ELE 214
Electronics Laboratory – I

Multistage Amplifier Design

Term Project – Spring 2021

STUDENT NAME: Mehmet Nurettin GÜNDÜZ

STUDENT ID: 21990887

TABLE OF CONTENTS

LIST OF FIGURES

The link for the fabrication video : https://youtu.be/BBcsB84B7sA

ABSTRACT

In this project, our goal is to design multistage audio amplifier. The input is an audio signal and at the output, the audio signal is amplified. The multistage amplifier should work between specific frequencies. First, we defined the problem and technical terms to clearify the project. Considering the requirements and the limits, we took the components in appropriate way. The numerical calculations helped us to determine the values of the components. Then, we simulate the circuit and predict the behavior of signal in the simulation programme. In this case, we used LTSpice as we did during the term. After testing the circuit several times, we specified the convenient configuration in order to obtain desired Av in each case. In the main part, we utilized from one BJT and one JFET transistor and in the middle of them, we put bandpass filter so that only the certain signals between low and high frequency limits are allowed to pass. Finally, the theoretical design is tested by setting up the circuit and getting the output at the speakers. The multistage audio amplifier is an useful project which drives us to get intended audio signal. In other words; by changing the frequencies, we can control the thickness of the sound. The input audio signal is also amplified, say strengthened, and released to the environment via speakers. This system is being used everywhere in our daily life. For example, we can adjust the voice according to the fit of our ears in the mobile phones or provide stronger sound from the guitars via speakers to the audience. The more detailed and functional system can be produced by adding necessary components or configurations.

INTRODUCTION

The problem is to produce multistage audio amplifier which takes the audio signal as an input, amplifies it by sifting out the inhibited frequencies and releases the sound. The working frequencies are specified according to the last two digits of the student ID number. The project is tested in LTSpice platform and the simulations are provided in the scope of the report. In the steps of the solution, the necessary terminology is explained and the theoretical calculations are shown. The information in the report is given with respect to project guideline. The requirements of the project are determined after finding the proper numerical values for the circuit components.

STATEMENT OF OBJECTIVE

1.PURPOSE: The aim of the project is to design a multistage audio amplifier by using cascade circuits with transistors. Besides, the system should work between certain frequencies specified with respect to student ID number. The project is needed for different cases including the sound amplification, frequency selection and volume control.

Definition of some important terms:

1.a. AMPLIFIER: A electrical device used for controling the input signal and providing stronger output signal. A voltage or a current can be amplified at desired ratio by adjusting the components in appropriate way. These devices are widely used in electronic circuits because in general, the input signal requires several operations before transferred at the output. The transistors are the main devices used in the amplifiers at very small sizes. The input signal is strengthened with the help of the power supply by minimizing the lost at the output.

1.b. BUFFER STAGE OF AN AMPLIFIER: In this stage, the elimination of the input and the other stage effects is done. In other words, the load current or voltage effects are minimi-mized as much as possible to purify the signal. By doing so, the unconformities between the circuit elements are avoided in order to deal with unique signal in the following stages.

1.c. EQUALIZER STAGE OF AN AMPLIFIER: The case in which the specified frequency ranges are obtained by using frequency selective circuits called filters. The loudness is controlled wiht respect to lower and higher cut-off frequencies so that the circuit can behave only between these frequencies. For example, a bandpass filter is constructed by two consecutive RC circuits in which the numerical values for the components are picked up by satisfying the equations f(low)=1/(2*pi*R*C) and f(high)=1/(2*pi*R*C).

1.d. GAIN STAGE OF AN AMPLIFIER: It is the stage in which the input signal is adjusted to obtain desired output signal without any distortion. Generally, this term is used in audio engineering by mentioning the dB of the output signal. The signal is taken between certain frequencies and after that, it is released by reinforcing the final signal in convenient ratio, which is called ‘gain’.

1.e. SINGLE STAGE AMPLIFIER: A amplifier which consists of one input and one output. The signal is strengthened with the help of the transistor. Since it only includes one stage, generally it is preferred to use with other amplifiers by connecting them in cascade manner. Understanding the behavior of the single stage amplifier is easy because it doesn’t require complicated mathematical analysis. The transistor and the other assistant components are adjusted in the way that the desired gain is obtained at the output.

1.f. MULTISTAGE AMPLIFIER: A amplifer which is formed by combining the several stages in single manner. The stages of the amplifier are cascade; the output of the one stage is the oinput of the next stage. The voltage or the current is controlled in overall circuit by setting up the convenient transistor configurations. The connection of the components in each stage is different depending on the purpose of the stage or the desired output signal. In general, a multistage amplifier is more widely used than the single stage amplifier because it provides more detailed operations on the signal.

1.g. THE ROLE OF CAPACITOR IN AMPLIFIER DESIGN: The capacitors are widely used in electronic circuits. Depending on the configuration, the aim of the capacitor usage can vary in terms of the connection or the magnitude. The capacitors are blocking the signal in DC analysis and transfering the signal in AC analysis. The main use of capacitors in the circuits is to determine the signal flow. In other words, the capacitors are acting as a filter by selecting the desired AC signal to pass and by preventing the unwanted DC signals. The name of the capacitor can differ according to its connection in the circuit; for example, By-Pass Capacitor or Coupling Capacitor. The capacitors are considered as a short circuit in AC analysis and a open circuit in DC analysis.

1.h. DIFFERENCES BETWEEN MOSFET, JFET AND BJT: The main difference is the way they are controlled. The FETs are voltage-controlled devices and the BJTs are current-controlled devices. The input resistance for the FETs is high while the input resistance for the BJTs is low. In general, the size of BJTs is bigger compared to the FETs. The BJTs generate higher noises than the FETs. The FETs are more sensitive devices than the BJT in terms of the handling requirements. The BJTs are easier to manufacture than the FETs; however, the BJTs are less popular than the FETs. The main difference between the JFET and MOSFET is their operation modes. The JFETs are used in depletion mode while the MOSFETs are used in both enhancement and depletion mode. The input impedance of the MOSFET is higher than the one for the JFET. The JFETs are easier to manufacture than the MOSFETs. Also, the MOSFETs are more sensitive devices while the JFETs are more robust devices.

2. DESCRIPTION:

2.A. PROBLEM: The problem is being solved by dividing into steps. Our input is an audio signal and at the output, we wish to get amplified audio signal. Before starting to solve the problem, it is important to know the necessary terminology in order to increase the understanding of the project. The first step is to do the theoretical calculations and make a decision about the features of the components. (magnitude, type, limits, size, exc.) Choosing the appropriate components let us achieve to the goal in each step. The theoretical calculations are the set of tests in which we are utilized from the characteristic equations of the components or the laws determining the relationship between the components. Solving the circuit means that we know the voltage and the current of the each element. After predicting the behavior of the signal, we test our assumption in the simulation programme. Meaning that in the second step, we set up the circuit on the simulation programme and look to te output waveform whether they are consistent with our calculations. If the errors or the distortions are occured, the necessary adjustments are done until we achieve to the goal in the related step. Changing the numerical values of the components, using another configuration or picking up different type of component might be the some logical solutions depend on the type of the problem. In the final step, we are trying to get the contol over an audio signal in real case. Setting up the circuit with convenient component models, we test the our amplifier by analyzing the changes on the sound at the speakers.

2.B. SYSTEM: The system consists of 5 stages and we are asked to design 3 of them. First, the input signal from the source is taken. In the second stage, the signal is buffed via using the JFET; meaning that the effects of the other components on the signal is eliminated. After the elimination, we are selecting the frequency of the signal. Therefore, a bandpass filter, which is combination of the high and low pass filters, is used in the third stage. In the fourth stage, the signal is transmitted to the gain stage. Here, the signal is amplified at the desired ratio with the help of the BJT transistor. Finally, the signal is released via speakers and the system is tested if it works well. In this case, we are given the circuit design to connect the speakers in suitable way. Basically, the concept is simple and the behavior of system is understandable by looking to the definitions and the calculations. The design is tested in LTSpice platform to predict the behavior of the system and come up with the proper configuration in each case. Before attempting to set up the circuit experimentally, we should ensure that the components with appropriate magnitudes are chosen and the theoretical calculations are make sense according to the input-output relationship in each stage.

3. INFORMATION ABOUT THE TRANSISTORS:

3.A. Reasons for selection:

- We are familiar with these transistor because we have used them in our experiments before.
- The numerical limits of the transistors are satisfying the calculations and the voltage-current interval where we work.
- The handling of JFET and BJT is easier than the handling of the MOSFET.
- The types of transistors are already available from our previous experiments.
- Their sizes are suitable for the design with the other components.

- Since we usually deal with n-channel or npn type transistors in daily life, it is logical to work with n-type transistors for providing clear comprehension.

3.B. Type numbers of the transistors:

JFET: BF245C N-channel transistor **BJT**: 2N2222 NPN transistor

3.C. Transistor parameters:

There are several parameters of the transistors and we can check them by looking to their datasheet. Several companies may produce same type of transistor with slight differences in the transistor parameters. The numerical values of the parameters determine whether the transistor is suitable for the design or not. Also, the parameters of the transistor specify the working conditions of the device. For example, the transistor may not be able to work after certain temperature or it may break down if we apply high voltage across its terminals. The set of specifications of the transistors are indicated in the datasheet and these datasheets are easy to attain on the different websites. Some important transistor parameters can be listed as follows:

for JFET:

- **Vgs:** The gate-source voltage which takes (-) values in the n-channel transistors.
- **Idss:** The maximum current flowing through the drain to the source when the Vgs=0.
- **Vp:** The pinch-off voltage between the Ohmic and Saturation Regions. In other words, it is the maximum voltage between the gate and the source when the drain current is zero.
- **g(m):** The transconductance is the constant rate of change in the drain current to the voltage between the gate and the source.
- **Av:** The voltage gain is the ratio between the output voltage and the input voltage.
- **r(d):** The dynamic output resistance is the rate of change of the drain-source voltage to the drain-source current.

for BJT:

- **Ic:** The current flowing through the collector and generally measured in milliamps.
- **Ib**: The current flowing through the base and generally measured in microamps.
- **Ie:** The current flowing through the emitter and generally measured in milliamps Also, it is the algebraic sum of the Ib and Ic. Approximately, Ic=Ie.
- **Vbe:** The treshold voltage between the base and the emitter which needs to be exceeded in order to run the transistor
- **a:** The alpha is the common-base current gain. Mathematically, it is Ic / Ie which is approximately equal to the 1.
- **b:** The beta is the common-emitter current gain. Mathematically, it is Ic / Ib and in general, it takes the values in terms of the hundreds.
- **h(i):** The input impedance in SSAC analysis.
- **h(f):** The forward transfer current ratio is expressed by Io / Ii.
- **h(o):** The output conductance whose inverse give the dynamic output resistance. (r(d)=1/h(o))
- **Av:** The voltage gain is the ratio between the output voltage and the input voltage.

