Booming Bass
EPO-1 Project
A4 2018-2019

January 18, 2019

Oussama Seddouki *4608690*
Gyan Ramdjas , *4702727*
Reinier van der Leer, *4947703*
Samuel Richardson, *4608690*
Robbin Poll, *4732324*
Gerben Mulder, *4793927*

Contents

1 Introduction

This report will describe the process of building and analyzing an audio amplifier with filters. The project is split up in a couple of steps. First of all a power supply had to be designed and built to power the system (chapter 2), then the provided speakers had to be tested and analyzed (chapter 3), in order to design and build filters that fit the acoustic frequency response of the specific speakers (chapter 4). A power amplifier had to be built as well to increase the volume of the speakers(chapter 5). Lastly a booming bass extension was built to improve the overall frequency response of the whole system. In figure 1.1 a block diagram shows how the different parts of the system will be put together. The black arrows show how the electrical power is distributed, the power supply will convert AC to DC, the diagram also shows that the filters are passive. The red arrows represent the audio signal, going through the booming bass extension and the power amplifier to the filters, which output the relevant audio frequencies to the speakers.

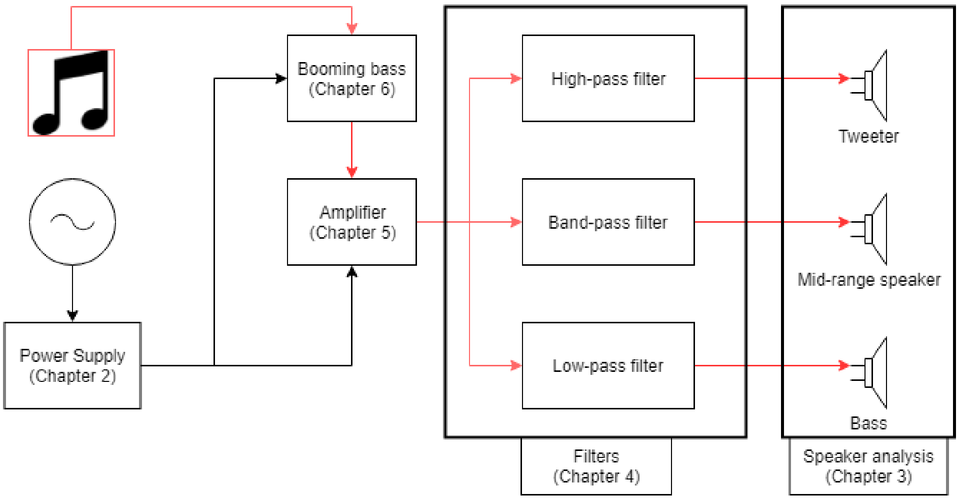

Figure 1.1: Block diagram of the audio system.

2 Power Supply

2.1. Introduction

2.1.1. The system requirements

In this report the design, analysis, dimensioning, simulation, building and measuring of a power supply is discussed.
The power supply had to be designed with the following requirements in mind:

1. A symmetric unloaded output voltage of ±22V DC.
2. With a load current of 1.0A, the output voltage is above ±19V DC.
3. With a load current of 1.0A, the maximum ripple voltage is 5 percent.
4. The capacitors must discharge in 2.5 minutes.

A transformer with the following specifications was provided:

- A primary voltage of 230V AC.
- A secondary voltage of 2 times 15V AC symmetric at nominal load with a center tap.
- Power: 80 VA.

This transformer will be connected to the mains socket on the primary side. The circuit is as follows: The

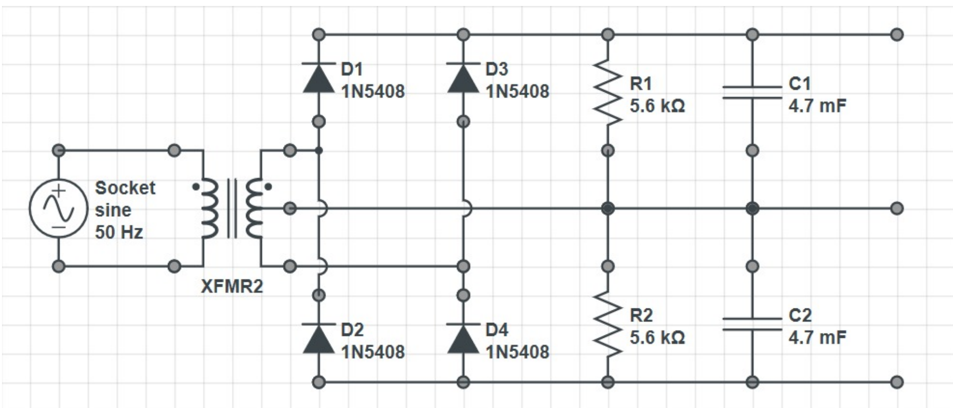

Figure 2.1: Power supply schematic.

circuit contains (besides the transformer and main socket) two resistors, two capacitors and 4 diodes forming a double sided rectifier with diode bridge.

2.1.2. The problem definition

The power supply circuit needs to meet certain requirements which are discussed in paragraph 1.1. These requirements can only be met if the right components with the right values are chosen. In order to let the power supply meet the specified requirements of the circuit, the following steps should be taken:

1. Calculate the values of the unknown components [R_1, R_2, C_1 and C_2]. This is done by answering the provided questions in [[4],Section 12.6.1, pp. 120-121].
2. Simulate the circuit in PSpice and determine whether the calculated values will provide a circuit with the specified requirements.
3. Build the circuit with the provided components.
4. Check if the circuit meets the requirements by taking measurements of the physical circuit and comparing it with the simulated circuit.Creating the power supply.

2.2. Creating the power supply

2.2.1. Design (methodology)

Elements in the circuit:

The power supply circuit is made up of nine elements: 4 Diodes, 2 Capacitors, 2 Resistors and 1 Transformer. Each of these elements plays a critical role in converting the 230 Volt AC from the wall outlet to a ±19 Volt DC to be used by the amplifier. The transformer transforms the 230 Volt AC signal to a 30 Volt (or 2x 15 Volt) AC signal. The capacitors hold charge so that the output voltage doesn't drop down immediately once the AC voltage swaps it's polarity. The diodes in turn prevent the capacitors from discharging once that once the AC signal reached peak value. The resistors R_1 and R_2 provide a desired discharge time of the capacitors.

The design steps :

If the desired output voltage of the power supply has to be ±22V DC then the secondary voltage of the transformer needs a peak value of at least 44V. This means the AC voltage needs to be $44/\sqrt{2} \approx 31.1$V AC minimum (taking the RMS value). Measuring the secondary voltage of the transformer gave the value of 34.65V AC, this is higher than the needed 31.1V AC calculated earlier. So the required input voltage of ±22V can be reached. Next is determined what the capacitance of the capacitor must be in order to reduce the ripple voltage to a maximum of 5 %. If the average voltage after the rectifier is assumed to be equal to ±19V at a load current of 1.0 the capacitance of the capacitor can be calculated as followed:

$$i_c = C\frac{dv}{dt} \qquad \text{rearranged to:} \qquad C_{1,2} = \frac{i\triangle t}{\triangle v} \tag{2.1}$$

The difference in voltage is determined by what the maximum and minimum voltage is with a ripple voltage of 5%. This is a maximum voltage of 19.95V and a minimum of 18.05V, this gives a difference in voltage of 19.95-18.05 = 1.9V. The difference in time is the difference in the values of time that correspond with the maximum and minimum voltage. In order to do that the function of the voltage with respect to time is used. Since it is an AC signal, it can be described as a sine function:

$$v(t) = V_m sin(\omega t) \tag{2.2}$$

$$\omega = 2\pi f \tag{2.3}$$

Where V_m=19.95V, f =50Hz and thus:

$$v(t) = 19.95 sin(100\pi t) \tag{2.4}$$

The maximum voltage of the capacitor is equal to the maximum voltage of the sine function. The minimum voltage of the capacitor is less obvious to calculate. Since the capacitor starts charging when the voltage of the sine function is higher than the capacitor, the corresponding times can be calculated by setting the voltages equal to the sine function. But since the sine function is an absolute sine function after the rectifier the symmetry of the sine can be used to calculate the corresponding time:

$$-18.05 = 19.95sin(100\pi t) \qquad \text{and thus:} \qquad t_{min} = 13.6 \quad \text{ms} \tag{2.5}$$

$$19.95 = 19.95sin(100\pi t) \qquad \text{and thus:} \qquad t_{max} = 5.0 \quad \text{ms} \tag{2.6}$$

The difference in voltage is calculated as followed: $\triangle v$= 19.95-18.05= 1.90 V. The time difference is then equal to $\triangle$ t= 13.6-5.0 = 8.6 ms.

Using the 1.90V, 8.6 **ms** and 1A current values and inserting them in to equation 2.1 the capacitance of the capacitors is calculated, resulting in a value of $C_{1,2}$ = 4.5**mF** . This is the minimum capacitance that can be used as the ripple voltage will be larger than 5 % if a lower capacitance is chosen (equation 2.1). The higher the capacitance the lower the ripple voltage will be. However, this is not a feasible option, as the capacitor pulls a higher current as the capacitance increases. Choose too low and the ripple voltage will be larger than 5%.

Choose to high and the current pulled by the capacitor will affect the secondary voltages of the transformator resulting in a possible unstable signal. The average current through the diode with the capacitor being 4.7 **mF** is about 0.75A and the peak current is initially about 35A but after the capacitor is charged about 10A. Doubling the capacitance to 9.2 **mF** the average current becomes 0.9A and the peak current initially 65A and after that 12A. These high currents are not to be desired and thus a capacitance of 4.7 **mF** is chosen.

The last requirement to consider is the discharge time. The capacitors must be able to discharged within 2.5 minutes when the power supply is not loaded. The discharge time of a capacitor is equal to 5τ [[1], Part III, pp. 122] where:

$$5\tau = 5R_{1,2}.C_{1,2} \qquad \text{[[1], Formula 10.5, pp. 105]} \tag{2.7}$$

so if 5τ has to be less than 2.5 minutes(150 secondes), $5R_{1,2}.C_{1,2}$ also has to be less than 150 seconds. The value of the resistors R_1 and R_2 can be calculated by dividing 15o secondes by $5C_{1,2}$ which results in a value of 6.7kΩ. This is the maximum value of the resistors because a larger resistance will result in a longer discharge time (equation2.5).Considering the provided resistors for the EPO Project and a maximum value of 6.7kΩ a value of 5.6kΩ was chosen for R_1 and R_2.

When determining which diodes were going to be used, it is important that they are capable of withstanding the inverse voltages. In this case the maximum reverse bias voltage over the diode won't be higher than the maximum voltage difference between the two signals. Meaning in reverse bias the diodes have to withstand the ±22V = 44V. In reality this is lower because the diodes themselves are not perfect conductors, this is further shown in the simulations. According to the datasheet of the diodes that were used (1N5408), the diodes are well capable of withstanding these voltages.

2.2.2. Simulations

Employed tools and steps:

The program PSpice [4] was used to simulate the power supply. The schematic as seen in figure 2.2 is used to simulate the designed power supply.
Before putting together the power supply, the design with the calculated dimensions of the components has to be simulated to get an idea of how the circuit reacts. If unexpected values appear or the requirements are not met, the values of the components can be reconsidered.
This simulation contains four voltage probes. The voltages at “Trafo+” and “Trafo-” in figure 2.1 are always equal to the specified voltages for the AC voltage sources. These probes are for visualization of the input voltage. The focus of the simulation is on “DC+” and “DC-”. These points are the output terminals of the power supply. Their values need to be within the required specifications.

The value of “DC+” is specified to be at least +19V and “DC-” is specified to be at least -19V relative to ground.

Schematic:

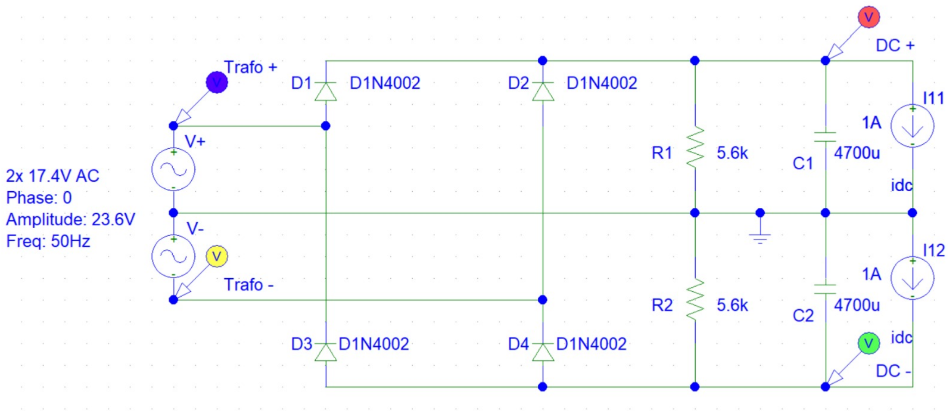

Figure 2.2: Schematic used for simulation under loaded condition.

The circuit contains the calculated component values and has been simulated with and without a load of 1A on both the terminals. The measured output of the transformator was 2x 17.4V AC instead of 2x 15V AC as stated in the manual [[1], pp. 120]. To have the simulation be as close as possible to the real circuit, the voltage sources are adjusted.

Simulation Results:

In Fig 2.3 the voltages in the unloaded power supply are shown. The peak values of the sine functions are 24.6V, and the resulting DC signal has a voltage of around ±23.5V. It also shows that the maximum reverse bias voltage over the diodes is equal to twice the voltage amplitude minus the voltage over the diodes themselves. The maximum reverse bias voltage is equal to around 48V.

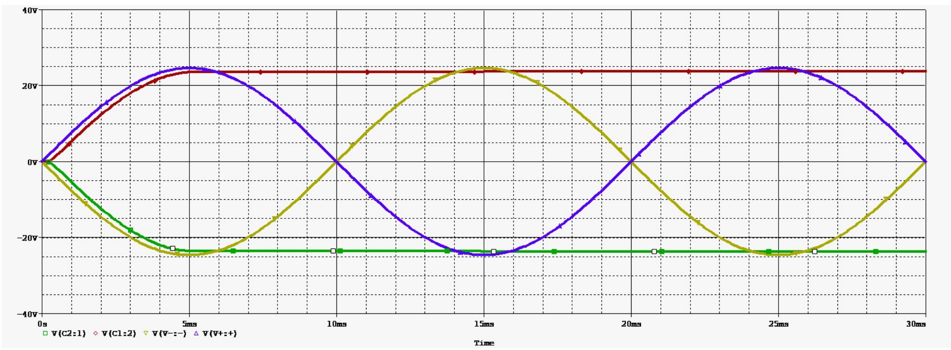

Figure 2.3: Voltages in the PSU without any load.

In Fig 2.4 the ripple voltage at the positive terminal with a load of 1 Amp (in red) is shown. Here is also shown that the maximum forward bias voltage over the diodes is around 1.2V.

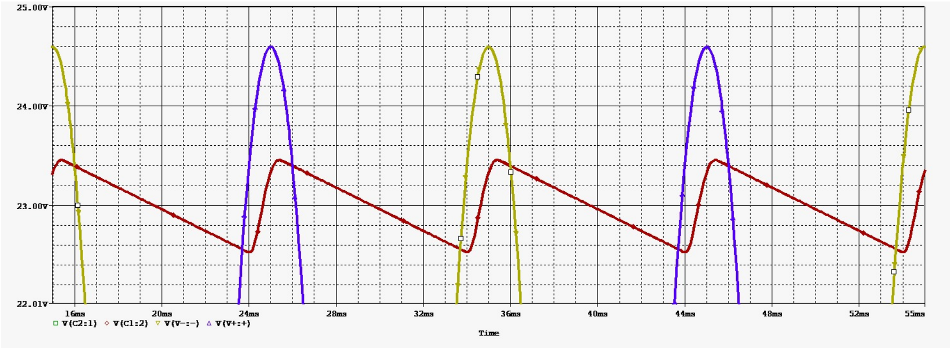

Figure 2.4: Ripple voltage at the positive terminal with a load of 1 Amp.

These results show that the design achieves the first requirement of a DC voltage of at least ±22V. Fig 2.3 shows that under a load of 1A the ripple voltage is: V_{gem}= 23.0V V_{max}= 23.5V
Using equation 10.4 from [[1],pp. 104] Ripple voltage(%) ={(23.5 - 23.0) / (23.0)} . 100 = 2.13%
The requirement was a maximum ripple voltage of 5%, so this was also achieved.

Comparison:
The results of the simulation match the expectations. But this was not true in the first simulation. A problem in the beginning was that the secondary voltage of the transformator acted differently to what is described in the manual. This is all explained by the fact that in the simulation the transformer is omitted. The transformer has been replaced by two voltage sources with a fixed amplitude. In reality the transformer doesn't have a fixed amplitude. The measured output voltage of just the transformer is equal to 2x 17.4V AC. The peak of this signal is equal to 23.50V. So when adjusted, the second simulation with the new values for the voltage sources does get the desired results.

2.2.3. Assembly of the power supply

choice of components:
A predetermined circuit was given with 1N5408 Diodes and soldering connections for the two capacitors and two resistors. As determined in equation 2.7 he capacitors should have a capacitance of at least 4.5 **mF**.The closest match of provided capacitors where capacitors with a capacitance of 4.7 **mF**.Because the value of the capacitors have changed the resistance of the resistors had to be recalculated. Using equation 2.7 with the right capacitance, R ≈ 6380Ω. Theprovided resistors that matched this value the closest and made sure the power supply still meets the requirements had a value of 5600Ω. Timing how fast the capacitors discharged with these resistors still gave a desired result.

2.2.4. Measurements

Procedures and Results:

The first value needed is the output voltage is of our transformer. The measured unloaded secondary voltage is 34.8V AC and with the center tap this means ±17.4V AC. Therefore the peak values are ±24.5V. Theoretically the capacitors can charge up to that value, but this doesn't happen as other elements in the circuit also influence the voltage. Lastly the voltages after the rectifier are to be measured to make sure the power supply still meets the requirements.

After the rectifier the voltage is measured 23.5V DC over the positive and ground terminal and -23.5V DC over the negative and ground terminal. This is the voltage without any load. Because the power supply needs to have a minimum output voltage of ±19V DC under a load of 1.0 Amp, a load of 1.0 Amp is applied and the voltages are remeasured. The measured voltages are now ±20.1 V DC. This meets the required minimum voltage of 19V DC.

The measured ripple voltage as seen in Fig 8.1 in the appendices is equal to

V_{max}=840 **mV** V_{pk-pk}= 1.58 **V**

These values are added to the DC voltage to get the real output voltage:
V_{max}= 20.1 + 0.84 = 1.58 **V** V_{DC}= 20.1 **V**

Using equation 10.4 from [[1] ,pp .104] {(20.94 - 20.1) / (20.1)} . 100 = 4.18% is calculated. This is below the maximum allowed 5 %.

The discharging of the capacitor at the positive terminal is shown in Fig 8.2 in the appendices. The figure clearly shows that after 2.5 minutes (6 div) the capacitor is negibly charged and is considered discharged.

Comparison

Comparing the final measurements with the simulation it shows that without any load on the power supply, the measured voltages over the terminals are equal to the simulated ones. However, with a load on the power supply the voltages drop under the simulated values. This difference can be explained by the fact that the transformer has a power of 80 VA, so the voltage is higher when the current is lower. Unloaded means no current, resulting in no voltage drop. A load of 1A however does affect the voltage output of the transformator. In the simulation we replaced the transformer with a normal AC voltage source. This voltage source will always put out the specified voltage, no matter the load. In reality this is different and that is seen in the difference of voltage across the terminals when loaded.

2.3. Conclusions and Recommendations

Comparing the achieved values with the requested requirements will give a proper image of how well the assignment was completed. As noted earlier in 1.1, the requirements were as follows:

- A total unloaded output voltage of ±22V DC.
- With a load current of 1.0A, the output voltage is at least ±19V DC.
- With a load current of 1.0A, the maximum ripple voltage is 5 percent.

Taking a look at the final measurements, the circuit achieved to meet all the requirements. The total unloaded output voltage is equal to ±23.5V or 47V over both terminals. This is above the ±22V required. Under a load of 1.0A, the measured voltage at the terminals are ±20.1V. As the minimum required voltage is ±19V, the power supply certainly delivers more than needed. And last but not least, the ripple voltage is less than 5%. This ensures a stable enough voltage for the amplifier. In conclusion one could say that the assignment was successfully completed.

3

Speaker analysis

The goal of Loudspeaker analysis is to establish the electrical and acoustical characteristics of each individual speaker driver. This information will later be used to design passive filters which are needed to constrain the output signal to the different speaker drivers to their designated, most optimal[1] frequency ranges.
In order to determine the electrical and acoustical characteristics of the speaker drivers, to be named the two woofers, the midrange driver and the tweeter, a number of measurements and calculations will be performed. To determine the electrical characteristics and find the equivalent circuit for each of the speaker drivers, first the impedance amplitude response for a wide range of frequencies is measured, and then calculations are performed to find the values of different elements in the equivalent circuits. To determine the acoustical characteristics, the output frequency response for the same range of frequencies is measured using a microphone.

3.1. Loudspeaker analysis

This section describes the process of analyzing the electrical and acoustical properties of the speaker drivers in the given loudspeaker.

3.1.1. Loudspeaker impedance model

A loudspeaker can be modeled as an electrical impedance as seen in figure 8.5. In this circuit:
R_e = DC resistance of the voice coil.
L_e = self inductance of the voice coil.
R_a = acoustic radiation resistance.
R_v = mechanical resistor loss.
R_p, C_p, L_p = electrical equivalent of the mass-spring system.

The impedance model is formed by adding together the electrical and mechanical properties of the speaker. The cone of the speaker is mounted to the frame and moves back and forth. This forms a mass-spring system that can be modeled as a parallel RLC circuit. The power dissipated in R_a is in reality the power which is transformed to sound, R_v is the power dissipated by mechanical losses due to friction. In practice R_v and R_a are much smaller than R_e, which is why they can be ignored to get the simplified circuit of figure 8.6.

