Jarkko Oksanen

A Dummy Load for Testing and Measuring Amplifiers

Metropolia University of Applied Sciences

Bachelor of Engineering

Electronics

Bachelor’s Thesis

1 March 2022

Abstract

Author:	Jarkko Oksanen
Title:	A Dummy Load For The Testing And Measuring Amplifiers
Number of Pages:	24 pages
Date:	27 March 2022
Degree:	Bachelor of Engineering
Degree Program:	Electrical and automation engineering
Professional Major:	Electronics
Supervisors:	Matti Fischer, Principal lecturer Daniil Danilov, Electronics design engineer

This thesis went through the process of designing a versatile resistive load used when testing amplifiers at Darkglass Electronics Oy. The device was intended for internal use in the company in the process of product development and quality assurance. The device that was designed in this thesis was intended to replaced the dummy loads used earlier to improve user safety and ease of use.

The design process of the device's impedance selection system, the instrumentation amplifier needed for separating ground potentials and the PCB design phase were explained in this document. The PCB was designed using Altium Designer software

Before building the final device the design is simulated and the functionality and different design aspects are analyzed according to the results of the simulations. The circuit was designed and simulated by using LTspice software.

Finally the performance of the device is evaluated by measuring signals going in and coming out and by analyzing them against the expectations.

This thesis will also give a quick look at some of the phases that go into a research and development project in the electronics industry. As an outcome of the thesis a device was created to be used in future product development processes to analyze the functionality and performance of new products.

Keywords: Dummy load, instrumentation amplifier !

Tiivistelmä

Tekijä:	Jarkko Oksanen
Otsikko:	Keinokuorma bassovahvistimien testaamiseen ja mittauksiin
Sivumäärä:	24 sivua
Aika:	27.3.2022
Tutkinto:	Insinööri (AMK)
Tutkinto-ohjelma:	Sähkö- ja automaatiotekniikka
Ammatillinen pääaine:	Elektroniikka
Ohjaajat:	Yliopettaja Matti Fischer Elektroniikkasuunnittelija Daniil Danilov

Opinnäytetyössä käytiin läpi bassovahvistimia valmistavan Darkglass Electronics Oy:n sisäiseen käyttöön tarkoitetun monikäyttöisen keinokuorman suunnitteluprosessi. Kyseinen keinokuorma suunniteltiin käytettäväksi tuotekehityksessä ja laadunvalvonnassa. Laite suunniteltiin korvaamaan aiemmin käytössä olleet raskaat ja vaikeasti käytettävät keinokuormat turvallisemmalla ja monikäyttöisemmällä mallilla.

Keinokuorman impedanssivalinnan, maasilmukan erotukseen käytetyn instrumentointivahvistimen ja piirilevyn suunnittelu käytiin läpi tässä opinnäytetyössä. Piirilevy suunniteltiin käyttäen Altium Designer -ohjelmaa.

Ennen varsinaisen laitteen rakentamista sen toimintaa analysoitiin simuloimalla suunniteltujen piirien toimintaa tietokoneohjelmalla. Simulointien yhteydessä kokeiltiin erilaisten kokoonpanojen vaikutusta lopputulokseen. Simulointeihin käytettiin LTspice-simulointiohjelmaa.

Valmiin laitteen toimintaa arvioitiin mittaamalla sisään menevien ja ulos tulevien signaalien arvoja ja vertaamalla niitä laskennallisiin odotuksiin.

Kokonaisuutena opinnäytetyö tarjoaa pienen katsauksen muutamiin elektroniikka-alan tuotekehitysprosessien työvaiheisiin. Työn tuloksena syntyi valmis laite, jota tullaan käyttämään osana tuotekehitysprosessia uusien tuotteiden ominaisuuksia mitattaessa.

Avainsanat: Keinokuorma, instrumentaatiovahvistin

Contents

List of Abbreviations

AC: Alternating current

ATX: Advanced Technology extended. A type of motherboard and chassis configuration for personal computers

CMR: Common mode rejection

CMRR: Common mode rejection ratio

PCB: Printed circuit board

SMD: Surface mounted device

THD: Total harmonic distortion

THD+N: Total harmonic distortion + noise

1. Introduction

The prototyping phase of designing a bass guitar amplifier requires a number of tests and measurements to be carried out. The measurements are often carried out while running the amplifier at high power. Using a speaker cabinet for this would make the testing procedure unnecessarily loud in an office environment. A speaker cabinet also lacks the connections for any measurement devices and a ground isolation which plays a key role in the validity of the readings gotten from the measurements. To get around this issue the speaker cabinet is replaced with a dummy load which has similar set of impedances and power handling capabilities as a speaker cabinet the amplifier is designed to drive. A dummy load is in essence a set of resistors that convert the electrical energy, that would otherwise be used to create sound in a loudspeaker, into heat [1]. This thesis will showcase the design, development and creation of a versatile dummy load that can be used for different types of music instrument amplifiers.

The key requirements for the end product were ease of use, safety and the ability to be used for variety of different amplifiers. The Darkglass Electronics product catalog consists of amplifiers that have been designed to be used with speaker cabinets with impedances varying from two to eight ohms. For this reason the dummy load needed to have a simple-to-use selection between different impedances. The requirements for power handling capabilities were set at 500 watts to make sure the load could be safely used for all the amplifiers produced by the company. The most powerful amplifiers in the Darkglass catalog put out 450 watts of RMS power when used with an eight ohm load [2].

For the dummy load to be usable it needed to have connecting terminals for measurement devices such as a digital multimeter, an audio analyzer and an oscilloscope. These connectors along with an impedance selector were placed on printed circuitboards which were designed as a part of the thesis project.

The whole set of resistors needed to be enclosed in a way that all the heat generated in the testing process could be dissipated as well as being safely contained so that no-one could come into contact with the heat source.

At the time of completing this thesis work the Covid-19 pandemic was causing a shortage of consumer electronics and electronics components. This affected the process of selecting the components used for the project, which made availability one of key factors in component selection.

2. Theory

2.1 Instrumentation Amplifier

An instrumentation amplifier's function is to amplify the difference between two input signals and reject signals that are common in the two inputs. The common mode rejection or CMR reduces noise in the signal being amplified.

Figure 1 shows a basic setup of an instrumentation amplifier.

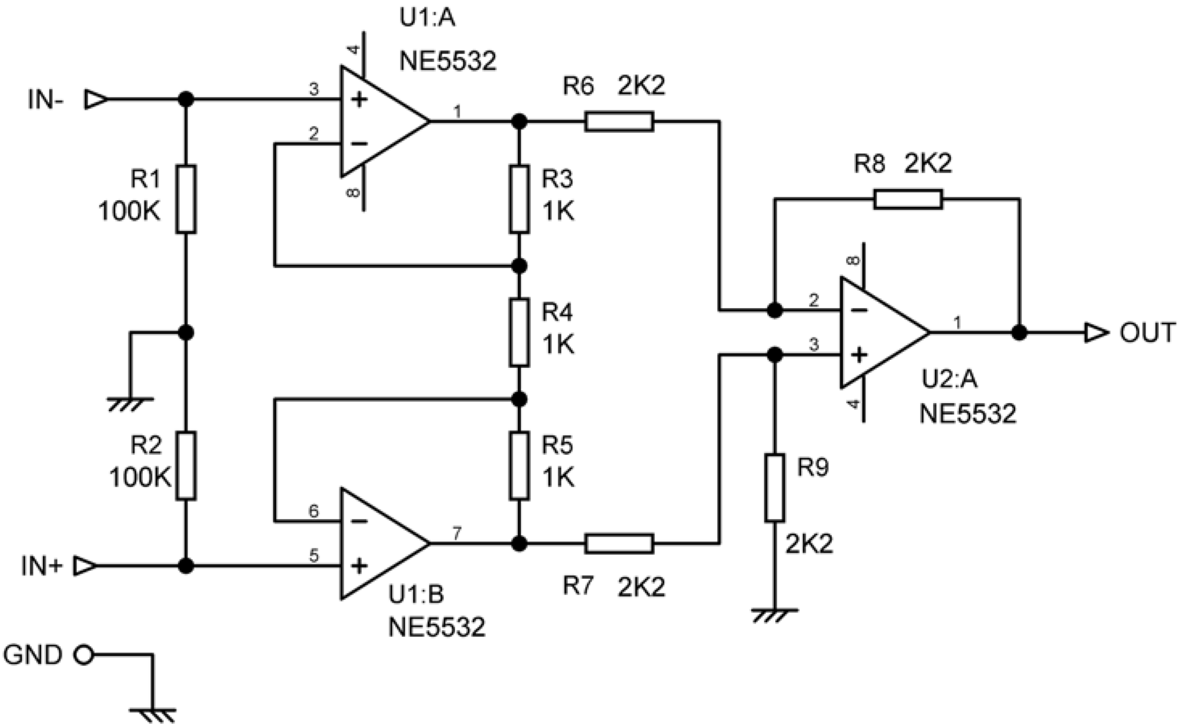

Figure 1. An instrumentation amplifier built around three operational amplifiers is depicted in the picture. [14]

