Stellenbosch
UNIVERSITY
IYUNIVESITHI
UNIVERSITEIT

forward together
sonke siya phambili
saam vorentoe

Electronics 315 - Practical 3

Xari Mouchtouris
23795824

Practical 3:

Theoretical Calculations:

Calculation of VCC

The audio amplifier has to be designed for the following undistorted power and for the following values below in table 1.

Parameter	Value
P_L	5 W
R_L	8
V_{RE}	1 V
$V_{BE(on)}$	0.7 V
$V_{EC5(sat)}$	0.2 V
β_2	50
β_1	100

Table 1: Design Values for Audio Amplifier.

Peak Amplitude of load voltage:

$$V_{op} = \sqrt{2 \cdot P_L \cdot R_L} = 8.944\,\mathrm{V}$$

Choose $V_{RE} = 1$ V and the voltage between Q2 and Q3: $2V_{BE(on)}$

$$V_{x(\mathrm{min})} = 2 \cdot V_{\mathrm{be\ on}} = 1.4\,\mathrm{V}$$

$$V_{x(\mathrm{NoInput})} = V_{x(\mathrm{min})} + V_{op} = 10.344\,\mathrm{V}$$

$$V_{x(\mathrm{Max})} = V_{x(\mathrm{NoInput})} + V_{op} = 19.29\,\mathrm{V}$$

$$V_{y(\mathrm{Max})} = V_{x(\mathrm{Max})} + 2 \cdot V_{\mathrm{BE(on)}} = 20.69\,\mathrm{V}$$

$$V_{CC} = V_{y(\mathrm{Max})} + V_{\mathrm{EC5(sat)}} + V_{RE} = 21.89\,\mathrm{V}$$

For simplicity and reliability in the operation of the circuit, we have selected $V_{CC} = 24$ V for further calculations.

Design of Current Source:

The current source should be capable of supplying the maximum base current i_{B1_max} at full load power, as well as the minimum collector current i_{C6_min} at full load power, in addition to providing the current through resistors R_1 and R_2.

$$i_{L(\text{Max})} = \frac{V_{op}}{R_L} = 1.11803\,\text{A}$$

Next, $i_{B2(Max)}$ can be calculated. From the TIP41C datasheet, we can see $\beta_2 = 50$:

$$i_{B2(Max)} = \frac{i_{E2(max)}}{\beta_2 + 1} = 21.922\ mA$$

Because, we can see $i_{B2(Max)} = i_{E1(Max)}$, we can calculate $i_{B1(Max)}$ as well. I chose $\beta_1 = 100$ for the calculations:

$$i_{B1(max)} = \frac{i_{E1(Max)}}{\beta_1 + 1} = 217.052\,\mu\text{A}$$

Choose $i_{C6(Min)}$ so that $Q6$ will stay on, thus I chose $i_{C6(Min)} = 1.5$ mA.

$$i_{C6(\text{max})} = i_{B1(\text{max})} + i_{C6(\text{min})} = 1.717\ mA$$

$$i_{B6(\text{Max})} = \frac{i_{C6(\text{Max})}}{\beta_6} = 17.17\,\mu\text{A}$$

Next, I chose $i_{R1} = 1$ mA. I chose I_{R1} » $i_{B6(Max)}$ so that the voltage division can be used in the design of the V_{BE}-multiplier.

$$\begin{aligned} I_{R_1} &= 1mA \\ I_{C5} &= i_{B1(Max)} + i_{C6(Min)} + I_{R1} = 2.717\ \text{mA} \end{aligned}$$

Thus $I_{C5} = 2.71$ mA, but I assumed a higher value of $I_{C5} = 3$ mA for the rest of the calculations. Now RE can be calculated as:

$$R_E = \frac{V_{RE}}{I_{C5}} = 333.33\,\Omega$$

Design the current source for β-stability with a chosen $\beta_5 = 100$.

Firstly, calculate R_{th} and V_{th}:

$$R_{th} = 0.1 \cdot \beta_5 \cdot R_E = 3333.33\,\Omega$$
$$V_{th} = V_{CC} - V_{RE} - V_{BE(on)} - \frac{I_{C5}}{\beta_5} \cdot R_{th} = 22.2\ V$$

Next, we can use R_{th} and V_{th}, to calculate the resistors of the current source, R3 and R4 below:

$$R3 = \frac{V_{CC} \cdot R_{th}}{V_{th}} = 3603.603\,\Omega$$
$$R4 = \frac{R3 \cdot \left(\frac{V_{th}}{V_{CC}}\right)}{1 - \left(\frac{V_{th}}{V_{CC}}\right)} = 44.44\,\mathrm{k}\Omega$$

Design of the V_{BE}-Multiplier

The minimum collector current for Q6 has been carefully chosen to ensure the stability of the transistor. Additionally, the current flowing through R_1 and R_2 consistently surpasses $i_{B6_{\max}}$, even under maximum load conditions. The total resistance $R_1 + R_2$ has been calculated based on the specified I_{R_1}, considering the activation of four transistors (two Darlington pairs). Subsequently, the ratio between R_1 and R_2 is determined from the VBE-multiplier design equation.

Firstly, we start by calculating the total resistance:

$$R_{Total} = \frac{4 \cdot V_{BE(ON)}}{I_{R1}} = 2800\,\Omega$$

Thus, R_{Total} is:

$$R_{Total} = R_1 + R_2 = 2800\,\Omega$$

We can solve for R_2 and R_1 in the following ways:

$$R_2 = \frac{R_{\text{Total}}}{4} = 700\,\Omega$$

$$R_1 = 3 \cdot R_2 = 2100\,\Omega$$

Design of R_{Bias}

When no input signal is applied, the voltage between the emitters of Q2 and Q3 will be $2V_{\text{EB(on)}} + V_{\text{OP}}$. This implies that the voltage across R_{Bias} will be V_{OP}. Since the input base currents of the Darlington pairs are negligible, the current flowing through R_{bias} will equal the current supplied by the current source (I_{C5}). Consequently, the value of R_{Bias} is determined from the provided voltage and current values.

$$V_{\text{Bias}} = V_{OP} + 2 \cdot V_{\text{BE(on)}} = 10.34\,\text{V}$$

$$R_{\text{Bias}} = \frac{V_{\text{bias}}}{I_{C5}} = 3448.09\,\Omega$$

Power ratings of components

Transistor Power Ratings

Calculate the maximum power dissipation of $Q5$ as:

$$P_{Q5(Max)} = I_{C5} \cdot V_{EC5(Max)} = I_{C5} \cdot (V_{\text{CC}} - 1 - 4 \cdot V_{\text{BE(on)}}) = 60.6\ mW$$

Calculate the maximum power dissipation of $Q6$ as:

$$P_{Q6(max)} = 4 \cdot V_{\text{be\_on}} \cdot (i_{C6\_max}) = 7.608 mW$$

The power dissipation for transistors $Q1$ and $Q4$ will be relatively low, and small-signal transistors should be adequate for them. We'll calculate the maximum average power dissipated by the power transistors, as derived previously. It's important to note that the maximum voltage swing is now $\frac{V_{CC}}{2}$, so the expression needs to be adjusted accordingly.

$$P_{Q1(Max)} = \frac{V_{\text{CC}}^2}{4 \cdot (\pi)^2 \cdot (\beta_2 + 1) \cdot R_L} = 35.760\ mW$$

$$P_{Q2(Max)} = \frac{V_{\text{CC}}^2}{4 \cdot (\pi)^2 \cdot R_L} = 1.824\ W$$

Resistor Power Ratings

The maximum power dissipation of R_{bias} will be relatively high when the maximum positive value of the input signal is applied. Therefore, it's important to use a resistor with an appropriate wattage rating.

$$P_{R_{\text{Bias}}} = \frac{(V_{\text{CC}} - V_{RE} - V_{EC5(sat)} - 4 \cdot V_{\text{BE(on)}})^2}{R_{\text{Bias}}} = 113.208\,\text{mW}$$