From the simplified model in figure 8.6, eq. 3.1 for the impedance as a function of the frequency can be derived:

$$Z(\omega) = \frac{j\omega L_p R_p}{j\omega L_p + R_p - \omega^2 L_p R_p C_p} + j\omega L_e + R_e \tag{3.1}$$

At high frequencies the first term approaches zero, so the model can be further simplified for frequencies well above the resonant frequency [figure 8.7]. In this case, eq. 3.1 can be simplified to eq. 3.2.

$$Z(\omega) = R_e + j\omega L_e \tag{3.2}$$

[1]Where the output amplitude is equal for all frequencies within the range

3.1.2. Formulas for calculation of component values

With the two models, the measurements and equations from [1, p. 137-144], the values of the components in the circuit can be derived as follows:

$$R_p = R_{peak} - R_e \tag{3.3}$$

$$C_p = \frac{1}{R_p B_\omega} \tag{3.4}$$

$$L_p = \frac{1}{C_p \omega_0^2} \tag{3.5}$$

$$L_e = \sqrt{\frac{|Z(\omega)|^2 - R_e^2}{\omega^2}} \tag{3.6}$$

R_e is the DC resistance of the voice coil and is measured using a multimeter. Values for R_{peak}, B_ω, ω_0 can be obtained from the impedance amplitude graphs as shown in figure 8.10. In order to find L_e, a point on the graph right of the resonance peak is chosen, for a frequency high in the relevant frequency range for the speaker. The value is then calculated using eq. 3.6, which is derived from eq. 3.2.

The impedance of a component at a given frequency ω can be calculated using eq. 3.7 - 3.9:

$$Z_{resistor} = R \tag{3.7}$$

$$Z_{inductor} = j\omega L \tag{3.8}$$

$$Z_{capacitor} = \frac{1}{j\omega C} \tag{3.9}$$

From these equations it follows that the impedance of a resistor does not depend on frequency, an inductor has a high impedance for high frequencies and a low impedance for low frequencies, and a capacitor has a high impedance for low frequencies and a low impedance for high frequencies. RLC circuits have a resonant frequency. Parallel RLC circuits have maximum impedance at the resonant frequency, while series RLC circuits have minimum impedance and zero phase at this frequency. For both series and parallel RLC circuits, the resonant frequency can be found using the following equation from the manual [1, p. 144]:

$$\omega_0 = \sqrt{\frac{1}{L_p C_p}} \tag{3.10}$$

Eq. 3.5 is derived from this equation, since the resonant frequency can be obtained from the measurement graphs, C_p is given by eq. 3.4, and L_p is to be found.

In our case, R_p, C_p and L_p form a parallel RLC circuit [fig. 8.6], so at the resonant frequency this circuit will have maximum impedance. This is explained by the manual [1, p. 143] A short summary: The inverse of impedance is admittance (symbol Y), just as conductance is the inverse of resistance.

$$Y = \frac{1}{Z} \tag{3.11}$$

Since R_p C_p and L_p are in parallel, their admittances can be added up; this results in eq. 3.12:

$$Y_{inductor} + Y_{capacitor} + Y_{resistor} = \frac{1 - \omega^2 L_p C_p}{j\omega L_p} + \frac{1}{R_p} \tag{3.12}$$

At the resonant frequency, the numerator of the first term becomes zero, thus the impedance of the parallel circuit will be equal to the resistance R_p at this frequency. Since the parallel circuit of R_p, C_p and L_p is in series with R_e, the total resistance at this frequency is given by eq. 3.13:

$$R_{peak} = R_p + R_e \tag{3.13}$$

From equation 3.13, eq. 3.3 is derived.

While this section describes the electrical phenomenon of resonance, we have to keep in mind that R_p, C_p and L_p are actually the electrical model of the mechanical mass-spring-system of the speaker cone, thus its resonance is actually mechanical in nature; the electrical model describes or emulates the behavior of the system.

3.2. Simulations

In order to be able to simulate the circuit of the speaker drivers, [eq. 3.1, 3.2] were derived as expressions for the impedance of the equivalent circuits in section 3.1.1.
Using these formulae, matlab code was written to simulate the circuits. See appendix 8.2.3 for the code to simulate model 1 [figure 8.6, eq. 3.1] and model 2 [figure 8.7, eq. 3.2] respectively. Figure 8.11 shows comparisons of the output of impedance simulations for the midrange speaker driver using the two different models. When compared in one graph, it is clearly visible that the absolute values of the two models converge around 300Hz [figure 8.11], and the phases converge around 1.5kHz. This corresponds with the notion in [1, p.143] that "At high frequencies, the impedance increases due to the self-inductance of the voice coil, L_e, to $Z_{L_e} = R_e + j\omega L_e$." More specifically, the impedance of L_e increases significantly and the combined impedance of R_p, C_p and L_p (the mechanical model) approaches zero above the resonant frequency, so the total impedance becomes that of R_e and L_e combined. This is shown in figure 8.12:
Of course, in order to make these simulations, measurements done on the actual speaker had to be analyzed. Figure 3.1 shows the impedance amplitude of the midrange driver with respect to the frequency, it compares the simulation with the measurement.

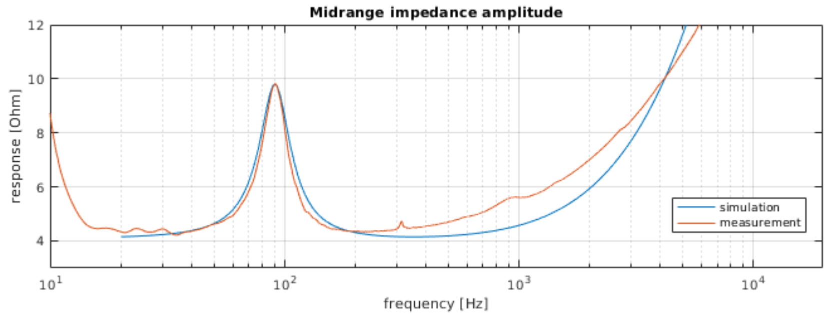

Figure 3.1: Comparison of the impedance measurement and simulation for the midrange driver.

In this graph, the result of the measurement and simulation look very similar, although the measured graph is less smooth than the simulated graph. The biggest difference, aside from the smoothness, is that the measurement graph shows a significantly higher impedance than the simulation between 300Hz and 3kHz. This is mostly due to magnetic coupling between the voice coil of the speaker and other surrounding metal parts [8]. This form of parasitic inductance is dependent on the mechanical structure of the speaker box, and is not included in the models given by the book, so we cannot simulate it.
Analyzing the tweeter presented a different challenge. When strictly following the instructions in the manual [1], the values calculated give a simulation result that differs greatly from the measurement [figure 3.2].
The source of these differences could be traced back to some oddities in the measurements. The measured DC resistance (R_e) of 4.0 Ω is different from the "baseline" impedance that can be seen in the graph. This baseline lies at approximately 4.3 Ω. Adjusting R_e like this required recalculating R_p, C_p, and L_p [eq. 3.3 - 3.6], which in turn prompted recalculation of L_e. After making these adjustments, the difference in bandwidth of the resonance peaks was the next big problem. Doing the simulation and comparison in Matlab, the impedance being calculated from the component values using eq. 3.1 and the code in appendix 8.2.3, it was possible to quickly iterate and find better matching values for the components of the simulation model. Increasing C_p from 26.3 μF to 100 μF decreased the width of the resonance peak to the point where it almost exactly matched the measured peak. The peak was now offset to the left with respect to the measurement by about 20 Hz, because the value of ω_0 depends on both L_p and C_p by eq. 3.10, and L_p had not been properly adjusted after increasing C_p. Increasing ω_0 in eq. 3.5 from 7603 rad/s ($1210Hz * 2\pi$) to 7720 rad/s ($1228.7Hz$) and recalculating L_p nicely aligned the peaks. Now the most significant difference between the simulation and measurement was the part where the impedance starts to increase more or less linearly and converges to the output of eq. 3.2. Between $\sim 3.5kHz$ and $\sim 18kHz$, the simulated impedance was lower than the measured value, with a maximum of ~ 0.5 Ω between 7 kHz and 14 kHz. To decrease this difference, the value of L_e was recalculated [eq. 3.6] with a reference point of 10 kHz and 5.90 Ω (previously 20.41 kHz and 8.25 Ω). This resulted in a value of L_e of 64.3 μH, while it was previously calculated as 56.3 μH. After all these adjustments, the simulation matches the measurement fairly well [figure 3.3].
The intersection of the impedance amplitude simulation and measurement in figure 3.3 is not at 10 kHz

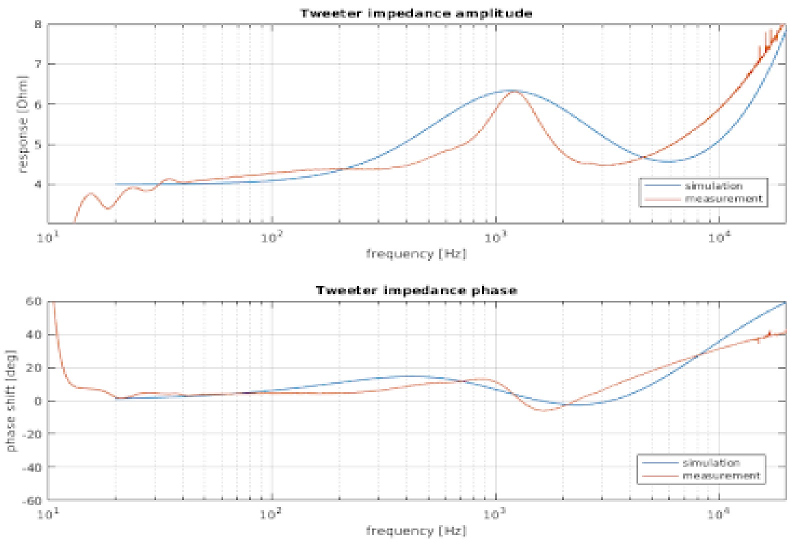

Figure 3.2: Comparison of the simulated and measured impedance amplitude and phase response of the tweeter. Most noticeable are the difference in bandwidth and the difference at high frequencies.

because at this frequency the first term of eq. 3.1 in the simulation is not yet zero. Graphs comparing the simulation and measurement results of the woofers can be found in Appendix 8.2.5.

3.3. Measurements

3.3.1. Measurement setup

In order to measure the frequency-dependent impedance of each speaker the MatLab program LS-Measure was used. The program generates white noise: the signal contains all frequencies, each frequency having the same amplitude. The signal is sent to the USB Sound card. The signal then passes through a voltage divider which consists of the unknown impedance (the speaker) and a resistor of $100\,\Omega$ used as reference. The reference resistor and the connections are located in a connection box which is given. The response is sent back to the PC and is then processed. The setup is shown in figure 8.8.
For the measuring the short distance acoustic response the LS-Measure program is used as well. Here a buffer amplifier is used to for the input of the speaker which is the output of the connection box. And the response signal is obtained by feeding the output of a microphone placed 2 cm away from the speaker back into the soundcard and thus the PC [figure 8.9].

First, measurements of known resistors are made to make sure the system works as intended, it was easier to calibrate with smaller resistors. Also measurements of an inductor of 1 mH and a capacitor of 1 µF were made. The results are as follows:
Impedance of the 1 mH inductor:
$-151i\,\Omega$ at 1 kHz
$-15i\,\Omega$ at 10 kHz
Impedance of the 1 µF capacitor:
$6.25i\,\Omega$ at 1 kHz
$69.28\,\Omega$ at 10 kHz

Calculating the expected values with eq. 3.8 and eq. 3.9:
$Z_L = j\omega L = j * 1000 * 2\pi * 10^{-3} = 6.28\, j\Omega$
$Z_L = j\omega L = j * 10000 * 2\pi * 10^{-3} = 62.8\, j\Omega$
$Z_C = \frac{1}{j\omega C} = \frac{1}{j*1000*2\pi*10^{-6}} = -159\, j\Omega$
$Z_C = \frac{1}{j\omega C} = \frac{1}{j*10000*2\pi*10^{-6}} = -15.9\, j\Omega$

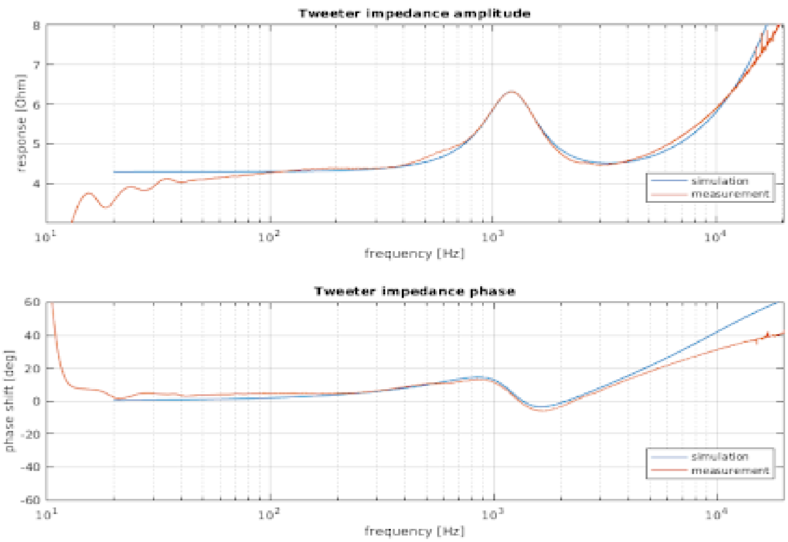

Figure 3.3: Comparison of the measured impedance amplitude and phase response of the tweeter and the adjusted simulation.

The calculations of the expected results are within 10% of the measurement results, which is acceptable, since the measurement setup is not perfect and also the capacitor and inductor are not ideal components and their actual values have non-negligible error margins.

3.3.2. Calculating component values from measurement results

Figure 8.10 shows a typical result of the LS-measure program. From this graph it is possible to obtain values to put in eq. 3.3 - 3.6. Take the tweeter as seen in figure 3.3 for example:
$R_e = 4.0\,\Omega$ $\quad B_\omega = 16387\,\text{rad/s}$ $\quad \omega_0 = 7603\,\text{rad/s}$ $\quad |Z| = 8.25\,\Omega$ at $\omega = 12.82\,\text{krad/s}$

$$R_p = R_{peak} - R_e = 6.32 - 4.0 = 2.32\,\Omega \tag{3.14}$$

$$C_p = \frac{1}{B_\omega R_p} = \frac{1}{16378 \times 2.32} = 26.30\,\mu\text{F} \tag{3.15}$$

$$L_p = \frac{1}{\omega_0^2 C_p} = \frac{1}{7603^2 \times (26.30 \times 10^{-6})} = 657.8\,\mu\text{H} \tag{3.16}$$

$$L_e = \sqrt{\frac{|Z(\omega)|^2 - R_e^2}{\omega^2}} = \sqrt{\frac{8.25^2 - 4.0^2}{(128.24 \times 10^3)^2}} = 56.27\,\mu\text{H} \tag{3.17}$$

Performing these calculations for all speaker drivers yields the values in table 3.1:

	Tweeter (adjusted)	Midrange	Woofer 1	Woofer 2
R_e (Ω)	4.0 (4.3)	4.1	7.1	7.1
L_e (H)	56.3 µ (64.3 µ)	349 µ	1.1m	1.32m
R_p (Ω)	2.32 (2.0)	5.72	15.4	11.4
C_p (µF)	26.3 (100)	987	304	367
L_p (H)	658 µ (186 µ)	3.12m	36.9m	15.4m

Table 3.1: Calculated values of the components for the model.

The adjusted values for the tweeter result from the adjustments described in section 3.2; the initially calculated values did not produce a simulation that reflected the measured behavior of the tweeter.

Resonance happens when the speaker cone vibrates at its natural frequency, this is a mechanical phenomenon. Figures 3.1 and 3.3 clearly show resonance peaks for the midrange driver and the tweeter respectively. For the first woofer as in figure 3.4, a peak is also very clear, but this graph shows two peaks. This second resonance peak is measured because there are two woofers in the system, so when testing one the other also starts vibrating. This also has to do with the bass reflex ducts being blocked, containing the air within the speaker box. Figure 3.4 also shows another small bump at 200 Hz, this is probably due to the mid-range speaker. For the calculations the highest of the two peaks is used.

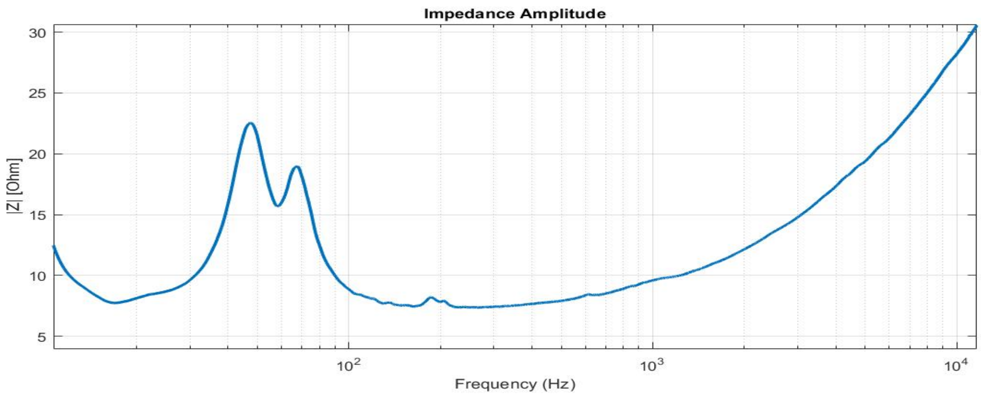

Figure 3.4: Absolute value of the impedance as function of the frequency, for woofer 1.

3.3.3. Acoustical measurement results

Alongside the electrical impedance measurements, acoustical short range measurements were also taken from the speaker drivers.
Figure 8.15 shows the acoustical frequency response of the midrange driver is most linear between ~200 Hz and ~2.4 kHz. This gives an indication of the most optimal range for the driver to be used in, and when the crossover frequencies have been chosen, these boundaries will be implemented using passive filters. Acoustic frequency response graphs of the other speaker drivers can be found in the appendix, section 8.2.5. Based on figure 8.16, the optimal range of the tweeter is determined as ~1 kHz to ~12 kHz, but since there is no speaker driver to handle even higher frequencies, it shall be used for frequencies above 12 kHz as well.
As for the woofers, per figure 8.17 and 8.18 the optimal range for the woofers is determined as ~60 Hz to ~2 kHz. Because this overlaps significantly with the range of the midrange speaker, and since both woofers have a fairly sharp dip in power response [fig. 8.17, 8.18] around 1 kHz, it seems better to use the woofers up to only 700 Hz, and because there are no other speaker drivers to handle frequencies under 60 Hz, the woofers shall also be used to display all lower frequencies. The acoustical phase response [figure 8.19] of the midrange driver is very different from the impedance phase shift. This might turn out to be a problem later on if two speaker drivers are out of phase at the crossover frequency. Acoustic phase response graphs of the other speaker drivers can be found in the appendix, section 8.2.5. The impulse response [figure 8.23] is not very interesting for our purposes, but it shows how quickly the speaker driver comes to a standstill after it has been given a single impulse. This says something about how over- or underdamped the mechanical mass-spring system of the speaker driver is. The midrange driver is fairly large and thus takes a few milliseconds to recover. Comparing this to the impulse response of the tweeter [figure 8.24], the tweeter recovers much more quickly because its cone is simply much smaller and lighter. Impulse response graphs of the other speaker drivers can be found in the appendix, section 8.2.5.

4

Passive filters

4.1. Introduction

The goal of Loudspeaker analysis was to establish the electrical and acoustical characteristics of each individual speaker driver, with these results the passive filters were able to be designed. How this is done can be read in this report, the main goal of project 'Booming Bass' is to design an audio system that reproduces a music signal. As explained in [Chapter speaker analysis] during this project a four-way speaker will be used. The complete frequency range of this system ranges from 20Hz until 20kHz. This is divided into three display regions, each of these regions is represented by a speaker element. The Woofer (or bass), Midtoner and Tweeter.

In order to obtain a good overall representation of the music signal, the speaker elements have to only display the selected region of the signal that suits within the range of that element. The Woofer, Midtoner, Tweeter are suitable for the low, mid and high frequencies respectively. If the speaker receives a signal that is not within their range, distortion of the signal and even damage to the speaker element can occur. To make sure this does not happen speaker filters are used.

4.1.1. Foreknowledge

The filter in an audio system is an electrical component which is designed to separate incoming audio signals. For EPO-1 it is required to design a passive three-way filter; this filter consists of three components; a Low Pass Filter (LPF), Band Pass Filter (BPF) and High Pass Filter (HPF).
The LPF lets frequencies lower than its cutoff frequency pass and blocks the frequencies higher than its cutoff frequency. The BPF only passes the frequencies within their band-width (explained in the previous chapter), it blocks the frequencies outside of this band-with. For the HPF the frequencies above its cutoff frequency are passed the ones below are reduced. This was a quick explanation of each filter and its function.
A passive filter only consist of passive components like a capacitor,inductor and resistor. These filters will generate three output signals to give each speaker a signal in its frequency range. The filters will generate

three output signals to each of the speaker elements. The sum of these signals must provide a flat acoustic transfer function[1]. To get a flat total transfer function the LPF and the HPF side of the BPF need to have the same slope so the sum of these slopes becomes 0. The same holds for the LPF side of the BPF. The electrical transfer function H(f) is the ratio between the output and the input signal as a function of the frequency(see equation 4.1). The difference between electrical- and acoustical transfer function is that the electrical transfer function gives the amplification and the phase of the speaker as a function of the frequency.
The acoustic transfer function as shown in equation 4.2 , "consists of two different types of signals you offer an electrical input signal and measure the resulting acoustic pressure signal P_{out}" [[1],p.149]. How to measure the acoustic pressure is discussed in the measurements chapter.

$$H_{electrical}(f) = \frac{U_{out}(f)}{U_{in}(f)} \tag{4.1}$$

[1] A flat transfer function means that the transfer is independent of the frequency.

$$H_{acoustic}(f) = \frac{P_{out}(f)}{U_{in}(f)} \tag{4.2}$$

4.1.2. Process

When designing a filter the frequency dependent impedance of the speaker elements has to be corrected with a Zobel network , when used in higher frequencies. This impedance only occurs at high frequencies this discussed more in depth in the section[Zobel network].There are 2 regions were the filters are crossing over, between the woofer and the midtoner and between the midtoner and the tweeter. For these regions the cut-off frequencies need to be determined. After determining the the cutoff frequencies the order of each filter can be chosen. Thereafter, the transfer functions need to be derived. If there is a Zobel Network needed this has to be determined. With these transfer functions and the potential use of a Zobel Network the individual components can be calculated. When the components are calculated, simulations can take place, these simulations result in bode plots of each individual filter and the power transfer of the whole circuit. If the filter simulations satisfy the needs they can be built. After building the filters its electrical transfer and acoustic transfer can be measured. There might be adjustments needed like another cutoff frequency or a volume adjustment. All of this will be explained in depth in the upcoming chapters.