The circuit of an instrumentation amplifier has three operational amplifiers. Two of the operational amplifiers are positioned on the input side, both of them receiving one half of the signal after the voltage divider on their positive inputs. These two operational amplifiers function as input buffers for the differential amplifier. The outputs feed the signal to the differential amplifier. A differential amplifier is the stage where the amplification of the buffered signal happens.

2.2 Common Mode Rejection Ratio

Common mode rejection ratio, commonly abbreviated as CMRR, is used to describe an amplifier’s ability to differentiate between incoming signal and noise. An amplifier with ideal balanced input is able to tell the difference between positive and negative input points, often referred to as IN+ and IN- respectively. The amplifier is intended to ignore voltages that are present at both input points at the same time, or in other words are in common mode. The amount of which the amplifier is able to hold off those voltages if referred to as common mode rejection ratio. That ratio is expressed in a dB value. [14]

In an ideal situation if IN+ and IN- were both supplied with the same voltage against the ground in same phase, the output of the amplifier would be zero. In a real-world scenario the value would likely be ranging from -20 dB with faulty balanced interconnection to -140 dB in a near optimal situation. [14]

3. Design

3.1 Overview of the System

A signal comparable to one from a bass guitar in voltage, frequency and current was generated by a signal generator and fed into the input of a bass guitar amplifier head. An amplifier head is a unit consisting of a pre-amplifier, possible effects and a power amplifier. The amplified signal coming out of the power amplifier is under normal conditions fed into a speaker cabinet. In a testing environment the speaker cabinet is replaced with a set of power resistors converting the energy into heat instead of sound.

An audio analyzer was placed to measure the signal over the power resistors to extract different readings and figures from the signal. The audio analyzer’s ground potential needed to be galvanically separated from the one on the amplifier. This was achieved by placing an instrumentation amplifier in between the two. The instrumentation amplifier circuit in this particular application was based around a Texas Instruments’ INA1650 integrated circuit. The INA1650 had two instrumentation amplifiers built in but in this project only one of them was used.

Because the power of the signal coming from the bass guitar amplifier was much higher than the INA1650 would have been able to handle, the signal had to be lowered. This was done by placing a voltage divider in parallel with the power resistors before the instrumentation amplifier.

In the case of Darkglass Electronics products the power amplifiers were two channel models that were bridged to function as a mono amplifier. This meant that the positive terminal of a load was connected to the positive of one channel and the negative to the negative of the other. This setup bypassed the common ground point in the middle. For this reason the measurement device also had to be connected over the common ground point which was the reason why the instrumentation amplifier was needed.

In the core of the system was the load that consisted of four eight ohm 300 watt power resistors that acted as a replacement for a speaker. From these power resistors four different loads could be selected by using two switches. The whole system was enclosed in an ATX PC-chassis.

The INA1650 requires power which was taken from an ATX power supply.

The negative voltage was taken from an inverting Ćuk converter built around a Texas Instruments LM2611 integrated circuit.

The instrumentation amplifier and inverter with their circuits, signal input and output, and the connector terminals for the wires to the power resistors were all built on one printed circuit board. The design of the PCB is explained later in this document.

The signal chain and power supply is depicted in Figure 1.

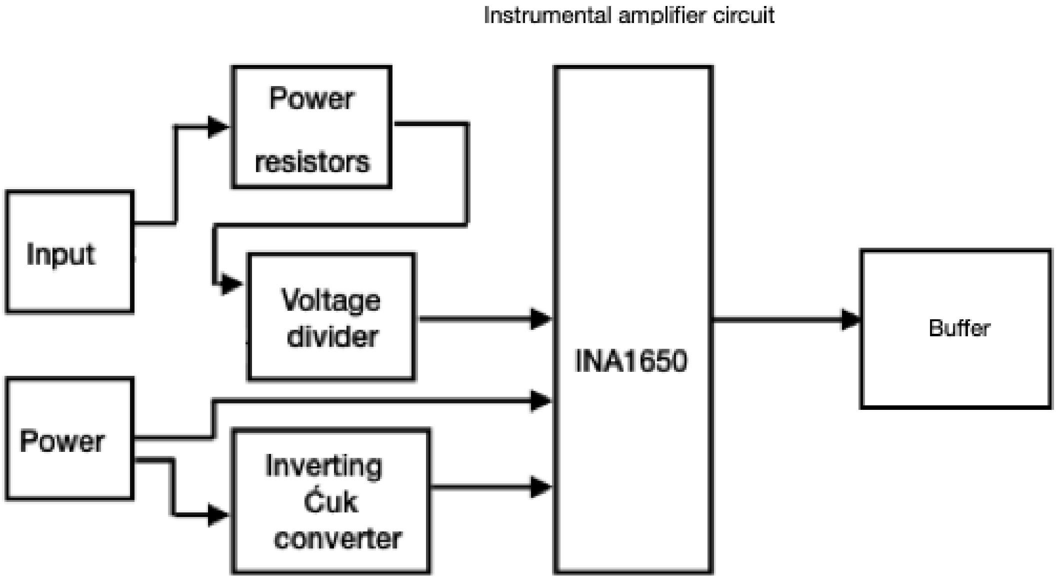

Figure 1. The logical flow of the signal through the whole system.

The input signal was fed to the power resistors from where it went to the voltage divider. From the voltage divider the signal flowed to the INA1650 and finally out from there to the output jack. An ATX power supply fed +12 V to the VCC of the INA1650 and the Ćuk converter which converted it to -12 V before it went to the VEE of the same chip. The INA165 fed the signal to the output jack through an output buffer.

3.2 Instrumentation Amplifier

After deliberating the options to create a galvanic separation between the load and the audio analyzer, the choice to use an instrumentation amplifier was made. This circuit was built around three operational amplifiers with seven resistors. Such a circuit was found readymade in an integrated circuit form.

The Texas Instrument INA1650 was chosen for its suitability for audio applications as well as being easily available. Other options were searched for but finding instrumentation amplifiers for audio use during the component shortage caused by the COVID-19 pandemic proved to be difficult.

Alternatives for the INA1650 existed, but their poor availability and inferior specifications made them fall short in comparison. One of the options considered was 1200-series Balanced Line Receiver IC from THAT Corporation. The 1200-series had inferior characteristics compared to the INA1650 in terms of total harmonic distortion and supply voltage range. The 1200-series did have a higher common mode rejection ratio, which was desirable in this application. This alone was nonetheless not enough for the choice to be made in its favor.

Common mode rejection ratio in a differential amplifier tells the amplifier's capability to separate signals in the inputs that are in-phase from the ones that are out-of-phase. In an ideal situation the ratio would be infinite. The in-phase, or common mode, signal has a very high voltage compared to the differential signal. In an example given in a Texas Instruments' manual for designing a three op-amp instrumentation amplifier the goal for the differential signal's input voltage is set at -0.5…+0.5 volts. In this example the common mode voltage was expected to be +-7 volts.[7]

3.3 Printed Circuit Board

The PCB was designed using Altium Designer based on the schematics simulated in LTSpice. The connections for the signal's input and output were placed on one side of the PCB so that it could be placed inside the ATX chassis' front panel. For connecting the power resistors there weree screw terminals placed on the opposite side of the board from the signal connections. Wires from the power resistors were connected to those terminals. The 12 volt power was supplied through a four-pin Molex Micro-Fit connector that was soldered on the PCB.