The resistors $R1$, $R2$, $R3$, and $R4$ are designed for currents in the low mA range, so they shouldn't dissipate a significant amount of power. Using 1/4 W or 1/2 W resistors should be sufficient.

$$P_{R1(Max)} = \frac{(3 \cdot V_{\text{BE(on)}})^2}{R_1} = 2.1\,\text{mW}$$

$$P_{R2(Max)} = \frac{(1 \cdot V_{\text{BE(on)}})^2}{R_2} = 0.7\,\text{mW}$$

$$P_{R3(Max)} = \frac{(V_{RE} + V_{\text{BE(on)}})^2}{R_3} = 0.802\,\text{mW}$$

$$P_{R4\_max} = \frac{(V_{\text{CC}} - V_{RE} - V_{\text{BE(on)}})^2}{R_4} = 11.189\,\text{mW}$$

The current through R_E will be relatively large, the low voltage across it should still allow 1/4 W or 1/2 W resistors to handle the power adequately.

$$P_{RE(Max)} = \frac{(V_{RE})^2}{R_E} = 3\,\text{mW}$$

Heat Sinks Design:

To ensure the power transistors don't exceed $T_{j,\text{max}}$ at maximum load power dissipation, we need to assess the effectiveness of the provided heat sink, considering a 1 mm thick aluminium plate. It's crucial that both transistors in a Darlington pair maintain the same temperature. Let's determine if the heat sink adequately reduces the junction temperature to an acceptable level.
The heatsink design values I took out of the Heatsink notes, can be seen below in table 3.

Heatsink Note Variables	Values
θ_{jc}	$1.92\,°\text{C}\,\text{W}^{-1}$
θ_{ja}	$62.5\,°\text{C}\,\text{W}^{-1}$
θ_{ja}	$6\,°\text{C}\,\text{W}^{-1}$
T_a	$25\,°\text{C}$

Table 2: Heatsink Note Design Values

We calculated previously $P_D = \overline{P_{Q2(\text{Max})}} = 1.82\,\text{mW}$. We can use this to calculate the junction temperature T_J without the heatsink:

$$T_J = P_D \cdot \Theta_{ja} + T_a = 138.75\,°\text{C}$$

Thus, we can see that without a heatsink, the junction temperature is quite high and very close to $T_{J,Max} = 150\,°\text{C}$.

The dimensions of the heatsink plate can be seen below in figure 1.

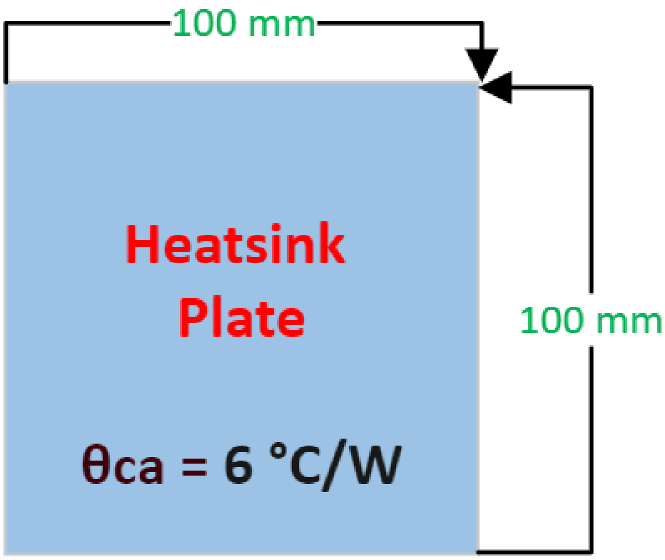

Figure 1: Heatsink Dimensions:

Now, that we know the dimensions of the heatsink, and θ_{ca} we can calculate the new junction temperature T_J with the heatsink:

$$T_J = P_D \cdot (\Theta_{jc} + \Theta_{ca}) + T_a = 39.4\,°\text{C}$$

Thus, we can see that with a heatsink, the junction temperature is much lower and far from $T_{J,Max} = 150\,°\text{C}$. The addition of using a heatsink on both TIP42C and TIP41C is a good decision, and would prevent the power transistors from burning out during operation of the power amplifier. By using the heatsink the temperatures will stay much lower, and ensure working operation of both power transistors and prevent any damager to the components.

Design of Low-Frequency cut-off:

A low-frequency cut-off was specified as $f_{Low} = 80$ Hz. Thus, the input capacitor (C1) and the output capacitor (C2) must be calculated for $f_{Low} = 80$ Hz.

Firstly, we need to calculate the equivalent resistance, that each capacitor ”sees”:

C2:

The resistance of C2 is already known, as we are using a $8\,\Omega$ speaker, thus the capacitor (C2) ”sees” $R_L = 8\,\Omega$.

C1:

The resistance ”seen” by C1 is challenging to calculate, thus NGSPICE will be used to calculate R_{C1}. The procedure we will follow is the following:

- Step 1: Apply V_{Input} of 1V
- Step 2: Measure the input current.
- Step 3: Use $R_{\text{Input}} = \frac{V_{\text{Input}}}{I_{\text{Input}}}$

The NGSPICE values I measured were the following:

- V_{Input}: 1 V
- I_{Input}: 0.000297826 A
- $R_{\text{Input}} = \frac{V_{\text{Input}}}{I_{\text{Input}}} = 3357.67\,\Omega$

Figure 3 below illustrates I_{Input} measured on NGSPICE.

```
No. of Data Rows : 508
ngspice 2 ->
x0 = 0.002875, y0 = 0.000297826
```

Figure 2: I_{Input} Measured on NGSPICE:

Now, we have to calculate which capacitor is dominant/non-dominant. The smallest equavelent resistance is the dominant capacitor. Thus, since C2, has the smallest equavelent resistance, it is chosen as dominant. C1 is the Non-dominant capacitor. Table 3 below illustrates capacitor design frequencies.

Capacitor	Non-Dominant/Dominant	Frequency
C1	Non-Dominant	$f_{ND} = \frac{80}{10} = 8$ Hz
C2	Dominant	$f_{Low} = 80Hz$

Table 3: Capacitor Design Frequencies:

Now, we have to calculate the capacitor value for C2:

$$C_2 = \frac{1}{2\pi R_L f_{\text{Low}}} = 248.68\,\mu F$$

Next, we can calculate the capacitor value of C1, with the R_{Input} we calculated from NGSPICE.

$$C_2 = \frac{1}{2\pi R_{Input} f_{\text{ND}}} = 5.925\,\mu F$$

Thus, C1 = 248.68 µF and C2 = 5.925 µF.

Design of Pre-Amplifier:

To design the Pre-Amplifier for the 5W amplifier, we firstly consider a non-inverting Op-Amp.

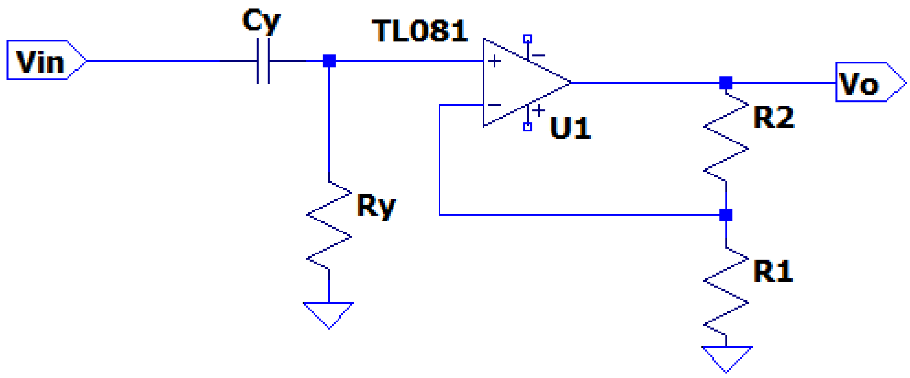

Figure 3: Non-Inverting Dual-rail Op Amp Circuit:

The gain of a Non-Inverting Op Amp is given by:

$$A_v = 1 + \frac{R2}{R1}$$

The Pre-Amplifier gain I want to design for is $A_v = 9.43$ V/V. The gain was chosen to ensure an output of 8.94 V at the output of the audio amplifier. Thus, I choose R2 = 100 kΩ, and then I can calculate R1 with the formula above. This results in a value of R1 = 11 862.40 Ω. The reason, why I chose R2 = 100 kΩ, is because it's a standard variable resistor value, and I chose that as a constant and calculated R1.