4.2. Design Methodology

4.2.1. Zobel network

General information

A Zobel network(ZN) is a Resistor and a Capacitor in series that are in parallel with the speaker element as shown in figure 4.1.

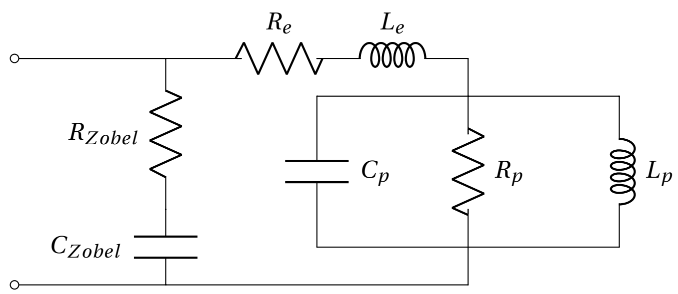

Figure 4.1: Model of a speaker element with Zobel Network

The reason this network is used is to correct the impedance of the voice coil Le as shown in figure 4.1.

> This is necessary because at high frequencies this can cause for the voltage to increase proportionally by the voltage division of the series impedance [[1], p.159].

Impedance correction is not needed for low frequencies because the impedance is nearly equal to the DC resistance R_e.

> When the ZN is applied the combination of impedance's of the speaker element and the ZN is completely real[2] and can be approximated as a resistance.[1]

Used Zobel networks

After taking a look at the impedance measurements done by the Loudspeaker analysis. The decision was made to use a ZN for the Midtoner and the Tweeter, this is based on the impedance measurements as shown in figure 8.27 and 8.28 in the appendix. As discussed in the previous part, a Zobel Network correction is not needed for low frequencies. That is why a ZN is not included in the LPF. Equations 4.3 & 4.4 that are used to calculate C-zobel & R-zobel are seen below and are proven during the tutorial Chapter 16 [1]

[2] Completely real means that there is no imaginary part present

$$C_{zobel} = \frac{L_e}{R_e^2} \tag{4.3}$$

$$R_{zobel} = R_e \tag{4.4}$$

	Midtoner	Tweeter
$R_{zobel}(\Omega)$	4.1	4.3
$C_{zobel}(F)$	33μ	3.38μ

Table 4.1: Calculated values Zobel network

Cutoff frequencies

The cutoff frequency is the frequency at which the transfer function is dropped to 70.7% of its maximum value [2]. Based on the frequency response & impedance measurements, of the three different speaker drivers the cutoff frequencies were picked.To not get the resonance peak as shown in figure 8.27 in our midtoner range, the cutoff frequency was set to 350 Hz for the low-pass filter and simultaneously for the high-pass of the band-pass filter. The cutoff frequency at the low-pass of the band-pass filter was set to 2000 Hz. The reasoning behind that was, the frequency response as shown in figure 8.29 is fairly flat. Furthermore the resonance peak as shown in figure 8.28 is before 2000 Hz, so the simplified speaker model can be used like discussed in the previous chapter.

Simplifying

The whole circuits shown in figure 4.1 will be represented by $Z_l s$ from now when using a Zobel network, to make the upcoming circuits less complicated. For the LPF and the BPF the same circuits but minus the Zobel network will be represented by Z_{ls}

4.2.2. Filter design

Low-pass filter

A LPF is designed to reduce the overlap between the band-pass and the low-pass filter. Every first order circuit either attenuates or weakens with 20dB per decade for frequencies lower that the cutoff frequency. Second order circuits attenuate with 40dB per decade by the 2 poles in the transfer function see equation 4.5.
To keep a overall phase consistent circuit, a second order low-pass filter is needed. A second order LPF consists of a Capacitor an inductor as seen in figure 4.2. The schematic of the actual circuit that is simulated is shown in figure 8.30.

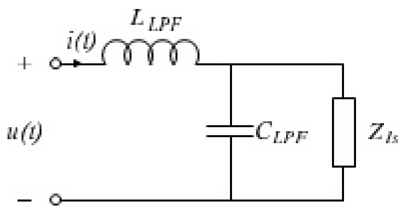

Figure 4.2: A model of a second order Low-pass filter

The output of a second order LPF can be described as:

$$U_{uit}(\omega) = \frac{1}{1 - \frac{\omega^2}{\omega_0^2} + j\frac{\omega}{Q\omega_0}} \cdot U_{in} \tag{4.5}$$

The resonance frequency (ω_o) and the quality factor (Q) [*Manuel* ,p.160] can be seen below:

$$\omega_0 = \frac{1}{\sqrt{L_{LPF} C_{LPF}}} \tag{4.6}$$

$$Q = Z_{ls}\sqrt{\frac{C_{LPF}}{L_{LPF}}} \tag{4.7}$$

The equations 4.6 & 4.7 with 2 unknown variables can be rewritten to find L_{LPF} & C_{LPF}. To obtain a maximum flat filter transfer Q is equal to $\frac{1}{\sqrt{2}}$.

$$L_{LPF} = \frac{Z_{ls}}{Q\omega_0} \tag{4.8}$$

$$C_{LPF} = \frac{L_{LPF}Q^2}{Z_{ls}^2} \tag{4.9}$$

High-pass filter

A High-Pass filter (HPF) is a kind of filter that is designed to pass all frequencies above the cutoff-frequency. This kind of filter works perfectly with the Tweeter. The reproduction of bass frequencies can damage this element. A first-order HPF is formed when the output of a Resistor-Capacitor(RC) is taken off the resistor. The second order HPF contains a capacitor in parallel with an inductor as shown in figure 4.3 and the voltage is taken of Zls. As instructed in the manual a HPF of the second order had to be constructed.The cutoff frequency was chosen at about 2000 Hz.

The output of a second order HPF can be described as:

$$U_{uit}(\omega) = \frac{-\frac{\omega^2}{\omega_0^2}}{1 - \frac{\omega^2}{\omega_0^2} + j\frac{\omega}{Q\omega_0}} \cdot U_{in} \tag{4.10}$$

The resonance frequency (ω_o) and the quality factor (Q) are derived from the equation 4.10 and can be seen below:

$$\omega_0 = \frac{1}{\sqrt{L_{HPF}C_{HPF}}} \tag{4.11}$$

$$Q = Z_{ls}\sqrt{\frac{C_{HPF}}{L_{HPF}}} \tag{4.12}$$

ω_0 and Q are known thus equations 4.11 & 4.12 can be solved for L_{HPF} & C_{HPF}.

$$L_{HPF} = \frac{Z_{ls}}{Q\omega_0} \tag{4.13}$$

$$C_{HPF} = \frac{1}{\omega_0^2 L_{HPF}} \tag{4.14}$$

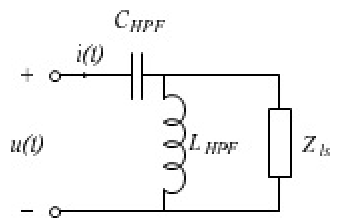

Figure 4.3: A model of a second order High-pass filter

Band-pass filter

A Band-Pass filter(BPF) is a kind of filter that is designed to pass all frequencies within a range of frequencies. The frequencies outside of this are being cancelled. This kind of filter is used for the mid toner or also named mid range woofer of a speaker. A RLC circuit provides a simple BPF-filter when the output is taken off the resistor. Another way to create a this filter is to combine both a Low-Pass filter and a High-Pass filter. If the cut-off frequencies are close together, the circuit has to be analyzed as one. But when the frequencies are far apart from each other (The cut-off frequency of the High-pass filter is more then 5 times bigger then the cut-off frequency of the low -pass filter), the circuit can be analyzed as two separate filters.

This also makes the calculations allot less difficult. The range of a BPF-filter depends fully on the characteristics of the circuit. The smaller the range that the circuit allows through, the higher the Q factor is. The equations needed to calculate the components for the BPF are (when Q = $\frac{1}{\sqrt{2}}$):

$$L_{B-HPF} = \frac{Z_{ls}}{\sqrt{2}\pi f_{c1}} \tag{4.15}$$

$$C_{B-HPF} = \frac{1}{2\sqrt{2}\pi Z_{ls} f_{c1}} \tag{4.16}$$

$$L_{B-LPF} = \frac{Z_{ls}}{\sqrt{2}\pi f_{c2}} \tag{4.17}$$

$$C_{B-LPF} = \frac{1}{2\sqrt{2}\pi Z_{ls} f_{c2}} \tag{4.18}$$

The resonance frequency (ω_o) of the the high and the low pass of the band pass filter can be seen below

$$\omega_0(B-LPF) = \frac{1}{\sqrt{L_{B-LPF} C_{B-LPF}}} = 2\pi f_{c1} \tag{4.19}$$

$$\omega_0(B-HPF) = \frac{1}{\sqrt{L_{B-HPF} C_{B-HPF}}} = 2\pi f_{c2} \tag{4.20}$$

with f_{C1} = 350 Hz and f_{C2} = 2000 Hz

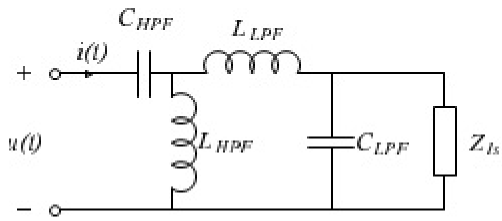

Figure 4.4: A model of a second order Band-pass filter

Component values

There are only certain values for resistors, capacitors and inductors available to use for this project. So there will be a deviation between calculated and actual values. See in the tables below the values for each filter:

	L_{LPF}(H)	C_{LPF} (F)
Calculated	4.56m	45μ
Used	4.7m	48μ

Table 4.2: Low-pass filter component values

	C_{HPF}(F)	L_{HPF}(H)
Calculated	14μ	4.5m
Used	15μ	4.4m

Table 4.3: High-pass filter component values

	C_{B-HPF}(F)	L_{B-HPF}(H)	L_{B-LPF}(H)	C_{B-LPF}(F)
Calculated	78.4μ	0.45m	0.46m	13.72μ
Used	80μ	2.8m	0.55m	15μ

Table 4.4: Band-pass filter component values

4.3. Simulations

4.3.1. LT Spice XVII simulations

After calculating the values for the components for each filter, it is necessary to check if the values give the right cutoff frequencies. LT Spice is used to simulate the filters, as shown in Appendix C. LT spice is a high performance simulation software with schematic capture and waveform viewer [3]. How LT spice works is, you have a blank sheet where you can remake the circuit you want. The options of different types of components is endless, we did not focus on the potential deviation that the components might have, because they are very small. A voltage source with a sinusoidal waveform is used that simulates the actual voltage that is going to the system when it is completely finished. It is expected to find the same cut off frequencies as the ones we based our calculations for the components on.

Low-pass simulation

The low-pass simulation was a pretty straight forward circuit, like mentioned before it consists of an inductor in parallel with a capacitor. The load that represents the speaker element of the woofer is Re, as shown in figure 8.30. The simulation, that is seen in figure 8.31 is almost the same as expected. The cutoff frequency was set to 350 Hz this is similar but with a slight deviation in the simulation 8.31. A decrease of 40dB/decade in the slope of the LPF indicates that the filter acts like a second order LPF

High-pass simulation

For the HPF we expected the cutoff frequency at 2kHz, after putting in the calculated values in the circuit shown in figure 8.34 it was found that the simulation matched what was expected. As showed in figure 8.35. An increase of 40dB/decade in the slope of the HPF, indicates that the filter act like second-order HPF.

Band-pass simulation

The BPF is the most complex circuit of all the filters, but with LT-spice remaking the circuit went fairly easy. The circuit that was made is shown in figure 8.32. The simulations for the BPF did meet our expectations when using the calculated values. The simulation is shown in figure 8.33. Both the low-pass side and the high-pass side of the band-pass filter have the cutoff frequency at around 350Hz and 2000Hz.

4.3.2. Matlab: Power Transfer

A simulation of the filters was also made so the effect of the different components could be tested. The Power Transfer as shown in figure 8.40 below, after looking at the graph that the most of it is flat and on the 0 dB line. In the beginning it wasn't flat in the mid section, there were a slight deviation plus a high amplitude in the mid section, but with changes to components and also using a volume adjustment(see the next paragraph) the BPF and transition from the LPF and to the HPF can be flatten. Also one thing to keep in mind is that the results of the simulation deviate from the real results. The real components in the filters aren't ideal and the simulation doesn't take the effect of the magnetic fields that inductors have on each other in consideration.
Four graphs are plotted, one for each filter (see figure 8.36 for the Low-pass filter, see figure 8.37 for the Band-pass filter, see figure 8.38 for the High-pass filter)and one for the total sum of the three filters and that by adding the transfer functions (complex and not absolute numbers).
The graph of the band-pass filter has a higher top than the other filters. this means that the midtoner produces a different volume at the same input voltage than the woofer and the tweeter. to solve this problem volume adjustment has to be applied. this is discussed in the following section.
The equation for the power transfer of a filter is given in equation 4.21.

$$G(\omega) = \left| \frac{U_{out}(\omega)}{U_{in}(\omega)} \right| \tag{4.21}$$

Volume adjustment

Volume adjustment is needed for the band-pass filter to acquire a flat total transfer. This means that the circuit as shown in figure 4.5 has to added to the band-pass filter.

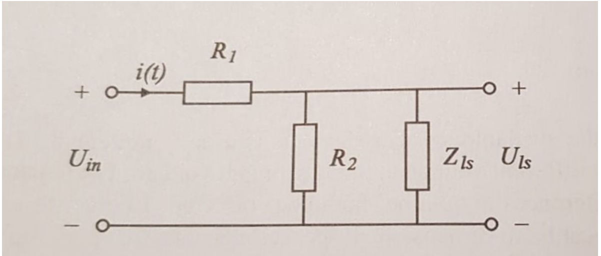

Figure 4.5: Volume adjustment

The circuit has to present an impedance to the filter which is equivalent to Z_{ls} and gives the desired damping. Equation 4.22 is used to adjust the volume.

$$\beta = 20\log_{10}\frac{Z_{ls}R_2}{R_1(Z_{ls}+R_2)+Z_{ls}R_2}(dB) \tag{4.22}$$

β represents the factor by which the power has to be adjusted expressed in db. Through this equation and the fact that the equivalent circuit should be equal to Z_{ls}, the values of R_1 and R_2 can be determined. These equations produce the values in table 5.

	$R_1\,(\Omega)$	$R_2\,(\Omega)$
Calculated values	1.3	30
Available	-	-

Table 4.5: values for the components of the volume adjustment

The power transfer with volume adjustment is depicted in Figure 8.40, for the matlab code used see Appendix F (Matlab code).

4.4. Measurements

4.4.1. Electrical Frequency Response

After simulating the individual filter and the complete power transfer it was time to finally build the filters and measure them. The first measurement is the electrical or electric frequency response. The electrical frequency response of the filters requires the following items:

- Computer (with LS-measure)
- jack-jack cables
- Buffer amplifier
- BNC cable

In order to measure the electrical frequency response, the setup that was used was the same as during the 'Frequency-dependent impedance measurements' done during the loudspeaker analysis.

> LS-measure generates an input signal which can be defined as a white noise, because it consists all frequencies with the same amplitude. The signal is then sent to the USB Sound card. The signal is further put through a voltage divider which consists of the unknown impedance (the speaker) and a resistor of 100 Ω used as reference. The reference resistor and the connections are located in a connection box which. is pre-given.The response and originally generated signal are sent back to the PC which is then processed.[Chapter Loudspeaker analysis]

Results of the Electrical frequency response

Low-pass filter has according to the measurement, as shown in figure 8.41 its cut off frequency at 312 Hz when the expected cut off frequency was 350 Hz. This deviation is 10.8% which is slightly bigger than 10 %, in [1] it said there can be a up to 10 % deviation. This part has to be adjusted.Because of the speakers internal components the measured cut off frequency can and most likely will deviate.

The high-pass filter measurement as shown in figure 8.42 does deviate a bit more in comparison to the LPF. The measured cutoff frequency is around 1770 Hz when the expected frequency was 2000 Hz. The band-pass filter looked very good in the simulation but when we measured the filter, its cutoff frequency on the low-pass side of the filter was around 1300Hz when we expected 2000Hz. This can be because of calculation errors or soldering errors. The errors are not yet found but will be included in the final report.
Considering that the exact components were not available and multiple components where used to correspond to the calculated values. The graphs can differ from the simulations due to the internal resistance of the components and also the mutual inductance of the inductors.

4.4.2. Acoustic measurements

During the acoustic frequency response measurements, the speaker elements and corresponding filters are being measured together. The Near field measurement will be performed with a microphone that is placed at a distance of 5cm from the speaker. And the far field measurement is at a distance of 1m. The measurement setup contains the following items:

- Computer (with LS-measure)
- jack-jack cables
- Buffer amplifier
- BNC cable
- Microphone
- Microphone amplifier

The measurement setup for the short and long distance acoustic frequency measurement is shown in chapter Speaker Analysis.

Results and discussion Acoustic Measurements

Near field measurements:
After measuring the electrical frequency response it was needed to measure each filter individually. The results of the LPF and HPF can be found in Appendix E. The BPF was not capable of being measured, because of the error during the electrical frequency measurements.
Looking at the HPF graph, it has a nice flat transfer from 2000 Hz, but it also has random peaks at lower frequencies. This can be because of the fact that the measurements are done in a noisy hall. The LPF has an low dB and irregular transfer from 20 Hz until 70 Hz, this is were the 'Booming-bass' will help out. Furthermore, the transfer as a whole seems to have a relatively low dB. This can be caused by not placing the microphone close enough or the volume was not turned high enough. These inconveniences might have influenced the measurements.
Far field measurement: The far field acoustic transfer as shown in figure 8.47 is not as flat as hoped. Particularly in the region between the LPF and the HPF of the BPF. The reason for this can be the phase difference of to filters. Also the sound in the noisy hall has to be taken into consideration, because the measurement is done at a long distance and the microphone picks up all the surrounding noises. But at higher frequencies starting from around 2000 Hz the acoustic transfer is rather flat. This means that the HPF works very good.

4.5. Conclusion and Recommendation

Three filters have been designed: a Low-pass filter (LPF), a Band-pass filter (BPF) and a High pass filter (HPF). The chosen cutoff frequencies were 350Hz for The LPF, 350Hz and 2000Hz for the BPF and 2000Hz for the

HPF. These were determined by looking at the acoustic frequency response. A Zobel network was presented to the speaker element to correct the frequency-dependent impedance. As can be seen when looking at the measured results, for each filter we have an electrical power transfer in which the curve holds a flat line. The results of the electric frequency are not exactly as expected en are different from the simulations due to the internal resistance of the components and also the mutual inductance of the inductors or soldering errors. Nevertheless the components available did not match with the calculated values. This would not have been a problem if there were enough components available to combine in order to acquire the needed value.

5 Power amplifier

5.1. Introduction

In the booming-bass project a power amplifier is designed, analysed, dimensioned, simulated, built and measured. This power amplifier ensures that the signal passing through the audio system that is designed in the booming bass project has a proper amplification in a desired frequency range. The amplifier discussed in this report had to be designed with the following requirements in mind:

1. The amplifier circuit has a non-inverting configuration.
2. The passband of the amplifier is 20 Hz - 40 kHz.
3. The voltage gain in the passband equals 25.
4. DC on the input signal should be blocked and may not be present at the output.
5. The DC offset of the op-amp may be maximally amplified by 1.

These requirements we're provided by the EE1L11 Manual [1, p. 124] First the composition of the circuit of the power amplifier must be determined. This is done using the provided general circuit from figure 17.6 in the EE1L11 Manual [1, p. 175] and the requirements. After designing the circuit, it's components must be dimensioned. This is done using calculations, these calculations can be roughly found by doing Tutorial 5 from the EE1L11 manual. After designing and dimensioning the circuit simulations are done in MatLab [2]. If the simulation results are desirable the circuit can be built and measurements are done to confirm the circuit copes with the requirements.

5.2. Design methodology

5.2.1. The Circuit

The circuit (Fig5.1) is made up of three parts. The first part is a passive high-pass filter, the second part a passive low-pass filter, and the last part an active non-inverting high-pass filter (Fig.5.2 Each of these circuits have an important role in filtering and/or amplifying the signal. Lastly a resistor and capacitor are used to control the mute function (Fig 8.55).

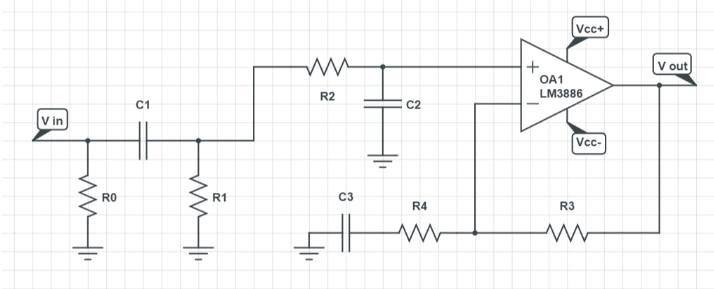

Figure 5.1: Complete amplifier circuit (without mute).

