = \frac{(5.54/\sqrt{2})^2}{8} = 1.92W$ These errors also could be explained by the lack of sufficient samples when simulating.

Simulation result of power supply circuit:

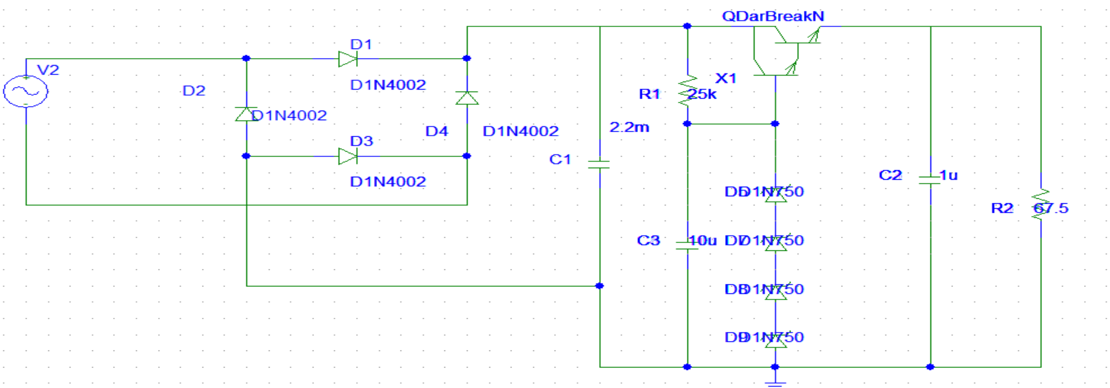

Figure 3.4: The design diagram of the Power supply circuit

In this design, V2 is used as transformer to provide 24V rms AC signal. 4 diodes(D1~D4) are used as a rectifier whose function is to convert AC to DC. C1 is used to smooth the voltage after the rectifying. The value of C1 is 2200uF, because larger value of capacitance could obtain more smooth waveform of voltage. 2200uF capacitor is large enough to obtain the smooth voltage. R2 , BJT transistor and 4 zener diodes(D1N750) are represent our LM117 IC. The function of them is regulator which outputs 15V DC. For each zener diode D1N750, the breakdown voltage is 4.7V. Thus, the breakdown voltage of 4 zener diodes in series is 18.8 V. R2 is used to limit the current flowing into the transistor which controls the output voltage. This current is amplified in the transistor. R2 is set to 25kΩ.Therefore, the 15V DC is obtained. C2 is used to smooth the regulated voltage. In addition, C2 also influences the transient response of the output voltage. Because C1 is large enough and the effect of smooth voltage is desirable, the value of C2 is unnecessary to be large. In order to obtain the quick transient response, 1uF is chosen. R2 represents the load which is the amplifier circuit. The calculation of R2 value is below:

$P_{amplifier-out} = 2W \qquad P_{amplifier-out} = 60\% P_{amplifier-in} \qquad Pout = P_{amplifier-in} = \frac{2}{0.6} = 3.33W$

$R2 = V^2 / Pout = 15^2 / 3.33 = 67.5\Omega$

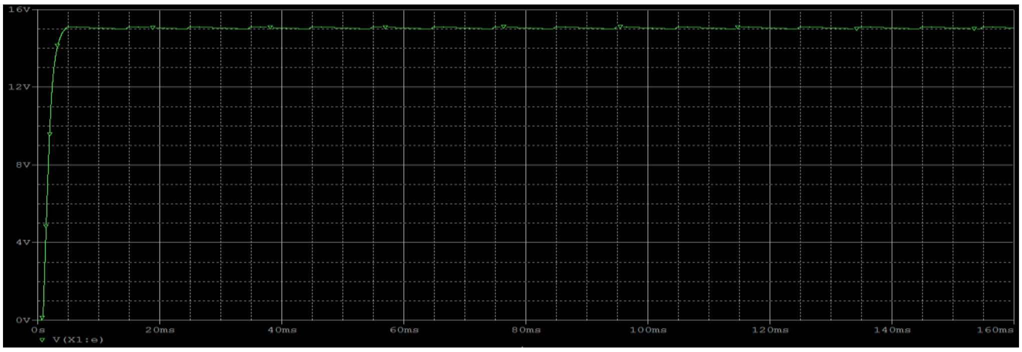

Figure 3.5: The simulation result of the power supply voltage.