4. COMMENTS ON THE SIGNAL STAGE AUDIO AMPLIFIER

In the previous sections, we have already mentioned the requirements for the design.The signal should be eliminated from effects of the other components, tested whether if it is between the certain frequencies, amplified at the desired rate and released via the audio device. In this whole process, we require cascade connection of two main transistors and a bandpass filter. Also, the necessary resistor should be placed in the project in order to obtain undistorted output signal. However, we cannot handle the problem if we would use a single stage audio amplifier because it consists of only one transistor. It means that we are allowed to do only one specific operation on the signal which is not sufficient to get desired output waveform. In general, we encounter the multistage amplifier configurations in the technological devices and the single stage amplifier configuration is usually used to show the behavior of the design in each stage individually. The multistage amplifier requires more complex analysis than the one for the single stage amplifier because the behavior of the components in the multistage amplifier may affect each other in terms of delay, frequencies, phase shift, voltage or current. For example, we worked with the single stage amplifier designs during the lecture in order to understand the concept in the configuration more clearly. The derivation of the equations is simple in the single stage amplifier because there aren't any other components to affect the parameters and outputs of the amplifier. The components in the multistage audio amplifier are said to be connected in cascade, meaning that the output of the one stage is the input of the another stage. It is important to note that the number of stages in the amplifier design depend on the number of the transistors used. To conclude, one who wants to design an audio amplifier should use multistage design instead of single stage because a single stage audio amplifier is insufficient for the requirements.

ANALYTICAL CALCULATIONS AND RESULTS

Brief Information

In this part, we analyzed the stages seperately and focused on the goals step by step. The derivation of the convenient circuits have been obtained through consecutive trials. The formulas and the configurations are based on the lecture notes given in the ELE230. Since we had no chance to build the circuit again and again with various components, we had to precisely be sure in order to work with ideal components in terms of magnitudes and types. Before analyzing the complete circuit, the goals in each stage should be reached. The calculations done by hand are shown with the picture and each component is described briefly. The order followed on the report is the same as the requirements indicated on the guideline.

1. BUFFER STAGE

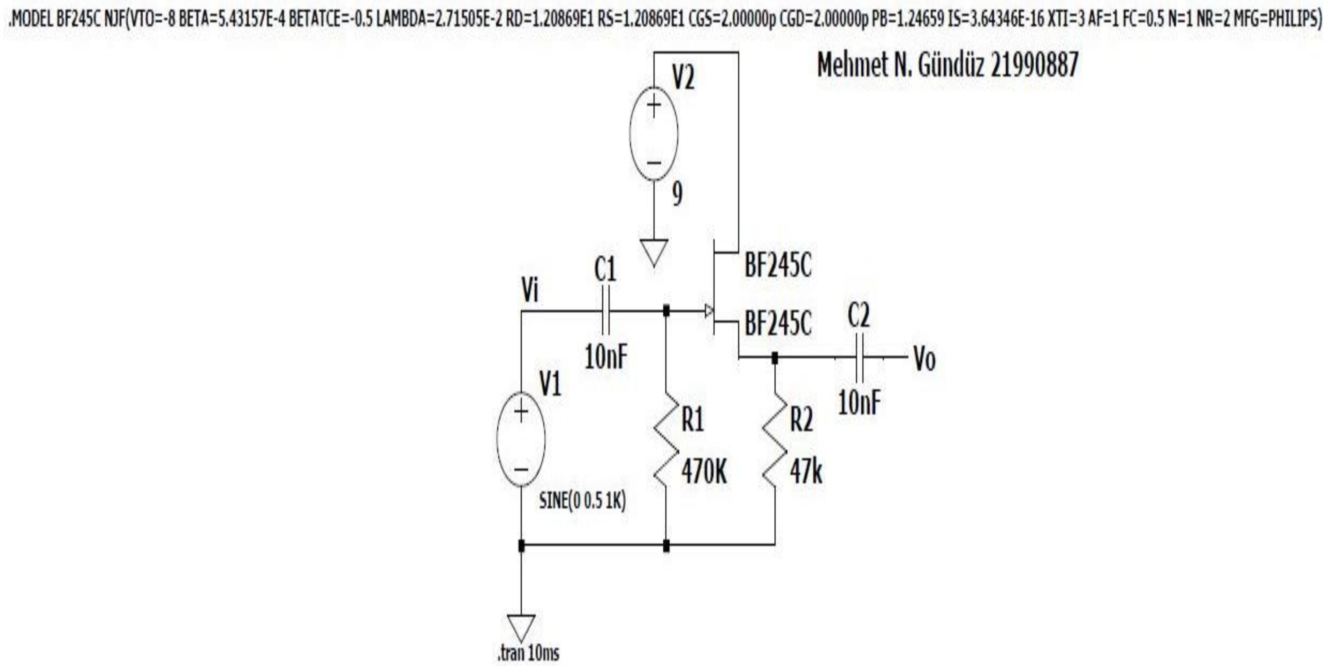


FIGURE 1.1.A: The circuit for the Buffer Stage

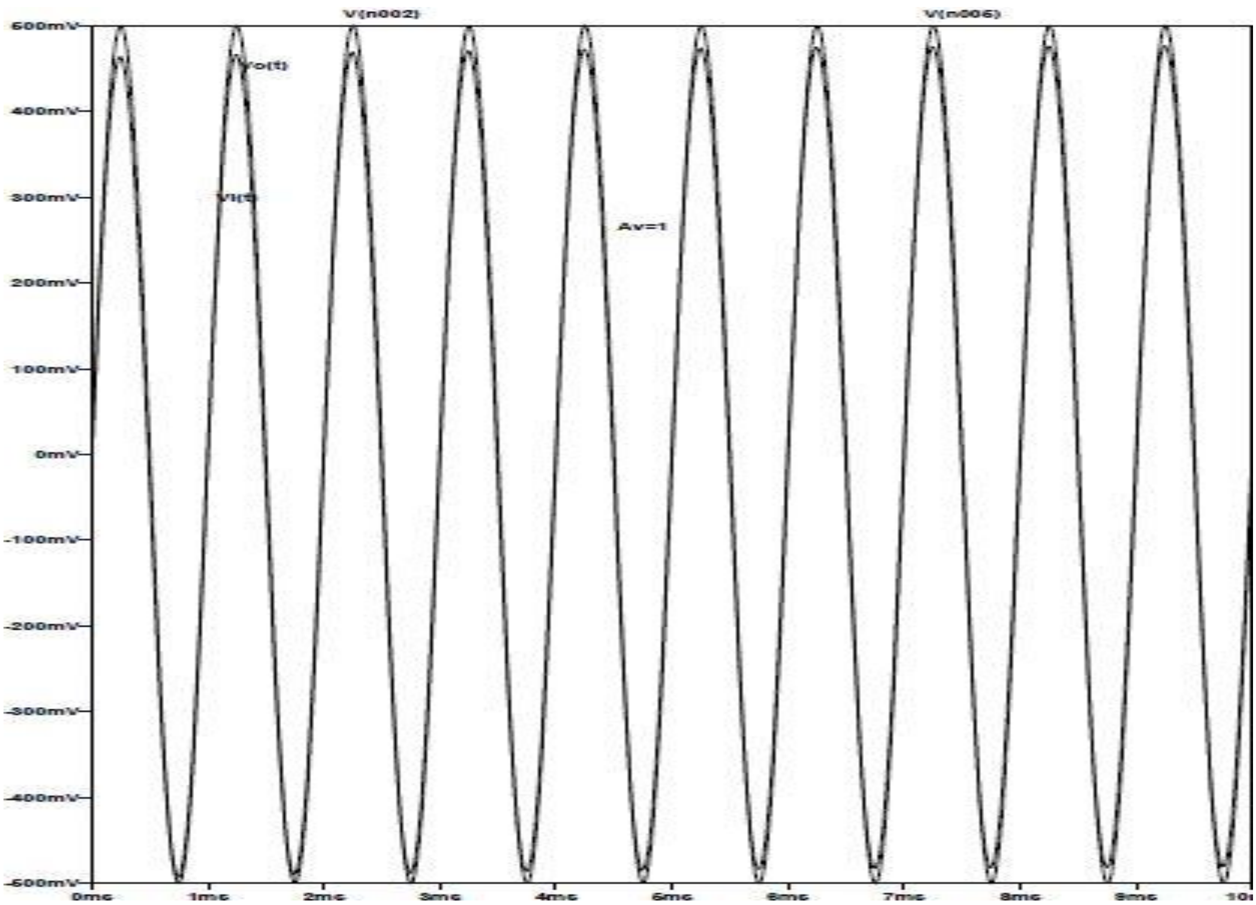


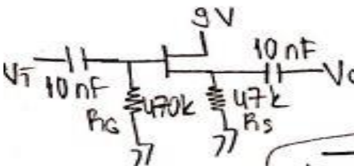
FIGURE 1.1.B: Vo-Vi vs t waveform for the Buffer Stage

1.a DESCRIPTION: We are expected to use JFET in order to get Av=1. This stage eliminates the effects of the other components on the audio signal. In other words, the signal is purified and the input voltage is protected till the output with the use of the transistor. Utilizing from the SSAC analysis, we decided to use Source-Follower Configuration in which voltage gain is approximately equal to 1. It is important to note that in the LTSpice, we set the VTO of the BF245C to -8 because we want to work as close as possibe to real case. Besides, we can control the output current while the Av value is remaining the same. In this case, the current gain value is calculated as Av*(Ri/Rs).

1.b COMPONENTS:
Vi: The sinusoidal source is generating AC signals and the circuit is observed with the procedures of SSAC analysis.
Vdd: The DC voltage source is providing a certain amount of voltage in order to turn-on the transistor.
C1 and C2: The capacitors are eliminating the effect of the AC signal.in DC analysis. Also, they allow the AC signal to pass through a circuit in SSAC analysis. Hence, they are called "Coupling" capacitor.
BF245C (JFET): The voltage controlled transistor which is the main component of the circuit. It determines the behavior of the circuit with the voltages across its terminals. (Drain-Gate-Source)
R1: The input resistance is very high because we have to minimize the voltage loss on the Gate as much as possible. Consequently, the current on the resistor is low.
R2: The source resistance is relatively low compared to R1 because we want to maximize the voltage between the Drain-Source by providing a current on this path.