To begin with resistor R0 makes sure induced voltages are grounded, preventing noise. Resistor R0 is given 220 kΩ. Resistor R1and capacitor C1form together the first filter, a high-pass filter. Resistor R2and capacitor C2form together the second filter, a low-pass filter. Resistors R3 and R4and the operational amplifier OA1 form together the amplifier, amplifying the signal. Capacitor C3forms a third filter with resistor R4 but its main purpose is to ensure unity gain for DC offset. The Vcc-, Vcc+ and ground terminals of the power supply are attached to the respective terminals on the amplifier circuit. The power is then supplied to the op-amp OA1 (Vcc+ and Vcc-) which is a LM3886 audio power amplifier with a mute.

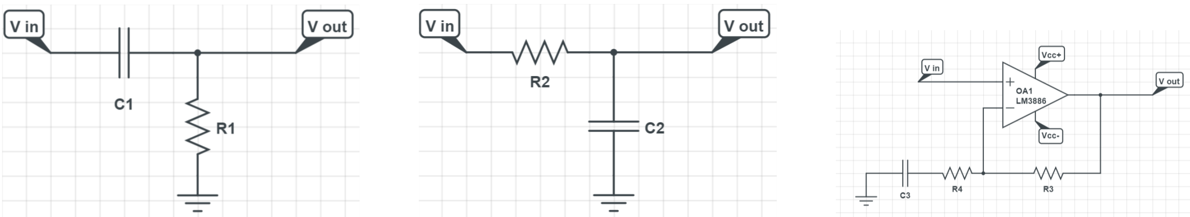

Figure 5.2: (from left to right) High-pass filter, Low-pass filter, Amplifier with high-pass filter.

Each circuit in (Fig. 5.2) acts differently at a given frequency. This can be visualized by their respective transfer function. The general equation for the transfer function of the voltage is:

$$H(\omega) = \frac{V(\omega)_{out}}{V(\omega)_{in}} \tag{5.1}$$

[[1], Eq. 14.2a, pp.613]

$V(\omega)_{out}$ & $V(\omega)_{in}$ can be calculated using nodal analysis. The transfer functions are then used to calculate the values of the components in the filters. (see derivations in Power amplifier Appendix A)

High-pass filter:

$$H(\omega) = \frac{j\omega R_1 C_1}{1 + j\omega R_1 C_1} \tag{5.2}$$

Low-pass filter:

$$H(\omega) = \frac{1}{1 + j\omega R_2 C_2} \tag{5.3}$$

Amplifier:

$$H(\omega) = 1 + \frac{j\omega R_3 C_3}{1 + j\omega R_4 C_4} \tag{5.4}$$

5.2.2. Calculating component values

The given specifications requires the passband to have a voltage gain of 25. Rearranging the general equation for a non-inverting op-amp gives the ratio of R3and R4for a gain of 25:

$$V_{out} = (1 + \frac{R3}{R4}) \cdot V_{in} \tag{5.5}$$

$$24 = \frac{R3}{R4} \tag{5.6}$$

[[[5]], pp. 23] recommends a resistance between 10kΩto 100k Ωfor R3. With the E12/E24 scale a few options are available (for all possible combinations see Power amplifier Appendix B):

$$R3 = 18k\Omega \;\; R4 = 750\Omega \;\; \frac{R3}{R4} = \frac{18000}{750} = 24 \tag{5.7}$$

$$R3 = 24\Omega \;\; R4 = 1k\Omega \;\; \frac{R3}{R4} = \frac{24000}{1000} = 24 \tag{5.8}$$

$$R3 = 36k\Omega \;\; R4 = 1.5k\Omega \;\; \frac{R3}{R4} = \frac{36000}{1500} = 24 \tag{5.9}$$

These three combinations give the desired gain for the amplifier. But because the amplifier is also a filter, the cut-off frequency has to be taken into account. Because of this, the value of capacitor C3, resistors R3and R4 need to be chosen together so that the cut-off frequency is well below 20Hz. The last requirement states that "The DC offset of the op-amp may be maximally amplified by 1." This means that the gain of the op-amp must be less than 1 for DC voltage. The cut-off frequency must therefore not be too low, otherwise DC voltage gets amplified. The values chosen are: R3 = 68k Ω, R4 = 2.8k Ω, C3 = 4.7F. The combination of these values gives:

Amplification:

$$1 + \frac{R3}{R4} = 1 + \frac{68000}{2800} = 25.29 \tag{5.10}$$

Cut-off frequency:

$$\frac{1}{2\pi \cdot 2800 \cdot 4.7 \cdot 10^{-6}} = 12Hz \tag{5.11}$$

This satisfies both requirements.
The high-pass filter has a cut-off frequency of $f_{-3dB} = \frac{1}{2\pi R1C1}$ and frequencies below 20Hz need to be suppressed, so f_{-3dB} = 20Hz and therefore $R1 \cdot C1 = \frac{1}{40\pi}$. Any combination will suffice. The parts chosen are: R1 = 24k Ω, C1=330nF and:

$$f_{-3dB} = \frac{1}{2\pi R1C1} = \frac{1}{2\pi \cdot 240000 \cdot 330 \cdot 10^{-9}} = 20.095Hz \tag{5.12}$$

The low-pass filter also has a cut-off frequency of $f_{-3dB} = \frac{1}{2\pi R2C2}$ but now frequencies above 40 kHz need to be suppressed, so f_{-3dB} = 40kHz and therefore $R2 \cdot C2$ =180000. Again, any combination will suffice. The parts chosen are: R2=1k, C2=3.9nF and:

$$f_{-3dB} = \frac{1}{2\pi R2C2} = \frac{1}{2\pi \cdot 1000 \cdot 3.9 \cdot 10^{-9}} = 40808Hz \tag{5.13}$$

The last part to consider is R5 (Fig. 8.55). Although not used for filtering noise or unwanted frequencies, it plays an important role for the amplifier.
This resistor connects the Vcc-to pin 8. This pin is used to mute the audio via a switch or in this case, in combination with C4=220nF, make the audio fade in and fade out, preventing popping sounds when turning on the amplifier. [[[5]], Fig. 44, pp. 15] shows the mute attenuation vs the current through pin 8. It shows that above around 0.8mA the audio is unaffected, but lower than that it starts to mute. The current of 1mA ensures the audio is not muted, so with the fact that the voltage at Vcc- is equal to 22V, the value of resistor R5 must be equal to $\frac{V}{I} = \frac{22}{10^{-3}} = 22k\Omega$.

5.2.3. Power dissipation

The power dissipated of the op-amp with the given values Rl = 8 Ω, Vcc = 44V = 2VDC, is equal to the total power dissipated minus the power dissipated by the load. So the first thing to be calculated is the load current. Because the output voltage is a sinusoidal signal, the RMS value can be used instead of the peak value. Using Ohm's law, the load current can be calculated and after that, the peak of the current signal. Again using Ohm's law the peak voltage is calculated and then at last with the supply voltage, the power dissipated:

$$I_{RMS} = \frac{V_{RMS}}{Rl} = \frac{4/\sqrt{2}}{8} = \frac{\sqrt{2}}{4}A \tag{5.14}$$

$$I_{top} = \frac{\sqrt{2}}{4} \cdot \sqrt{2} = \frac{1}{2}A \tag{5.15}$$

$V_t = 4$V

With these values and the given formula for power dissipation in [[7]] the result is:

$$P_{amp} = \frac{Vt}{Rb} \cdot (\frac{2V_{DC}}{\pi} - \frac{Vt}{2}) = \frac{4}{8} \cdot (\frac{44}{\pi} - \frac{4}{2}) = 68.12W \tag{5.16}$$

The obtained value is a bit higher, but still similar to the stated 68W in the title of [[5]]. This power produces a lot of heat and it can't be dissipated fast enough by the package itself. A heat sink needs to be attached to the package to ensure an operating temperature below the maximum of 85C.

5.3. Simulations

Simulations for this circuit are done with MatLab code that simulates the overall transfer function of the circuit. For determining the code a few reference points have to be made. The transfer function of the op amp shown in (Eq.5.4) is only true when the op-amp is assumed to be an ideal op-amp. In reality this is not the case. (Fig.5.3) shows the real circuit of the op-amp with the given specifications, this model will be used for the simulation: $R_{in} = 200k\Omega$ $\quad R_{out} = 10\Omega$ $\quad$ A = 10^5 $\quad V_d = V_+ - V_-$

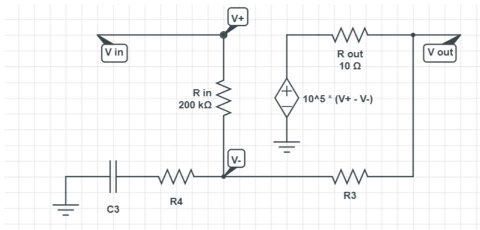

Figure 5.3: Real op-amp circuit.

This way the transfer function for individual parts of the circuit can be determined referring to the operational amplifiers, high pass filter and the low pass filter. Then the function for the complete circuit is determined by combining these transfer functions in Matlab. The code can be found in Power amplifier Appendix A. Running this code provides some of the following results:

In figure 8.53(Power amplifier appendix G) can be seen that the amplification within the specified bandwidth is as specified by the requirements. In the figures provided in Power amplifier Appendix D it can be seen that a -3dB bandwidth was also achieved.

5.4. Measurements

5.4.1. Measurements setup

The setup used for measuring the output of the amplifier consists of:

1. Oscilloscope
2. Function generator
3. Power supply
4. Amplifier

The power supply that was designed previously, is connected to the amplifier circuit. The output of the function generator is connected to both the amplifier and the oscilloscope. This way the function generator's output signal can be measured alongside the amplifier's output signal, which is also connected to the oscilloscope. The oscilloscope is set up in such a way that the peak to peak values of both signals are visible, these values are then used to calculate the gain.

5.4.2. Measurement results

Measurements are done over the whole required frequency spectrum at a low resolution. Only around the cutoff frequencies, f = 20Hz and f = 40kHz, a higher measuring resolution is used.
All measurement results are put into a table (Power amplifier Appendix D) and when put together in a graph with the simulated response (Fig.8.54 in Appendix G) it becomes clear that the real circuit behaves almost exactly as simulated. The graph shows that at frequencies between 100Hz and 40kHz the gain is close as simulated. Thus being 25. Outside those frequencies the amplification goes to 1. That means any DC offset created by the op-amp won't be amplified. The cutoff frequencies are also clearly visible, creating the desired pass-band. The lower and higher frequencies are suppressed relative to the passband. Table (5.1) shows the design requirements and the measured results of the built amplifier.

Required specifications	Measured results
For f = 20Hz-40kHz, A=25	Gain for 20Hz <f <40kHz are near 25
For f <20Hz A<25	Gain for f <20Hz drop below 17
For f >40kHz A<25	Gain for f >40kHz drop below 18
DC may not be present on the output	Voltage over Vout to ground is zero
DC offset amplification is max: 1	Gain for f <5Hz are equal to 1

Table 5.1: Specifications vs Results

5.5. Conclusions and Recommendations

In conclusion, the requirements of the circuit are compared with what was achieved. First the amplifier circuit had to have a non-inverting configuration. This was achieved as can be seen in the schematic of the circuit (figure 2.1). DC signals are filtered out. The calculated and therefore achieved cut-off frequencies were a bit off, 20Hz vs 20.095Hz and 40kHz vs 40.808kHz, but both are less than 2% off their target frequency. The DC offset created by the op amp has a gain of 1, this means the DC offset won't be amplified as is requested. The voltage gain in the passband is 24.62, again a bit off the target but also less than 2% off. In the end the amplifier has its shortcomings, but it comes really close to achieving the desired results. The cut-off frequencies could have been closer to the target by choosing different sets of resistors and capacitors. Because the components had more functionalities, choosing the right values became more complex. Not being limited by the E12/E24 series of components for example would definitely give better solutions. The resulting amplifier is close enough to what is required to not implicate the functionality of the device.

6

Booming bass

As explained in the manual **??**p. 125, section 12.6.5]coursemanual, each woofer together with the loudspeaker cabinet forms a second order high-pass filter, with corner frequency of 60 Hz [figures 8.17, 8.18]. Thus, below a frequency of 60 Hz, the frequency response declines with 40 dB per decade, or 12 dB per octave. This means the bass rendering of the system is limited, and so the last challenge of EPO-1 is to build a component, to be named "Booming Bass", to be added to the existing system, with the goal of extending the bass range by one octave.

6.1. Passive

To extend the bass range by one octave, the cutoff frequency must be moved from 60Hz to 30Hz. This means the transfer function of the booming bass should have a rolloff of 40 dB per decade or 12 dB per octave between 30 and 60 Hz. Above 60 Hz, the transfer should remain constant.
This requirement can be achieved either by applying a passive second order low pass filter with a cutoff frequency of 30 Hz. To prevent this filter from cutting off all frequencies above 60 Hz, it should have a second corner frequency at 60 Hz, keeping the transfer at a constant -12dB after the second corner frequency. The disadvantage of a passive Booming Bass is that the overall output power decreases, since all frequencies above 60 Hz are damped by -12dB. The advantage of a passive solution, however, is that it is relatively simple to build. The filter can be placed either before or after the Power amplifier, but placing it before the Power amplifier allows the usage of smaller and cheaper components, making the Booming Bass more compact.

6.2. Active

Another option is to apply an active filter or bass booster, using an OP-AMP[1]. This has the advantage of not losing power at higher frequencies, because the active filter can amplify the signal at frequencies below 60 Hz instead of damping the signal above 30 Hz. To achieve this, a high-pass filter consisting of a resistor and a capacitor is placed in the feedback loop of the OP-AMP. The capacitor is in parallel with a resistor to limit the gain. This gives the circuit in figure 9.1. At frequencies well above the corner frequency, the capacitor C_1 will behave as a short circuit between the output and inverting input of the OP-AMP, making the circuit behave as a voltage follower. At frequencies well below the corner frequency, the C_1 will behave like an open circuit, making the gain of the circuit equal to $1 + \frac{R_1}{R_2}$.

6.3. Execution

Due to the advantages of being compact and not losing power at higher frequencies, an active solution was chosen. The proposed OP-AMP circuit of figure 9.1 will be worked out.
The maximum gain of the circuit is limited of the power supply. The power supply cannot deliver more than $\pm 22V$, and since the Power amplifier has a gain of 25, the input signal for the Power amplifier cannot be more than $\frac{22}{25} = 0.88V_{pp}$. The maximum input signal is given to be $400mV_{pp}$, so the gain of the Booming Bass cannot be higher than 2.2.

[1] Operational Amplifier

Using a first order high-pass filter in the feedback loop of the OP-AMP will result in a maximum rolloff of 20dB/decade or 6dB/octave. The current rolloff of the woofer, however, is twice that. Adding a second high-pass filter in series gives a maximum rolloff of 40dB/decade or 12dB/octave, which is our goal. An example circuit is shown in figure 9.2. This, however, resulted in unexpected behavior of the circuit. The gain exhibited a bump which was higher than the DC gain from R_1 and R_2. Also, the corner frequency shifted to a higher frequency. Multiple TA's[2] and a tutor could not clearly explain this, and the partial explanations some of them gave were different from each other. The transfer function, when plotted, did not exhibit this behavior either. For this reason, the values of the components were from this point found by trial and error, adjusting values and simulating the circuit to test if it had the desired effect. Also, a high pass filter was added to the input to prevent DC from being amplified by the circuit.

The final circuit schematic can be seen in figure 9.3. The gain of the simulated circuit with respect to frequency is shown in 9.4.

[2]Teacher Assistants

7 Conclusion

To conclude this project a final look is taken on the achieved results. The different parts have been constructed an thoroughly tested. The power supply met all the requirements and achieved a stable DC signal to power the system. The behaviour of each of the speakers were tested to then create the suitable filters for a level frequency response. The amplifier and the booming bass component made sure the power is high enough for a good volume and an even wider frequency range respectively. Combining all these elements gives a system specifically made for the given speakers.

Looking back on the course of development of this system it becomes clear that a lot could have done better, timing and communication were not always on point. Also deadlines were almost never followed and resulted in some stressful moments. A small group of only six people made it hard to finish everything on time. But everyone pulled through and put in all their effort until the end. This resulted into a completed and successful project where all assignments were completed, and the working system is the result of all of that.

Bibliography

[1] Student Manual Lab Courses EE Semester 1
Dr.ir. G.J.M. Janssen ,Dr.ir. J.F. Creemer, Dr.ir. D. Djairam, Dr.ir. M. Gibescu, Dr.ing. I.E. Lager, Dr.ir. N.P. van der Meijs, Dr.ir. S. Vollebregt, Dr.ing. B. Roodenburg, Dr. J. Hoekstra, Dr. S. Izadkhast, Ing. X. van Rijnsoever,

[2] Fundamentals of Electric Circuits
Christopher K. Alexander/ Matthew N.O. Sadiku,
Sixth edition, 2017

[3] LTSpice Information
www.analog.com/en/design-center/design-tools-and-calculators.html

[4] OrCAD Systems Corporation, OrCAD PSpice, Demo Version 9.1, Web Update 1, OrCAD Inc.

[5] Texas Instruments. (May 1999 - revised 2013). LM3886 Overture™ Audio Power Amplifier Series. Retrieved from http://www.ti.com/lit/ds/symlink/lm3886.pdf

[6]Cleve Moler, MatLab, R2018b, Mathworks

[7] G. de Graaf. Januari 2018. Power dissipation LM3886 “Eindversterker”. Retrieved from https://brightspace.tudelft.nl/d2l/le/content/126277/viewContent/1017820/View

[8] K. Thorborg. Traditional and Advanced Models for the Dynamic Loudspeaker. Videbaek, Denmark, 2012.

8 Appendices

8.1. Power supply

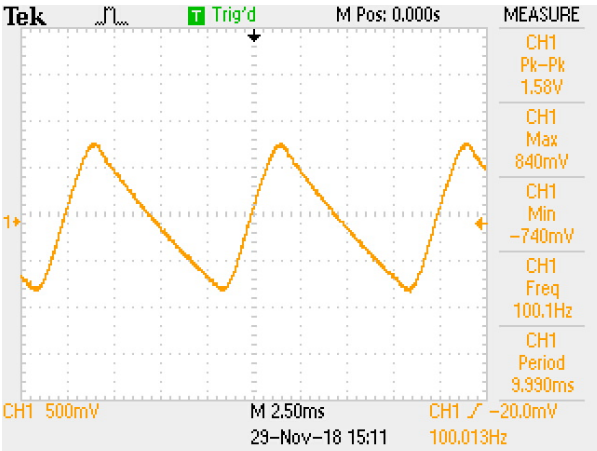

Figure 8.1: Measured ripple voltage at the positive terminal with a load of 1 Amp

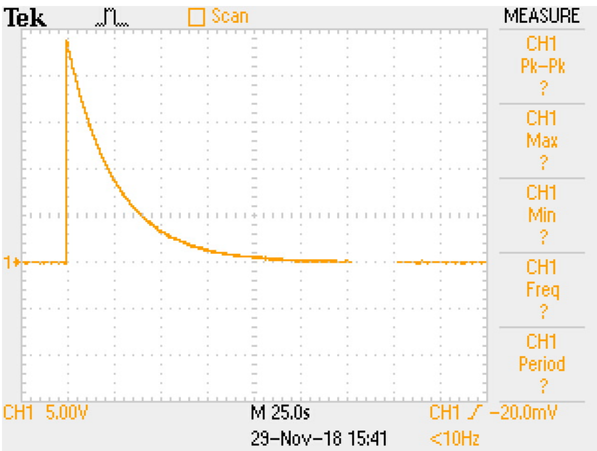

Figure 8.2: Discharging of the capacitor at the positive terminal.

8.2. Speaker analysis

8.2.1. Appendix A: Loudspeaker photos

Figure 8.3: The speaker we received. The four speaker drivers can be seen, from top to bottom: the midrange driver, tweeter, woofer 1 and woofer 2.

Figure 8.4: The type plaque on the back side of the speaker. Underneath it, the upper bass reflex duct is visible with a Styrofoam ball stuffed in it.