Designing the layout of the PCB was done according to the layout guidelines in the IMA1650's and LM2611's datasheets. The datasheets had example layouts for different applications and recommendations of which components should be placed near each other to avoid unwanted noise and signal losses in the circuit. [11][12]

The recommended placements of the components on the PCB are shown in the datasheet [11] as pictured in Figure 2.

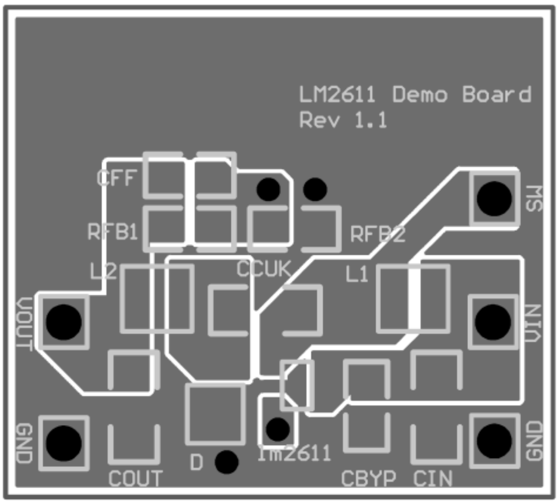

Figure 36. Example Layout Top

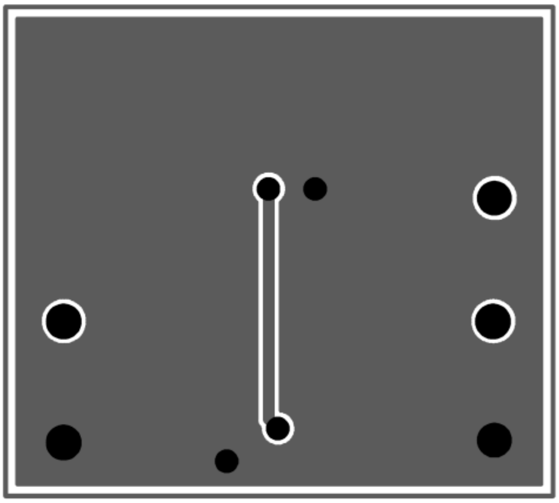

Figure 37. Example Layout Bottom

Figure 2. Example layout given in the datasheet. [11]

Layout of the component placements, the copper layers' layouts and drilling holes for wire connections to the PCB are shown on the top example. The bottom example shows a routing option for one of the traces. [11]

Holes for mounting screws were placed in the corners of the board so it could be mounted in the place of a 5.25” drive slot. With this setup the whole dummy load unit could be placed on or under a desk and be used without the need for moving it around when changing to a different measurement device or amplifier.

The printed circuit board was two sided and thus had a ground plane on both sides of it. This was required in the datasheets of both the INA1650 and the LM2611. It also resulted in a more stable ground connection throughout the board. The two sided board also helped to avoid traces from crossing each other which in turn made designing the board easier. The traces relocated to the other side of the board were connected to each other with vias. Vias are copper plated holes that go through the board making a galvanic connection through the board [12]. A grid of vias were also created throughout the board to ensure a stable ground connection for all the components. This method is called via stitching.

The input and output jacks were placed on the front of the PCB for easy accessibility. The SMD components were then placed around the corresponding jacks according to the recommendations in datasheets [11][12] and PCB design practices. The layout can be seen in Figure 3.

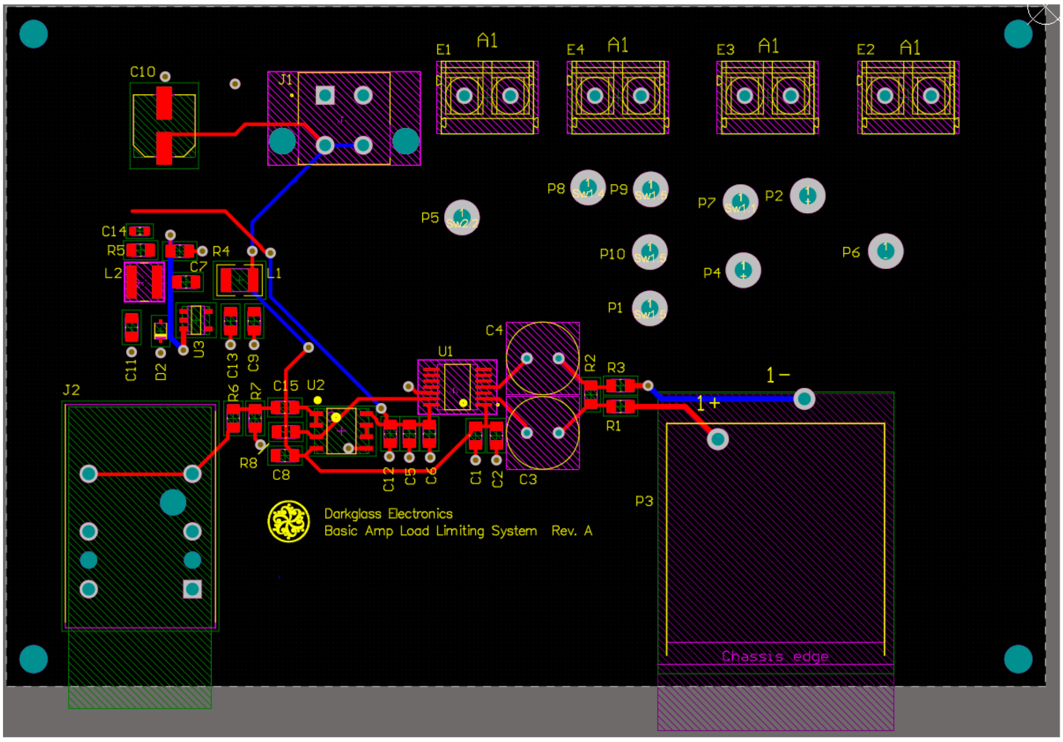

Figure 3. Layout of the PCB

The grouping of the components and the traces are visible in this picture. The red traces are on the top side of the board and blue ones on the bottom side.

Components on the PCB were connected to one another by 0.7 mm wide traces on the copper layer. This width was chosen as the optimal balance between efficiency and space economy. The traces had to be as wide as possible while still narrow enough to fit between the tight spaces between the components. For most of the board this width was used for consistency and simplicity.

The traces to and from the screw terminals were created as large polygons instead of copper traces used elsewhere on the board to ensure they can handle high currents. The PCB being the core of the circuitry, measures were taken to ensure that it would not be damaged if a 900 watt amplifier was run into the dummy load at full power. The traces through which the highest powers go through were designed as polygons on the copper plate of the board. On these polygons the narrowest point was 5.2 mm, length 38.8 mm and thickness of the copper was 0.04 mm.

3.4 Negative Voltage

The power source used to power the circuits was a power supply unit for an ATX specification setup [9]. The ATX power supply originally intended to be used in the setup had no -12 V output built in it. This is why an inverting Ćuk converter was designed. Due to issues with availability of the power supply, another model was opted for. The newly selected model had the -12 V supply so the Ćuk converter became obsolete.

The inverting Ćuk converter was designed around Texas Instruments’ LM2611. The layout for the inverting circuit was found in the LM2611’s datasheet [11].

The schematic was implemented in Altium Designer to create a PCB around the chip. The example gave the values for all the components, but two of the resistor values needed to be calculated to have the VCC voltage match the VEE voltage.

The circuit of the Ćuk converter can be seen in Figure 4.