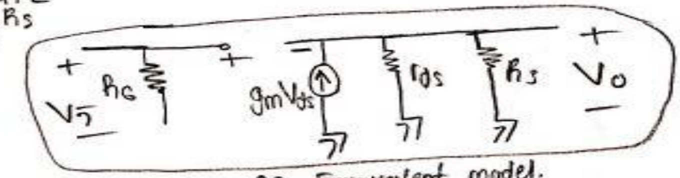
1.c CALCULATIONS:

BUFFER STAGE



$A_V=\frac{V_o}{V_i}=\frac{g_m(R_S||r_{ds})}{1+g_m(R_S||r_{ds})}\cong 1$, $A_I=\frac{R_G}{R_S}A_V=\frac{R_G}{R_S}$

For BF245C;
I_{DSS} min-max=12-25 mA
$V_P=-8$ V



SSAC Equivalent model.

★ $R_G>>R_S$

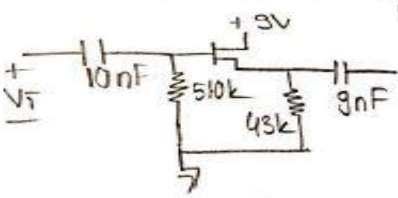
In DC Analysis, we can ignore R_G.

$V_{GS}=-I_D R_S \Rightarrow V_{GS}=-I_D(47k)$

$I_D=I_{DSS}\left(1-\frac{V_{GS}}{V_P}\right)^2 \Rightarrow I_D=I_{DSS}\left(1+\frac{V_{GS}}{8}\right)^2$

We couldn't determine the specific value of I_{DSS}. Actually, our main goal is to get Av=1 and we have done it by SSAC Analysis. As a general fact, we took $R_G>>R_S$.

Parameter Sweep



★ After changing the components by amount %10, we still have $A_V\cong 1$ because $R_G>>R_S$.

$V_{GS}=-I_D(43k)$

$I_D=I_{DSS}\left(1+\frac{V_{GS}}{8}\right)^2$

Same arguments are also true for this case. Our goal is reached without encountering any error at AV.

1.d PARAMETER SWEEP: After changing the values of the components by %10, we didn't notice any significant difference with the preceding case. As we can clearly see in the waveform, the Av=1 as long as we apply input voltage in such way that we obtain undistorted swing at the output. The effect of the capacitors remain same and still we have R1>>R2 which means that we can ignore the input resistance in DC analysis.

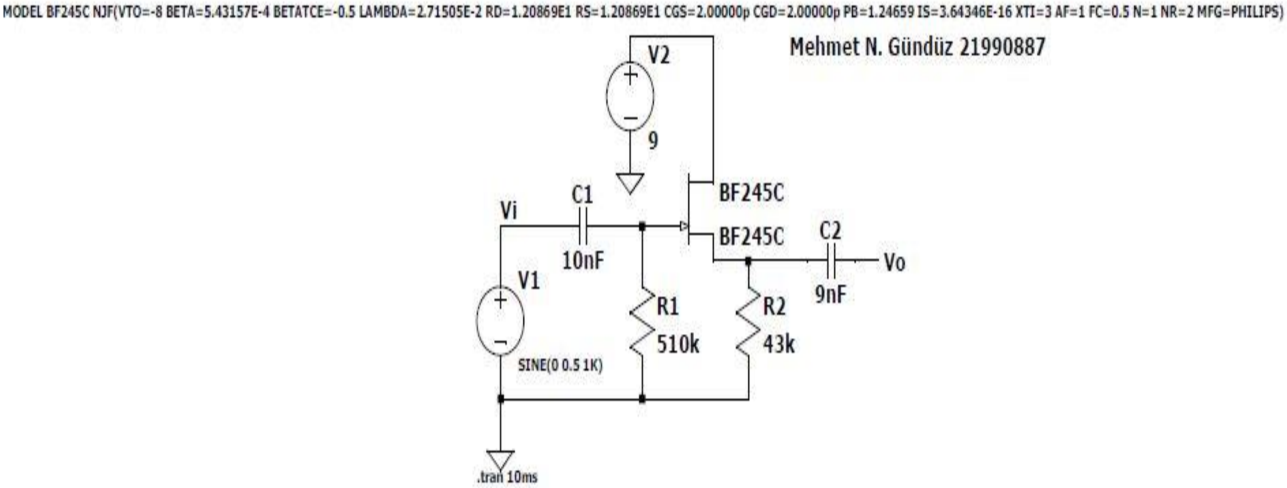


FIGURE 1.1.C: The circuit for the Buffer Stage (Parameter Sweep)

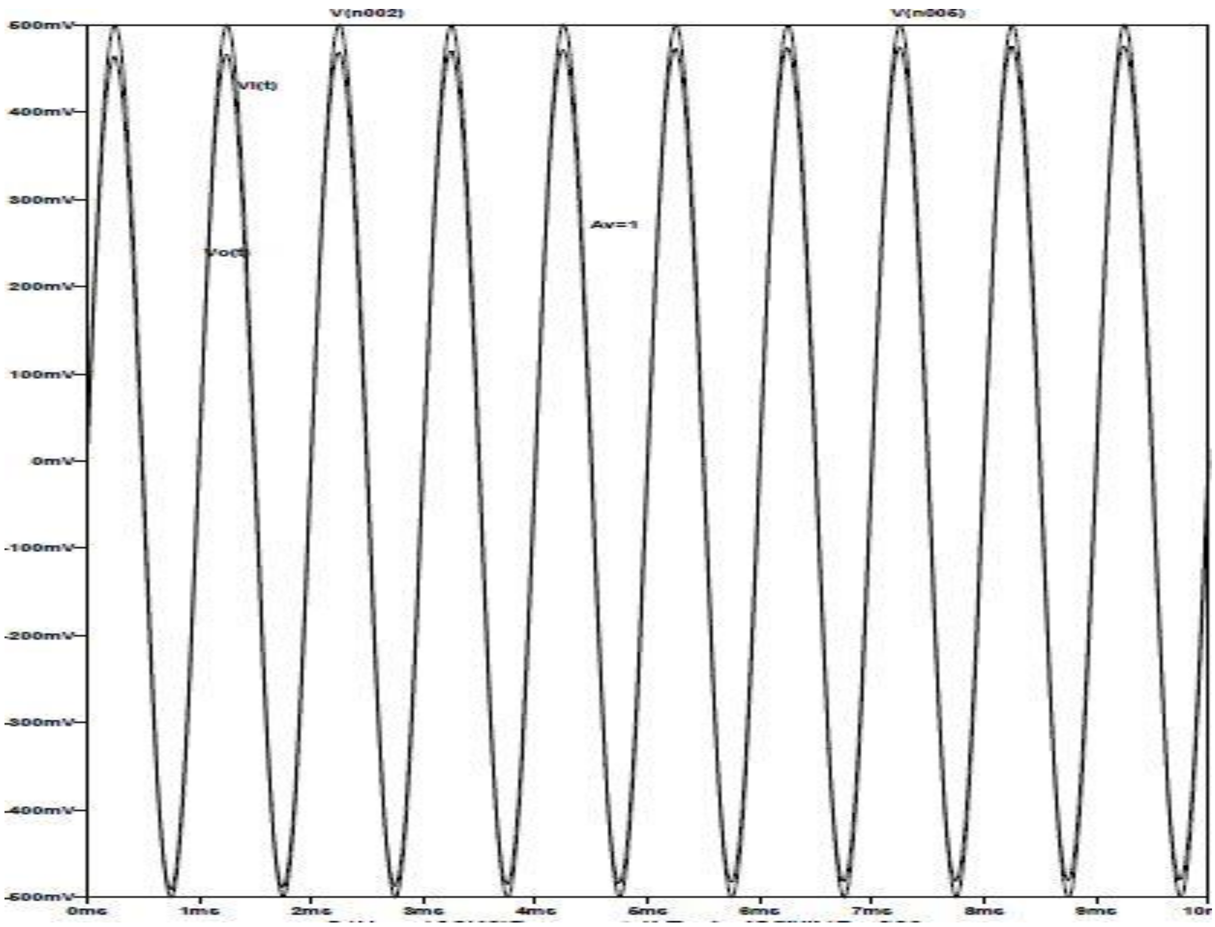


FIGURE 1.1.D: Vo-Vi vs t waveform for the Buffer Stage (Parameter Sweep)

