ENGR 301-01
Dec 17 2019
Moshe Stanylov

FINAL PROJECT: LOW NOISE HEADPHONE AMPLIFIER

Objective:
In this project I have designed and implemented a Low Noise Headphone Amplifier. The purpose of this project was to gain a better understanding of the design process of printed circuit boards and to practice common engineering tasks such as drafting a schematic, generating a pcb layout from a schematic, generating a bill of materials and soldering components to a PCB.

Design:

Schematic: The circuit is composed of three smaller circuits, a power supply and two mono audio channels.

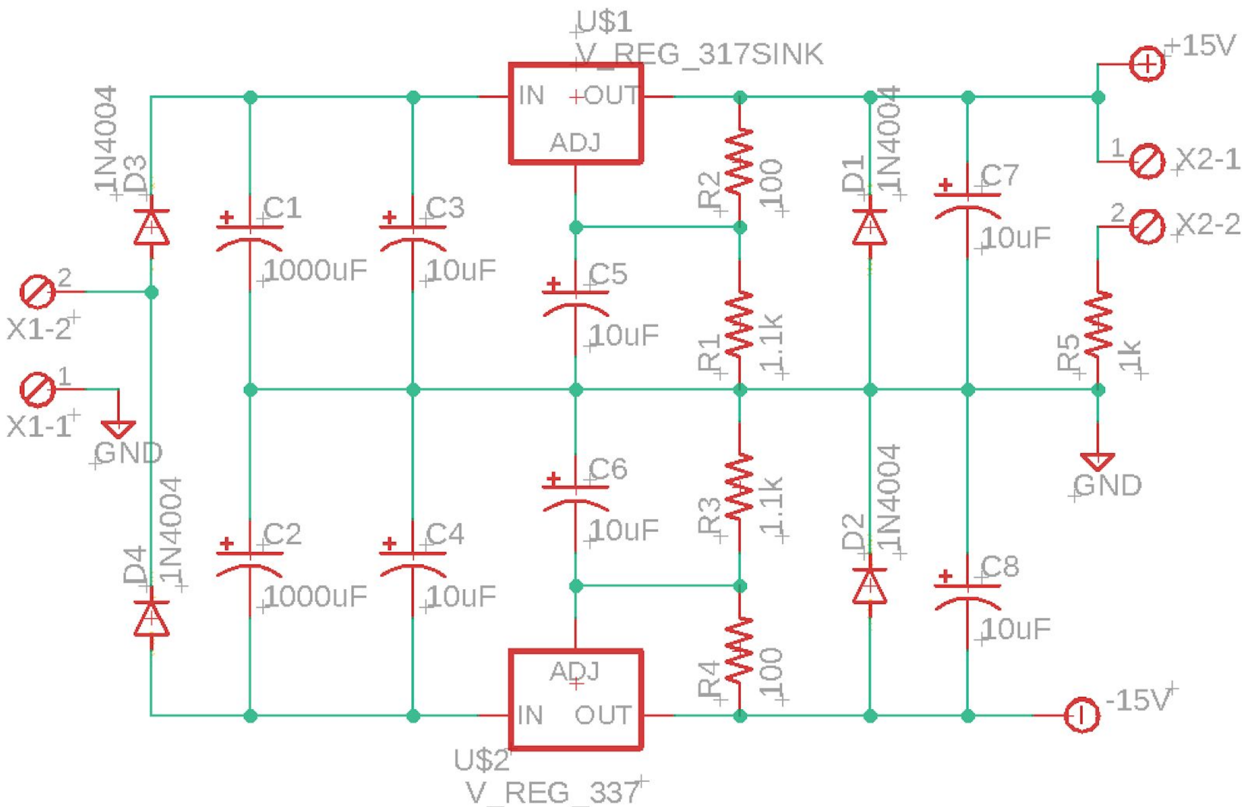

Fig.1 Power supply

The power supply in Fig.1 receives $15V\,AC$ at the input terminal labeled X_{1-2} from a $120V - 15V\,AC/AC$ transformer and outputs a $-15V\,DC$ and a $+15V\,DC$ at the output terminals labeled $-15V$ and $+15V$. It achieves the rectification by using a combination of diodes, capacitors, resistors and voltage regulators.