In theory, this non-inverting amplifier works, but unfortunately we can't use this Op-Amp design. We can't use a dual-rail voltage of ±24 V. Thus, the circuit should be changed to only use a single input voltage of 24V. Figure 4 below illustrates the new non-inverting Op-Amp design that will work as a Pre-Amplifier.

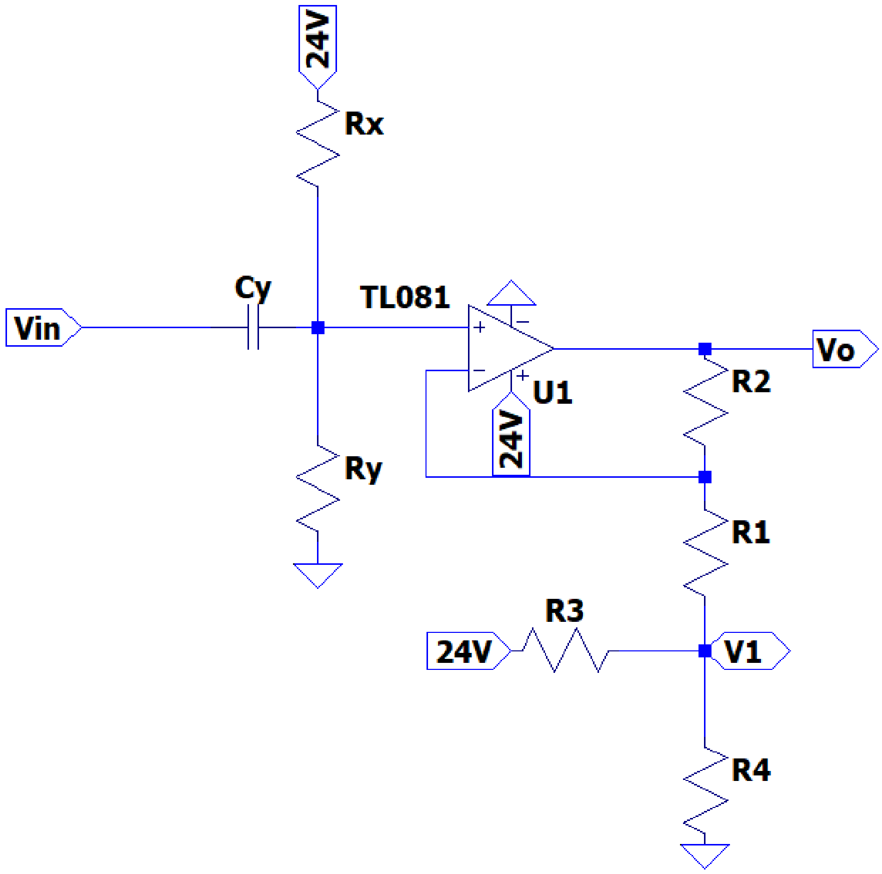

Figure 4: Non-Inverting Single Input Op Amp Circuit:

Firstly, we choose $R_x = R_y = 330\,\mathrm{k\Omega}$. This value is chosen very large, to achieve a larger input resistance.

We already calculated R2 $= 100\,\mathrm{k\Omega}$ and R1 $= 11\,862.40\,\Omega$. These resistors stay the same, although the design changed to a single input Op-Amp circuit.

The node V1 should be $\frac{V_{CC}}{2}$, thus we use a voltage divider. R3 and R4, can be chosen as $1\,\mathrm{k\Omega}$. The only requirement for R3 and R4, is that it should be much less than R1, which in this case is true. R3, R4 are $10\,862.40\,\Omega$ less than R1, which is valid.

Finally, the capacitor Cy should be calculated. Using the input resistance of $R_{In} = R_x||R_y = 165\,\mathrm{k\Omega}$, we can calculate Cy as:

$$C_y = \frac{1}{2\pi R_{In} f_{\mathrm{ND}}} = 120.57\ nF$$

The Pre-Amplifier Circuit with the final values can be seen in figure 12 below.

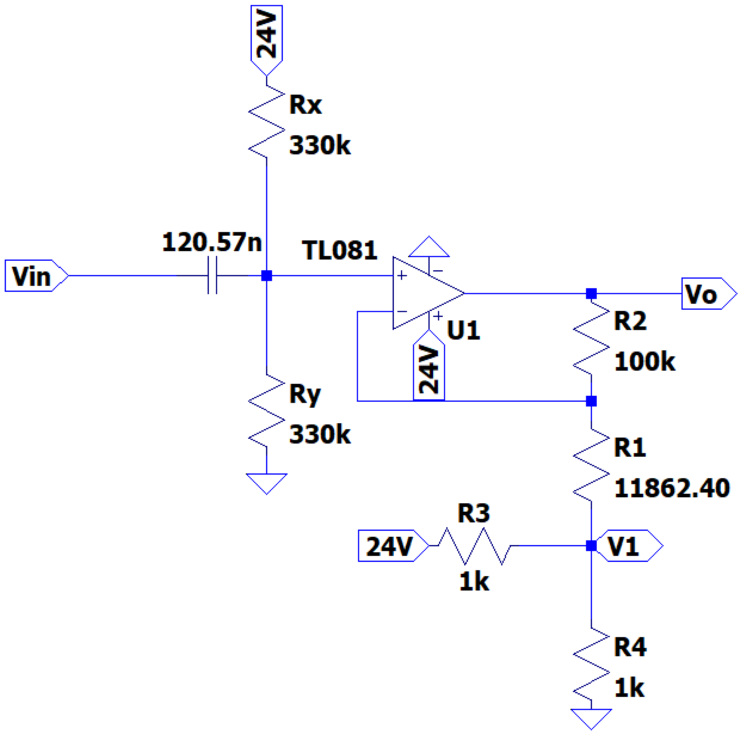

Figure 5: Non-Inverting Single Input Op Amp Circuit with final values:

The working Pre-Amp above, will work with a single input voltage of 24V. The only change still to make, is that because there are two channels from the AUX cable, we have to connect both channels to the Non-Inverting pin of the Op-Amp. For my design I chose to put a $100\,\Omega$ resistor in series with each channel from the AUX port. By connecting the two channels, in parallel with each other, our input voltage, won't be 1V any more, but 0.5V. Thus, this would cause our A_v to drop to 4.715 V. To prevent, this we have to adjust R1, to $\frac{R1}{2} = 5931.2\,\Omega$. This, would yield the same gain at the output of the pre-amplifier as before, but now only with both left and the right channel connected to the Pre-Amp.

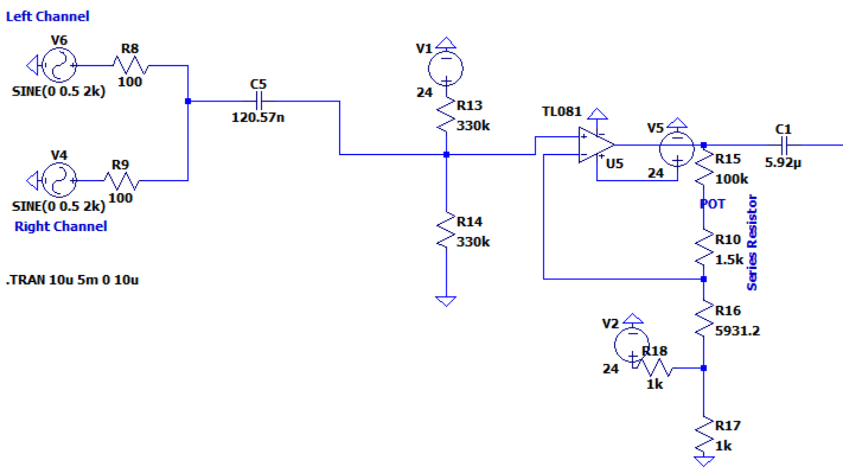

Figure 6: Adjust Pre-Amplifier with both channels from Aux

SPICE Analysis:

The audio amplifier circuit with the nodes labelled can be seen below in figure 7. These nodes were used in NGSPICE, to simulate the designed audio amplifier.