From the figure 3.5, it can be concluded that the output voltage is 15V DC. There are small ripples on the figure which could be ignored. The transient response is quick.

Figure 3.6: The layout of the choice on the PCB.

The discussions about the choice of the circuit of the PCB layout:

When the circuit is working, the temperature of LM117 and TDA2050 would be increasing. To protect our circuit, the connected with heat-sinks are necessary for them and they are laid on the side of the board. The components of the circuit should be distributed equally on the PCB to make the board tidy. The signals should be always flowed from the input to the output. The components of certain modules should be placed closer. For The extra space should be reserved for big components in order to easily maintain them.

Section 4: Experimental testing

Set of experimental values of components and complement simulations:

For the audio amplifier, Most of the components' value are based on the datasheet of the TDA2050 (Figure 2.1) and the design of the circuit (Figure 3.1). R1,R2 and R3 are 22kΩ, C1=2.2uF. C2=100uF. C3=000uF. C4=22uF, C5=100nF, C6=470nF. However, there are some limitations in our laboratory so that accurate values of some components could not be accessed. In design circuit, R5 is set to 18748.6, 18kΩ is chosen to use on the PCB. Hence, $Gain = 1 + \frac{R5}{R4} = 1 + \frac{18000}{680} = 27.47$. C7 is 497uF in the design circuit. In order to ensure the lower cut off frequency less than 40 Hz, the chosen value of C7 must be larger than 497uF. In that case, the 680uF capacitor is chosen . The load resistor for testing is 7.3Ω. The new cut off frequency is $f = \frac{1}{2\pi RC} = \frac{1}{2\pi \times 7.2 \times 680uF} = 32.51Hz$. For the power supply circuit, Most of the components' value are based on the datasheet of the LM117(Figure 2.2) and the design of the circuit(Figure 3.4). C1 whose function is smoothing the voltage after the rectifying is 1000uF 63V. Actually, 2200uF capacitor could have a better effect on smoothing. However, the breakdown voltage of 2200uF capacitor at the laboratory is 35V. For our transformer, it provides 24V rms. The peak amplitude voltage is 33.94V which is very close to 35V. Sometimes the voltage from the transformer could be larger than 35V to some extent and destroy our capacitor. In order to protect the whole circuit, the 1000uF 63V capacitor is chosen for higher breakdown voltage. C2=1uF for quick transient response. C3 is 10uF. Our output resistor R1 is 220Ω. The value of variable resistor is 0~5KΩ. To obtain 15V DC output, the variable resistor is set: $Vout = 1.25(1 + \frac{R2}{R1}) + I_{ADJ} R2$.

Iadj is small which could be ignored. $R2 = (\frac{15}{1.25} - 1) \times 220 = 2420\Omega$.

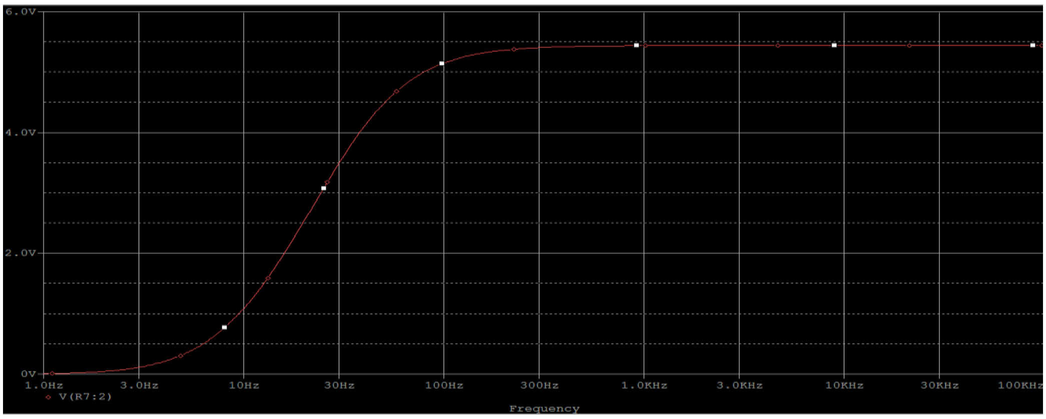

Figure 4.1: The simulation results of new set of the audio amplifier on the PCB

From figure 4.1, the voltage at flat region is 5.43V. $gain = \frac{5.43V}{198mV} = 27.42$ It is close to our design. The lower cut off frequency is 34.93Hz when the voltage is 3.84V.