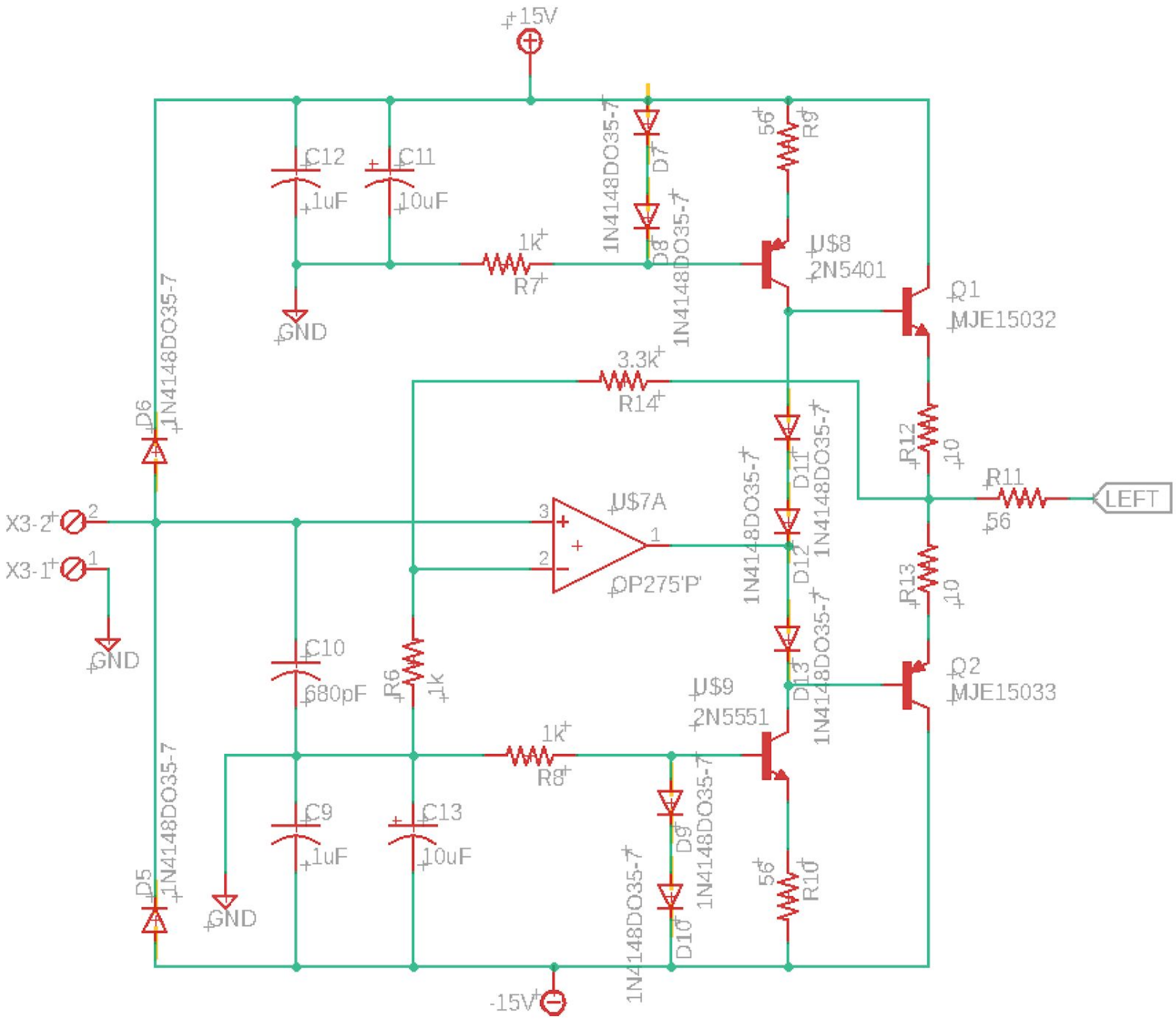

Fig.2 Left audio channel

Since this is a stereo amplifier, it is crucial to get the gain and signal to noise ratio of each individual mono channel to match. This is why the left and right audio channel schematics are exactly the same. There will still be some small differences since the tolerance of the components can’t be 0% but this difference is hardly noticeable.