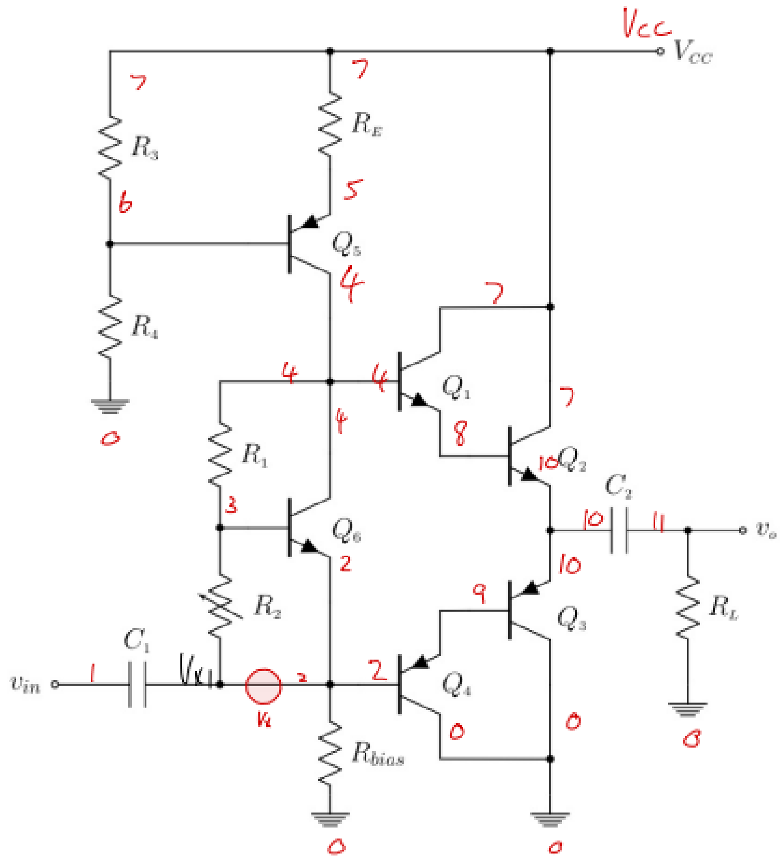

Figure 7: Audio Amplifier with SPICE node labels

The output voltage graph of the audio amplifier can be seen below, with a maximum value of 8.93 V, which is quite close to the calculated value. The input at Vin = 9.43 V, which gives an undistorted output voltage Vo wave. This proves that the audio amplifier works as designed. Figure 8 below illustrates the output voltage wave and figure 9 illustrates the input voltage wave from the Pre-Amplifier.

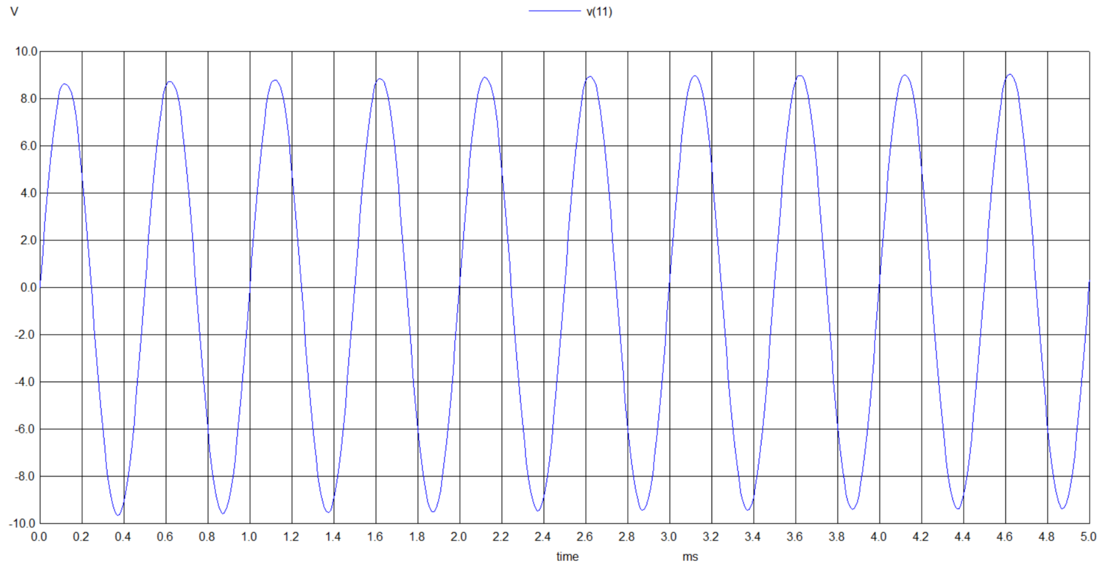

Figure 8: Undistorted Output Voltage V_O) of Audio Amplifier

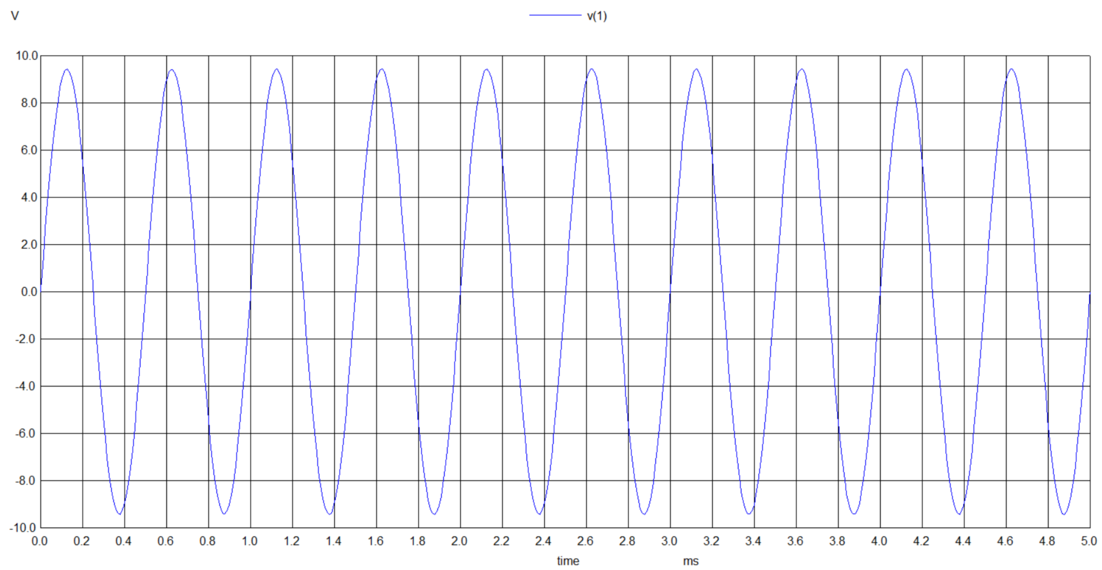

Figure 9: Applied Input Voltage from Pre-Amplifier (V_{In})

The gain of the audio amplifier is calculated as:

$$A_v = \frac{V_o}{V_{\text{in}}} = \frac{8.93}{9.43} = 0.947\,\text{V/V}$$

The SPICE gain of the audio amplifier is calculated as $A_v = 0.947$ V/V by using the equation above. From figure 8, it is visible that 5 W undistorted power is delivered to the audio amplifier.