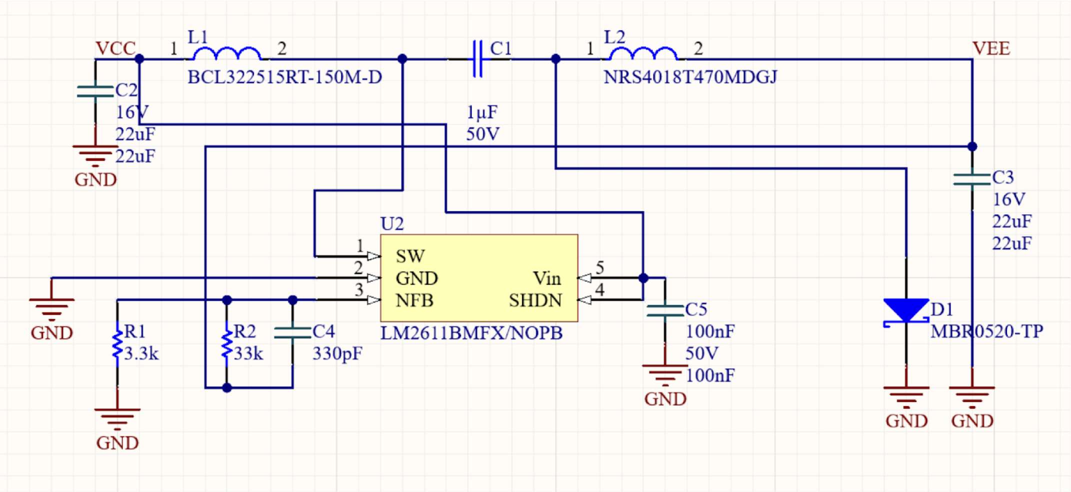

Figure 4. An LM2611-based Ćuk-converter was designed to supply negative 12 volt supply for the instrumental amplifier.

3.5 Voltage Divider

The INA1650 chip was operating with a +12 V as VCC and -12 V as VSS from the ATX power supply and the LM2611 inverting circuit. The datasheet for the INA1650 stated that recommended input voltages should be VCC - 2 V on the positive side and VSS + 0.25 V on the negative VSS side. These figures meant that the +45 V and -45 V needed to be lowered to at least +10 V and -11.75 V respectively. To lower the voltages a voltage divider was placed between the signal source and the INA1650 in parallel with the power resistors. Because of the two-sided signal input the voltage divider needed to be built with three resistors to get both of the voltages down. Despite the asymmetric maximum values of the INA1650's input, the divider was kept symmetrical since the signal from the amplifier and the signal needed out from the circuit were symmetrical to the ground level. The resistor values were selected to leave some headroom in the signal to avoid unwanted distortion in all situations. [11]

The structure of the voltage divider and voltage drops can be seen in Figure 5

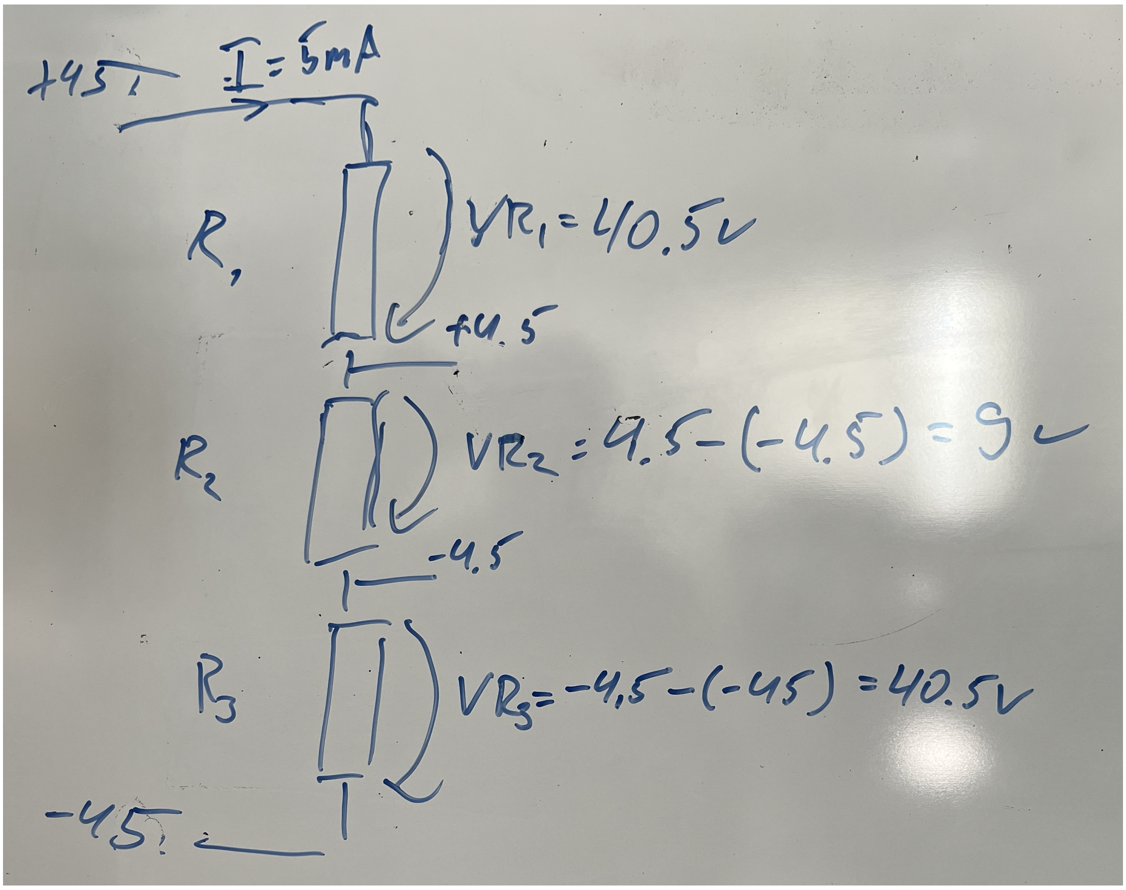

Figure 5. The structure and the values for the voltage divider

The current used in the calculations was 5 mA which was half of the maximum current the INA1650 was rated for. The input voltages on each side were lowered to one tenth of the original. Hence the voltage over R_2 was the difference between the remaining voltages after the first resistors on positive and negative sides.

$$VR_2 = 4.5\ V - (-4.5\ V) = 9\ V \qquad (1)$$

In this calculation the variables were:

V_i = input voltage from V1

R_1 = resistance of the resistor closest to the positive voltage source
R_2 = resistance of the middle resistor
R_3 = resistance of the resistor closest to the negative voltage source
V_o = output voltage of the voltage divider
P = power
I = current
Imax = maximum current the INA1650 can handle

When calculating the values for the voltage divider, the power was set to be 500 watts and the power resistor load four ohms.

Power was current multiplied by voltage and current was voltage divided by resistance

$$P = IV \tag{2}$$

$$I = V/R \tag{3}$$

This meant that power was voltage squared divided by resistance

$$P = V^2/R \tag{4}$$

Which meant that the voltage was

$$V^2 = PR \tag{5}$$

So when

$$P = 500\ \mathrm{W} \tag{6}$$

$$R = 4\ \mathrm{oms} \tag{7}$$

Then

$$\mathrm{Vi} = \sqrt{500 \cdot 4} = \sqrt{2000} \approx 44.7\ \mathrm{Vp} \tag{8}$$

To simplify the calculations this value was rounded to 45 Vp. The same magnitude voltage was also present on the negative side. For simplicity's sake the voltage was lowered to one tenth of the original in the voltage divider. The voltage needed to be lowered from 45 V and -45 V by 40.5 V on both sides. This meant that there was 9 volt potential over R_2.