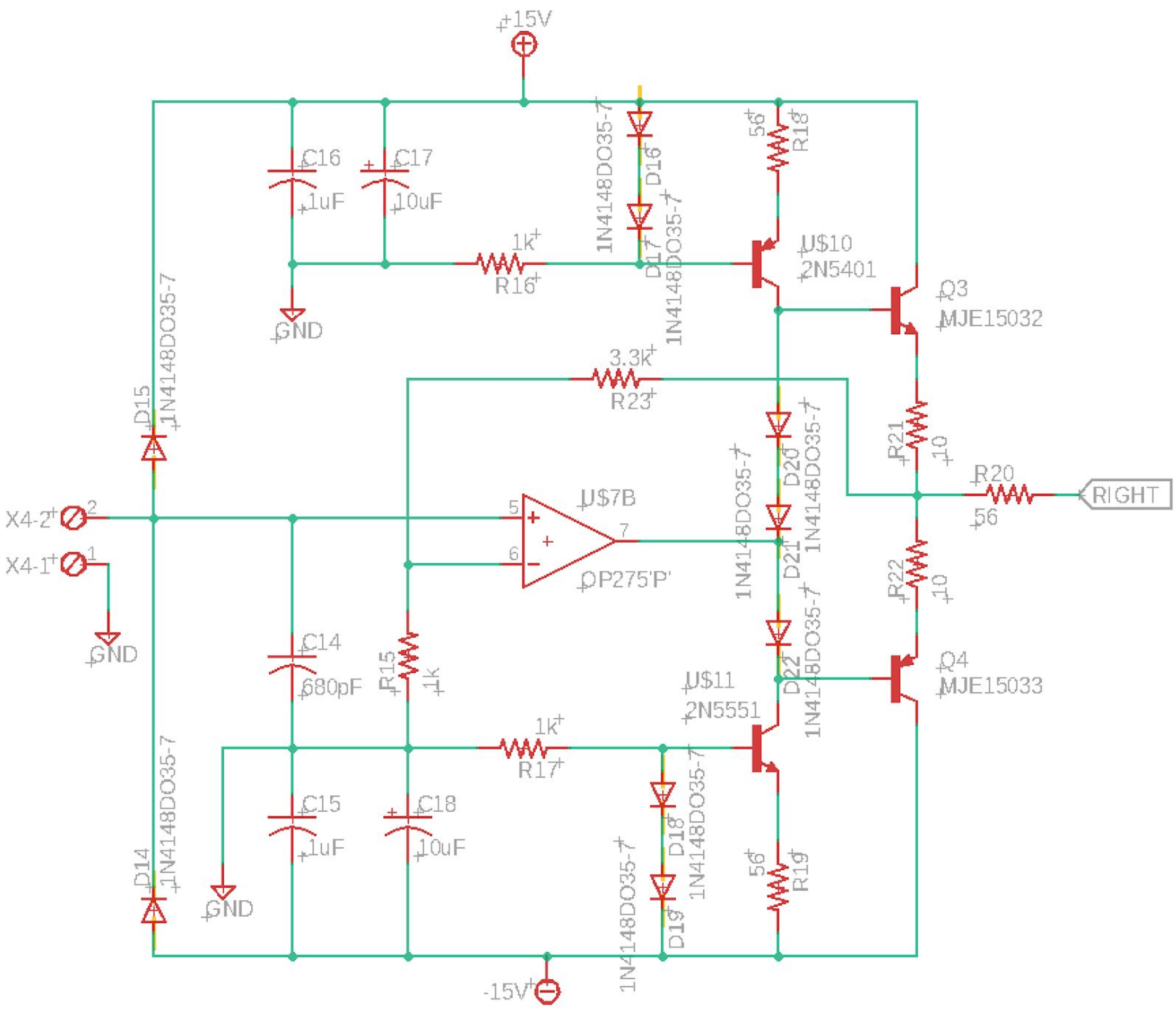

Fig.3 Right audio channel

Fig.4 supply rails for OP275

The left and right audio channels both receive an audio signal and amplify it using a non inverting amplifier. The operational amplifier that is used in the circuit is a OP275, with two channels sharing the same IC.
Dividing the entire schematic into three portions helped later with the board layout since I was able to easily separate the power supply from the signal path.

Board: While working on the board layout I made sure that the packages that I use in the layout match with the components that I will be ordering. I also tried to keep things consistent and use components of different values from the same series.