If a high voltage is applied for example $V_{In} = 10.5$V, we notice on figure 10, that the output voltage signal starts to distort on top, since the audio amplifier wasn't designed for such a high input voltage.

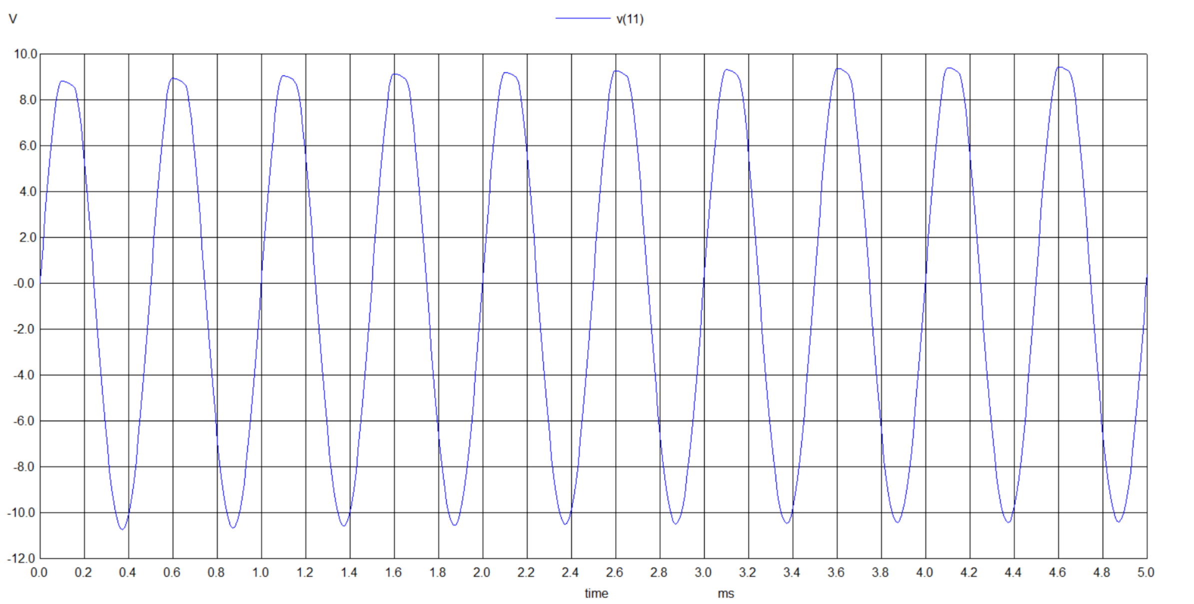

Figure 10: Output Voltage with a high input voltage (Distortion)

Frequency Analysis with Spice:

Figure 11 below illustrates the frequency response of the audio amplifier with a Vin = 9.43 V.

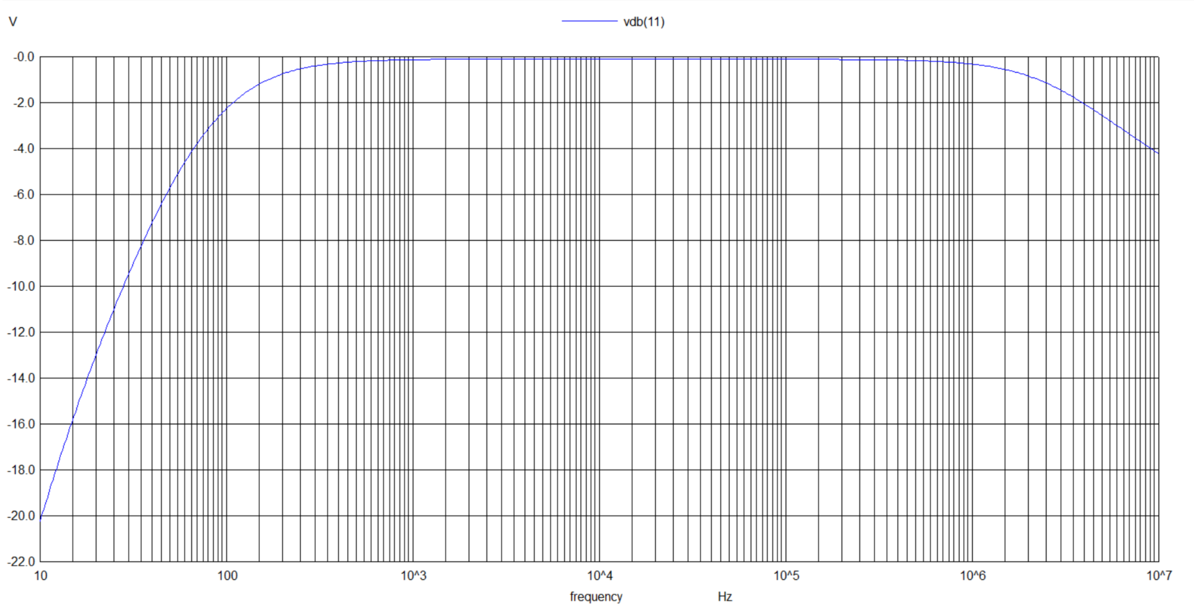

Figure 11: Frequency response of Audio Amplifier (NGSPICE)

The low and high cut-off frequencies from NGSPICE can be seen below in table 4.

	f_{High}	f_{Low}
Theoretical	N/A	80 Hz
Simulated (Spice)	10 MHz	80 Hz

Table 4: Frequencies calculated and measured on SPICE

The low cut-off frequency of 80Hz from SPICE matches our theoretical design value of f_{Low} = 80 HZ. Thus, this illustrates that the audio amplifier frequency behaves as expected and what was designed for.

Pre-Amplifier Test:

The Pre-Amplifier circuit was built on LTSPICE, to identify if the output voltage of the preamplifier is correct with the theory. See figure 12 below for the built circuit in LTSPICE.

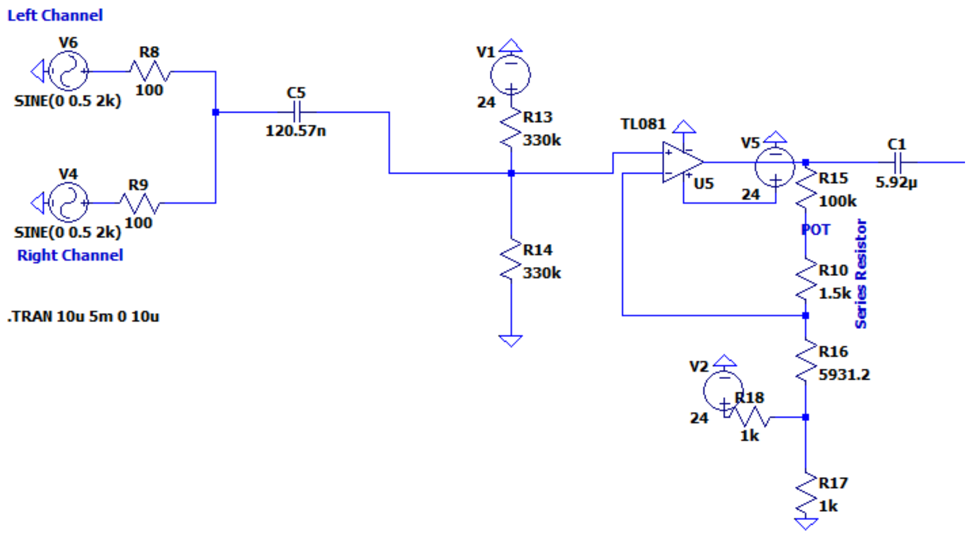

Figure 12: Pre-Amplifier for Audio Amplifier (LTSPICE circuit as designed above)

The circuit was tested by applying an input sine wave of 0.5V for both left and right channel of the AUX on LTSPICE and measuring the output voltage to identify if it amplifies the input signal from the Audio Jack to the correct voltage and if it swings the output voltage around 12V, instead of 0 V. Figure 13 below illustrates the output voltage of the Pre-Amplifier.