$I_{MAX} = 5mA$ (9)

$R_{TOT} = R_1 + R_2 + R_6$ (10)

$V = 90\ V$ (11)

The currents through the resistors were:

$I_1 = 40.5 / R_1$ (12)

$I_2 = 9 / R_2$ (13)

$I_6 = 40.5 / R_6$ (14)

And because the current was the same throughout the circuit

$I_1 = I_2 = I_3$ (15)

The resistances could be calculated from the known values

$40.5 / R_1 = 40.5 / R_6 = 9 / R_2$ (16)

$R_1 "9 = R_2 "40.5$ (17)

$R_1 / R_2 = 40.5 / 9 = 4.5$ (18)

Which was to say that:

$R_1 = 4.5 " R_2 = R_6$ (19)

When

$R_{TOT} = V/I$ (20)

So

$R_{TOT} = 90 / 0.005 = 18000$ ohm (21)

$R_{TOT} = 4.5R_2 + " R_2 + 4.5R_2 = 10R_2$ (22)

$10R_2 = 18000 \longrightarrow R_2 = 1.8$ kohm (23)

$R_{1,6} = 4.5 " 1.8$ kohm = 8.1 kohm (24)

Based on these calculations 8.2 kohm and 1.8 kohm resistors were chosen for the circuit.

The voltage coming out from each side of the voltage divider was calculated by multiplying the voltage of the input by the quotient of the resistance of the middle resistor divided by the sum of the resistances of all three resistors. The voltage between IN+A and IN-A was the sum of this calculation for both sides.

In Figure 6 the resistors of the voltage divider are named R1, R2 and R6.

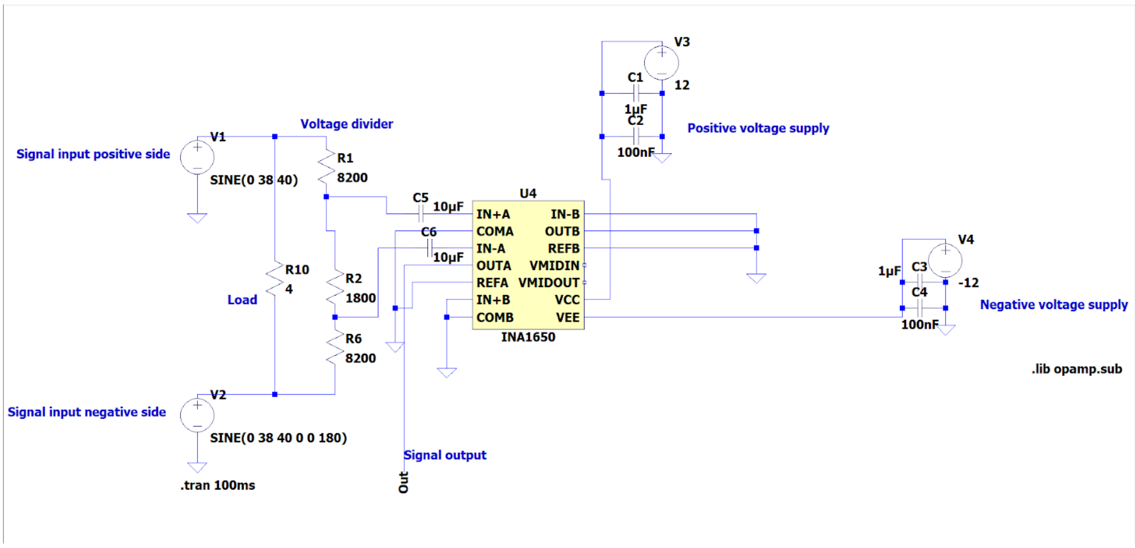

Figure 6. Pictured above are the signal input, voltage divider, INA1650 IC, signal output and power supply for the IC in relation to each other.

The signal was fed into the load from two sine wave generators, V1 and V2, both giving out 45 volts with 180° phase difference between them. This created a 90 volt potential over the four ohm resistor R10. The voltage divider consisting of resistors R1, R2 and R6 was placed in parallel with the R10 resistor which represented the power resistor load in the simulation. R1 and R6 had resistance of 8.2 kiloohm and R2 1.8 kiloohms. The attenuated signal was taken from between resistors R1 and R2 as well as between R2 and R6. The attenuated signal from between R1 and R2 went into the INA1650's IN+A pin and from between R2 and R6 into the IN-A pin. Both of the aforementioned signals were filtered with 10 µF AC-coupling capacitors for better low frequency CMRR performance [10].

3.6 Connectors and Power

Power needed to be fed to the circuit in order for it to function. The power was taken from an ATX power supply’s 12 volt output [9]. ATX power supply was equipped with a Molex Micro-Fit pin and socket -type connectors. The power supply had a 24-pin and a four pin connectors.

Signal was fed in through a 6.3 mm phone jack as it would on a speaker cabinet. The same type of connector was used for the output for the audio analyzer.

The wires from the power resistors were connected to screw terminals that were soldered to the PCB. This was meant to be helpful if the type of the power resistors needs to be changed in the future.

The switches controlling the impedance selection were connected to the PCB via wires that were soldered onto the board so that the switches could be easily placed outside the chassis.

3.7 Power Resistors

The resistors were a central part of a dummy load and needed to be selected according the needs of the particular application. When designing a load for testing audio amplifiers, choosing resistors that resembled in values the speakers they were replacing made it easier to compare different measurements to real-life environment. The resistor used in this project was Vishay RBEF03008R000KFB00. This specific model was chosen because of its power handling capability and impedance values as well as availability on the time frame the project required. In addition to this, the model had a tubular design which made directing air through the resistors easy and effective.

An alternative to the Vishay resistor was TE Connectivity HSC3004R0J, but it was not chosen as using that model would have required a more complicated cooling system, which would have increased the weight and overall complexity of the device unnecessarily.

The Vishay RBEF03008R000KFB00’s dimensions were 215.9 mm by 28.6 mm which made them fit conveniently inside an ATX chassis that was designed to house an ATX motherboard which was 305 mm high and 244 mm wide.

The resistors were mounted in a horizontal position inside the chassis so that the built-in fans could blow air through them.

The profile and mounting points of the Vishay RBEF03008R000KFB00 are depicted in Figure 7

Figure 7. The Vishay RBEF03008R000KFB00 power resistor [16].

The Vishay RBEF03008R000KFB00 power resistor has a tubular profile with a wire wound around a ceramic tube. On both ends of the tube is a metal band that extends out as a mounting point for the resistor.#

3.8 Transformer versus operational amplifiers

One way to separate the grounds from each other is to use an isolation transformer. A transformer would have been a simple way to achieve separation of the grounds of the amplifier and the dummy load as it would have transferred electrical power through induction between two coils. This method was forgone as it would have added unwanted weight to the unit.

3.9 Wiring

A good reference point to start from when choosing the wires for this project was the ones used inside the Darkglass speaker cabinets.

Darkglass’ 900 watt amplifier heads utilize ICE Power 700AS1 power amplifiers. These D-class amplifiers are capable of putting out a signal with maximum voltage of 76 volts and maximum current of 30 amperes. The maximum continuous power of these amplifiers is 710 watts when connected to 230 V and 50 Hz mains. The maximum peak power is 900 watts. The total efficiency is 84% when the load is 4 Ohms and power 700 watts. The amplifier is able to run at this power only for a limited amount of time. The dummy load is intended to be used at lower power for a prolonged periods of time.[3]

According to American Wire Gauge (AWG) table of wire gauges, 30 A current requires a minimum on AWG 7 wire size [4]. This size of copper wire equals to a conductor diameter of 3.66522 mm and cross section of 10.6 mm^2 [5].

3.10 Connectors

To connect an amplifier and all the measurement devices to the dummy load, the load needed to have appropriate connectors on it. All the external connections were placed on a connector interface that was placed in the front side of the PC chassis.

Darkglass amplifiers and speaker cabinets have Speakon connectors manufactured by Neutrik. The connectors accept both the Speakon connectors and regular 1/4” phone connectors widely used in music equipment. One 1/4” jack was assigned to be used with headphones to get an audible form of the signal for subjective analysis.

The signals were to be analyzed by an audio analyzer and an oscilloscope. The audio analyzer used at Darkglass Electronics was Audio Precision’s model APx515 B-series.

3.11 Enclosure

As the device was intended for internal use in the company only, the outward appearance was not a high priority. A chassis of a personal computer was proposed for this purpose for its size and easy maneuverability and convenience of being able to power the device with a regular ATX power supply.