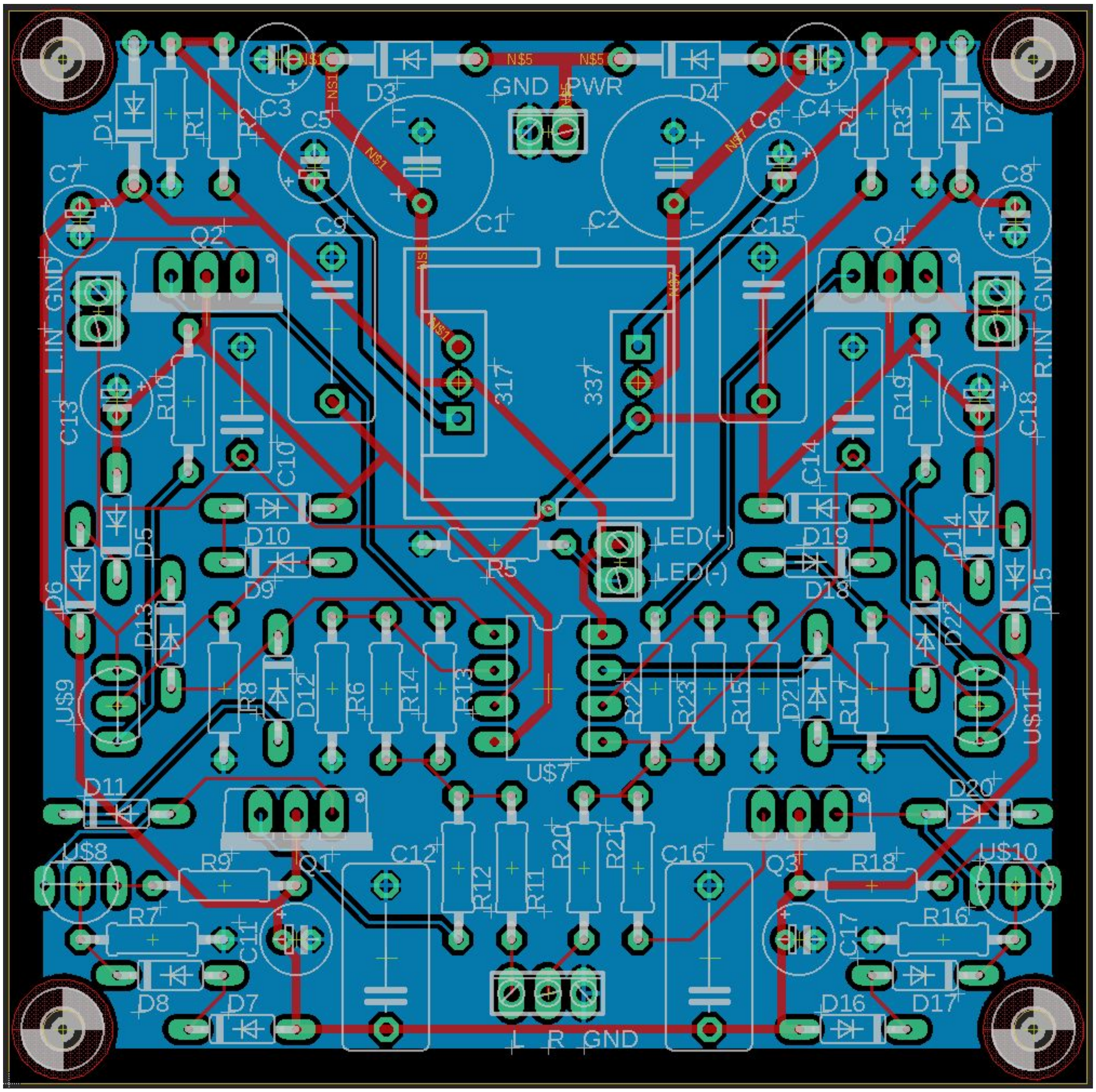

Fig.5 Board layout in Eagle

I’ve placed the power supply components in the upper corner of the board far from the signal path. By doing so I reduced the $60hz$ noise that can be caused by induction, since the wires carrying $15VAC$ are far from the wires that carry audio signal.
Another improvement to the layout was to define net class parameters, I’ve defined the wires carrying $15VAC$ to be the thickest, the wires carrying $\pm 15VDC$ thick and the rest was set to default.
The majority of the back panel was used as a ground plane. This helps to further reduce noise.
For the connections from the PCB to external components I used terminal block plugs, it helped with keeping the design clean and flexible when it comes to connecting and disconnecting external components.
The final stage of the PCB layout was to create labels for the terminals and to make sure all of the component labels are placed in a way that is easy to read.