2. EQUALIZER STAGE

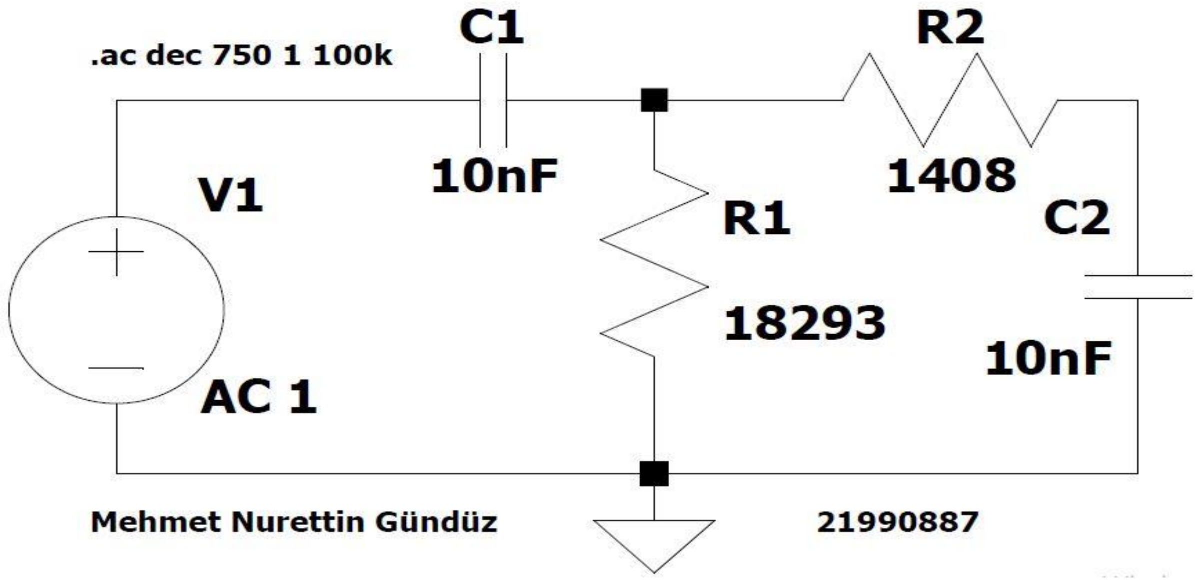


FIGURE 1.2.A: The circuit for the Equalizer Stage

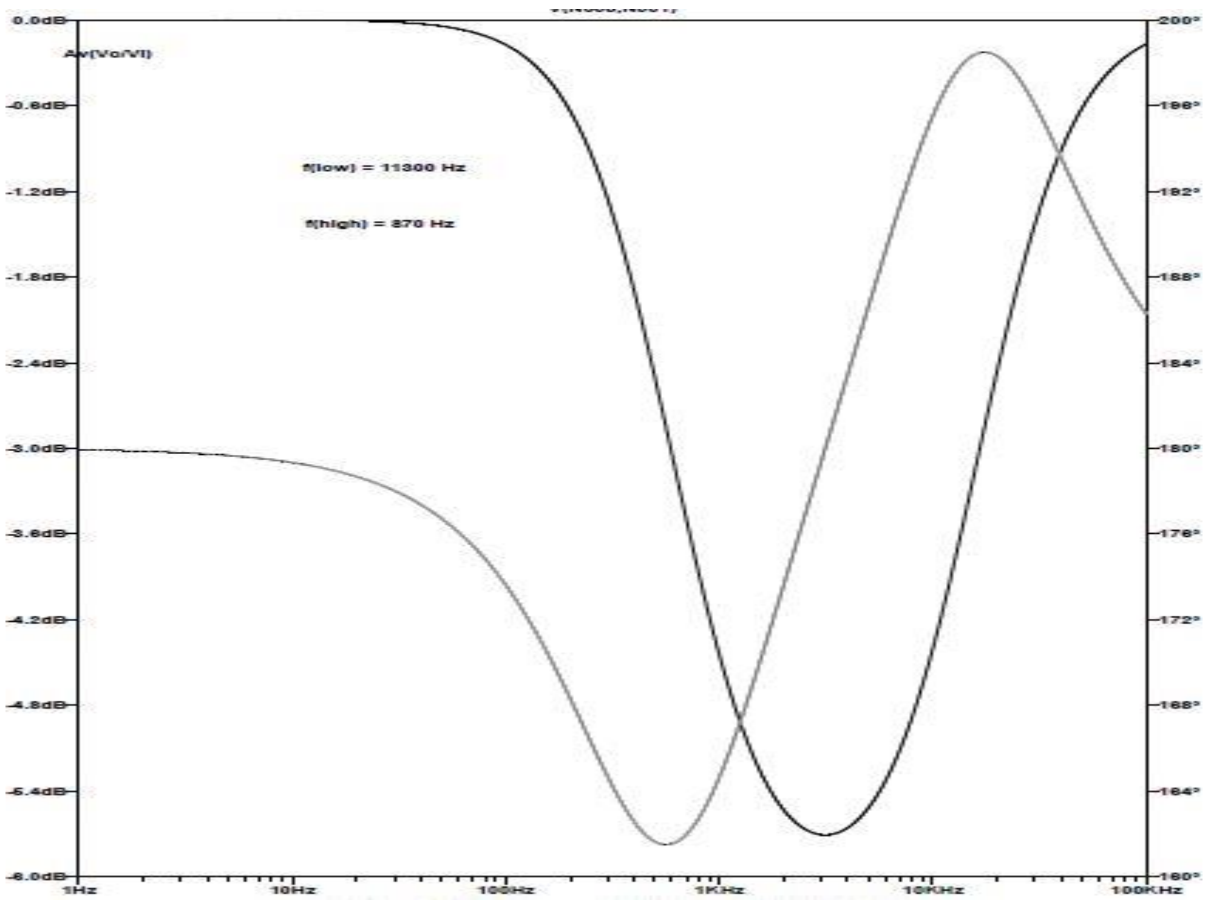
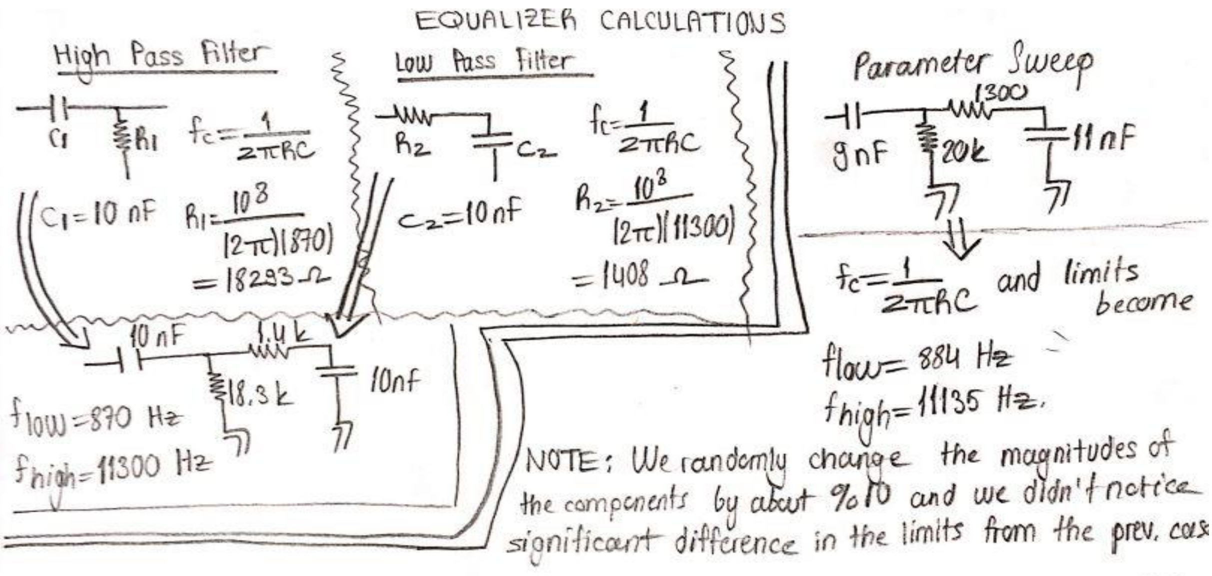


FIGURE 1.2.B: The Bode-Plot for the Equalizer Stage

2.a DESCRIPTION: In this stage, we are expected to design band pass filter which selects the signals between certain frequencies. The band pass filter is the cascade combination of the RC High and Low Pass filters. The band pass filter is a kind of passive filter and with the help of the potentiometer, we can change the frequencies transmitted via filter. The adjustable frequencies are determined with respect to student ID number. We can get information from the Bode Plot about the reaction of the components. It is important to note that in the bode-plots, we also provided the phase angles at the right vertical scaling.

2.b COMPONENTS:
Vi: The AC power supply is used to test the frequency interval whether is in the desired range. The signal is low and applied incrementally.
C1-R1: The circuit components of high pass filter which together determine the f(low) limit. It means that signals which have greater frequencies than f(low) are allowed to pass. Since 10 nF capacitors are commercially available, our calculations are based on this value.
C2-R2: The circuit components of low pass filter which together determine the f(high) limit. It means that signals which have lowerr frequencies than f(high) are allowed to pass. Since 10 nF capacitors are commercially available, our calculations are based on this value.

2.c CALCULATIONS:



2.d PARAMETER SWEEP: We changed the values of the components by about %10 and directly obtain different f(low) and f(high) limits. As the formula suggests, f=1/(2piRC), the frequency values depend on the components. The new frequency limits are slightly different than the previous ones. However, the change didn't affect the limits significantly and we still have a control on the boundaries because we are using potentiometer. It is important to note that in the simulations, we can either use step command on the resistors in order to obtain a potentiometer.

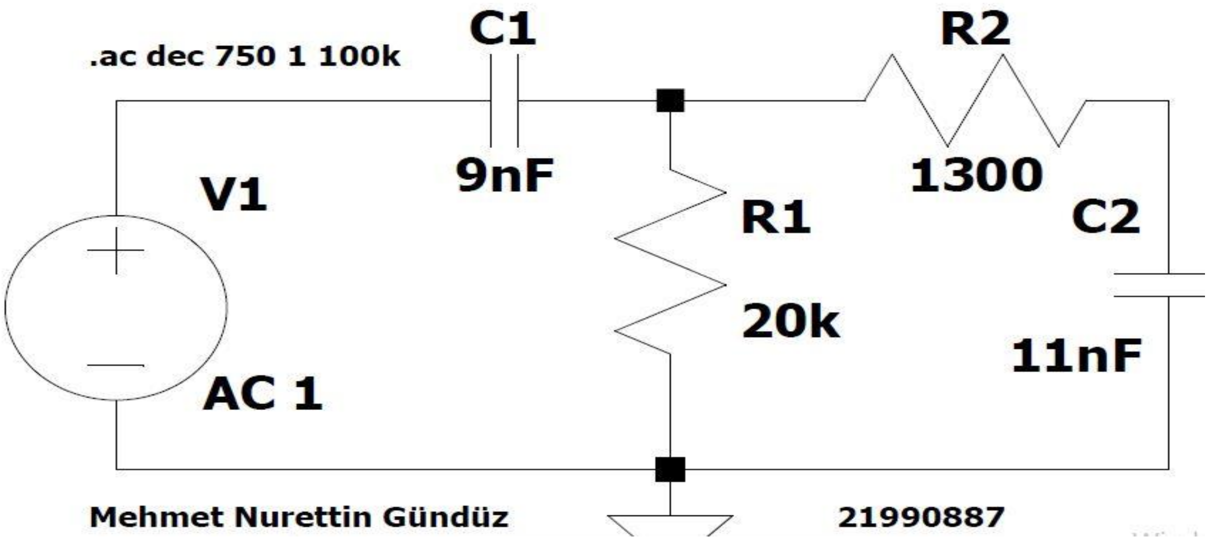


FIGURE 1.2.C: The circuit for the Equalizer Stage (Parameter Sweep)