3.12 Load Selection

The dummy load was needed to substitute speaker cabinets and their combinations of different impedances. For this reason four different impedances were required to cover the most common setups. Darkglass amplifiers are designed to be compatible with cabinets ranging from two to eight ohms of impedance. 16 ohm load is also common when two cabinets are connected to one amplifier.

After considering several different configurations of four and eight ohm power resistors the choice was made to utilize four eight ohm power resistors and have two DPDT switches to control the load selection. With this setup only one input was needed. This was particularly convenient since in that case only one Speakon connector was needed to accommodate both a phone jack and a Speakon plug.

The power resistors were connected in a series-parallel configuration where the switches could be used to either connect or disconnect series and parallel settings.

The different configurations of the power resistors could be selected with two dual pole dual throw, or DPDT switches. Depending on the position of the switches the load was either two, 4, 8 or 16 ohms. The different configurations are shown in Figure 8.

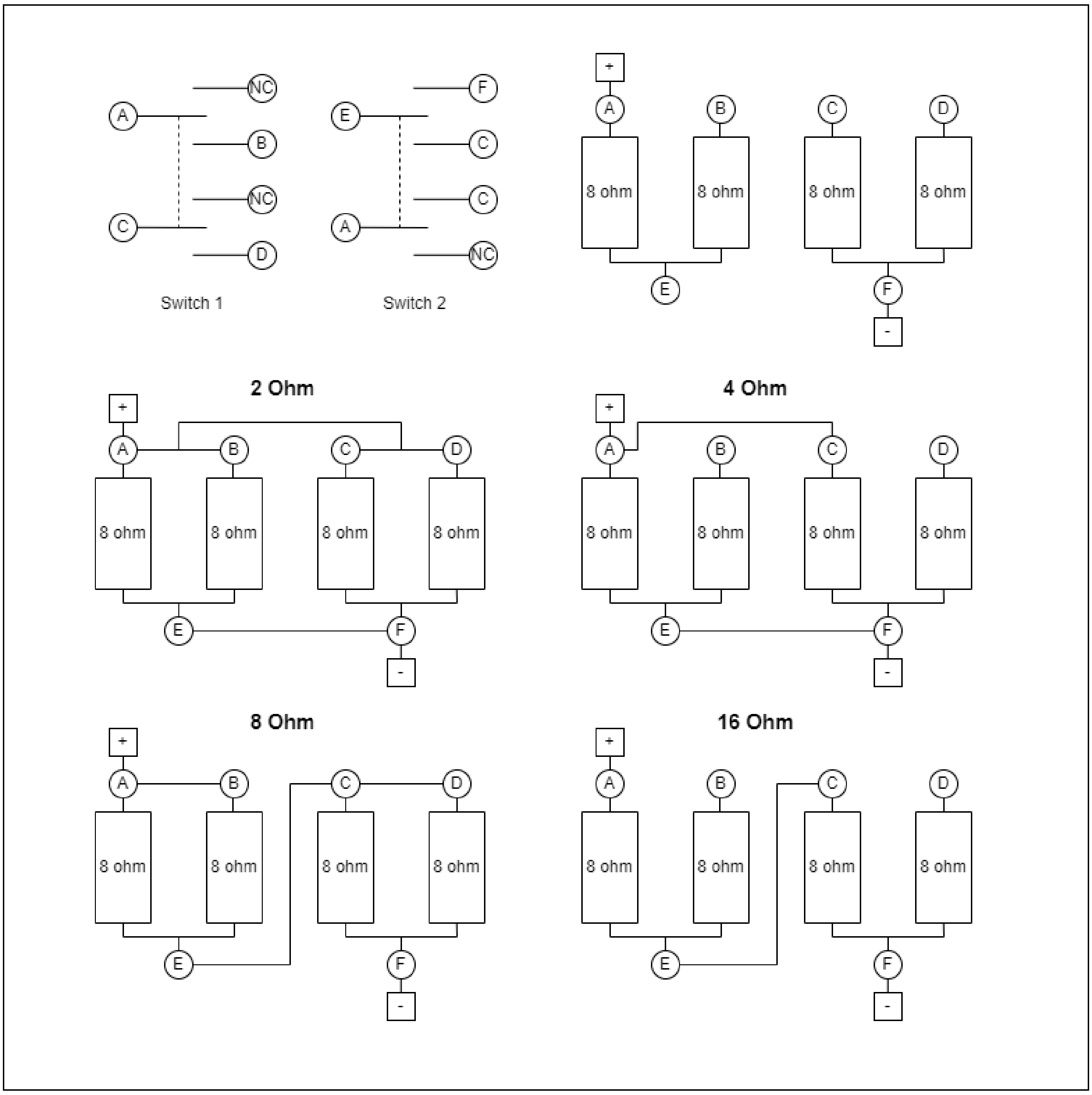

Figure 8. The configurations of the power resistors

Two DPDT switches allow four confirmations to be selected by engaging or disengaging series and parallel connections. When Switch 2 is in up position, Switch 1 selects between two and four ohms. When Switch 2 is moved to down position, Switch 1 selects between eight and 16 ohms.

4. Simulation

The circuit of the instrumental amplifier was implemented in LT Spice to test outcomes of different combinations of components without the need to build a physical circuit for that purpose. Different setups and wirings were tested to find the right components and connections needed to suit the needs of this project. The schematic first created in LT Spice was copied to Altium Designer to design a PCB for it.

Some of the simulations were run by feeding a +38 V and -38 V sine wave to the circuit consisting of the voltage divider and the INA1650 instrumentation amplifier

Figure 9 shows the two sine waves coming out of the voltage divider before going into the INA1650.

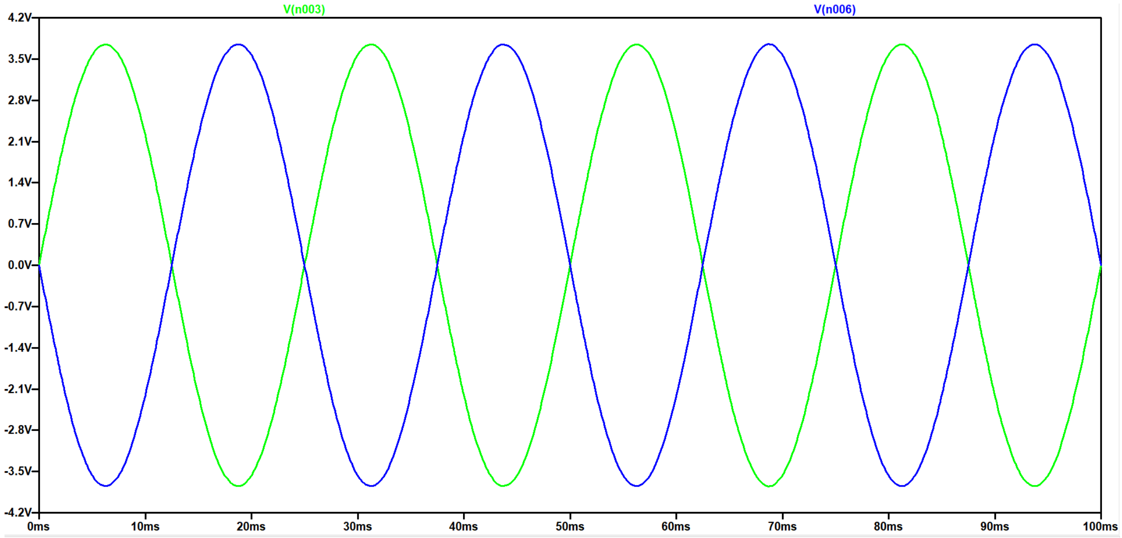

Figure 9. The blue and green lines represent the reduced signals after the voltage divider. The +38 V and -38 V voltages of the signals are lowered to +3.8 V and -3.8 V.

In the simulation the impedance value of the main load was set to four ohms.

5. Measurements

The PCB was connected to a Darkglass AlphaOmega 900 bass guitar amplifier, Audio Precision APx515 audio analyzer and a set of four four ohm power resistors with power rating of 300 W. The PCB was powered with the ATX power supply. The APx515's signal generator was used to supply the AlphaOmega 900 with sine wave signal.