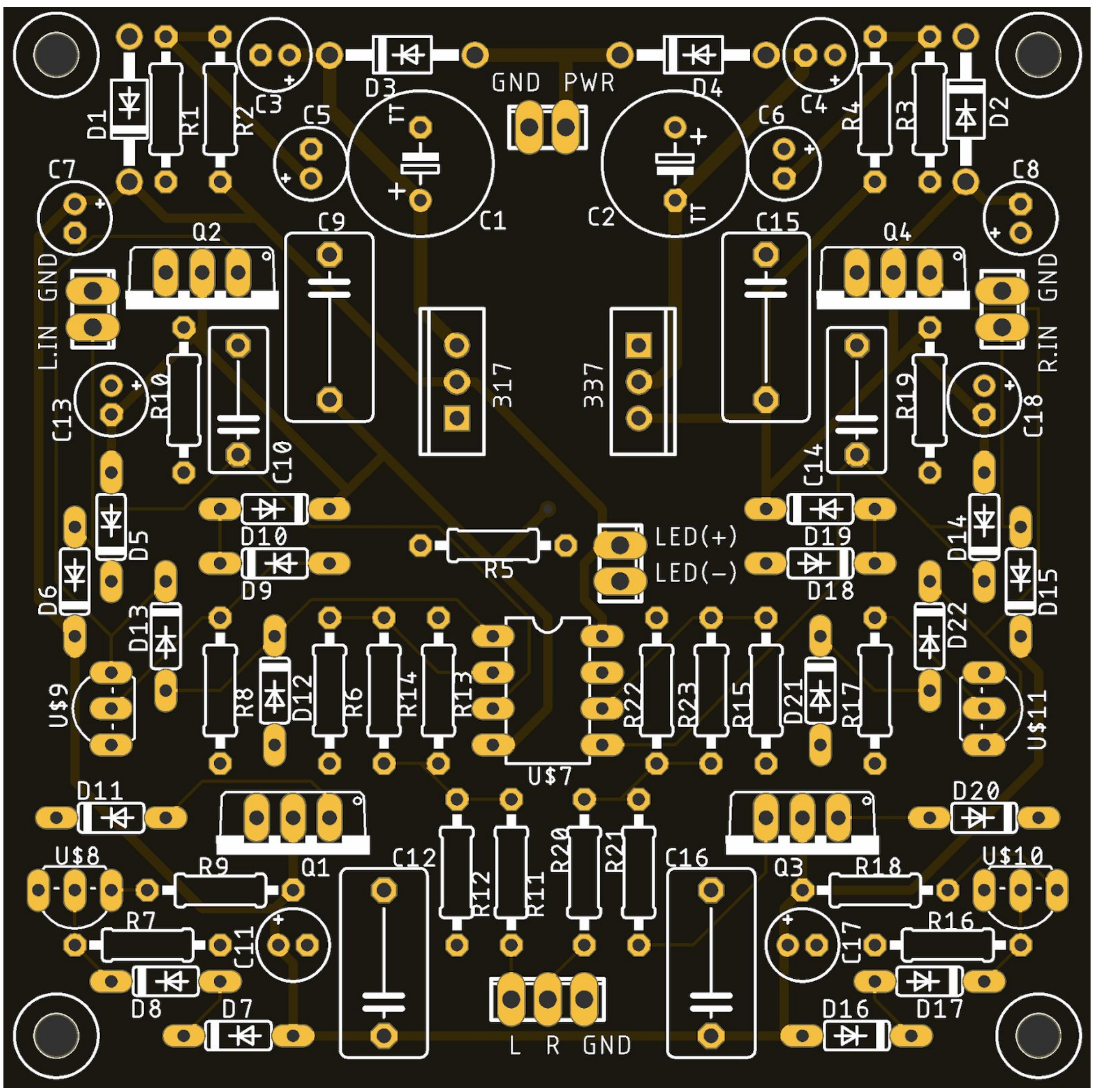

Fig.6 Board preview in Eagle

Components List:

Description	Part number	Quantity
Metal Film Resistors - $100\Omega \pm 1\%$	MBB02070C1000FCT00	2
Metal Film Resistors - $1.1k\Omega \pm 1\%$	MBB02070C1101FRP00	2
Metal Film Resistors - $1k\Omega \pm 1\%$	MBB02070C1001FC100	7
Metal Film Resistors - $56\Omega \pm 1\%$	MBB02070C5609FC100	6
Metal Film Resistors - $10\Omega \pm 1\%$	MBB02070C1009FCT00	4
Metal Film Resistors - $3.3k\Omega \pm 1\%$	MBB02070C3301FC100	2
Aluminum Electrolytic Capacitors - $1000\mu F \pm 20\%$	EEU-FP1E102	2
Aluminum Electrolytic Capacitors - $10\mu F \pm 20\%$	EEU-FR1H100	10
Film Capacitors - $0.68\mu F \pm 5\%$	PHE450HK3680JR17	2
Film Capacitors - $0.1\mu F \pm 5\%$	PHE450HA6100JR17	4
Diodes - Rectifiers	1N4004GP-E3/73	4
Diodes - General Purpose	1N4148TR	18
Bipolar Transistors - PNP	2N5401	2
Bipolar Transistors - NPN	2N5551	2
Bipolar Transistors - NPN	MJE15032G	2
Bipolar Transistors - PNP	MJE15033G	2
Linear Voltage Regulators	LM317T/NOPB	1
Linear Voltage Regulators	LM337T/NOPB	1
Potentiometers - $10k\Omega$	PDB182-E420K-103A	1
Op Amp - Dual Bipolar/JFET	OP275GPZ	1
Fixed Terminal Blocks - 2 Positions	1725656	4
Fixed Terminal Blocks - 3 Positions	1725669	1
1/4" TRS	NYS230	2