8.2.2. Appendix B: Speaker impedance models

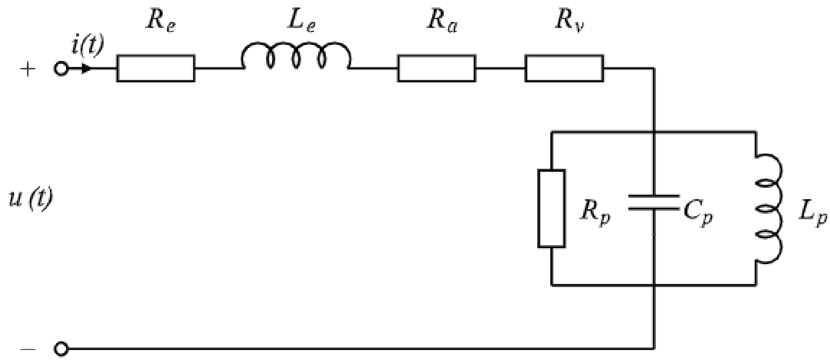

Figure 8.5: Impedance model for a speaker element.
[? , figure 14.3, p. 142]

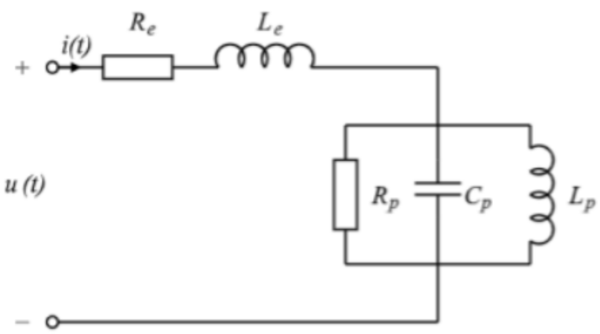

Figure 8.6: Simplified impedance model for a speaker element.
[? , figure 14.4, p. 142]

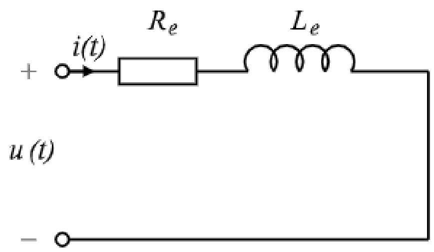

Figure 8.7: Simplified impedance model for frequencies above the resonant frequency.
[? , figure 14.6, p. 143]

8.2.3. Appendix C:Matlab code

Code to simulate impedance model 1

```
f = 2*logspace(1, 4, 100);  % array with frequencies 20 - 20,000 Hz
w = 2*pi*f;                 % Hz to rad/s

Re = 4.1;       % Ohm
Le = 349.4e-6;  % microHenry
Rp = 5.72;      % Ohm
Cp = 986.6e-6;  % microFarad
Lp = 3.121e-3   % milliHenry
Z = ((1i*w*Lp*Rp)./(1i*w*Lp + Rp - w.*w*Lp*Rp*Cp)) + Re + 1i*w*Le;

Z_A = abs(Z);       % array with impedance for all frequencies in f
dFi = angle(Z);     % array with phase shift (rad) for all frequencies in f
```

Code to simulate impedance model 2

```
f = 2*logspace(1, 4, 100);  % array with frequencies 20 - 20,000 Hz
w = 2*pi*f;                 % Hz to rad/s

Re = 4.1;       % Ohm
Le = 349.4e-6;  % microHenry
Z = Re + 1i*w*Le;

Z_A = abs(Z);       % array with impedance for all frequencies in f
dFi = angle(Z);     % array with phase shift (rad) for all frequencies in f
```

8.2.4. Appendix D: Figures related to measurements

Measurement setups

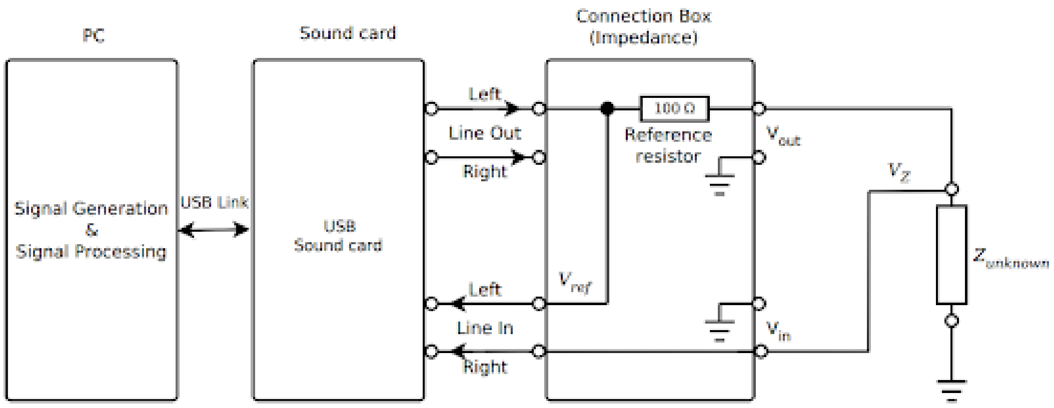

Figure 8.8: Measurement setup for measuring an unknown impedance. [? , figure 14.10, p. 146]

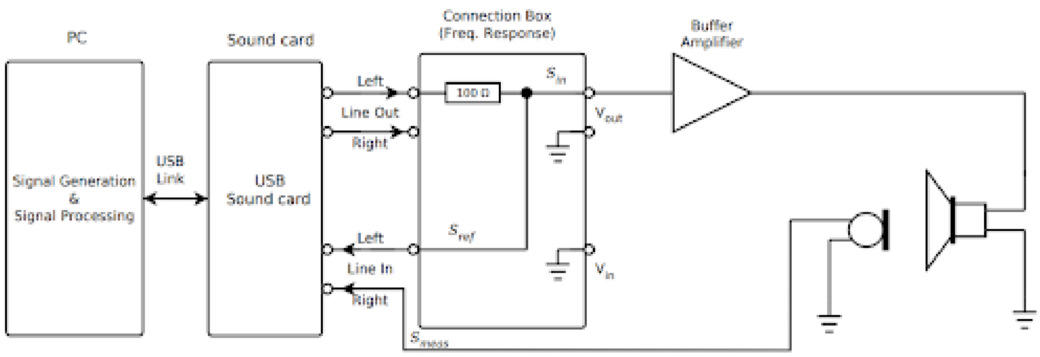

Figure 8.9: measurement setup for short distance acoustic measurement. [? , figure 15.5, p. 153]

Impedance model

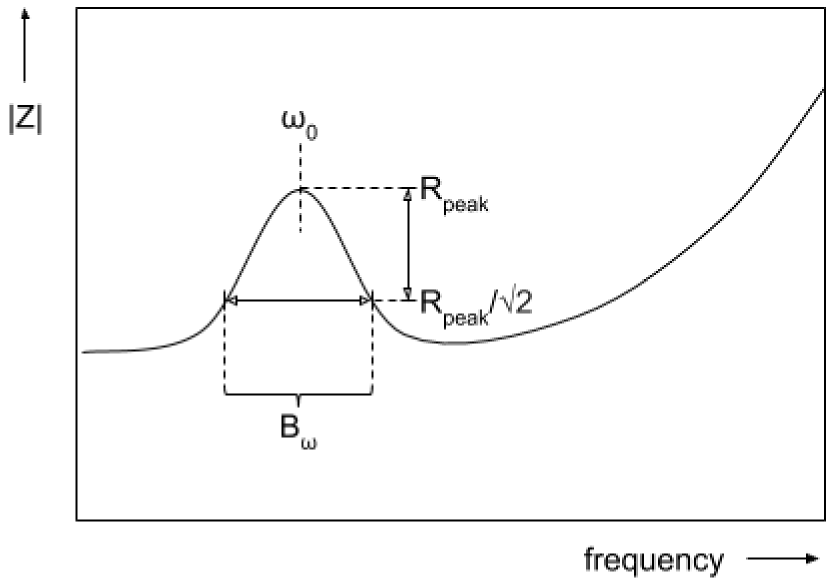

Figure 8.10: Typical impedance amplitude graph from a measurement, showing important parameters.

8.2.5. Appendix E: Matlab figures

Simulation comparisons

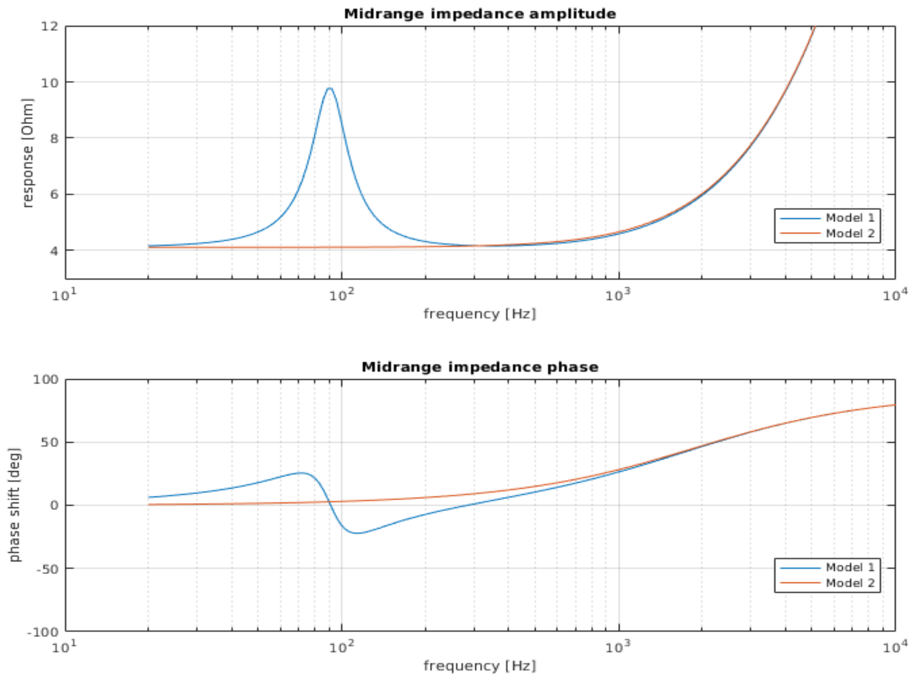

Figure 8.11: Impedance amplitude and phase graphs from the impedance simulation of the midrange speaker driver, comparing model 1 and model 2.

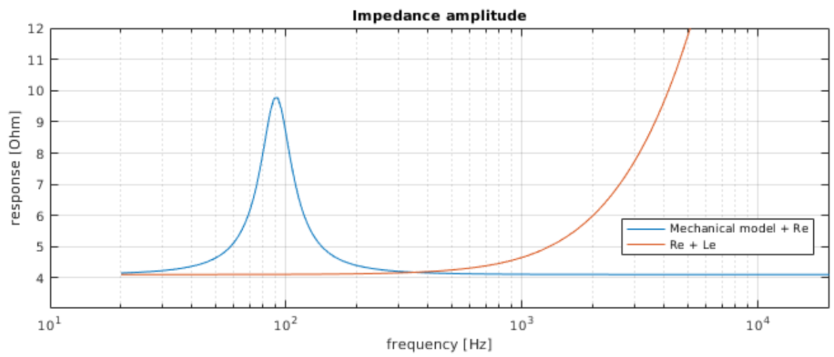

Figure 8.12: The combined impedance from the mechanical model approaches zero and the impedance from the self-inductance L_e increases significantly above the resonance frequency. This simulation was made using the values for the midrange driver.

Impedance simulation and measurement comparison of the woofers

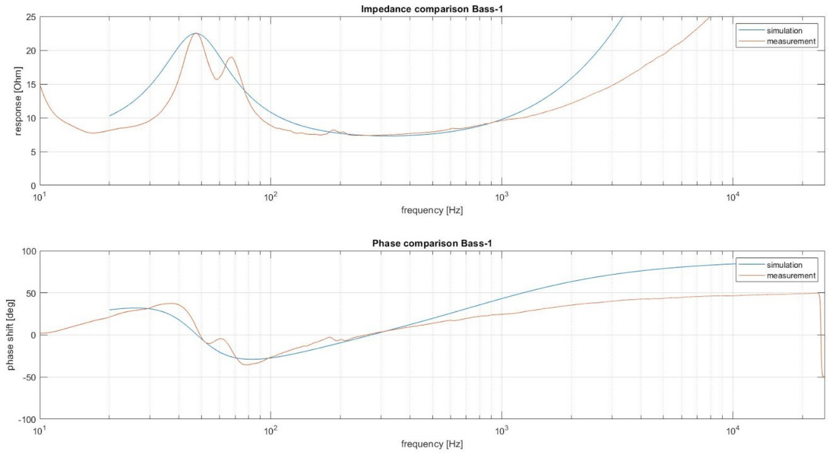

Figure 8.13: Comparison of measured and simulated impedance amplitude and phase of woofer 1

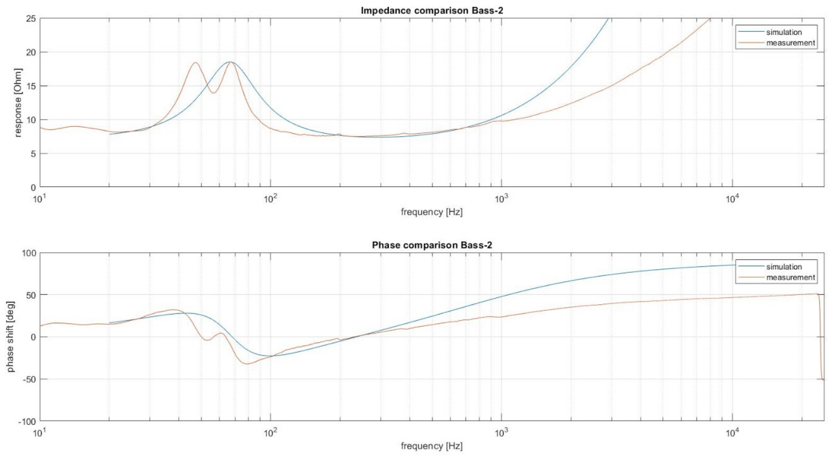

Figure 8.14: Comparison of measured and simulated impedance amplitude and phase of woofer 2

Acoustical frequency response graphs of the tweeter and woofers

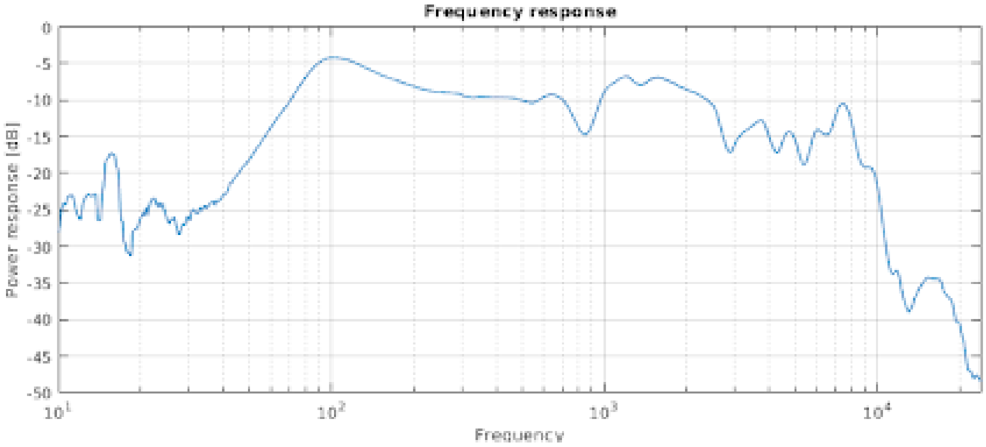

Figure 8.15: Acoustical frequency response of the midrange driver.

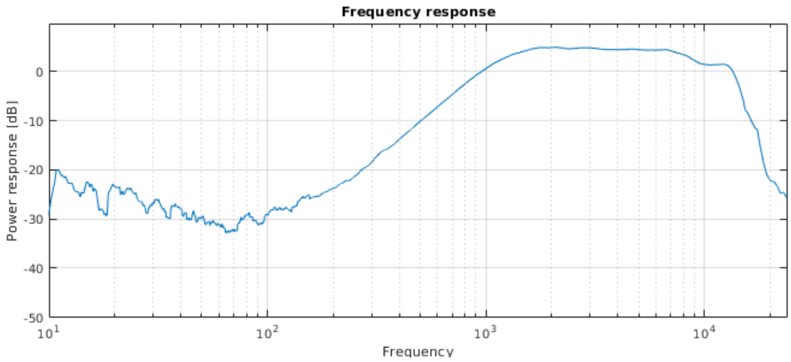

Figure 8.16: Acoustical frequency response of the tweeter

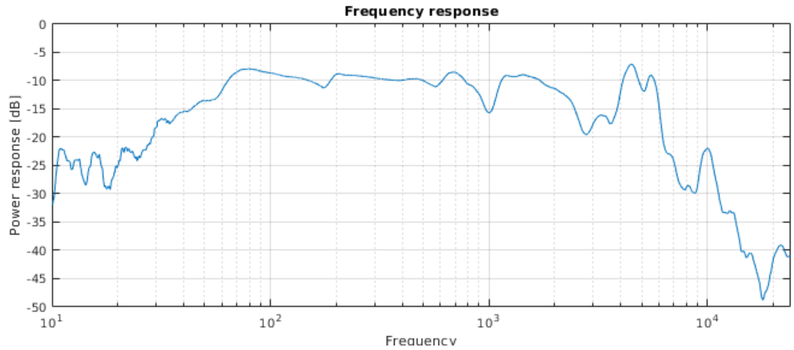

Figure 8.17: Acoustical frequency response of woofer 1

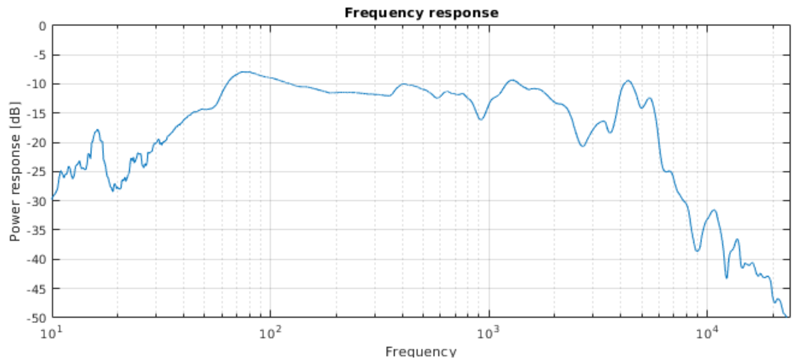

Figure 8.18: Acoustical frequency response of woofer 2

Acoustical phase response graphs of the tweeter and the woofers

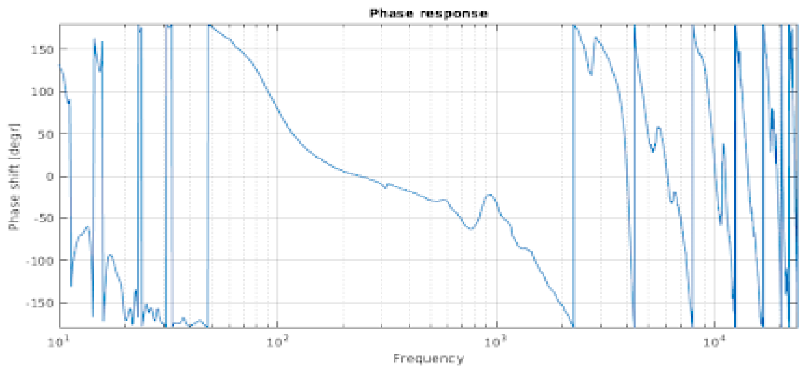

Figure 8.19: Acoustical phase response of the midrange driver.

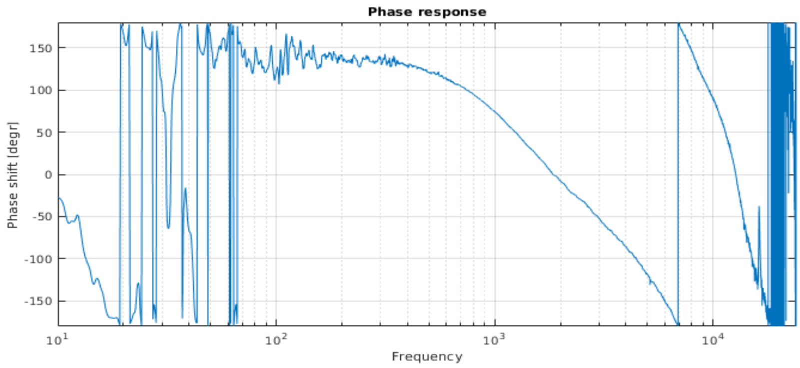

Figure 8.20: Acoustical phase response of the tweeter

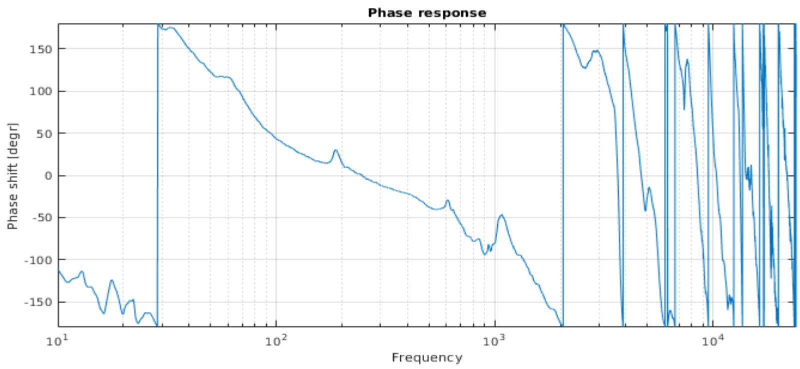

Figure 8.21: Acoustical phase response of woofer 1

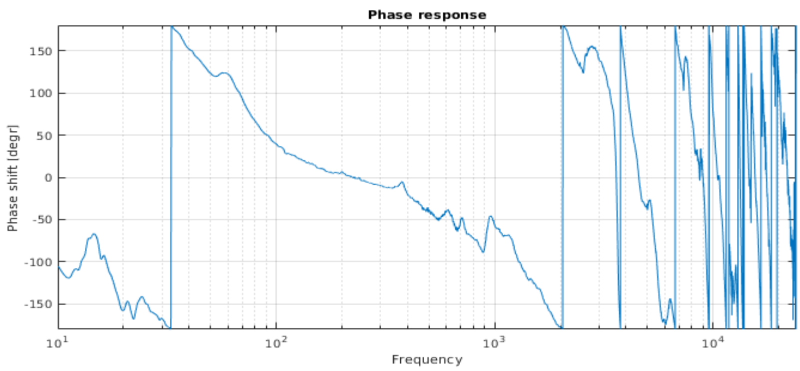

Figure 8.22: Acoustical phase response of woofer 2

Impulse response graphs of the speakers

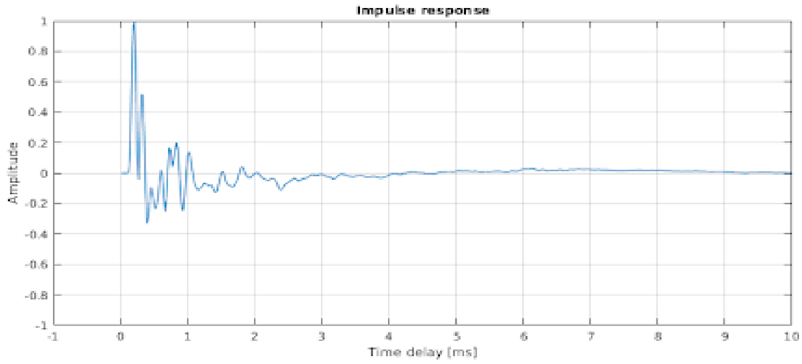

Figure 8.23: Impulse response of the midrange driver.