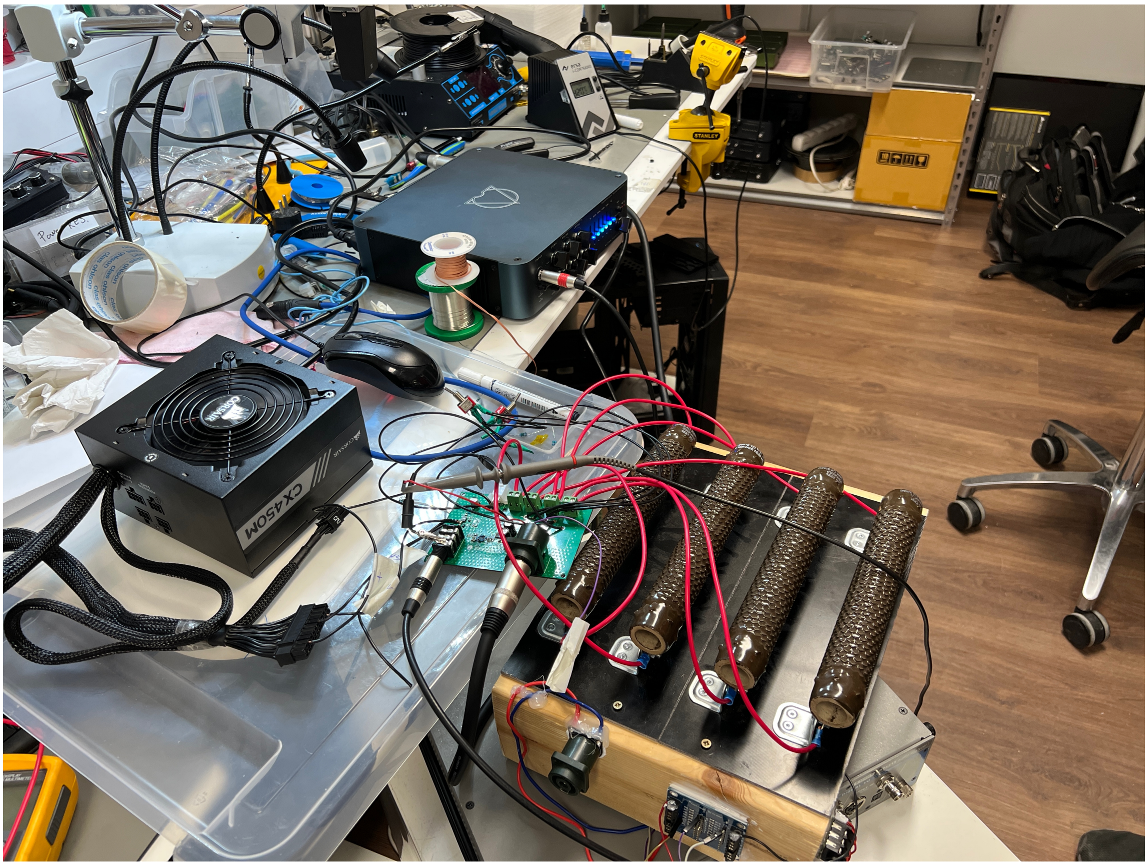

Figure 10. The setup for the preliminary test of the PCB

An ATX power supply powers the PCB with the voltage divider and instrumentation amplifier. Power resistors are connected to the PCB by red wires. The audio analyzer lies under the pack of power resistors. The audio analyzer feeds a test signal to the amplifier which can be seen at top middle of the picture with blue LED's. The amplifier's output feeds the amplified test signal to the PCB.

The APx515's signal generator was giving out a sine wave 2 Vpp at 1000 Hz. This signal was then amplified in the AlphaOmega to 10 Vpp. The voltage divider then lowered the signal to 1 Vpp.

In Figure 11 the input of the PCB and the output of the signal generator are visually comparable.

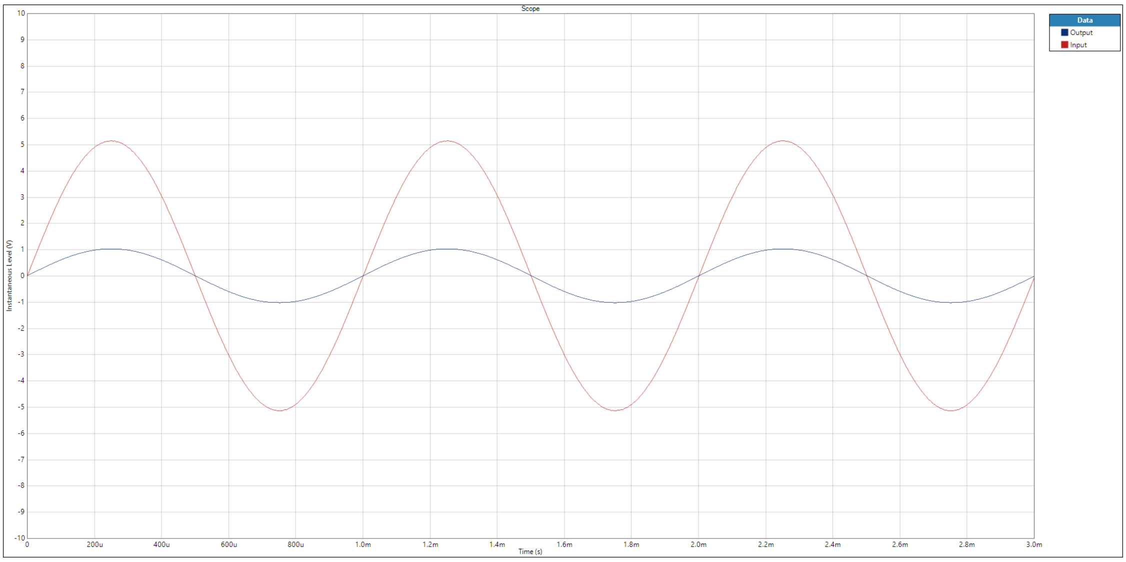

Figure 11. Input of the PCB and output of the signal generator.

The blue line varying from +1 V to -1 V is the signal coming from the APx515 signal generator and going in to the AlphaOmega. The red line represents the amplified signal the AlphaOmega puts out.

From Figure 12 the functionality of the voltage divider can be seen as the voltage coming out from the voltage divider is right around 1 Vpp.

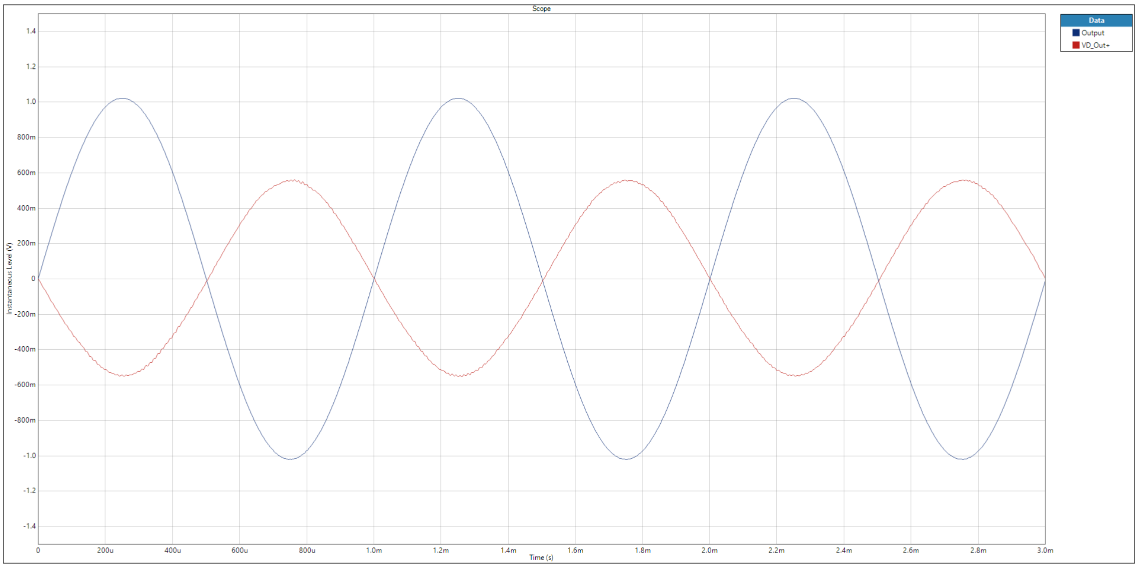

Figure 12. The output signal of the signal generator and the reduced signal after the voltage divider.