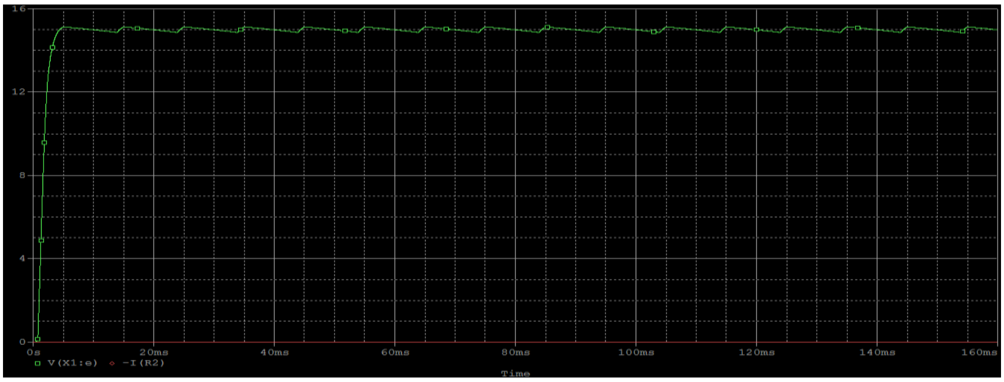

Figure 4.2: The simulation results of the new set of the power supply circuit on the PCB

From figure 4.2, Although the ripple smoothed by 1000uF capacitor is larger than that of 2200uF capacitor, the output voltage is still 15V DC with a small ripple which could be ignored.

Experimental results:

Gain:

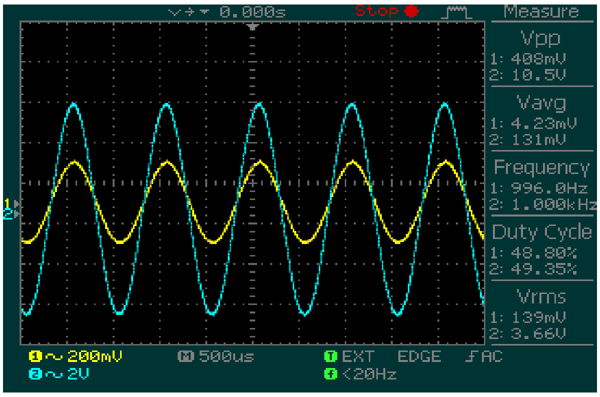

Figure 4.3: The experimental result of the input and output signal at frequency=1KHz

In the figure 4.3, frequency is chosen 1KHz to examine the gain at flat region. From the figure 4.3, it can be concluded that the shape of the waveform is very similar to the simulations results Figure 3.3. The voltage of the input is 139mV rms and the voltage of the output is 3.66V, so the $gain = \frac{3.66V}{139mV} = 26.33$. Compared with the ideal results of the simulations gain=28.57, the error is 7.8%. This is main because the feedback resistor R5 is not 18748.6Ω. Compared with results of the choice on the PCB. The design gain is 27.47, the error is 4.14%. The error may be caused by the imprecise measurements of the oscilloscope, a certain distortion of the TDA2050 and the inaccuracy of the resistance and capacitance of the components in the circuit. In addition, when the TDA2050 is working, the amplifier would be heating. The power dissipation would be decreased if the temperature is higher.

Lower cut off frequency (bandwidth):

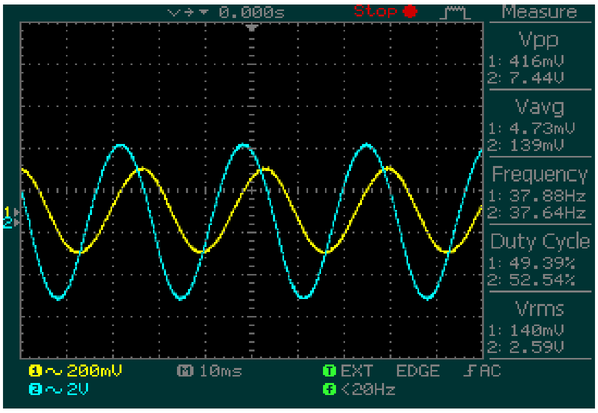

Figure 4.4: The experimental result of cut off frequency.