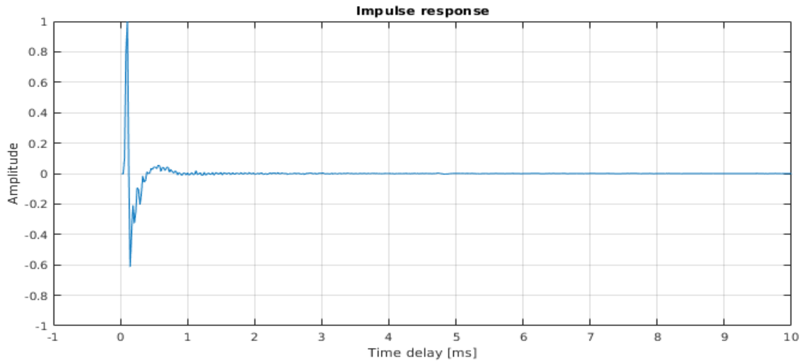

Figure 8.24: Impulse response of the tweeter

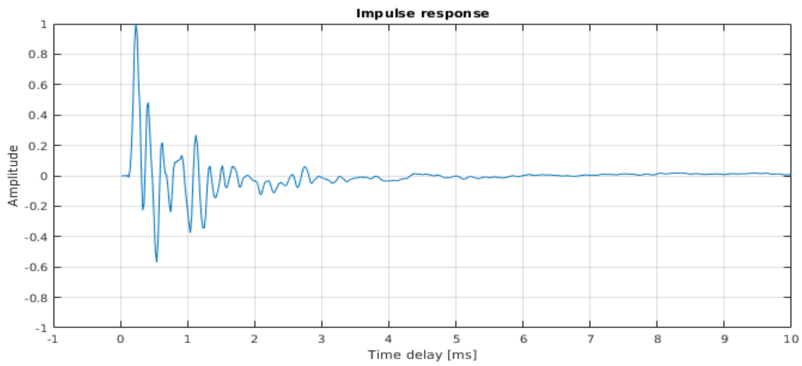

Figure 8.25: Impulse response of woofer 1

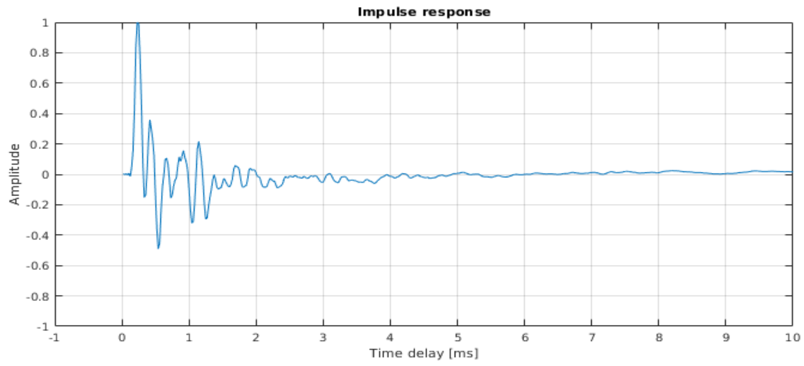

Figure 8.26: Impulse response of woofer 2

8.3. Passive filters

8.3.1. Appendix A: Impedance measurements

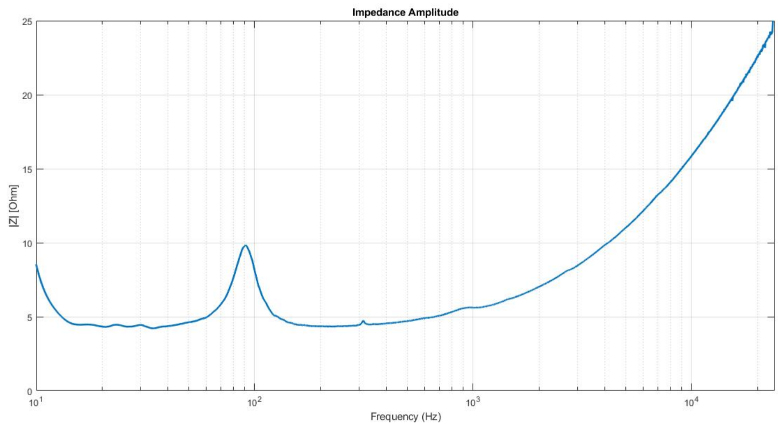

Figure 8.27: Impedance measurement Midtoner

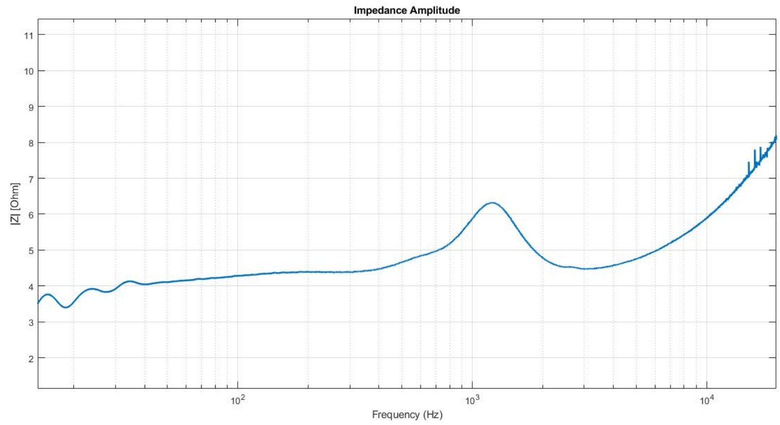

Figure 8.28: Impedance measurement Tweeter

8.3.2. Appendix B: Frequency response measurements

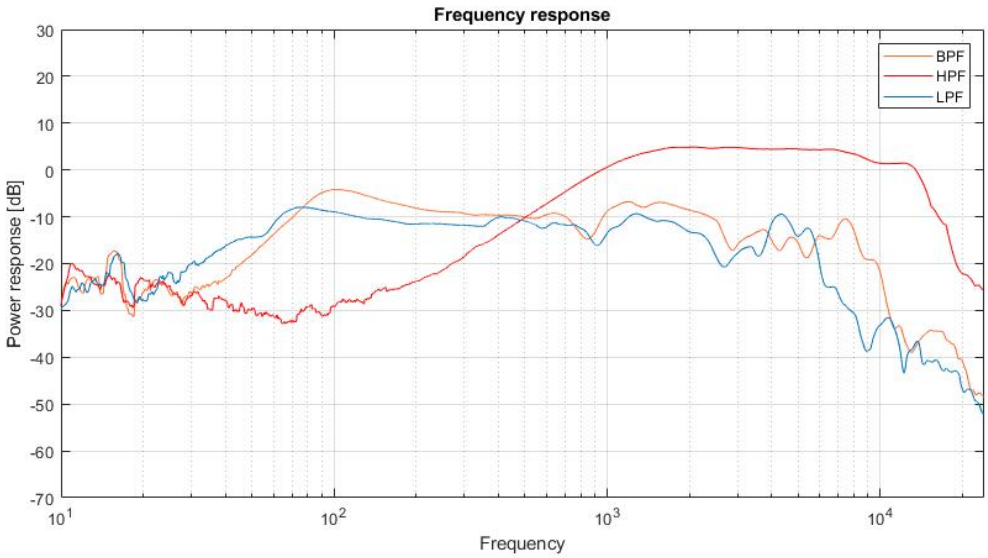

Figure 8.29: Frequency-response measurement Woofer

8.3.3. Appendix C: LT spice XVII and Matlab simulations

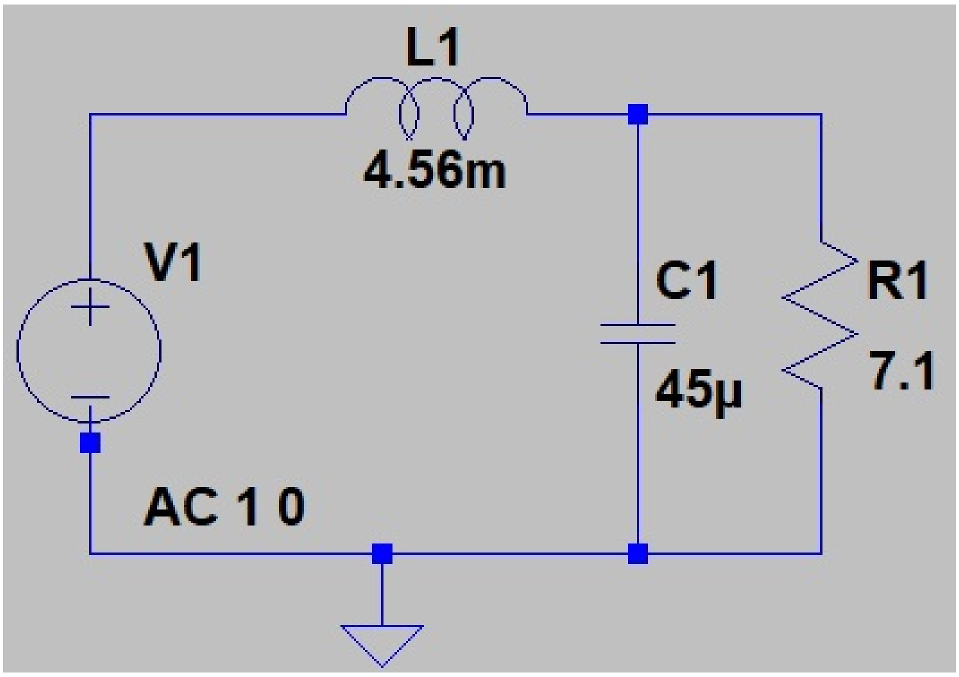

Figure 8.30: Low pass filter circuit

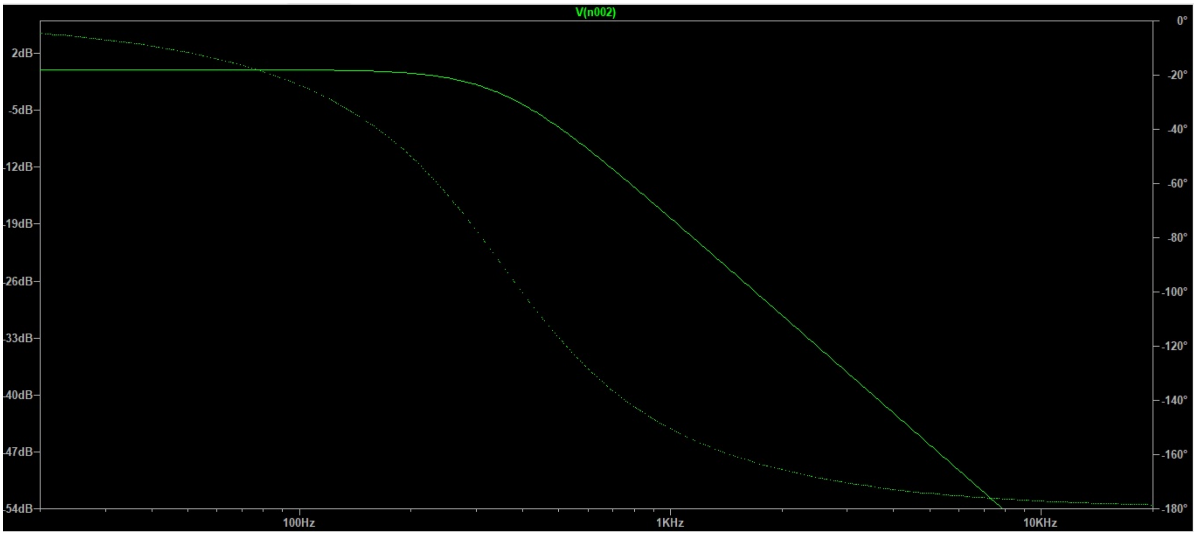

Figure 8.31: Low-pass filter simulation

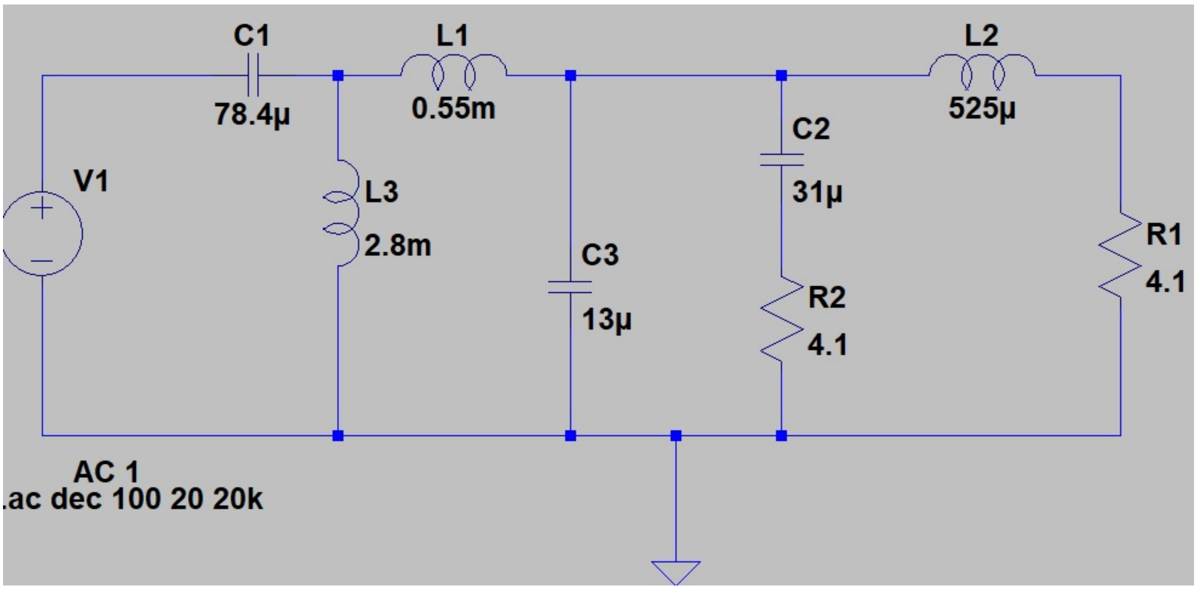

Figure 8.32: Band pass filter circuit

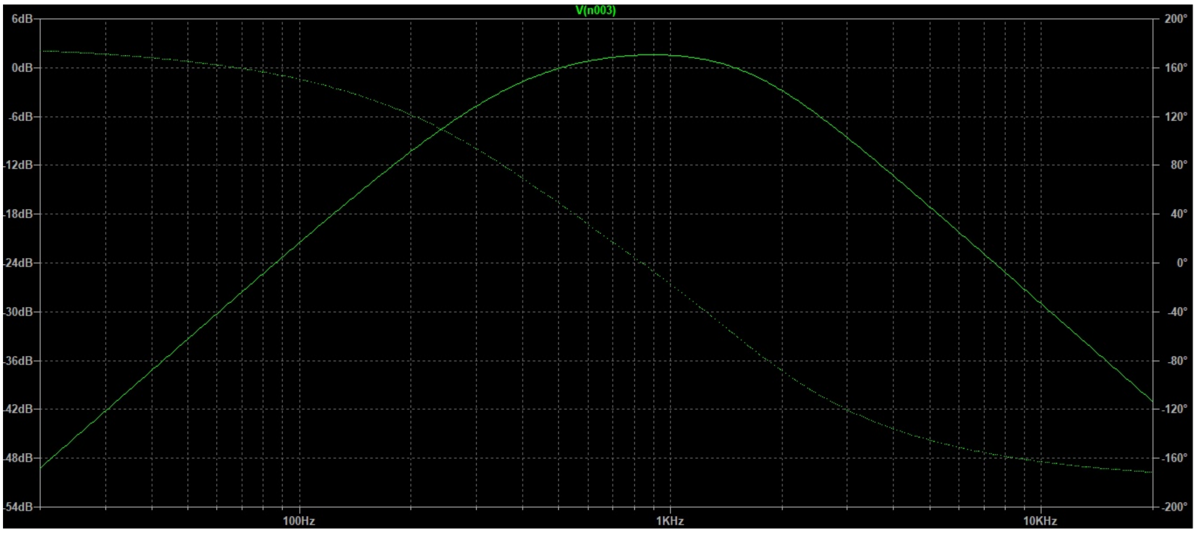

Figure 8.33: Band pass filter simulation

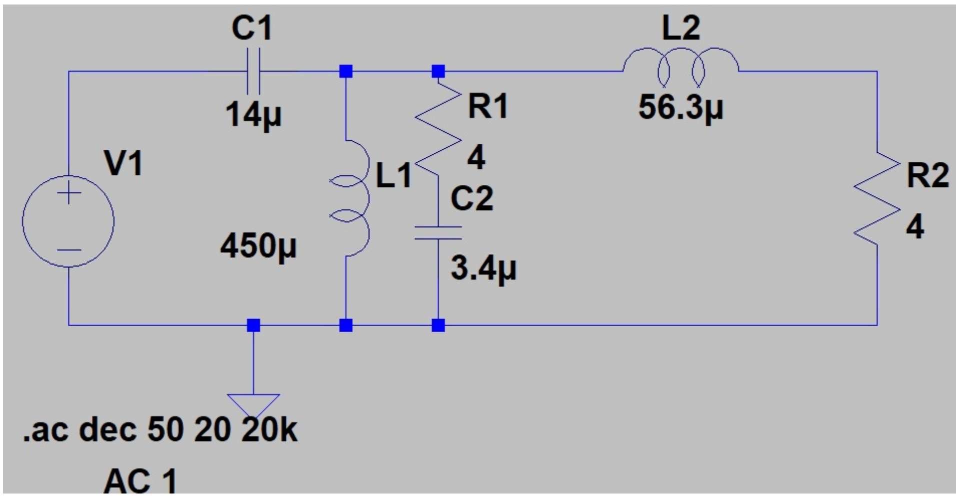

Figure 8.34: High pass filter circuit

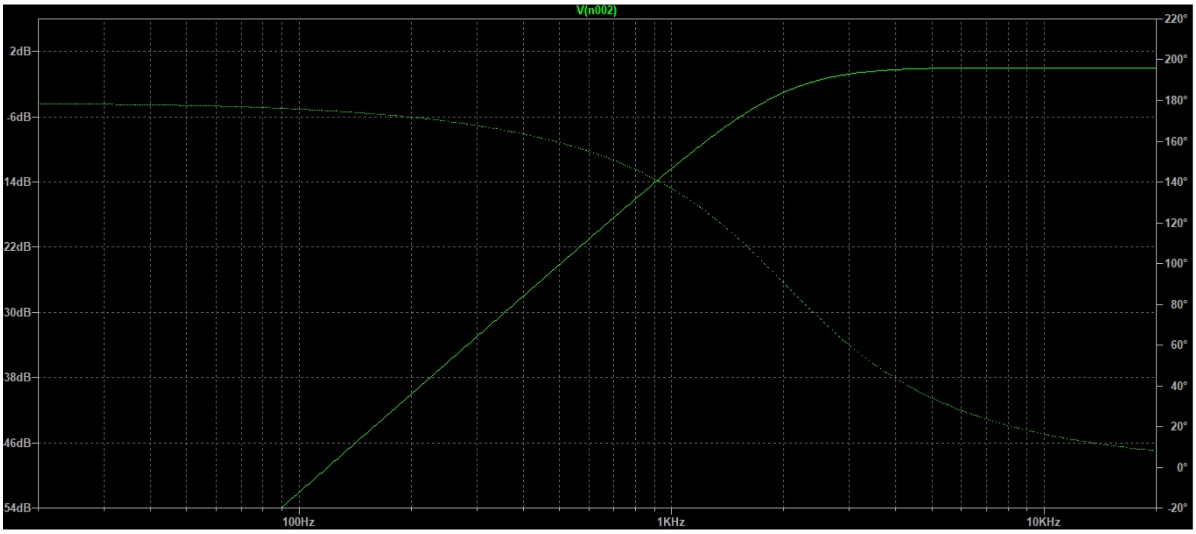

Figure 8.35: High pass filter simulation

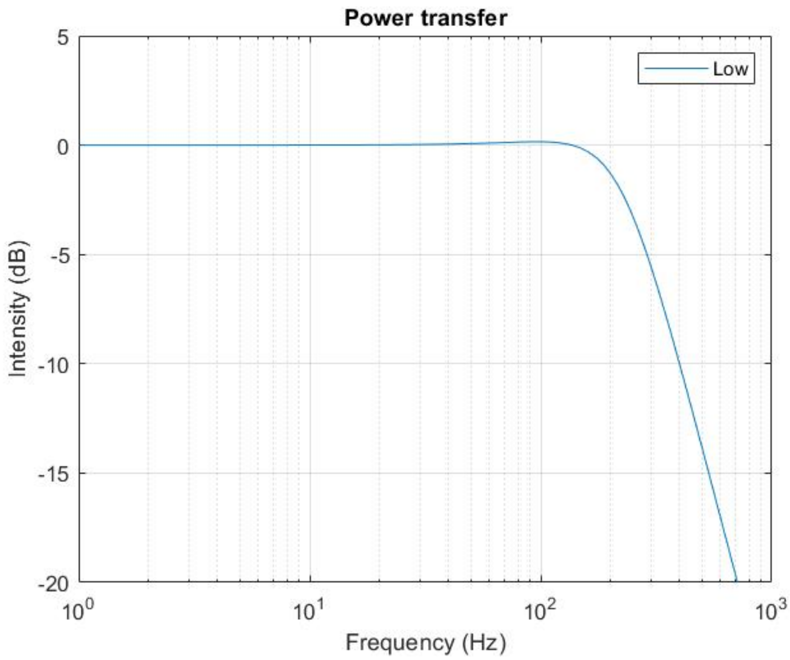

Figure 8.36: Low pass filter simulation

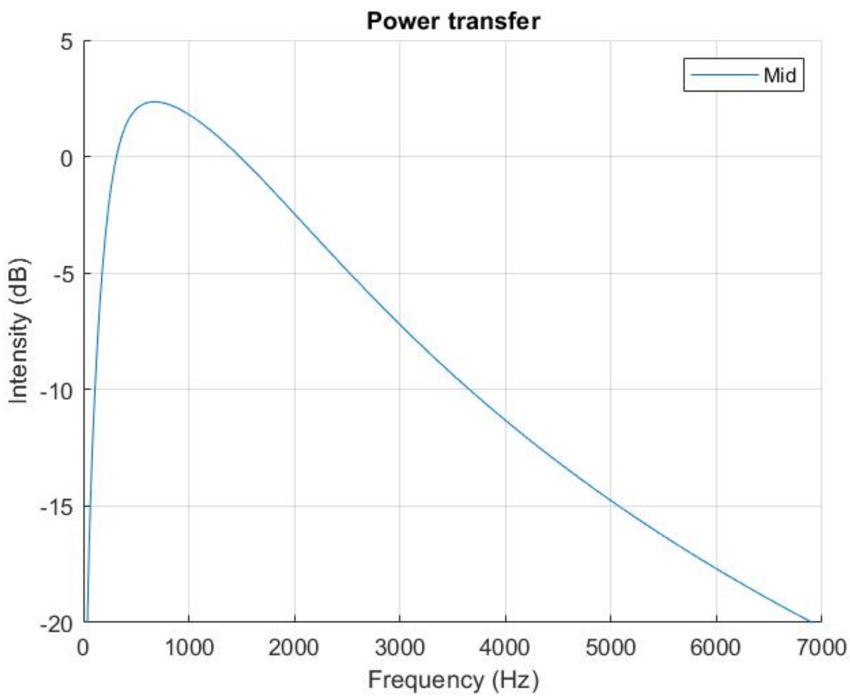

Figure 8.37: band pass filter simulation

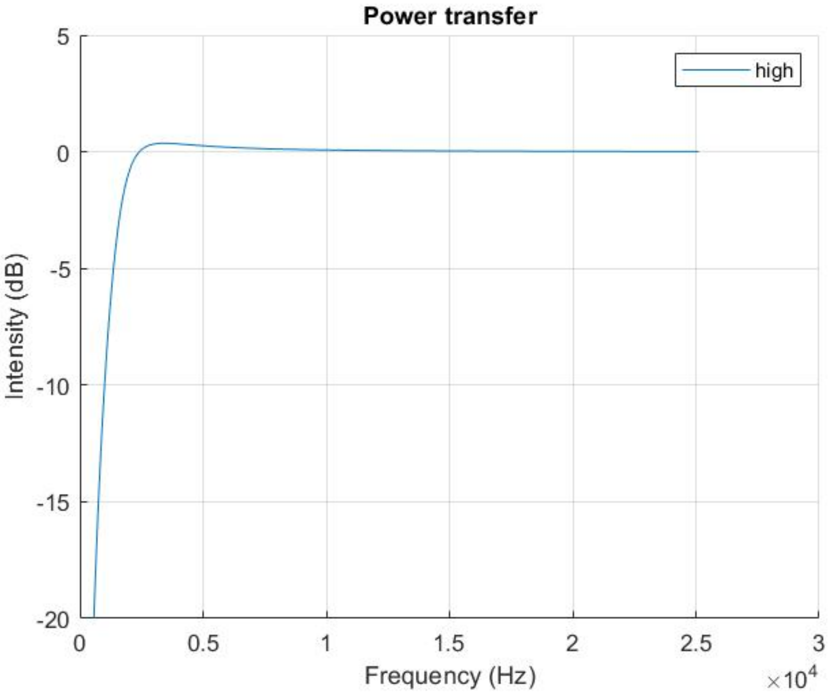

Figure 8.38: high pass filter simulation

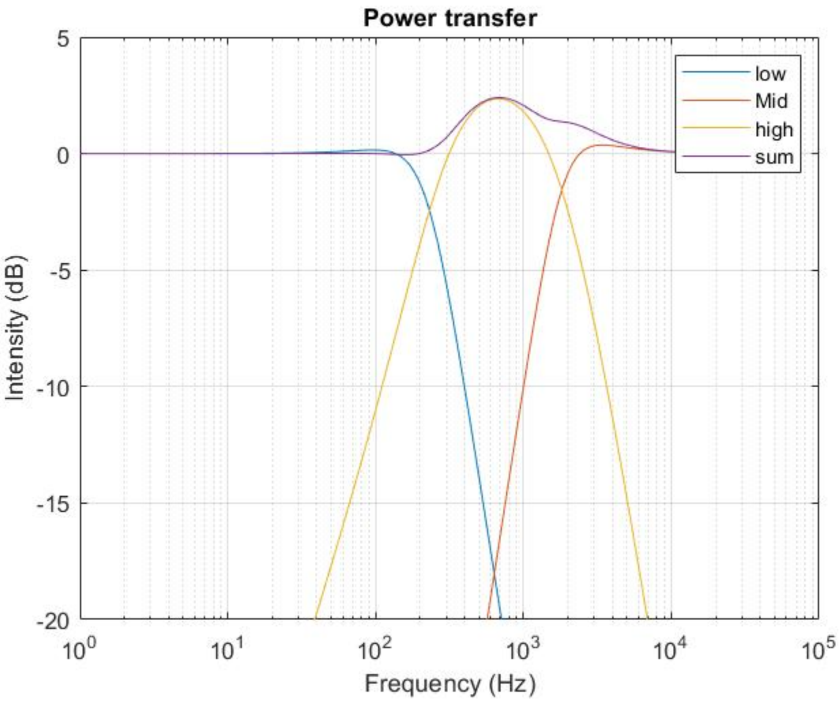

Figure 8.39: Power Transfer simulation

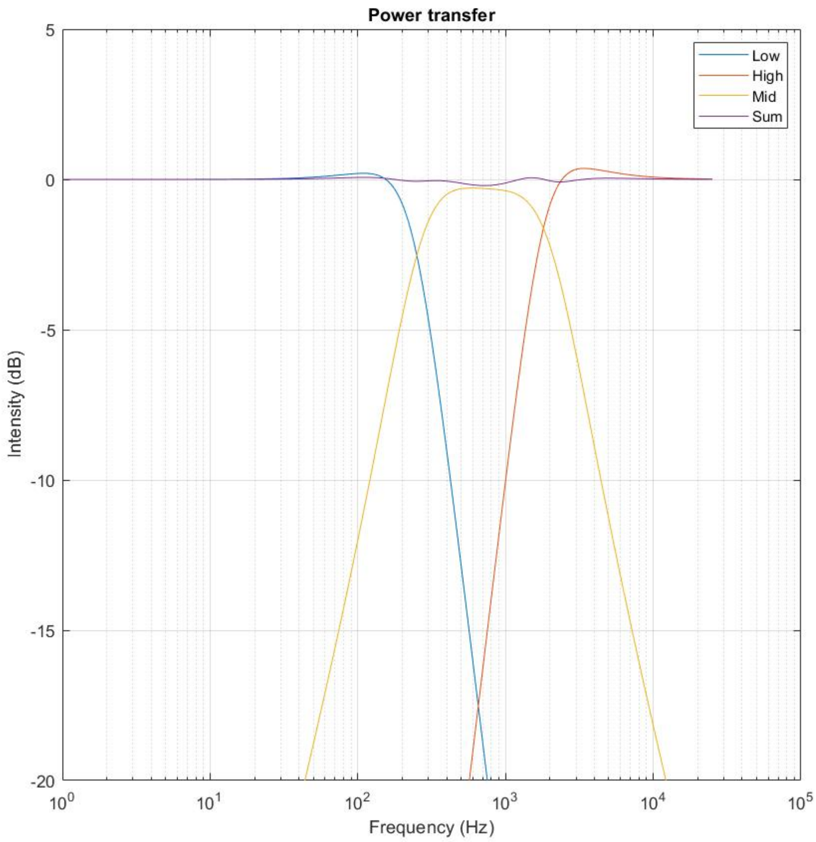

Figure 8.40: Power Transfer simulation with volume adjustment

Volume adjustment :

The equations for the values of the resistors are derived by keeping the following requirements in mind:

1. The added network has to give the desired damping.
2. The network should present a impedance to the filter, which is equivalent to Z_{ls}.

From the first requirement and figure 4.5 the following equation is obtained:

$$Z_{ls} = \frac{Z_{ls}R_2}{Z_{ls} + R_2} + R_1 \tag{8.1}$$

In equation 8.1 R_1can be isolated which results in the following equation:

$$R_1 = \frac{Z_{ls}^2}{Z_{ls} + R_2}(\Omega) \tag{8.2}$$

Equation 8.2 can be implemented in Equation 4.22:

$$\beta = 20\log_{10}\frac{Z_{ls}^2}{Z_{ls}^2 + Z_{ls}R_2}(dB) \tag{8.3}$$

From 8.3 R_2 can be isolated and solved since β and $Z_{ls}$are known:

$$R_2 = \frac{10^{\frac{\beta}{20}}Z_{ls}}{1 - 10^{\frac{\beta}{20}}}(\Omega) \tag{8.4}$$

8.3.4. Appendix D: Electrical frequency response

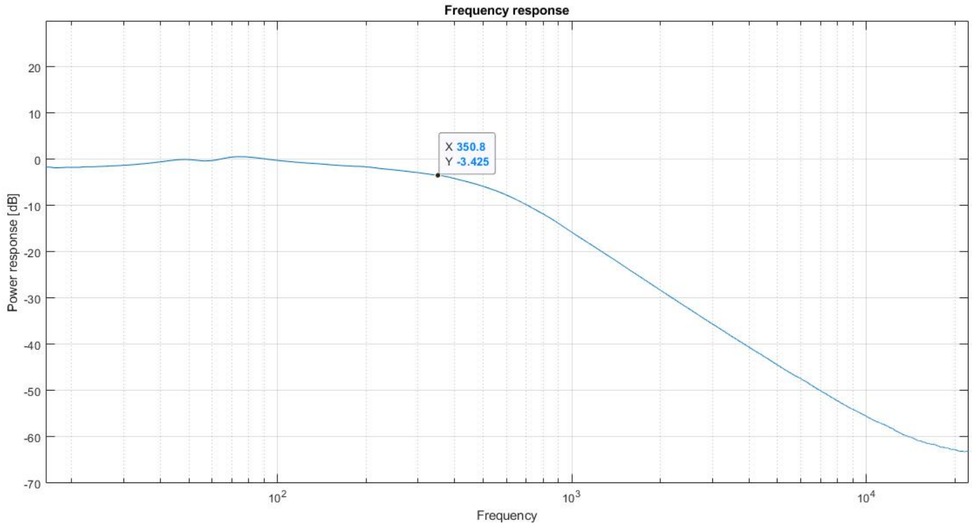

Figure 8.41: Electrical response Low-pass filter

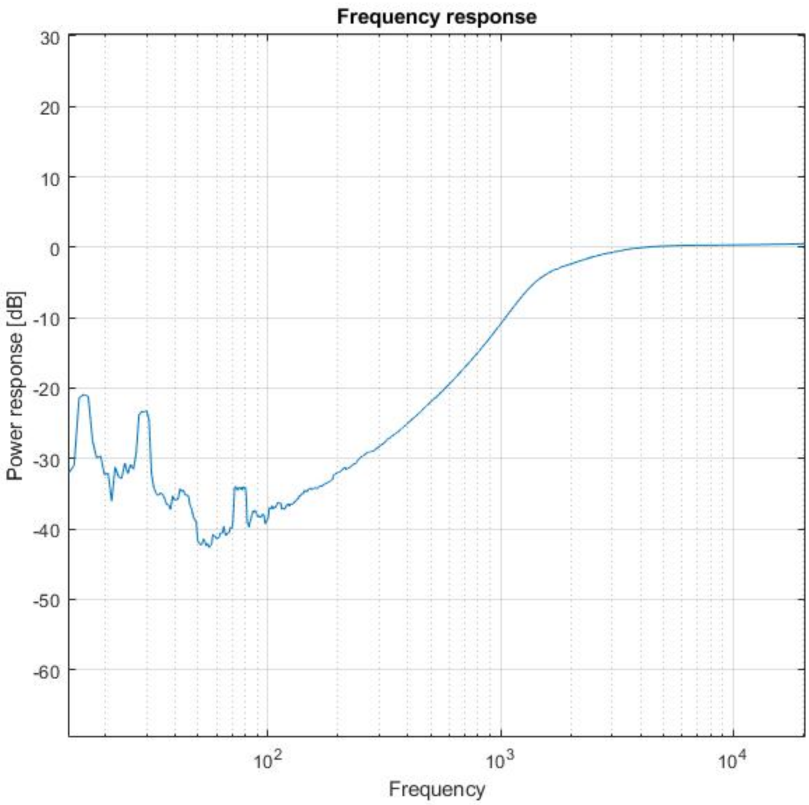