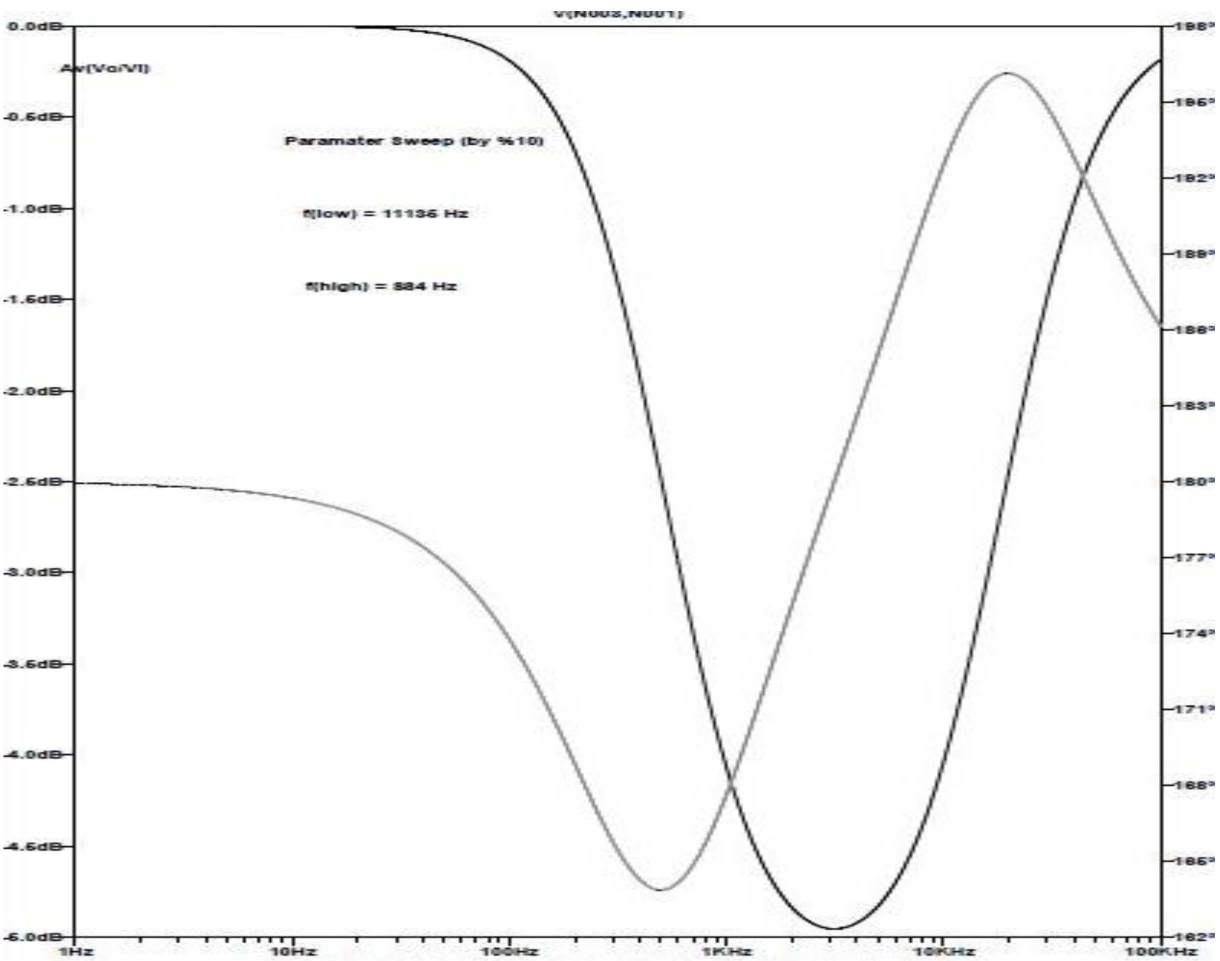


FIGURE 1.2.D: The Bode-Plot for the Equalizer Stage (Parameter Sweep)

3. GAIN STAGE

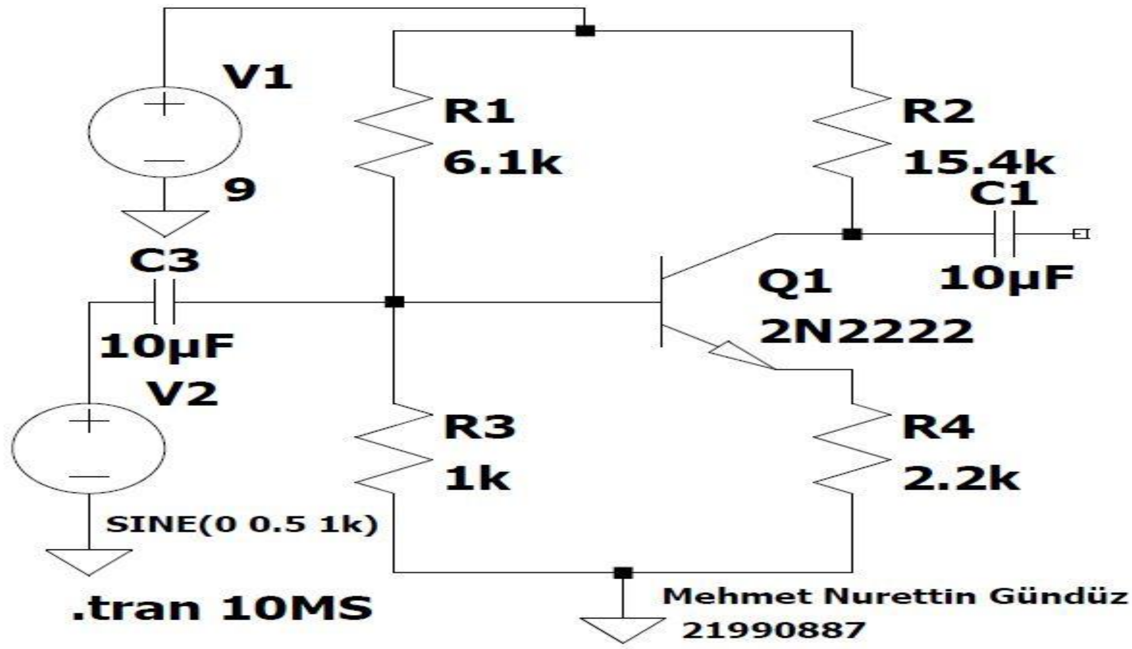


FIGURE 1.3.A: The circuit for the Gain Stage

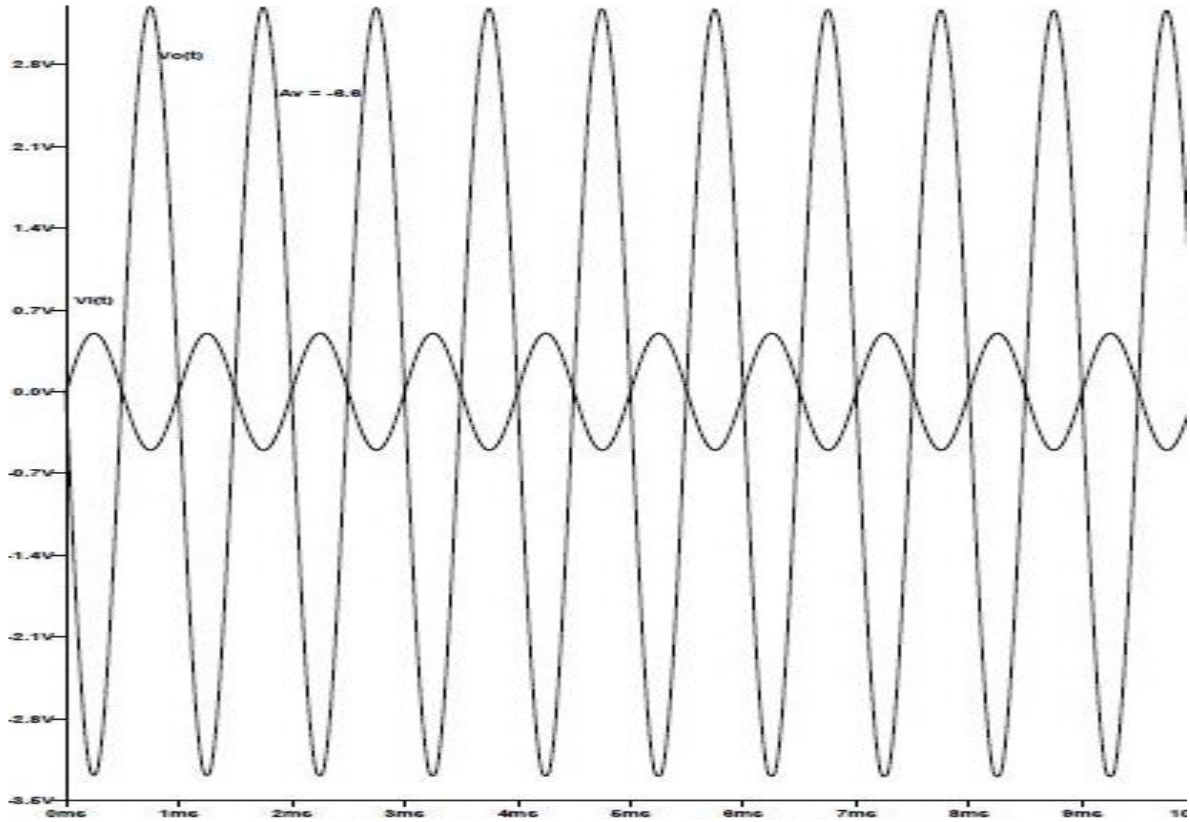


FIGURE 1.3.B: The Vo-Vi vs t waveform for the Gain Stage

3.a DESCRIPTION: Our goal is to design an amplifier with Av=-7 using BJT transistor. The BJT is a current controlled device and generally n-type BJTs are preferred. After analyzing the configurations given in the lecture notes, we found that Voltage-Divider Unbypassed Emitter Bias Configuration is the most convenient model because it has simple Av expression. Our Av value should be -7 but we got it as -6.6 in the simulation. Since we wanted to use components which we already have, we adjusted the system accordingly. The (–) sign in the Av expression states that there is a phase difference between the input and the output.