According to the Figure 4.3, the output voltage at flat region is 3.66V if the input voltage is 140mV. Therefore, the voltage at cut off frequency should be $\frac{3.66}{\sqrt{2}} = 2.59V$. In the figure 4.4, the output voltage is 2.59V with the input voltage 140mV at frequency=37.88Hz. This value satisfy the design of the project whose cut off frequency is less than 40Hz. Compared with the simulation result with the ideal components, this value is less than 42.62 Hz. This is because the load resistor is 7.2 Ω and the output capacitor is 680uF. Compared with the design circuit on the PCB, the cut off frequency should be 34.93Hz, the difference may be caused by the following issues: The inaccuracy of the output capacitor, which means the tolerance of capacitor is large, and the measurement error of the oscilloscope could affect the result.

Output voltage of Power supply circuit:

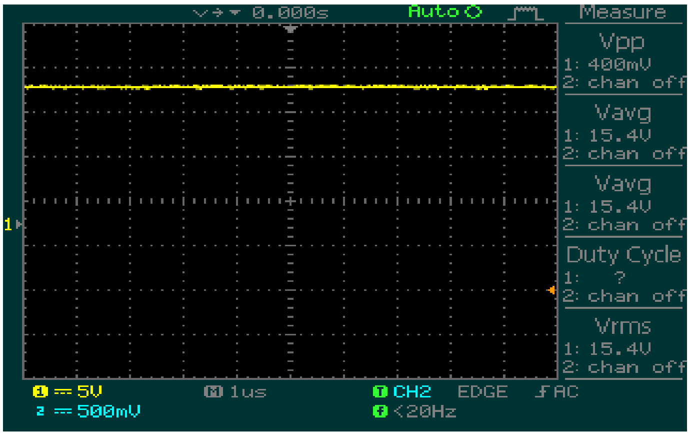

Figure 4.5: The result of the power supply circuit.

From the figure 4.5, the output voltage of our power supply circuit is 15.4V DC. The output voltage is also measured by the multi-metre is 14.98V. Compared with our simulation result, the figure is very similar. The Vpp is 400mV which shows the ripple of the smoothing. This value is small which could be ignored, even though the 1000uF capacitor replace the 2000uF capacitor. However,the error is main caused by the imprecise measurement of the oscilloscope and the effect of the LM117 when the temperature is becoming higher.

Saturation of the audio amplifier:

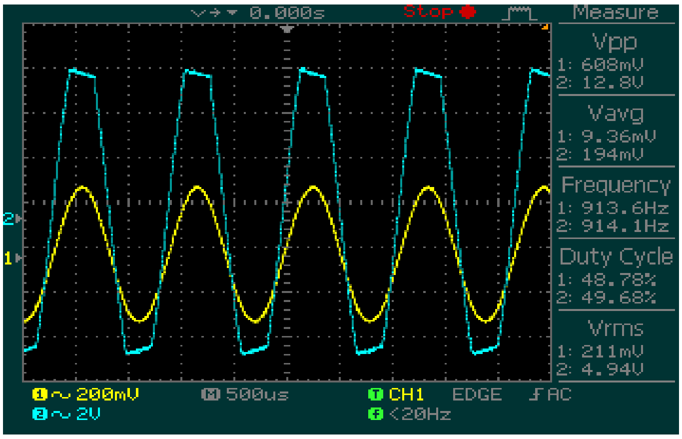

Figure 4.6: The result of the saturation of the audio amplifier

From the figure 4.6, it can be concluded that the maximum input amplitude is 304mV and minimum input amplitude is -304mV. The maximum output amplitude is 6.4V and minimum is -6.4V. Our theoretical output maximum amplitude is 7.5V.

Matlab test:

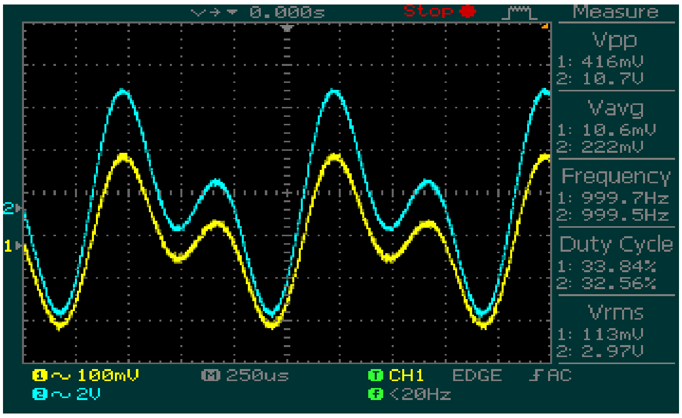

Figure 4.7: The result of using Matlab with frequency=1KHz+2KHz signal

From the figure 4.7, the result is tested by using Matlab to create signals. The shape of this signal is sharper. The $Gain = \frac{2.97V}{113mV} = 26.28$. It is similar as previous test. This means our gain is stable at flat region. Hence, it can be concluded that the design circuit could be regarded as linear circuit.