Figure 8.42: Electrical response high-pass filter

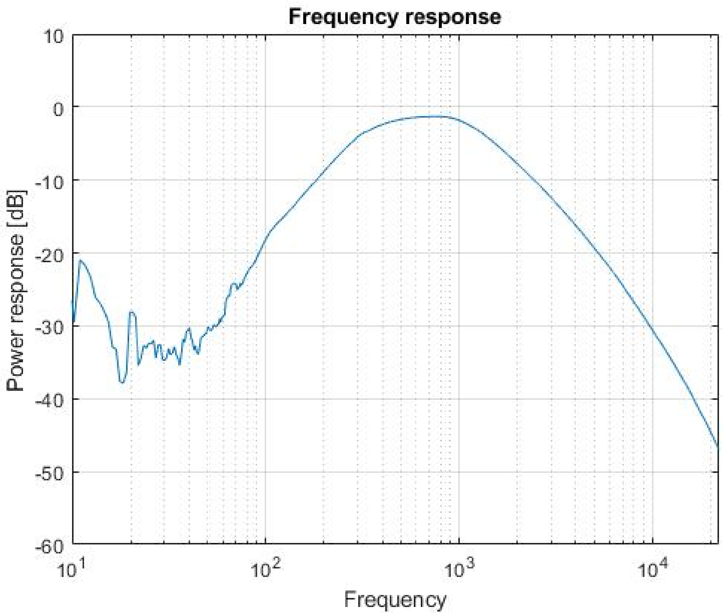

Figure 8.43: Electrical response band-pass filter with Zobel network

8.3.5. Appendix E: Acoustic frequency measurement filters

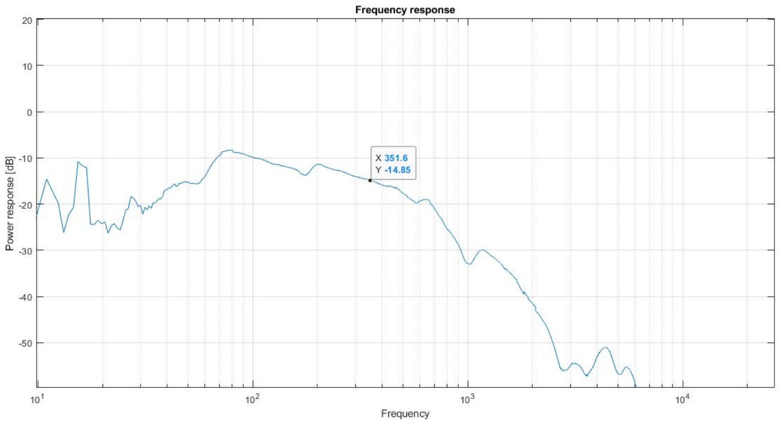

Figure 8.44: Near field acoustic transfer low-pass filter

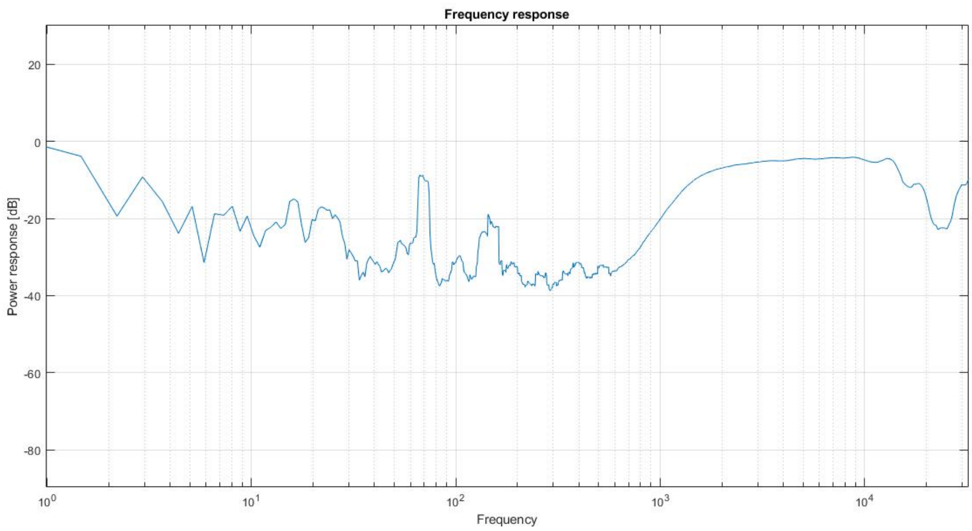

Figure 8.45: Near field acoustic transfer high-pass filter

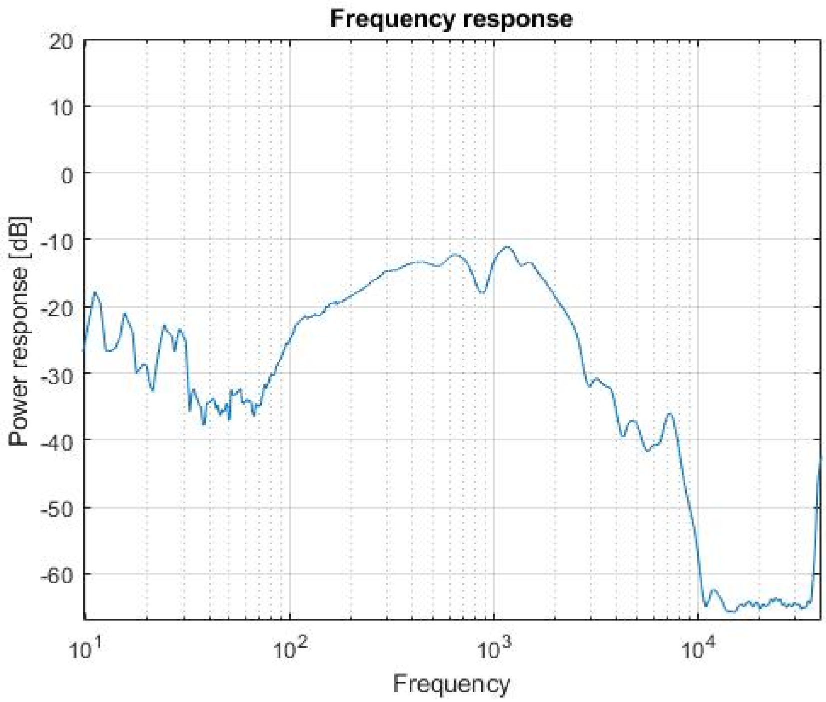

Figure 8.46: Near field acoustic transfer band-pass filter

8.3.6. Appendix F: Matlab code

```
 % Simulation on filter values and at what frequency and what voltage.
 f = logspace(0,4.4,1000);
w = 2.*pi.*f;
%lowpass filter
L_LPF = 6.46e-3;
 R_lsl = 7.1;
C_LPF = 130e-6;
%highpass filter
C_HPF = 17e-6;
L_HPF = 0.45e-3;
 R_lsh = 4.3;
%bandpass filter
C_LPF_mid = 10.5e-6;
L_LPF_mid = 0.555e-3;
C_HPF_mid = 98e-6;
L_HPF_mid =3.6e-3 ;
 R_lsb = 4.1;
%impedance L
ZC_LPF =1./(1j.*w.*C_LPF);
ZL_LPF = 1j.*w.*L_LPF;
 Z_eql = ((R_lsl.*ZC_LPF)./(R_lsh + ZC_LPF))+ ZL_LPF;
%impedance H
ZC_HPF =1./(1j.*w.*C_HPF);
ZL_HPF = 1j.*w.*L_HPF;
Z_eqh = ((R_lsh.*ZL_HPF)./(R_lsh + ZL_HPF))+ ZC_HPF;
%volume adjustment
R1= 1.3;
R2= 30;
%impedance MID
```

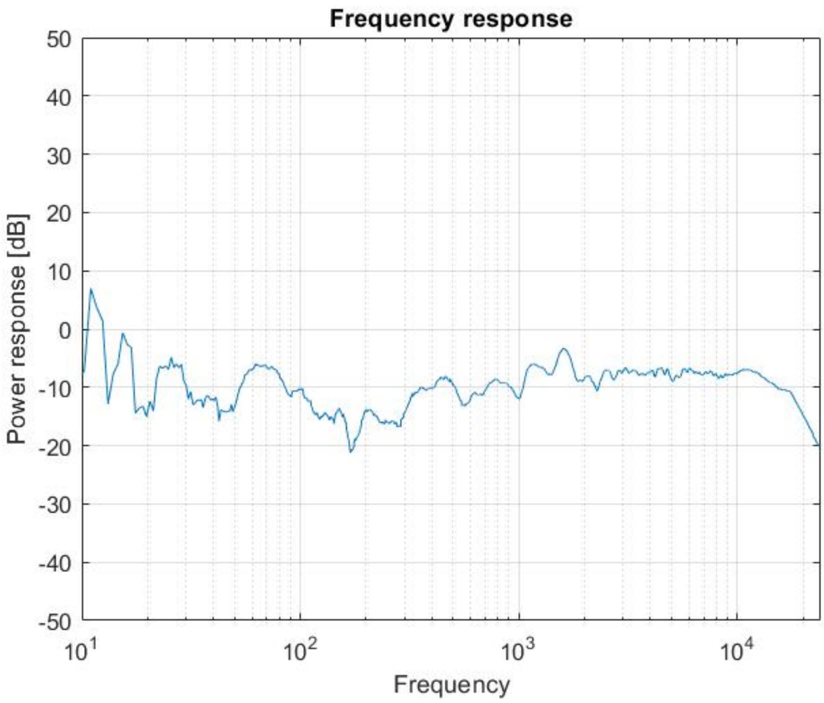

Figure 8.47: Far field acoustic transfer

```
ZC_LPF_mid = 1./(1j.*w.*C_LPF_mid);
ZL_LPF_mid = 1j.*w.*L_LPF_mid;
ZC_HPF_mid = 1./(1j.*w.*C_HPF_mid);
ZL_HPF_mid = 1j.*w.*L_HPF_mid;
Z_out = (R_lsb.*R2)./(R_lsb+R2);
Z_volume = Z_out + R1;
Z_1= (Z_volume.*ZL_HPF_mid)./(Z_volume + ZL_HPF_mid) + ZC_HPF_mid;
Z_2=(Z_1.*ZC_LPF_mid)./(Z_1 + ZC_LPF_mid)+ZL_LPF_mid;
Z_11= (Z_volume.*ZC_LPF_mid)./(Z_volume + ZC_LPF_mid) + ZL_LPF_mid;
Z_22=(Z_11.*ZL_HPF_mid)./(Z_11 + ZL_HPF_mid)+ZC_HPF_mid;
Z_eq1 = ((R_lsb.*ZC_LPF_mid)./(R_lsb + ZC_LPF_mid))+ ZL_LPF_mid;
Z_eq2 = (Z_eq1.*ZL_HPF_mid)./(Z_eq1 + ZL_HPF_mid);
Z_eqm = Z_eq2 + ZC_HPF_mid;

%The portion of the impedance over which should be meausured
Z_low_measure =(R_lsl.*ZC_LPF)./(R_lsh + ZC_LPF);
Z_high_measure =(R_lsh.*ZL_HPF)./(R_lsh + ZL_HPF);
Z_measure_mid = (Z_eq1.*ZL_HPF_mid)./(ZL_HPF_mid + Z_eq1 + ZL_LPF_mid);
%Decibels from the different filters
Db_l = 20*log10(abs(Z_low_measure./Z_eql));
Db_h = 20*log10(abs(Z_high_measure./Z_eqh));
Db_m = 20*log10(abs(Z_out./Z_22));
%Plots of the Db function to the frequency
semilogx(f, Db_l);
hold on;
semilogx(f, Db_h);
semilogx(f, Db_m);
semilogx(f, 10*log10(((Z_low_measure./Z_eql).^2) +((Z_high_measure./Z_eqh).^2) + ((Z_out./Z_22).^2)));
%Plot specifications
legend('Low','High','Mid','Sum');
```

```
grid on;
ylim([-20, 5]);
hold off;
xlabel('Frequency␣(Hz)');
ylabel('Intensity␣(dB)');
title('Power␣transfer');
```