3.b COMPONENTS:
Vi: The voltage source providing small signals for the circuit. By doing so, we can test the stage whether we have desired gain at the output.
R1-R3: Together, they form the Rbb which is the base-equavalent resistance. We can control the base current by adjusting the base resistance.
R2(Rc): The collector resistance is component which is used to control the Ic current. Also, we adjust the voltage gain by changing the value of Rc.
R4(Re): The resistance which avoids the direct connection to the ground at the emitter terminal. Also, it affects the Av value as it is given in the expression.
C1-C3: The coupling capacitors are eliminating the effect of the AC signals. In DC analysis, they are considered as open circuit; so we can analyze the DC equavalent circuit easily.
Vcc: It is the DC Power Supply which is providing 9V voltage for our amplifier in the project.
BJT: The main component of this stage is a current controlled transistor which is called BJT. The parameters for a specific type of BJT are given in the datasheet and also found in the calculations. In this project, we preferred one of the most widely used model, 2N2222 BJT.

3.c CALCULATIONS:

GAIN STAGE CALCULATIONS

Mehmet Gündüz
21990387

$h_{ie} = \frac{\gamma}{I_{BQ}}$

★ I set up the circuit and found the values at the Q point as: (DC Equavalent)

$h_{fe} = \beta_{ac} = 200$ (for 2N2222)

$I_{BQ} = 1.41\,\mu A$

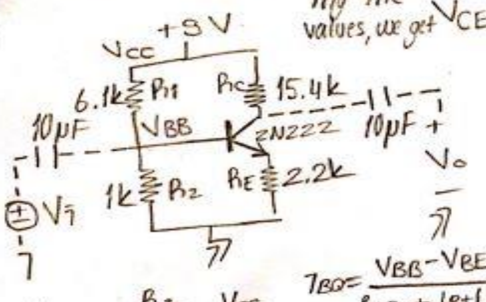
$1/h_{oe} = \frac{V_A + V_{CEQ}}{I_{CQ}}$

after inserting the values, we get $I_{CQ} = 0.29\,mA$, $V_{CEQ} = 3.87\,V$

$h_{ie} = \frac{26\,mV}{1.41\,\mu A} = 18439.72$

$h_{fe} = 200$

$1/h_{oe} = \frac{100 + 3.87}{(0.29)(10^{-3})} = 358172.41$



$R_E(h_{fe}+1) \geq 10 h_{ie}$ satisfied, so $A_V = \frac{-R_C}{R_E}$

$\Rightarrow \frac{-15.4k}{2.2k} = \underline{\underline{-7}}$

★ We have utilized from the unbypassed emitter common-emitter configuration

In the simulation, I got $A_V = -6.6$ which is nearly close to -7.

$V_{BB} = \frac{R_2}{R_1 + R_2} V_{CC}$

$R_{BB} = R_1 // R_2$

$I_{BQ} = \frac{V_{BB} - V_{BE(ON)}}{R_{BB} + (\beta+1) R_E}$

$V_{CEQ} = V_{CC} - I_{CQ}(R_C + R_E)$

$I_{CQ} = \beta I_{BQ}$

3.d PARAMETER SWEEP: We changed the magnitudes of the components by about %10 and observed the difference at the output. As it is clearly shown in the graph, Av value is 4.5 and this result is close to the theoretical result, which is 5. In the general Av expression, the effects of the other components are present. Therefore, randomly changing the values of the components can take us away from the goal. For safety, we should remain with our exact values except Rc and Re which are in the Av=-Rc/Re expression.

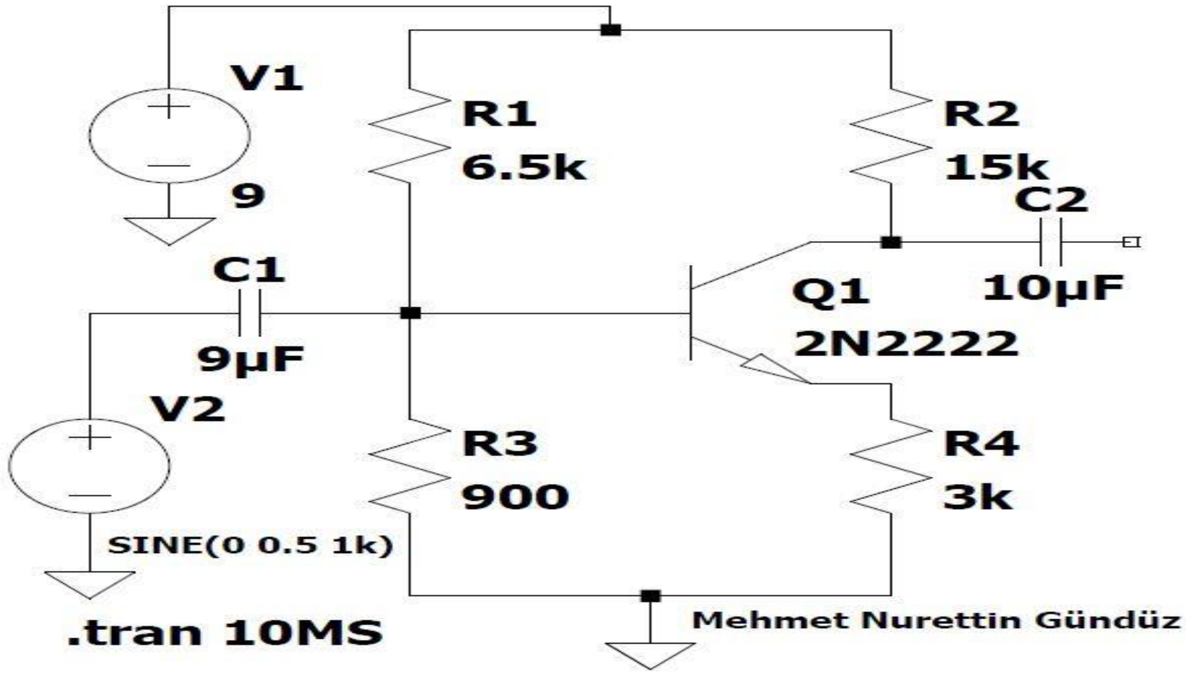


FIGURE 1.3.C: The circuit for the Gain Stage (Parameter Sweep)

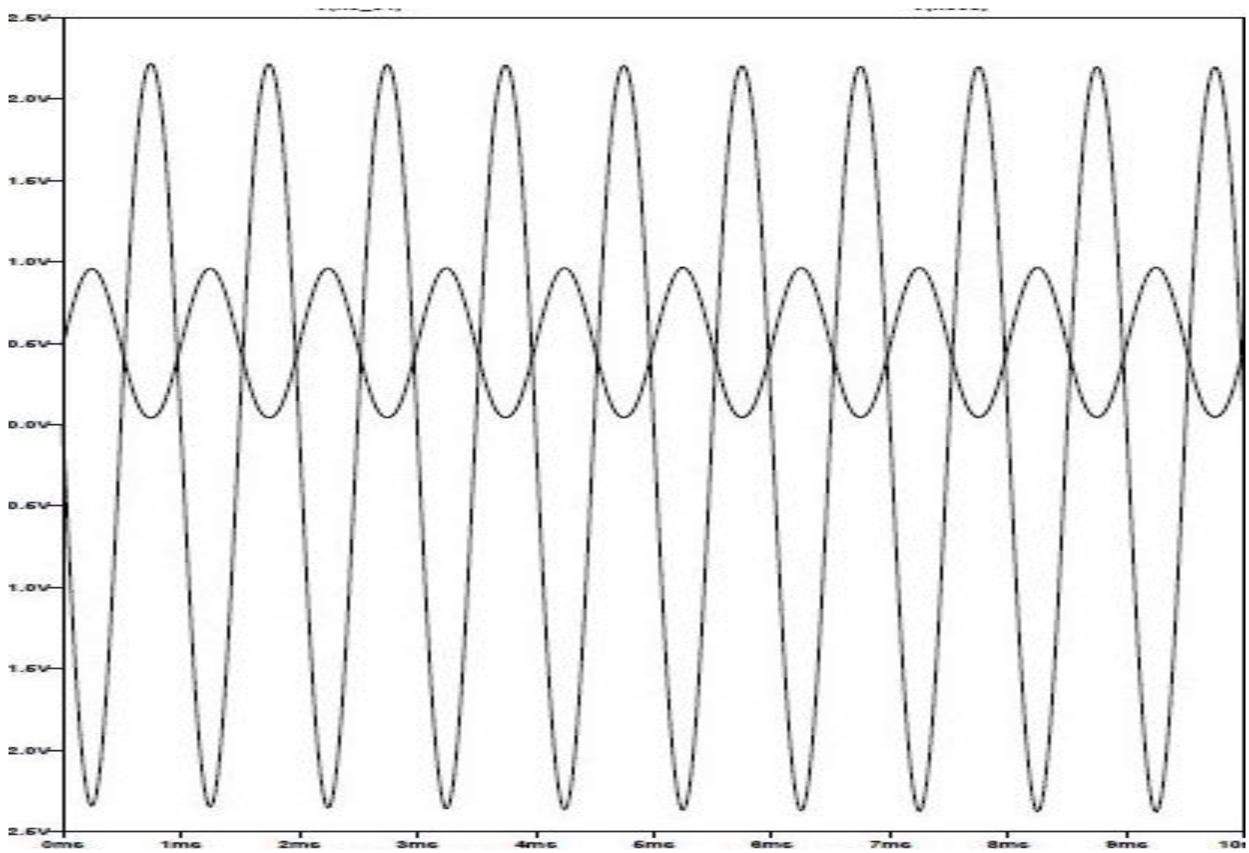


FIGURE 1.3.D: The Vo-Vi vs t waveform for the Gain Stage (Parameter Sweep)

4. COMBINATION OF THE STAGES

=> After analyzing each stage one by one, we are ready to combine the circuits and come up with the general design. The multistage audio amplifier takes a audio signal as an input and provides a sound at the output via speakers. We will discuss the results after observing the graphs, waveforms and response of the circuit.