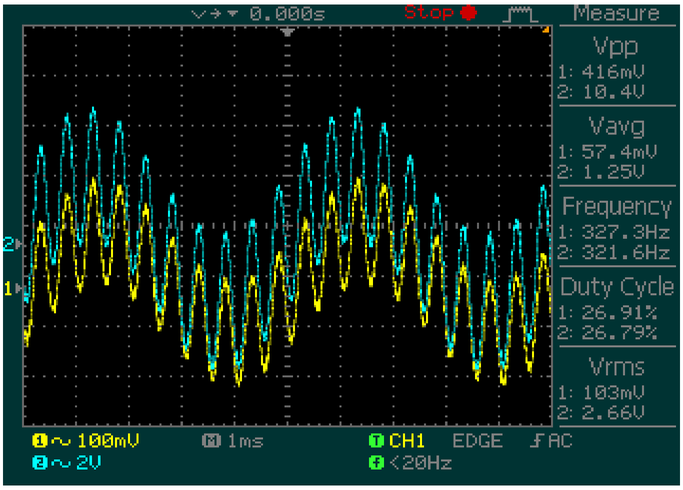

Figure 4.8: The result of using Matlab with frequency=200Hz+2kHz signal

From the figure 4.8, the result is tested by using Matlab to create signals. The shape of this signal is rougher. The $Gain = \frac{2.66V}{103mV} = 25.83$. It is similar as previous test. This means our gain is stable at flat region. Hence, it can be concluded that the design circuit could be regarded as linear circuit.

Discussion about the test methods:

Signal generator vs written Matlab files:

The signal generator could provide stable signal whose frequency, amplitude and shape could be easily to adjust on the menu of signal generator. However, the signal generator could only provide several types of signal such as square wave and sine wave. Written Matlab file could provide different kinds of signal. It also could generate signals with complex functions such as sum of independent sinusoidal signals. In addition, it is convenient to provide sweep signal to check the circuit. However, the output of the generated signal by matlab is from the computer. It may have distortion with high frequency signals and noises with low frequency signals. The amplitude of the output signal by using matlab is more difficult to be adjusted by the computer volume control than that of testing by signal generator.

Multi-meter vs oscilloscope:

For multi-meter, it is simple and convenient to use to measure the value of the resistors, current and DC/AC voltage at different point. The multi-meter could also check the problem of the circuit such as short circuit and open circuit.

For oscilloscope, it could measure the instantaneous value of the signal and show the waveform on the screen. The frequency, Vavg, Vrms,Vpp,Vmax are Vmin of the signal could be read from the oscilloscope directly. However, the oscilloscope could only measure the voltage.

Section 5: Conclusion

For this project, techniques of using Pspice to simulate the circuit is addressed. The simulations have an significant effect on the convenience to design the circuit. The experimental results are matched with the simulation results to some extent. The gain is 26.3 in practical, which is very close to our design. The error is main because the limitation to obtain the exact value in the laboratory. In that case, the output power would be less than the required power dissipation. The lower cut off frequency is 37.88Hz tested by 7.2Ω load resistor, which satisfies the design requirement that the minimum frequency of the passband is 40Hz. Several complex functions created by the matlab is used to test our circuit. The circuit could be regarded as linear circuit at passband. The output voltage of power supply circuit is 14.98V DC which is close to the simulation and satisfies the design requirement. The ripple of the output voltage is 400mV Vpp which could be ignored. However, the upper cut off frequency is not achieve in the experiment, because the value of required capacitor is small and there is no place to make the capacitor parallel with the feed back resistor R5 on the PCB. In that case, our circuit do not have upper cut off frequency. Despite of the lack of upper limitation of the passband, the quality of the electro-acoustic transducers is terrific when applying a mp3 song from the Internet to the circuit.

Appendix:

1. The datasheet of TDA2050
2. The datasheet of LM117
3. Lecture notes of introduction of Electronic construction Project