Fuse - $1A$	0ADAP1000-RE	1
Toggle Switch - SPDT ON-ON	100SP1T1B1M1QEH	1
Power Supply Jack - $5.5 \times 2.1mm$	B07MPS3QD8	1
Aluminum Enclosure - $88 \times 38 \times 100mm$	LKHZ002	1

Table 1 Component List

Board Testing:

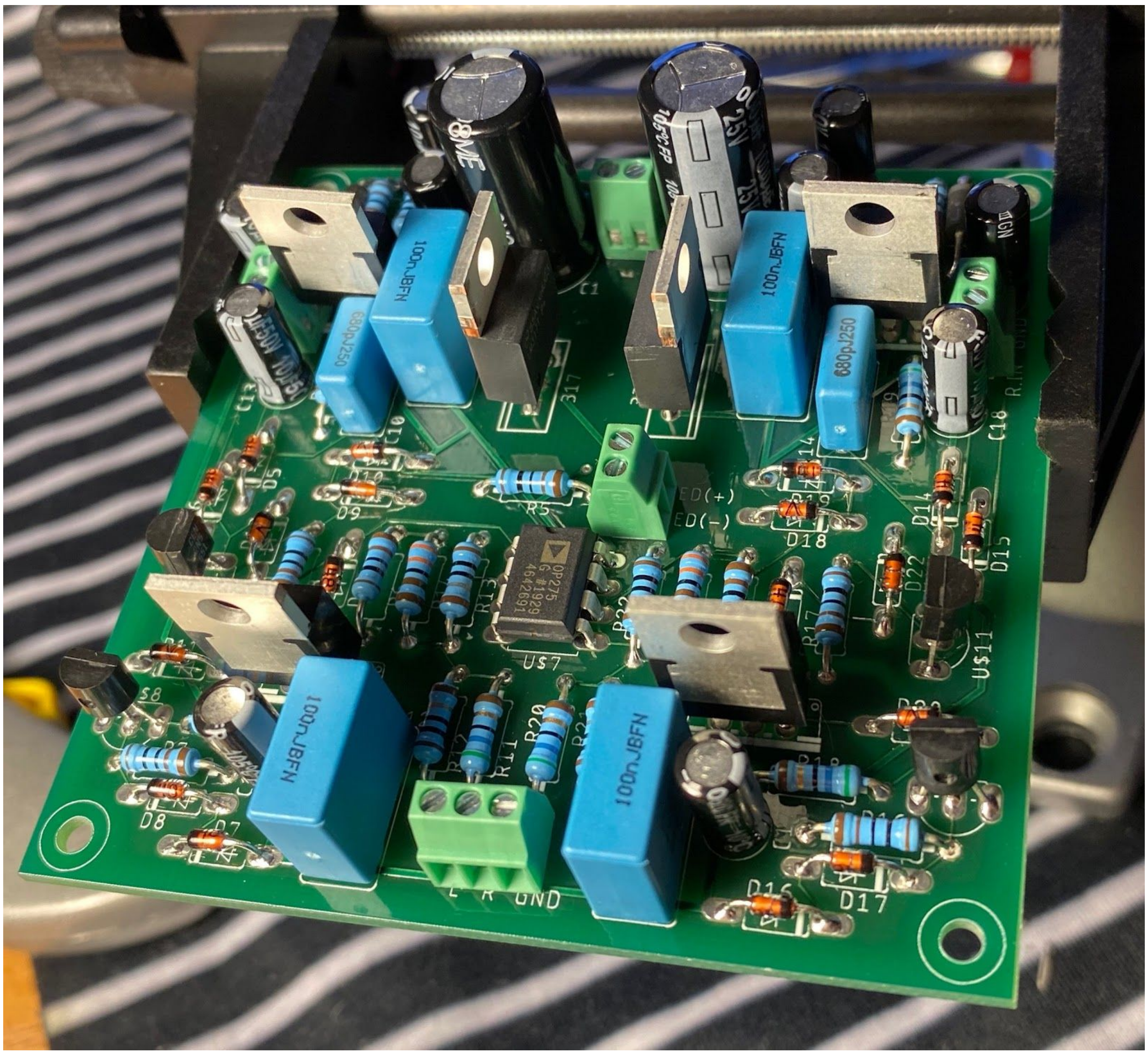

Fig.7 Board

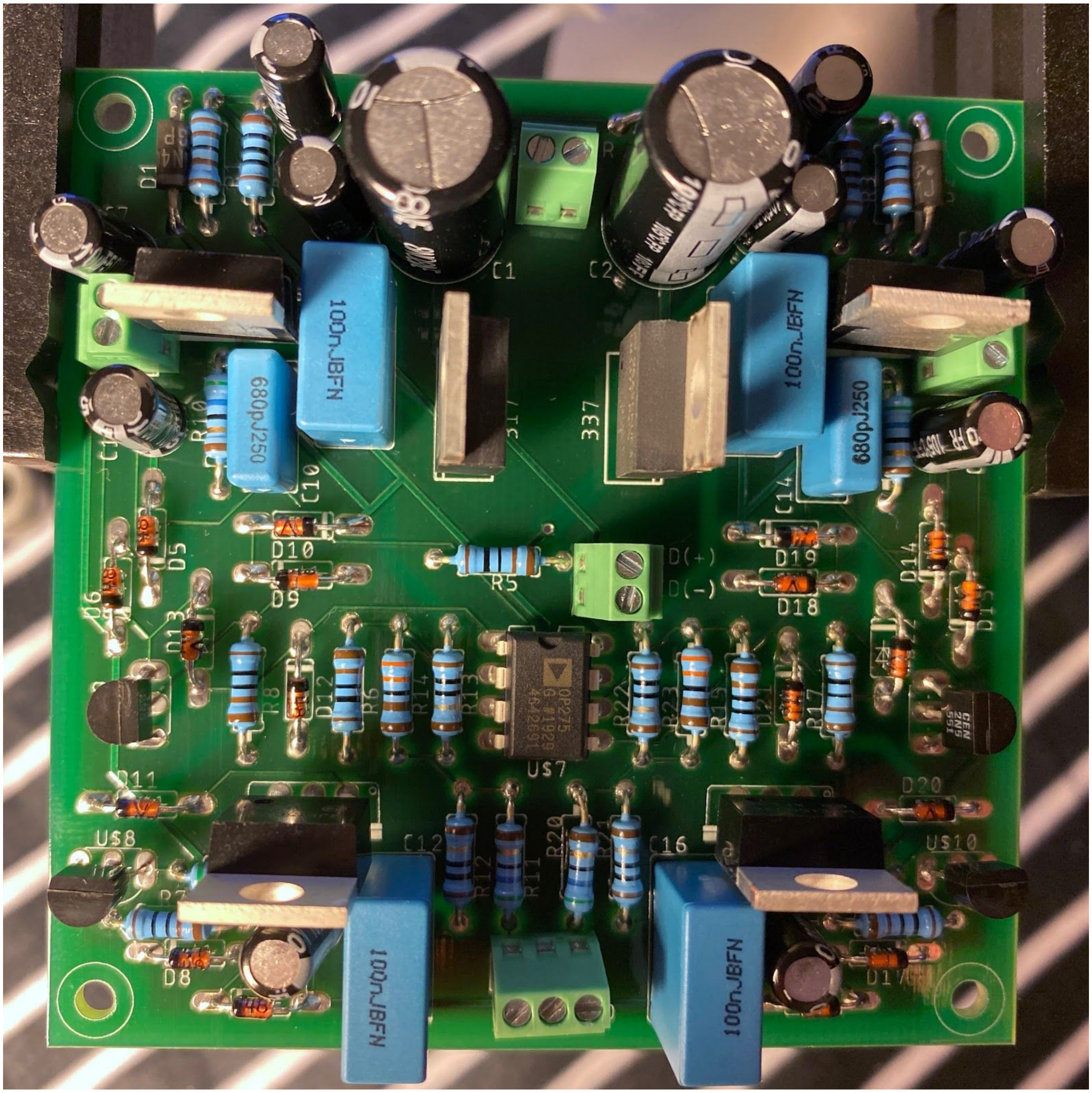

Fig.8 Board top

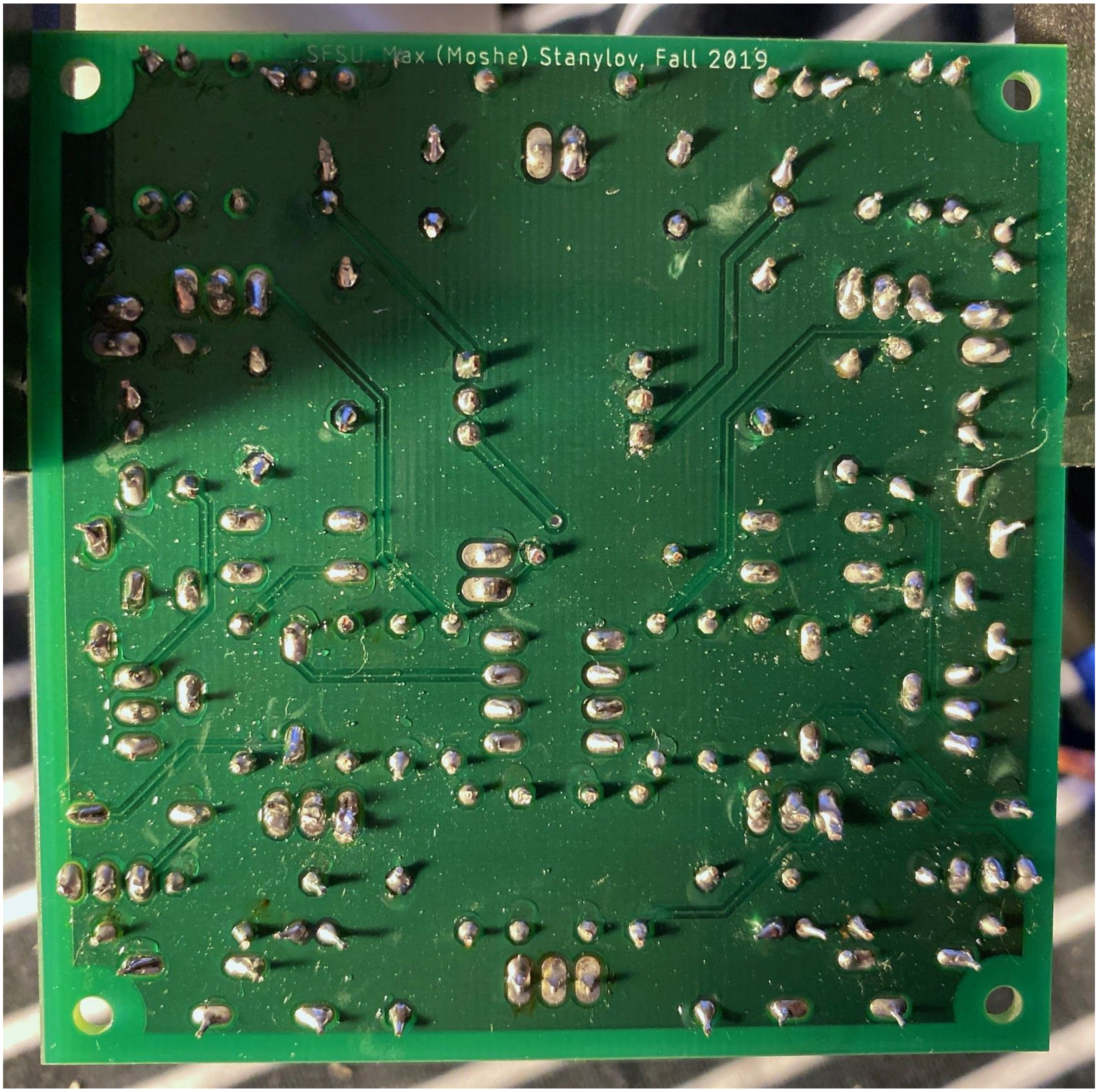

Fig.9 Board bottom

I've tested the board using a number of different inputs. The first test was done with an Audio Technica turntable, I've used an RCA to 3.5mm adapter in order to connect the output of the turntable to the input stage of the headphone amplifier and tested the potentiometer to see that I can control the volume and that the left and right channels are consistent. Everything worked but the potentiometer was wired backwards with the clockwise rotation decreasing the volume. Interchanging the connections on lugs 1 and 3 fixed the problem.

I then repeated the test twice using a laptop and a bass guitar as inputs, the circuit worked with both. The bass guitar was coming only from one channel since the output jack on a guitar is mono, this can be fixed by using an adapter.

Conclusion:
This project helped my understanding of the design and implementation process of printed circuit boards. It provided me with a lot of useful experience with Eagle. I learned how to create a board layout that takes into consideration the requirements of the design. Using terminal blocks for external components and paying attention to details such as package sizes of components, lead spacing and component placement made the assembling stage quick and easy.

Additional information:

Fig.10 Enclosure