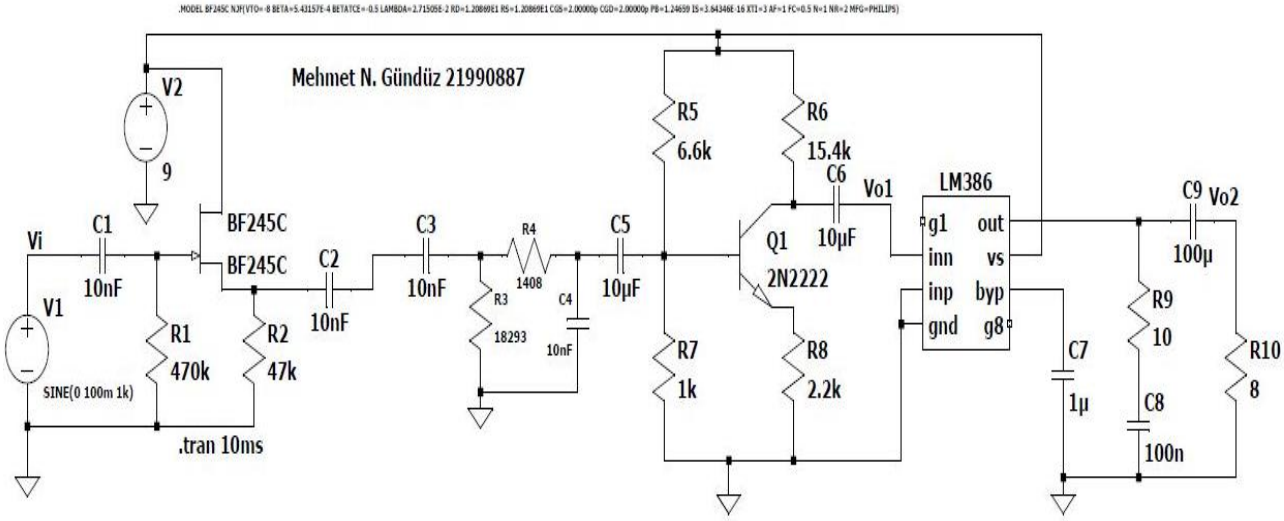


FIGURE 1.4.A: The circuit for the combination of the stages

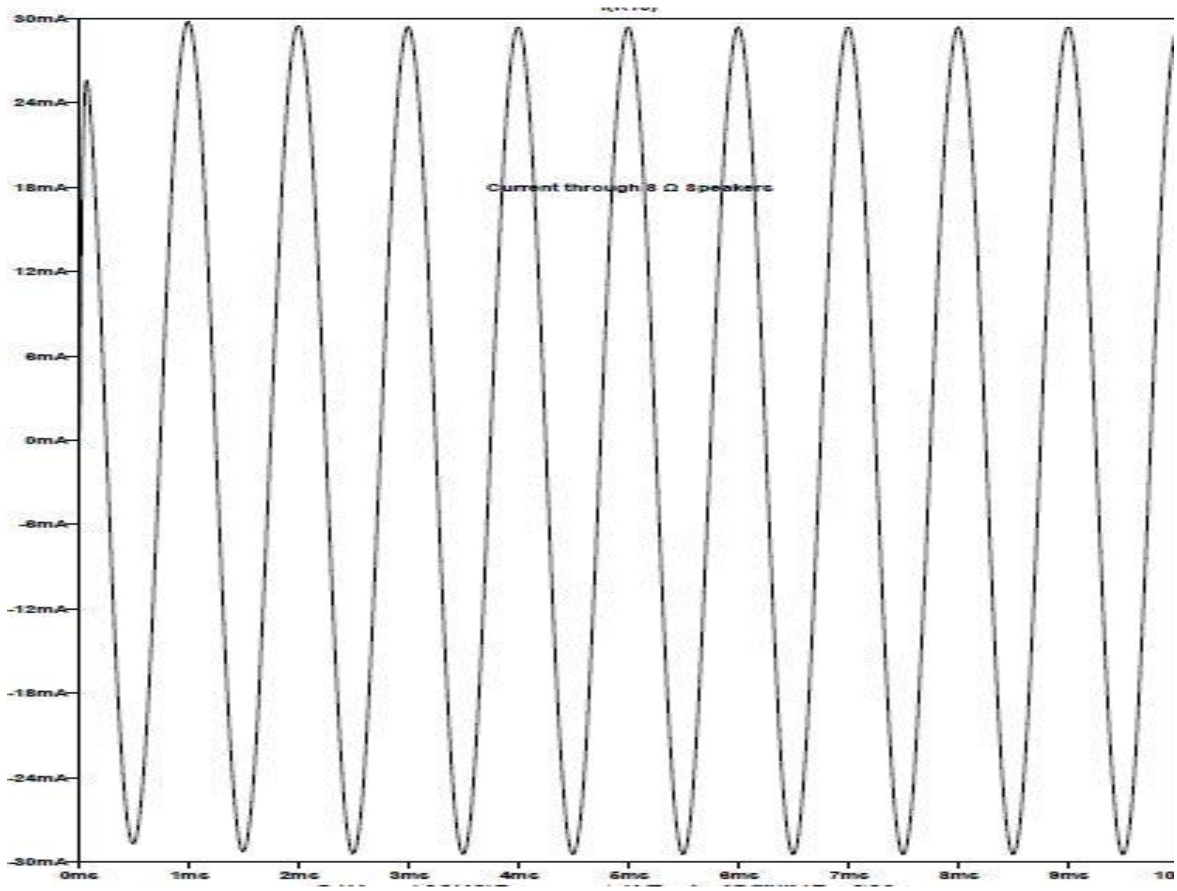


FIGURE 1.4.B: The waveform for the current flowing through 8 Ohm speaker

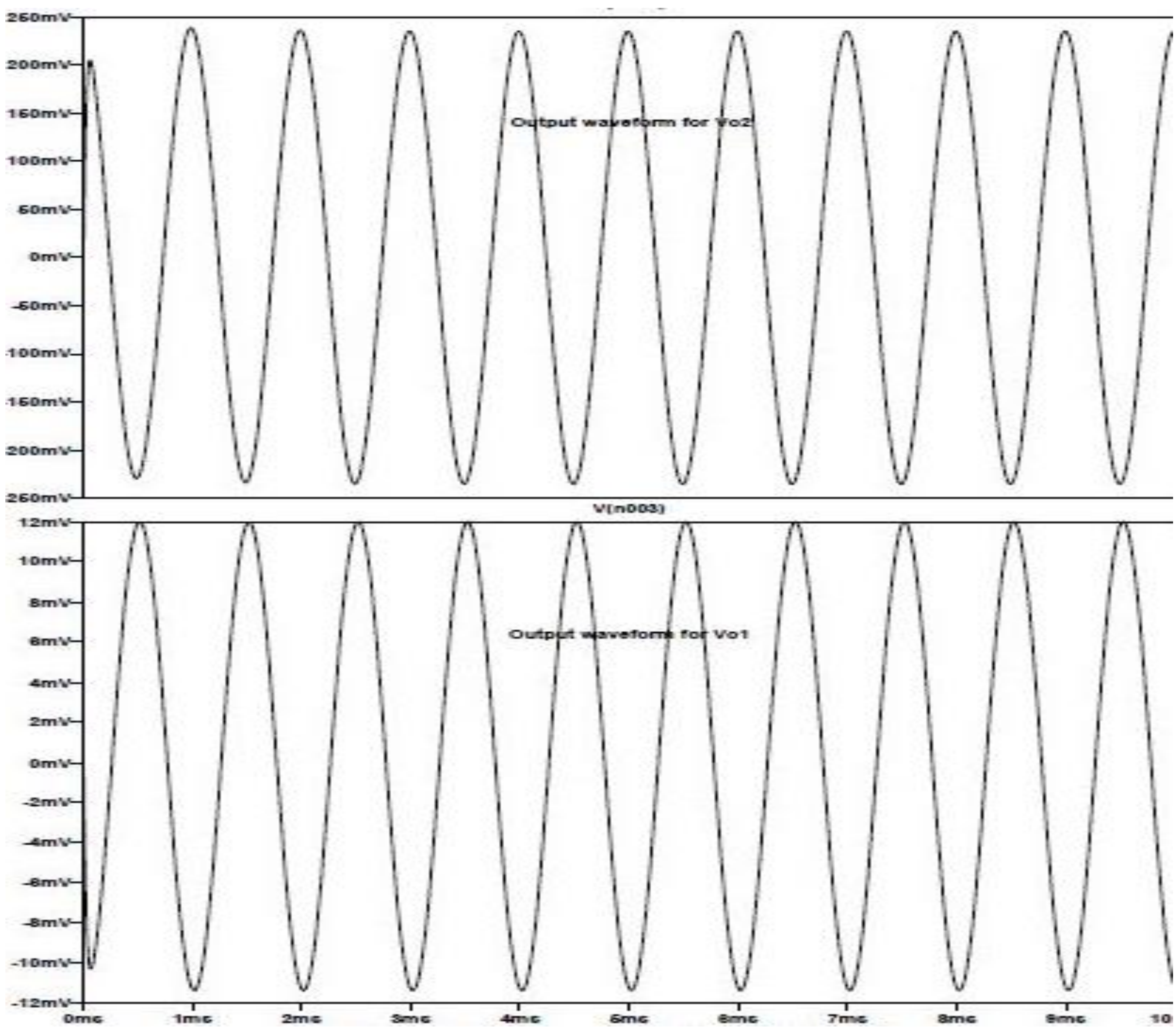


FIGURE 1.4.C: The output voltages for Vo2 and Vo1, respectively
(Note: The nodes Vo2 and Vo1 are indicated on the Fig. 1.4.A)

Comment on the results

We directly connected the stages from their input and output terminals. Also, we used the suggested output configuration in order to make the output more understandable. This whole configuration is said to be "Cascade". We applied AC signal with the amplitude 0.1 V and investigated the output terminals Vo1 and Vo2, respectively. The Vo1 node is the output of the gain stage and the input of the LM386 integrated circuit, as well. Here, unfortunately we couldn't get the desired voltages because of the voltage drop across the bandpass filter. Although we ignored the internal impedances of the capacitors in the calculations, it has significantly affected the voltage gain in the complete circuit. As we can understand from the output waveforms, we get 12 mV voltage by applying 100 mV voltage at the input. However, with the help of LM386, which is low-voltage audio amplifier, we were able to increase the voltage delivered to 8 Ohm Speaker at the node Vo2. The node Vo2 is the voltage of the speaker as it is seen at the rightmost part of the circuit. By looking the output waveform of the Vo2, we can notice that the amplitude is nearly 250 mV and it is the 2.5 times the input. In this case, the role of the LM386 is important because it helps us to hear the sound. Without the LM386, we cannot hear any sound on the speaker because we couldn't obtain enough voltage. Besides, the current flowing through the speaker is measured between the appropiate interval and the output waveform form the current is given in the Fig.1.4.B. If we apply higher input signals, say 1-1.5 V, we can observe distortions at the output and it will immediately affect the sound by causing unwanted noises. At the beginning, our aim was to set the gain Av = -7 but we are able to get Av = -2.5 thanks to LM386. That's why, we should also consider the impedances of the capacitors in order to avoid the loss in the voltage gain. The passive bandpass filter in the middle of the circuit uses the most of the voltage because we can observe higher voltage values at the output by removing the capacitors and the resistors in the bandpass filter. Nevertheless, there isn't any distortion at the output waveforms and the components seem to be adequate for delivering the audio signal to the speaker. The final design can be fabricated on a single breadboard because it doesn't occupy large space. Testing the circuit will improve our knowledge in the real case.