8.4. Power amplifier Appendix

8.4.1. Appendix A: Derivations

The derivations for the transfer functions:

High-pass filter:

$$\frac{V_{out} - V_{in}}{1/j\omega C1} + \frac{V_{out}}{R1} = 0 \tag{8.5}$$

$$V_{out} \cdot (R2 + 1/j\omega C1) = V_{in} \cdot R2 \tag{8.6}$$

$$\frac{V_{out}}{V_{in}} = \frac{R1}{R1 + 1/j\omega C1} = \frac{j\omega R1C1}{1 + j\omega R1C1} \tag{8.7}$$

Low-pass filter:

$$\frac{V_{out} - V_{in}}{R2} + \frac{V_{out}}{1/j\omega C2} = 0 \tag{8.8}$$

$$V_{out} \cdot (R2 + 1/j\omega C2) = \frac{V_{in}}{j\omega C2} \tag{8.9}$$

$$\frac{V_{out}}{V_{in}} = \frac{1}{(R2 + 1/j\omega C2) \cdot j\omega C2} = \frac{1}{1 + j\omega R2C2} \tag{8.10}$$

Op-amp:

$$\frac{V_{out} - V_{in}}{R3} + \frac{V_{in}}{R4 + 1/j\omega C3} = 0 \tag{8.11}$$

$$V_{out} \cdot (R4 + 1/j\omega C3) = V_{in} \cdot (R4 + R3 + 1/j\omega C3) \tag{8.12}$$

$$\frac{V_{out}}{V_{in}} = \frac{R4 + R3 + 1/j\omega C3}{R4 + 1/j\omega C3} = \frac{1 + j\omega R4C3 + j\omega R3C3}{1 + j\omega R4C3} = 1 + \frac{j\omega R3C3}{1 + j\omega R4C3} \tag{8.13}$$

8.4.2. Appendix B

Gain at different value combinations of R3 and R4. Used for determining the component values of the amplifier circuit.

R3(Ω)	R4(Ω)	Gain	R3(Ω)	R4(Ω)	Gain	R3(Ω)	R4(Ω)	Gain
10k	390	25.64	22k	910	24.18	47k	2.2k	21.26
10k	430	23.25	22k	1k	22.00	51k	2.2k	23.18
11k	430	25.58	24k	1k	24.00	51k	2.4k	21.25
11k	470	23.40	27k	1.1k	24.55	56k	2.4k	23.33
12k	470	25.53	27k	1.2k	22.50	56k	2.7k	20.74
12k	510	23.53	30k	1.2k	25.00	62k	2.7k	22.96
13k	510	25.49	30k	1.3k	23.08	62k	3k	20.67
13k	560	23.21	33k	1.3k	25.38	68k	3k	22.67
15k	560	26.77	33k	1.5k	22.00	68k	3.3k	20.61
15k	620	24.19	36k	1.5k	24.00	75k	3.3k	22.73
16k	620	25.81	39k	1.6k	24.38	75k	3.6k	20.83
16k	680	23.53	39k	1.8k	21.67	82k	3.6k	22.78
18k	750	24.00	43k	1.8k	23.89	82k	3.9k	21.03
20k	820	24.39	43k	2k	21.50	91k	3.9k	20.33
20k	910	21.98	47k	2k	23.50	91k	4.3k	21.16

Figure 8.48: Table with possible combinations of resistors to get a gain of 25.

8.4.3. Appendix C: Matlab code

Matlab code used for simulating the amplifier circuit.

```
%{
Part of circuit:    HPF                          LPF              OP-AMP

Transfer function:  (jwR1C1)/(1+jwR1C1)          1/(1+jwR2C2)     1 + jwR3C3/(jwR4C3)
Numerator:          [R1C1, 0]                    1                [(R3+R4)C3, 0]
Denumerator:        [R1C1, 1]                    [R2C2, 1]        [R4C3, 1]
%}

%HPF
num1 = [24000 * 330e-9, 0];
den1 = [24000 * 330e-9, 1];
a = tf(num1, den1);

%LPF
num2 = 1;
den2 = [1000 * 3.9e-9, 1];
b = tf(num2, den2);

%OP-AMP
num3 = [(68000 + 2800) * 4.7e-6, 0];
den3 = [2800 * 4.7e-6, 1];
c = tf(num3, den3);

d = a*b*c; %transfer function of the whole circuit

%Change units from rad/s to Hz
options = bodeoptions;
options.FreqUnits = 'Hz';

%make bodeplots for a, b, c or d
bodeplot(d, options);
```

8.4.4. Appendix D: Bode plots

Bode plots of the filters with -3dB frequencies displayed.

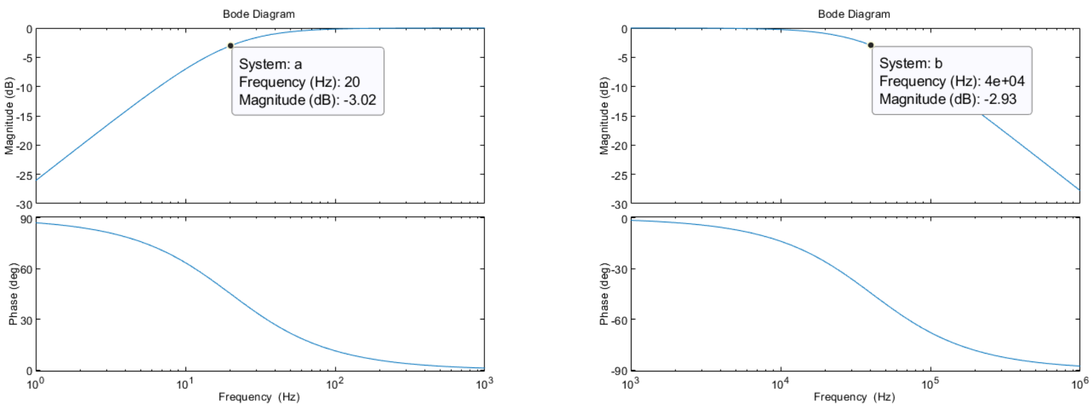

Figure 8.49: (from left to right) Bode plot of high-pass filter, Bode plot of low-pass filter

8.4.5. Appendix E: Measured voltages and gain

Frequency	Input Pk to Pk Voltage	Output Pk to Pk Voltage	Vout/Vin
5	0.12	0.4	3.33333333
10	0.176	1.28	7.27272727
15	0.188	2.24	11.91489362
17	0.188	2.56	13.33333333
18	0.192	2.68	13.67346939
19	0.196	2.8	14.28571429
20	0.2	2.96	14.8
21	0.2	3.08	15.4
22	0.2	3.2	16
25	0.2	3.44	17.2
50	0.208	4.56	21.92307692
75	0.208	4.8	23.07692308
100	0.208	4.96	23.84615385
200	0.208	5.08	24.42307692
500	0.208	5.12	24.61538462
1000	0.208	5.12	24.61538462
2000	0.208	5.12	24.61538462
5000	0.208	5.08	24.42307692
10000	0.216	5.08	23.51851852
20000	0.216	4.72	21.85185185
30000	0.212	4.28	20.18867925
38000	0.216	3.92	18.14814815
39000	0.216	3.88	17.96296296
40000	0.216	3.8	17.59259259
41000	0.212	3.8	17.9245283
42000	0.212	3.76	17.73584906
43000	0.212	3.72	17.54716981
44000	0.212	3.68	17.35849057
45000	0.212	3.64	17.16981132
50000	0.212	3.44	16.22641509

Figure 8.50: Measured voltages and voltage gain

8.4.6. Appendix F: Oscilloscope images

Images of oscilloscope measuring the input and output signals of the amplifier.

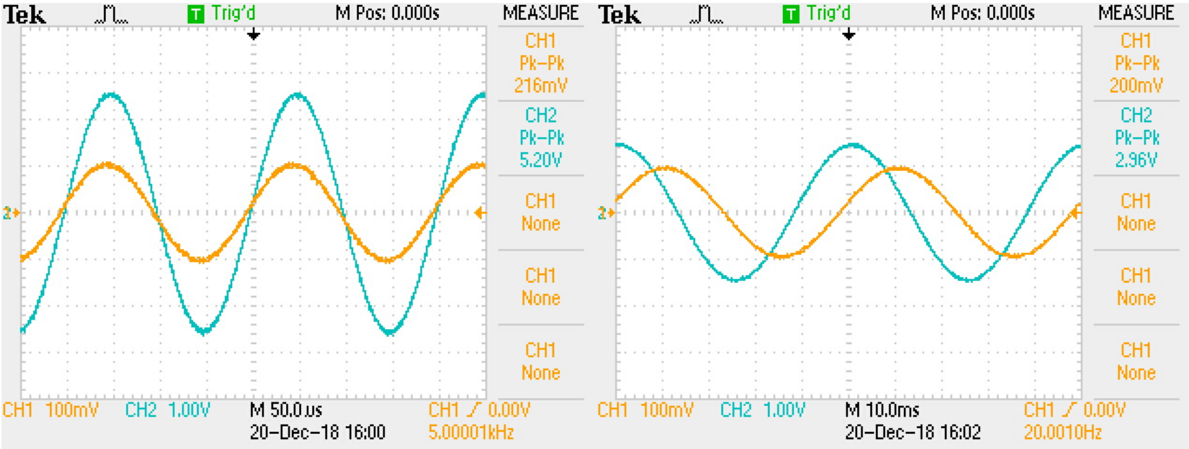

Figure 8.51: (Left to right) Oscilloscope screenshot at 5kHz, Oscilloscope screenshot at 20kHz

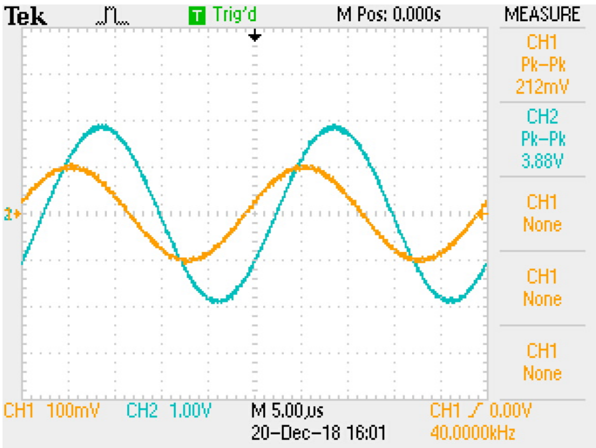

Figure 8.52: Oscilloscope screenshot at 5kHz

8.4.7. Appendix G: Miscellaneous

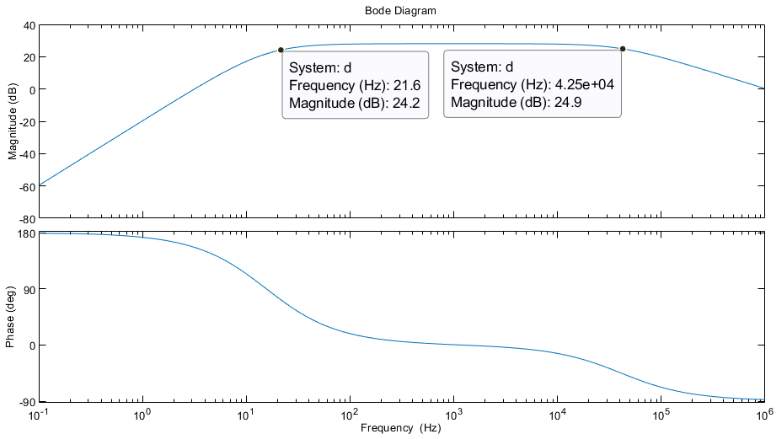

Figure 8.53: A bode plot of the circuit transfer function

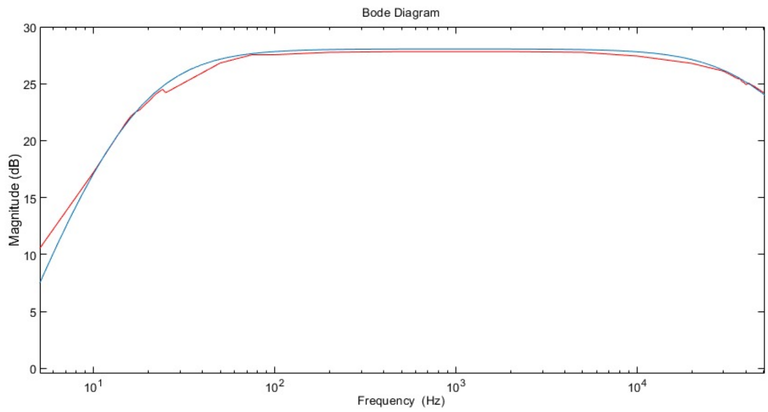

Figure 8.54: Voltage gain vs Frequency in Hertz

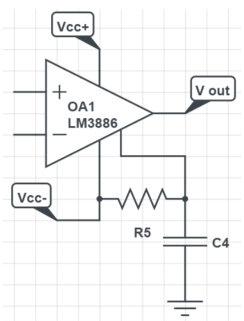

Figure 8.55: Mute circuit

9

OrCAD circuits and simulations

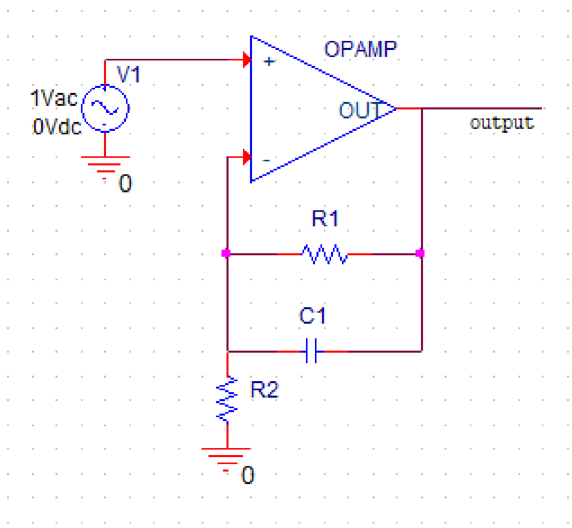

Figure 9.1: OP-AMP with first order high-pass filter in its feedback loop.

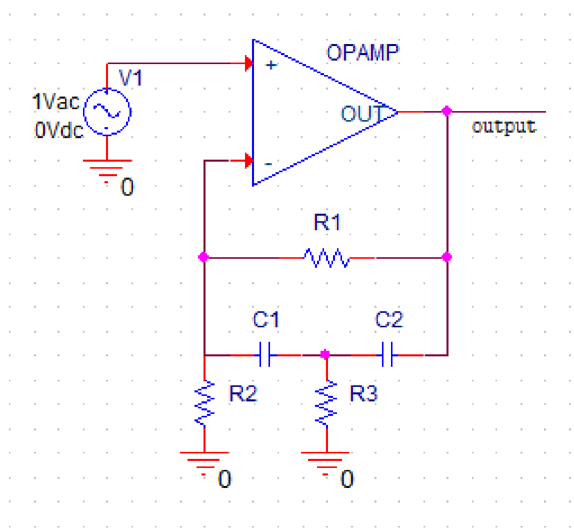

Figure 9.2: OP-AMP with second order high-pass filter in its feedback loop.

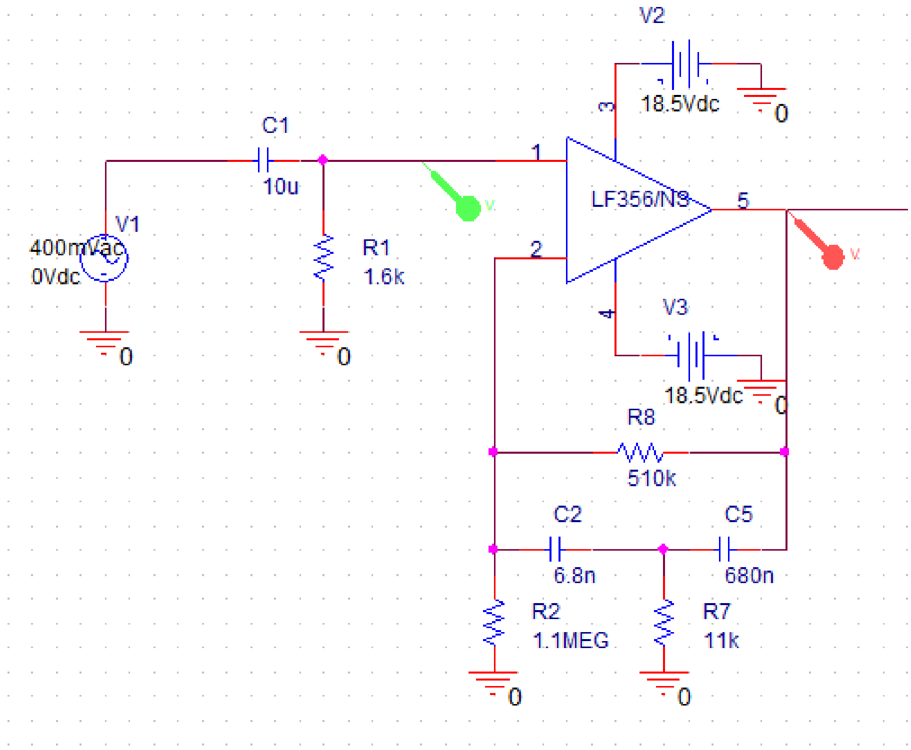

Figure 9.3: OP-AMP with second order high-pass filter in its feedback loop.

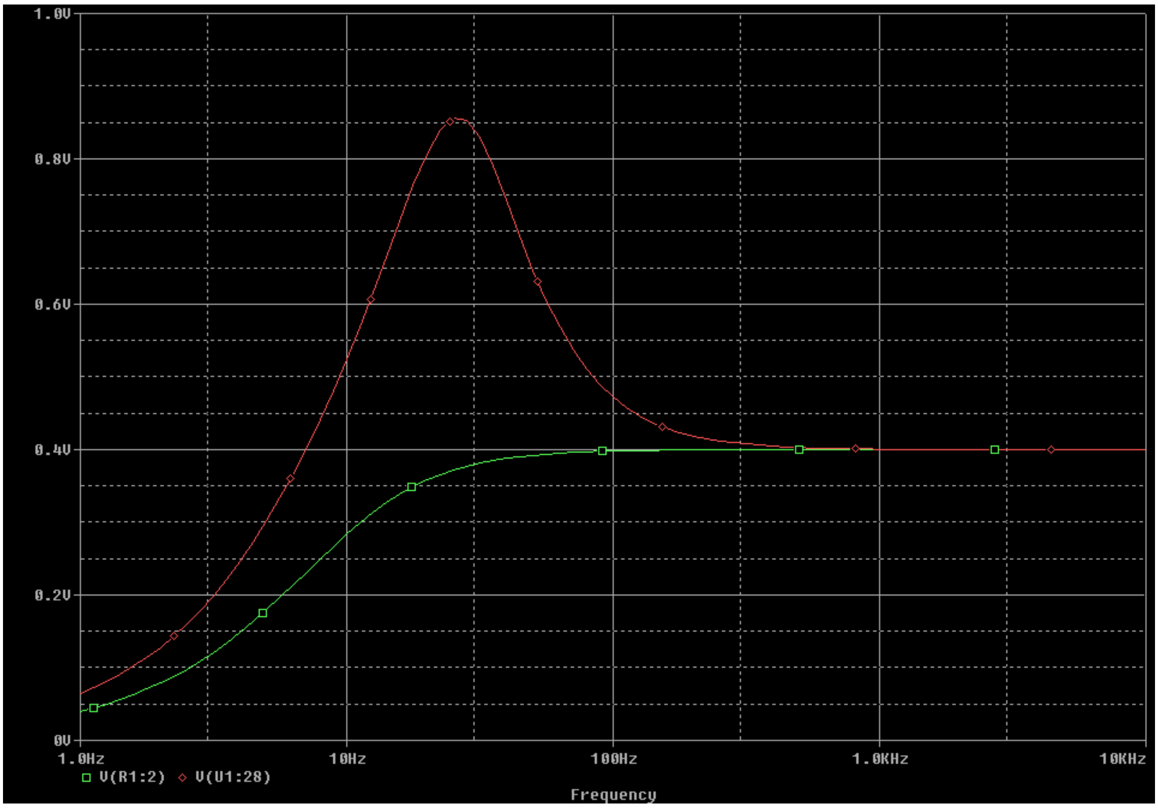

Figure 9.4: OP-AMP with second order high-pass filter in its feedback loop.