The values on the graphs were not all exactly at marks they represent as the output from the AlphaOmega was adjusted by hand and thus was not exactly accurate.

The total harmonic distortion levels from the amplifier and the PCB were drawn as a graph in in Figure 13.

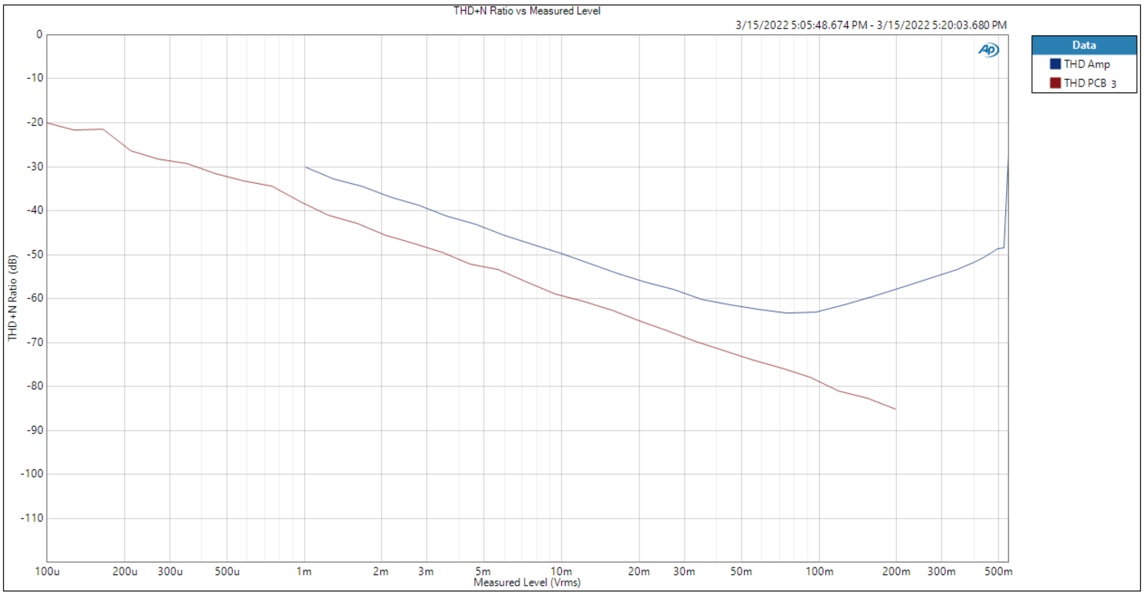

Figure 13. The THD+N levels of the amplifier output and the PCB.

The blue line represents the total harmonic distortion and noise in the out coming signal of the amplifier and the red line shows the similar measurement from the PCB.

The total harmonic distortion was measured from both the output of the AlphaOmega and the output of the PCB created in this project. Similar measurements were also made by feeding the signal directly to the PCB from the audio analyzer's signal generator.

THD is the ratio between the sum of all the powers of the harmonic components and the power of the signals fundamental frequency. THD+N measures total harmonic distortion and the noise compared to the fundamental frequency.

Conclusions

The project started off with the goal of replacing the old cumbersome dummy loads in use in the company with one easy-to-use all purpose unit. In the initial stage of setting the required specifications of the unit, the goals were intentionally set very high. Some of the originally specified features had to be left out in the process.

The original time frame allocated for completing this project was three weeks. It was obvious from the beginning that it was not going to suffice. By the end of those three weeks the time was extended first by a week and then by another. All in all the design and testing of the dummy load was completed in around five weeks. At that point the PCB had been designed and ordered from a manufacturer in China, all the components on the bill of materials had arrived and the circuit was put together so that it could be tested. The final assembly was seen as a lower priority in the time crunch as that was more of mechanical designing instead of electronics.

The functionality of the PCB and the system as a whole was tested and the results were satisfactory. All the intended measurements could be taken from the device and all the all the different parts of the system worked together as expected. Powering the PCB from the ATX power supply and fitting all the parts inside the PC chassis turned out to go according to the plan.

The project was brought to a point where the device can be assembled by anyone without deeper knowledge of the design or circuit, so the project can be seen as a success from the design point of view.

Some design errors are to be corrected in the PCB layout for future revisions and some new functions can be added. The device was originally meant to have a headphone amplifier built in but that part was left out due to time constraints. On the mechanical side of the design there are some tasks to be done. One of these is the designing of mounting brackets for the PCB. Said brackets will be designed and 3D printed later in the near future.

The main takeaway from the thesis overall is that a research and development project requires careful planning when it comes to setting expectations for schedules. !

References

[1] American Radio Relay League. 1968 Radio Amateur Handbook p.22

[2]User manual of Darkglass Electronics Microtubes 900 v2. Available from: <https://www.darkglass.com/app/uploads/2019/01/M900v2-Manual-v3-Website.pdf> [Accessed 6.4.2022]

[3] ICE Power 700AS1 data sheet. Available from: <https://icepower.dk/download/1652/> [Accessed 6.4.2022]

[4] Amplified Parts. Wire Gauge Guide. Available from: <https://www.amplifiedparts.com/tech-articles/wire-gauge-guide>

[5] American Wire Gauge. Available from: <https://www.altanatubes.com.br/Blog/AWG/ASTM-B258.pdf > [Accessed 6.4.2022]

[6] APx515 B Series Audio Analyzer user manual. Available from: <https://www.ap.com/download/apx515-installation-instructions-and-full-specifications-2/?wpdmdl=5868> [Accessed 6.4.2022]

[7] Texas Instruments. Analog Engineer's Circuit: Amplifiers. Available from: <https://www.ti.com/lit/an/sboa282/sboa282.pdf?ts=1644502523646&ref_url=https%253A%252F%252Fwww.google.com%252F> [Accessed 6.4.2022]

[8] Audio-Technica ATH-M50x Specifications. Available from: <https://www.audio-technica.com/en-us/ath-m50x> [Accessed 6.4.2022]

[9] ATX Specification version 2.2. Intel Corporation. Available from: <https://web.archive.org/web/20120725150314/http://www.formfactors.org/developer/specs/atx2_2.pdf> [Accessed 6.4.2022]

[10] Texas Instruments INA1650 datasheet. Texas Instruments 2016 Rev. 2018 . Available from: <https://www.ti.com/lit/ds/symlink/ina1650.pdf?ts=1646317590545&ref_url=https%253A%252F%252Fwww.ti.com%252Fproduct%252FINA1650> [Accessed 6.4.2022]

[11] Texas Instruments LM2611 datasheet. Texas Instruments 2001 Rev. 2015 . Available from: <https://www.ti.com/lit/ds/symlink/lm2611.pdf?ts=1646314683333&ref_url=https%253A%252F%252Fwww.google.com%252F> [Accessed 6.4.2022]

[12] PCB Via An I-Depth Guide to PCB Vias and PCB Design. Guy Shemesh 2015. Available from: <https://www.epiccolo.com/articles/pcb-vias-guide#via-types> [Accessed 6.4.2022]

[13] Fundamentals of Physics Extended (10th edition). David Halliday, Robert Resnick, and Jearl Walker p.754

[14] Small Signal Audio Design (Third edition). Routledge 2020. Douglas Self

[15] Op Amps For Everyone. Newnes 2017. Bruce Carter, Ron Mancini

[16] Vishay web page. Available from: <https://www.vishay.com/resistors-fixed/list/product-31807/> [Accessed 6.4.2022]