FABRICATION-VIDEO

The link for the fabrication video : https://youtu.be/BBcsB84B7sA

In the design, we tried to use the present components from the experiments. Therefore, our configurations and calculations are adjusted accordingly. After purchasing all necessary components, we set up the circuit in the convenient environment which is isolated from the electricity and other conducting materials. However, since a human body is already a good conductor, the components are certainly affected from the static electricity. The behavior of the circuit in each stage is described in the "ANALYTICAL CALCULATIONS AND RESULTS" Section. The input signal is taken from the radio station via aux cable and it's connected to the circuit by tucking up the aux cable to the jumper. Before starting to set up the circuit, we connected the – and + terminals in the both sides of the breadboard to take advantage of the space. Unfortunately, the circuit is seems to be complex because we required too many jumpers and one should look to the connections in detail and as close as possible in order to understand the logic of the connections. Also, at the end of the circuit, we placed the LM386 and suggested circuit but instead of connecting the speakers directly to the end of the circuit, we touch the terminals of the speaker by controling the jumpers manually. The output sound on the speaker included some amount of noise due to possible saturation of the transistors. We were able to hear the radio station at low quality and the music should be listened carefully.in order to understand the lyrics of the song. We have changed the transistors, worked in other isolated environment, rebuilt the complete design, tested the each component whether it works properly and used other voltage sources with different amplitudes but we couldn't find a solution for the problem. The audio signal should be amplified clearly if the theoretical design holds also for the fabrication. The transistor voltages were higher than the ones observed in the simulations and calculated theoretically. We tried to reduce the Vds and Vce by using additional resistors; however, we couldn't avoid the saturation on the transistors and the distortion at the output signal, as well. At least, the values for the components and the chosen configurations were approximately true because the signal is transmitted to the speaker safely and the sound of the music is heard at low levels. The voltage of the speaker is measured as 125 mV which makes sense compared to output waveforms obtained in the simulation. Similarly, the current is measured as 1.1 mA and this value is in the range of the output waveform for the current obtained in the simulation. We have changed the configurations slightly for several times in order to get desired output signal. Besides, some of the resistance values are obtained by connecting several resistors in series. For example, 6.6k Ohm resistor isn't commercially available, so we connected two 3.3k resistor in series. These connections can be noticed by analyzing the circuit in detail. As we mentioned before, the calculations are based on the 10 nF capacitors and naturally, we got high impedances especially on the bandpass filter. Therefore, the voltage gain Av significantly dropped at the output as expected. The measurements for the outputs Vo1 and Vo2 were close to numerical values observed in the simulation but the output voltages of the transistors, which are Vds and Vce, were higher than the simulation and theoretical results. These differences will probably lead to distortion at the output signal because the ideal output waveform cannot directly resemble to the ones observed in the simulations. If we have used an oscilloscope as an voltage source, we would have a chance to see the phase difference between the inputs and the outputs. However, since we use radio signal as an input. we don't know the exact characteristics of the signal. It's important to note that we didn't use a potentiometer in the bandpass filter because we have limited amount of the space on the breadboard. Instead, we used exact resistances in the high and low pass filters as we calculated theoretically. The potentiometers can be placed on the circuit by adjusting them properly but the slight differences on the resistance values will lead to fluctuations on the low and high cut off frequencies. The time constant for the series RC circuit is calculated by multiplication of the R and C and even one them is changed slightly, the f=1/(2*pi*R*C) may be significantly different. Our input signal has suitable frequency for the transitio both in the simulation and fabrication. The bandpass filter is a frequency selective circuit which sets a specific frequency range for the input signal by controlling the f(low) and f(high).

CONCLUSION

The project is designing a multistage amplifier following the instructions given in the project guideline. The report is prepared according to the order mentioned in the guideline.In the beginning of the report, one can get brief information by reading the abstract and the introduction. We started by providing necessary terminology; such as technical terms and analytical explanations. The definitions are written in a way that anyone who reads them can easily understand the basic functions of the components and the stages. Also, the main components of the design, which are called transistors, are described in terms of their parameters and working principals. The reasons for the selection of the transistor types are given in detail. The questions in the guideline are answered one by one systematically. The need of multistage amplifier instead of single-stage amplifier is stated in the "Comment" part under the "Statement of Objective" section. Then, we observed the each stage seperately and tried to come up with the accurate configurations. Since we are only responsible to design the stages indicated in the red boxes; for the output, we took the suggested circuitry in the guideline. In order to clarify the design, we have provided subsections in the order of "Description, Components, Calculation, Parameter Sweep" under the each stage. One can easily notice the logic under the design by following the topics and the instructions. Firstly, at the Buffer Stage, we utilized from Source Follower Configuration which sets the Av=1. Our main goal in this stage is to eliminate the effects of the unwanted components on the signal. As we calculated theoretically, we obtain desired output waveform in the simulation. After the parameter sweep, we didn't see any difference in the Av value because the numerical values remained within the limit in which we can employ the approximate analysis. Secondly, at the Equalizer Stage, the f(low) and f(high) frequencies are calculated in accordance with the student ID number and the formula stated in the guideline. The band-pass filter is the combination of the low-pass and high-pass filters. Within the scope of the course, we are dealing with the RC filters. We took the 10 nf capacitors as a reference because of their commercial availability. The theoretical calculations are hold by simulating the design with appropiate small signal. The simulation is done by frequency sweep analysis and the resultant graph is called bode-plot. We can observe the frequency response interval of the circuitry and the phase angle of the signal in the given figures. We changed the numerical values of the components by about %10 and realized that the frequency limits are slightly differed. Besides, this fact is described by calculating the exact differences explicitly and these differences are said to be ignorable. Thirdly, at the Gain Stage, we used n-type BJT transistor and unbypassed emitter configuration because of the easiness of the Av expression. After satisfying the certain conditions for eliminating some parameters in the calculation of the Av, we picked up the Rc and Re values to get the expected Av value which is 7. However, after the theoretical calculations of the Av and the simulation of the design accordingly, we came up with Av=6.6 value with the %95 achievement of the goal. Then, we did parameter sweep and revealed the effect of the each resistive component on the Av. Although the Av value seems to be related with only Rc and Re, the process didn’t go as we expected. The gain value is calculated as 4.6 whereas it should be 5. In this case, the achievement of the goal is about %92 and it means that randomly changing the other components will drive us away from the goal. It is important to note that we picked up the components by firstly considering the presence from the labarotary courses. Also, we adjust the designs according to the commercial availability of the components. For example, since we already have 470k, 47k, 10uF, etc. components available from the courses, our calculations are based on these values in some cases. In the fabrication part, we set up the circuit on a single breadboard. The design looks like complex because of the too many jumpers. However, the combination of the – and + terminals at the both sides of the breadboard is needed in order to ease the connections. The input signal is taken from the computer via aux cable as a radio signal. The frequency of signal was suitable for the transition on the whole filter. In the bandpass filter, we used exact resistance values instead of potentiometer because we had limited amount of space. We could also use potentiometers if we want to adjust the low and high cut off frequencies manually. Because of the high impedances of the capacitors, our voltage gain Av significantly dropped. For example, our aim was to set the gain to -7 in the gain stage but we could delivered a signal to the speaker with the Av = -2.5. However, with the use of the low-voltage amplifier component LM386, at least we were able to get a sound at the output. Also, the sound at the speaker contained a noise which affects the quality of the sound in a bad way. The different kind of possible solutions are tried in order to get clear sound but we couldn’t understand the exact reason of the problem.We have rebuilt the circuit for several times, changed some of the important components such as transistors, isolated the environment completely and applied various voltages from the input but we could only reduce amount of the noise instead of completely getting rid of it. The output currents and voltages made sense in terms of the numerical range obtained in the simulation. However, the transistor voltages were higher than the ones measured in the simulation. We tried to reduce the Vds and Vce values by adding transistors at the drain and collector terminals, respectively. By doing so, we reduced both the noise and the sound level at the same time. The main reason for the noise is likely the saturation of the transistors because of the high voltages. Unfortunately, I couldn’t show my problem to an expert in the distance learning. Besides, I controlled the connection of the speaker by hand instead of directly connecting it to the circuit. I bought different speakers and tested the sound on them one by one by touching the ends of the jumper to the terminals of the speaker. The sound was audible but not clearly understandable. It’s also important to note that since human body is already a good conductor, the sound may be affected by the static electricity of the human body. In this case, our precautions, such as isolated environment, were sufficient to avoid this kind of problems. In my opinion, the project was beneficial for us to get general information about the low-voltage designs. I understood the response of different configurations better and learnt how to read a datasheet. The transistor parameters have direct effect on the selection and the usage because one should pick up a transistor considering the numerical limits and physical facts. After dealing with the derivation of the expressions like Av or Idq for several times, I increased my speed in the calculations of the parameters. Although the project seemed to be hard at the first sight, the guideline helped me a lot in the development of the design. The most exciting part was the fabrication of the circuit because in each case, I wondered the response of the necessary adjustment. Although I couldn’t get clear sound at the speaker, hearing a sound at limited quality was sufficient to have fun. However, spending several hours on the problem made the process boring after a certain point. I am looking forward to improve my knowledge about the Electronics Course by working on the different circuits ,which can be easily set up at home, in the summer. Additionally, I still have some questions about my design and I will ask them after the pandemic. I am keeping the design ready and I will probably work on it again in a comfortable time by changing the JFET and BJT configurations. As we always see during term, having a sufficient knowledge about the hardware of the circuit is important in different fields of the engineering.

LIST OF REFERENCES

[1] R. L. Boylestad and L. Nashelsky, Electronic Devices and Circuit Theory, Pearson, 2012, (11th ed.).

[2] Lecture Notes of ELE230

[3] ElectronicsTutorials.com

[4] Wikipedia.com

[5] ti.com (Texas Instruments)

[6] Direnc.net

[7] Learningaboutelectronics.com

[8] Youtube.com

[9] ScienceDirect.com

[10] ElectronicsHub.com