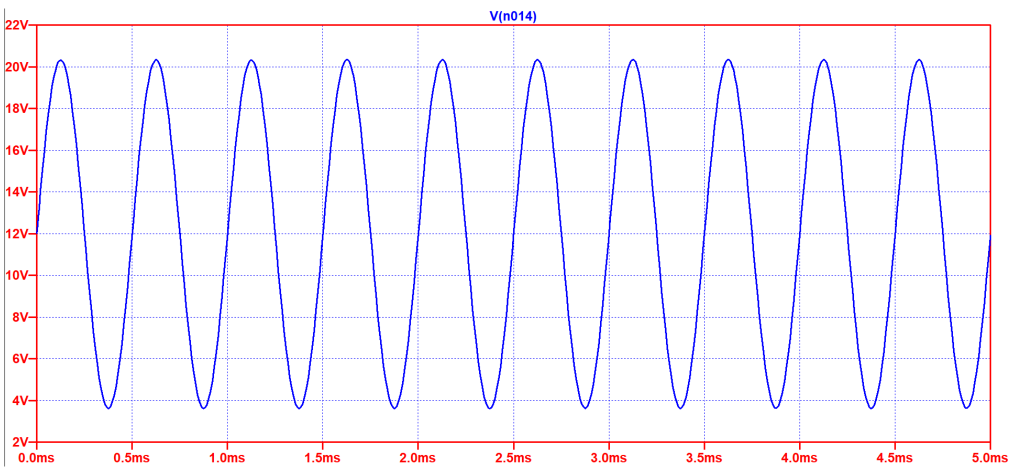

Figure 13: Output of Pre-Amplifier (LTSPICE)

From figure 13 above, it's visible that the output voltage from the Pre-Amplifier swings around 12V, instead of 0V. If we measure the output voltage, we see that it's quite close to the 9.43V. Thus, the pre-amplifier would supply the correct input voltage to the audio amplifier input. Thus, the Pre-amplifier design works as intended.

Heatsink and Temperature SPICE test:

We can make use of the NGSPICE function **.Temp Temp Value** to evaluate the audio amplifier output voltage at different temperature. This would prove that using a heatsink would be beneficial to the audio amplifiers and to protect the power transistors.

Firstly, I will evaluate the audio amplifier at a low temperature of 39.6 °C (With Heatsink). Figure 14 below illustrates the output of the audio amplifier at this temperature.

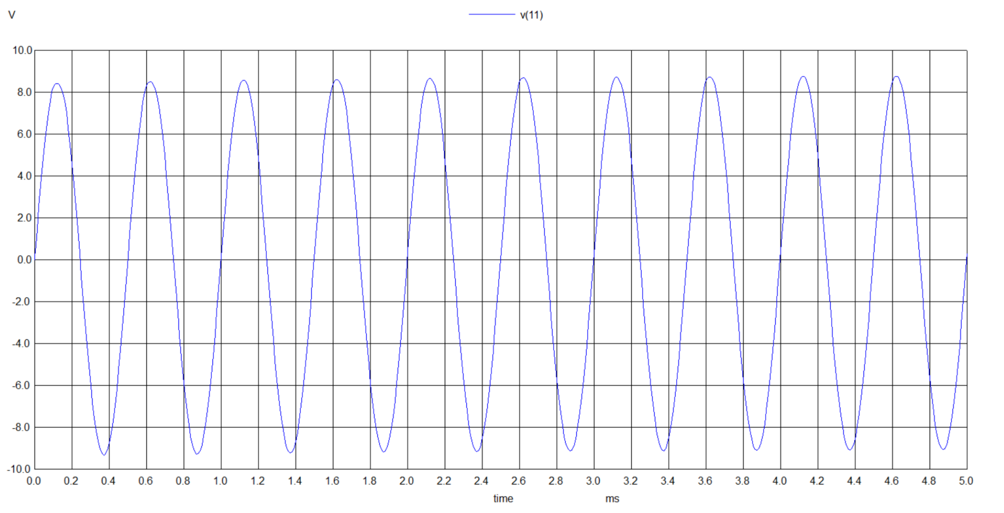

Figure 14: Output of Audio Amplifier at 39.6 °C

As we can see, the output of the audio amplifier is still undistorted output voltage. Thus, using the heatsink the temperature is low, and the circuit is at full operation.
Secondly, I will evaluate the audio amplifier at a high temperature of 139 °C (Without Heatsink). Figure 15 below illustrates the output of the audio amplifier at this temperature.

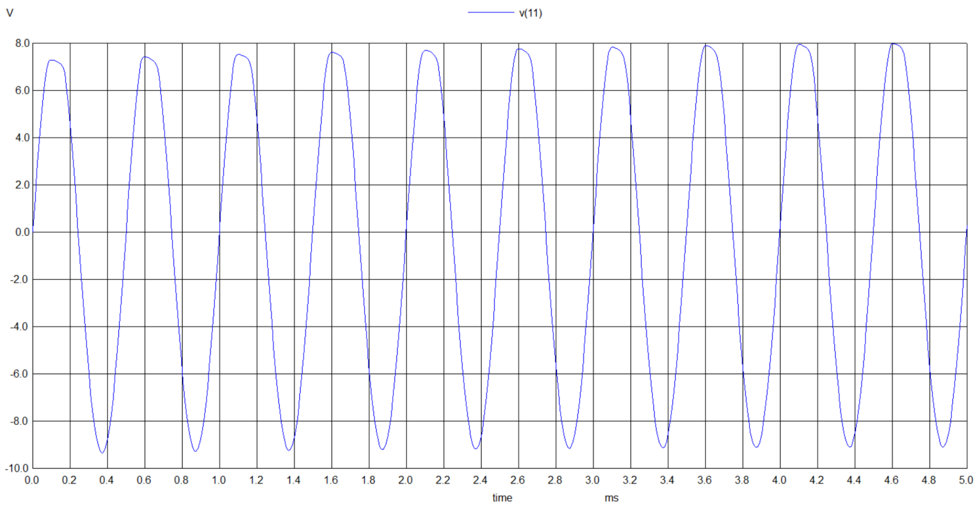

Figure 15: Output of Audio Amplifier at 139 °C

As we can see, the output of the audio amplifier has visible distortion at the top of the output voltage. Thus, opting to run the circuit without heatsinks will cause distortion to your circuit. Take note, although the circuit still runs on NGSPICE, in a real operating circuit at such high temperature the TIP42C and TIP41C would mostly likely burn out after running for too long at such temperatures.

Full Circuit Test on LTSPICE:

LTSPICE was used in conjunction with NGSPICE.The full circuit with the Pre-Amplifier included, was built on LTSPICE to test if it works. By doing this, we can compare it to the NGSPICE results.Figure 16 below illustrates the full schematic of the audio amplifier built in LTSPICE.

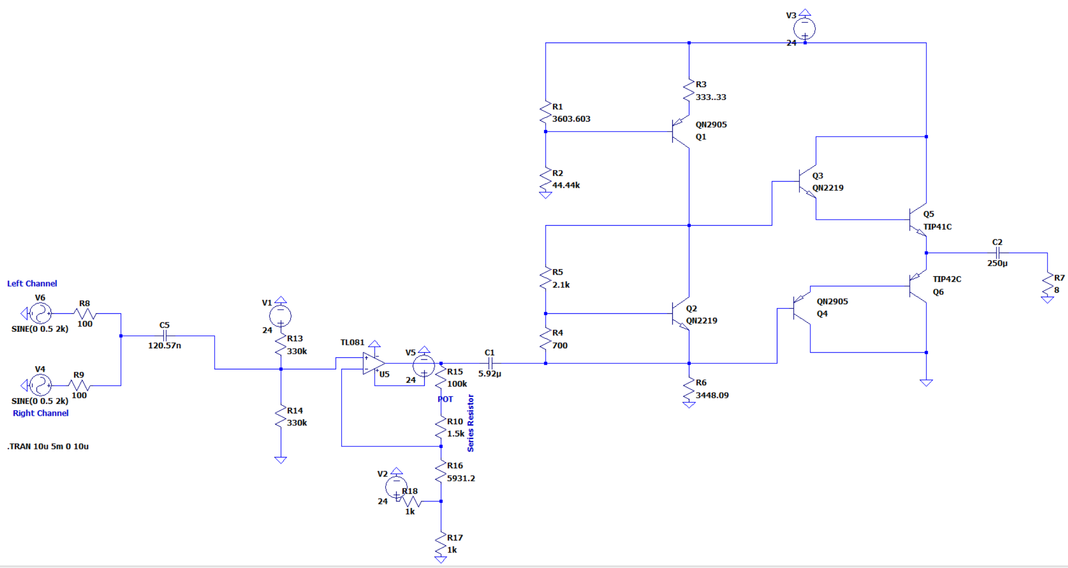

Figure 16: Full Schematic of Audio Amplifier (LTSPICE)

The output voltage of the LTSPICE audio amplifier can be seen below in figure 17.

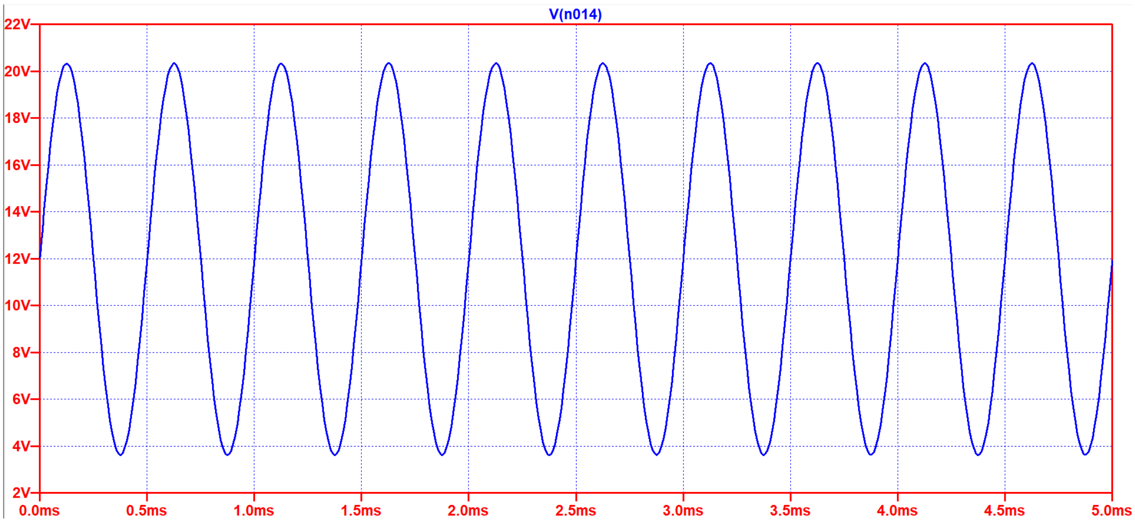

Figure 17: Output Voltage of LTSPICE (Full Circuit)

As can be seen above, the audio amplifier also works on LTSPICE. The output is the same as on NGSPICE, only the difference is that the output swings around 12V, instead of 0 V. The reason for this, is because the pre-amplifier, only uses a single input voltage of 24V, thus the pre-amplifier was designed to swing around 12V.

NGSPICE Code:

```
Electronics 315  Practical 3 - Xari Mouchtouris 23795824
Vin 1 0 AC 1 sin(0 9.12 2k)
C1 1 Vx1 5.92u
Rbias 2 0 3448.09
Vx Vx1 2 0
*Vbe-multiplier Design
R1 4 3 2.1k
R2 3 2 700
Q6 4 3 2 QN2219
* Current source Design
R3 7 6 3603.603
R4 6 0 44.44k
RE 7 5 333.33
Q5 4 6 5 QN2905
Vcc 7 0 24
*Power Transistors
Q1 7 4 8 QN2219
Q2 7 8 10 TIP41C
Q4 0 2 9 QN2905
Q3 0 9 10 TIP42C
C2 10 11 248.68u
RL 11 0 8
.MODEL QN2219 NPN (IS=15.2F NF=1 BF=105 VAF=98.5 IKF=.5
+ ISE=8.2P NE=2 BR=4 NR=1 VAR=20 IKR=.225 RE=.373 RB=1.49
+ RC=.149 XTB=1.5 CJE=35.5P CJC=12.2P TF=500P TR=85N)
* Motorola 30 Volt .8 Amp 300 MHz SiNPN Transistor 04-11-1991
*PINOUT TO-39 3 2 1
.MODEL QN2905 PNP (IS=3.81E-13 NF=1.0 BF=260 VAF=114
+ IKF=3.6E-01 ISE=5.85E-11 NE=2.0 BR=4 NR=1.0 VAR=20
+ XTB=1.5 RE=1.2E+00 RB=4.8E+00 RC=4.8E-01
+ CJE=4.6E-11 CJC=1.9E-11 TF=7.9E-10 TR=2.1E-08)
* 40 Volt 0.60 Amp 200 MHz SiPNP Transistor 08-06-1990
.model TIP41C npn Is=457.5f Xti=3 Eg=1.11 Vaf=50 Bf=156.7 Ise=1.346p Ne=1.34
+ Ikf=3.296 Nc=.5961 Xtb=2.2 Br=7.639 Isc=604.1f Nc=2.168
+ Ikr=8.131m Rc=91.29m Cjc=278.7p Mjc=.385 Vjc=.75 Fc=.5 Cje=433p
+ Mje=.5 Vje=.75 Tr=1.412u Tf=37.34n Itf=35.68 Xtf=1.163 Vtf=10 Rb=.1
.model TIP42C pnp Is=66.19f Xti=3 Eg=1.11 Vaf=100 Bf=137.6 Ise=862.2f
+ Ne=1.481 Ikf=1.642 Nc=.5695 Xtb=2 Br=5.88 Isc=273.5f Nc=1.24
+ Ikr=3.555 Rc=79.39m Cjc=870.4p Mjc=.6481 Vjc=.75 Fc=.5
+ Cje=390.1p Mje=.4343 Vje=.75 Tr=235.4n Tf=23.21n Itf=71.33
+ Xtf=5.982 Vtf=10 Rb=.1
.TEMP 39.6
.TRAN 10u 5m 0 10u
*.AC DEC 100 010 10meg
.control
run
* Transient analysis
*plot v(1)
*plot i(Vin)
plot v(11)
*Frequency analysis
*plot vdb(11)
.endc
.end
```

Figure 18: NGSPICE Code for Audio Amplifier Analysis

Python Code:

All my hand calculations for the audio amplifier design were verified by writing python code to calculate all the values. See the code below and the results that the python code gives:

```python
# Practical 3 - 23795824
import numpy as np
PL = 5       # maximum of 5 W undistorted power
RL = 8       # 8 Ohm Load (SPEAKER)
VRE = 1      # 1V voltage drop over VRE (This is a choice) [V]
VBE_on = 0.7 # VBE_on value [V]
VEB_on = 0.7 # VEB_on value [V]
VEC5_sat = 0.2 # VEC5(sat) value [V]
# Calculations for Vcc
Vop = np.sqrt(2*PL*RL)
print(f'Vop: {Vop} V')
Vx_min = 2*VEB_on
print(f'Vx_min: {Vx_min} V')
Vx_no_input = Vx_min + Vop
print(f'Vx_no_input: {Vx_no_input} V')
Vx_max = Vx_no_input + Vop
print(f'Vx_max: {Vx_max} V')
Vy_max = Vx_max + 2*VBE_on
print(f'Vy_max: {Vy_max} V')
VCC = Vy_max + VEC5_sat + VRE
print(f'VCC: {VCC} V')
VCC = 24   # Choose 24V for VCC
print(f'VCC (rounded): {VCC} V')
# Design Current Source
IL_max = Vop/RL
IE2_max = IL_max
print(f'IL_max = IE2_max: {IE2_max} A')
Beta2 = 50    # From TIP41C datasheet (Beta1 = 50)
Beta1 = 100   # Choose Beta1 = 100
IE1_max = IE2_max/(Beta2 + 1)
IB2_max = IE1_max
print(f'IB2_max = IE1_max: {IE1_max} A')
IB1_max = IE1_max/(Beta1 + 1)
print(f'IB1_max: {IB1_max} A')
Ic6_min = 0.0015   # Choose this current [A]
Ic6_max = Ic6_min + IB1_max
print(f'Ic6_max: {Ic6_max} A')
IR1 = 0.001 # Choose IR1 current (IR1 >> IB6_max) [A]
IC5 = IR1 + Ic6_min + IB1_max
print(f'Ic5: {IC5} A')
IC5 = 0.003  # Choose 3mA for IC5 instead of calculated value [A]
RE = VRE/IC5
print(f'RE: {RE} ohms')
```

(a) Code 1 for calculations

```python
# B-Stability Design
Beta5 = 100
Rth = 0.1*Beta5*RE
print(f'Rth: {Rth} ohms')
Vth = VCC - VRE - VBE_on - IC5/Beta5*Rth
print(f'Vth: {Vth} V')
R3 = Rth*VCC/Vth
print(f'R3: {R3} ohms')
R4 = (Rth*R3)/(R3 - Rth)
print(f'R4: {R4} ohms')
# Design of the Vbe multiplier
R1_R2 = 4*VBE_on/IR1
print(f'R1 + R2 = {R1_R2} ohms')
# Coefficients matrix
A = np.array([[1, 1],
              [1, -3]])
# Constants matrix
B = np.array([R1_R2, 0])
# Solve the system of equations
X = np.linalg.solve(A, B)
# X[0] will be R1 and X[1] will be R2
R1, R2 = X[0], X[1]
print(f"R1 = {R1} ohms \nR2 = {R2} ohms")
# Design Rbias
Vbias = Vop + 2*VBE_on
print(f'Vbias: {Vbias} V')
Rbias = Vbias/IC5
print(f'Rbias {Rbias} ohms')
# Transistor power ratings
PQ5_max = IC5*(VCC - 1 - 4*VBE_on)
print(f'PQ5_max {PQ5_max} W')
PQ6_max = (Ic6_max + IR1)*4*VBE_on
print(f'PQ6_max {PQ6_max} W')
PQ2_max = np.square(VCC)/(4*np.square((np.pi))*RL)
print(f'PQ2_max {PQ2_max} W')
PQ1_max = np.square(VCC)/(4*np.square((np.pi))*(Beta2 + 1)*RL)
print(f'PQ1_max {PQ1_max} W')
# Resistor power ratings
PR3_max = (np.square(VRE + VBE_on))/R3
print(f'PR3_max {PR3_max} W')
PR4_max = (np.square(VCC - VRE - VBE_on))/R4
```

(b) Code 2 for calculations

Figure 19: Python Code for calculations

```python
print(f'PR4_max {PR4_max} W')
PRE_max = (np.square(VRE))/RE
print(f'PRE_max {PRE_max} W')
PR1_max = (np.square(3*VBE_on))/R1
print(f'PR1_max {PR1_max} W')
PR2_max = (np.square(1*VBE_on))/R2
print(f'PR2_max {PR2_max} W')
PR_bias = (np.square(VCC - VRE - VEC5_sat - 4*VBE_on))/Rbias
print(f'PR_bias {PR_bias} W')
# Heat sink design
Theta_jc = 1.92 # Thermal resistance from datasheet [C/W]
Theta_ja = 62.5 # Thermal resistance from datasheet [C/W]
Theta_ca = 6    # Thermal resistance from datasheet [C/W]
print(f'Theta_jc {Theta_jc} C/W')
print(f'Theta_ja {Theta_ja} C/W')
Ta = 25 # Ambient temperature [Celsius]
PD = PQ2_max
Tj = PD*Theta_ja + Ta
print(f'Tj (without sink) Close to 150 Celsius: {Tj} Celsius')
Tj = PD*(Theta_jc + Theta_ca) + Ta
print(f'Tj (with sink) Far from 150 Celsius: {Tj} Celsius')
# Design of Low-Frequency Cut-off
f_low = 80 # Frequency [Hz]
VIN_SPICE = 1 # Input voltage [V]
IIN_SPICE = 0.000297826 # Input current [A]
RIN = VIN_SPICE/IIN_SPICE
print(f'RIN: {RIN} ohms')
C2 = 1/(2*np.pi*RL*f_low)
print(f'C2: {C2} F')
f1 = f_low/10  # Decade Lower
C1 = 1/(2*np.pi*RIN*f1)
print(f'C1: {C1} F')
# Design of Pre-Amp
Gain = 9.43
print(f'Gain_pre: {Gain} V/V')
Rx = 330000  # Choose Rx for preamp as 330 KOhm
Ry = Rx
print(f'Ry_pre = Rx_pre: {Ry} Ohm')
R1 = 1000000
print(f'R1_pre: {R1} Ohm')
R2 = 11862.40
print(f'R2_pre: {R2} Ohm')
fy = f1 # Fy for preamp
Rinput = 1/(1/Rx + 1/Ry)
print(f'Rinput_pre: {Rinput} Ohm')
Cx = 1/(2*np.pi*Rinput*fy)
print(f'Cx_pre: {Cx} F')
```

Figure 20: Python Code Calculations Code 3

The results of the python code can be seen below in figure 21

```
Vop: 8.94427190999916 V
Vx_min: 1.4 V
Vx_no_input: 10.34427190999916 V
Vx_max: 19.288543819998317 V
Vy_max: 20.688543819998316 V
VCC: 21.888543819998315 V
VCC (rounded): 24 V
IL_max = IE2_max: 1.118033988749895 A
IB2_max = IE1_max: 0.021922235073527353 A
IB1_max: 0.00021705183241116l9 A
Ic6_max: 0.00171705183241116 2 A
Ic5: 0.00271705183241116 2 A
RE: 333.3333333333333 ohms
Rth: 3333.333333333333 ohms
Vth: 22.2 V
R3: 3603.603603603604 ohms
R4: 44444.44444444434 ohms
R1 + R2 = 2799.9999999999995 ohms
R1 = 2099.9999999999995 ohms
R2 = 699.9999999999999 ohms
Vbias: 10.34427190999916 V
Rbias 3448.0906366663867 ohms
PQ5_max 0.0606 W
PQ6_max 0.007607745130751253 W
PQ2_max 1.82378130556208 W
PQ1_max 0.03576041775611921 W
PR3_max 0.0008019749999999998 W
PR4_max 0.011189025000000026 W
PRE_max 0.003 W
PR1_max 0.0021 W
PR2_max 0.0007 W
PR_bias 0.11600623131726025 W
Theta_jc 1.92 C/W
Theta_ja 62.5 C/W
Tj (without sink) Close to 150 Celsius: 138.98633159763 Celsius
Tj (with sink) Far from 150 Celsius: 39.44434794005167 Celsius
RIN: 3357.665213916851 ohms
C2: 0.00024867959858108647 F
C1: 5.925060010160853e-06 F
Gain_pre: 9.43 V/V
Ry_pre = Rx_pre: 330000 Ohm
R1_pre: 1000000 Ohm
R2_pre: 11862.4 Ohm
Rinput_pre: 165000.0 Ohm
Cx_pre: 1.205719265847692e-07 F
```

Figure 21: Python Code Results for audio amp