Fuzz Pedal Design Project

Kyle Schaefer

General Engineering

ENGR 462

March, 2014

Table of Contents

Objective

The purpose of this project was to design and build a high quality, yet affordable, analog “fuzz” guitar pedal kit to be sold to other members of Cal Poly’s Audio Engineering Society. The Audio Engineering Society (AES) has hosted pedal-building projects in the past, but my project partner Michael Twardochleb first had the idea to build an original kit to be officially affiliated with the club. Michael had built other pedals of his own in the past, making him an excellent partner to have for this project. Our design was based on the classic Vox Tonebender’s circuit, which was in turn based on the Dunlop Fuzz Face pedal. Once a basic working fuzz circuit was assembled, we modified the design and components in order to suit our sound preferences and then discussed the inclusion of additional features. We also hosted AES meetings specifically for our project in order to generate interest and get input on what features people wanted to see. After settling on a design, we ordered a small run of prototype boards along with components and stompbox enclosures, built & tested them, and then presented them to the club before taking orders for an actual production run.

“What is a fuzz pedal?”

A fuzz pedal is an effects unit for guitar that is used to create distortion in the signal, and is typically housed in a “stompbox” casing which is connected between the guitar and the amp. One will often include dials for changing certain parameters such as volume, gain and tone, as well as a main foot switch used to simply activate or deactivate the effect.

Distortion is caused by boosting the signal past the point of clipping to create a distinctively rough & noisy sound, although the term “fuzz” generally describes a more adaptable brand of distortion that can retain more of the clarity and subtlety of the original signal. The 1966 Fuzz Face pedal, popularized by Jimmy Hendrix, was arguably the first major fuzz pedal to hit the market. Countless other models have been built and sold since then, but at the core of nearly all designs is a pair of transistors used to overdrive and boost the signal. Classic fuzz pedals, prized for their warm vintage sound, employed the use of germanium for these transistors, but a lot of more modern pedals have converted to silicon instead. The supposed superiority of one transistor type over the other is a commonly debated topic among fuzz pedal enthusiasts.

Figure 1: Various fuzz pedals

Design Considerations

One of the first design considerations that came up was whether our design would use the classic-style germanium transistors, or the more modern silicon types. Despite the fact that some guitarists prefer the sound of germanium, we decided to go with silicon for a variety of reasons. Germanium transistors are significantly more expensive than silicon, are generally less consistent in terms of tolerance and quality, and often react to environmental factors such as temperature and humidity. Jimmy Hendrix himself was known to bring multiple units of the same fuzz pedal on tour because they all behaved differently in various temperature situations. We felt that silicon transistors would be much more practical for our purposes, and we were confident that we could make a great sounding pedal regardless of our choice of transistor.

However, the unstable nature of germanium transistors was only one of the reasons for the wide range of variability seen in classic fuzz pedals - many of the other circuit components from the original designs were made to an inferior tolerance compared to parts that are available today. So, then, it made sense for us to use lower-tolerance components than the original designs in order to avoid this problem.

Another important design factor to keep in mind throughout our project would be the cost of producing our kits. We wanted to be efficient and economical with our choice of parts and production methods in order to keep our retail price low. High quality fuzz pedals can often cost hundreds of dollars, but we wanted our kits to be well under $100. Our initial estimates, roughly based on the online prices of parts and enclosures, placed our pricing range at somewhere from $40-$70.

Beginning the Design

The first step of our project was to simply build the ToneBender circuit as described in its diagram, but with silicon transistors as previously stated. We did this on a temporary breadboard setup, with a variety of transistors and other parts at our disposal. We had it wired up to a set of 1/4 inch input and output jacks so that we could actively test it through Michael's guitar and amp setup, with sound and tone in mind from the very beginning. After some testing and experimentation, we determined that the circuit elements most worth thinking about switching were the transistors and the input capacitor. Different transistors had a very noticeable effect on the tone, and different capacitor sizes modified the level of bass response which, was important because we wanted our pedal to have a nice, full sound without getting overly muddy or "boomy" in the lower bass range.

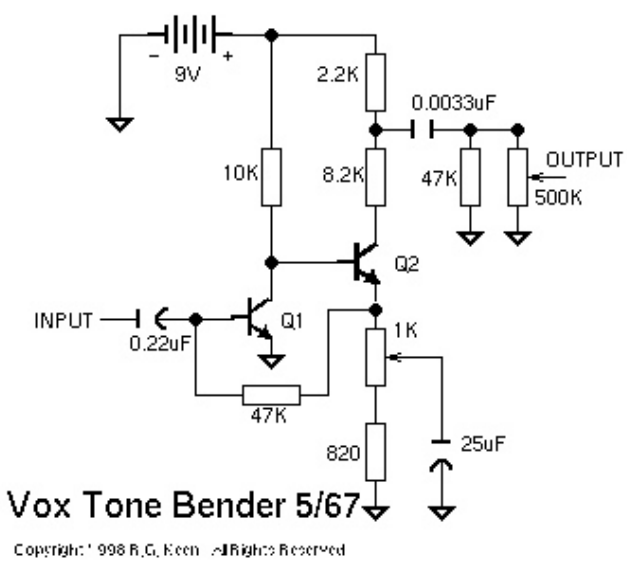

Figure 2: Vox ToneBender schematic

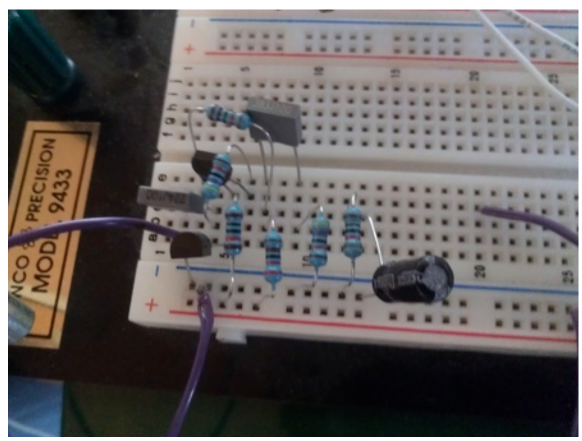

Figure 3: Our breadboard circuit

It quickly became clear that our choice of transistors would be one of the most important design decisions we would encounter. We tried using a variety of models with different levels of gain, and we found that those with particularly high gain caused some undesirable effects - particularly high noise levels and feedback tendency. High gain did help to produce a bright, full sound, but we found that it could reach a point where it created so much noise that the pedal would be practically unusable. It was necessary, then, to draw a line between what we considered to be a good maximum amount of distortion (since the user would always still have the option to reduce it with the built-in potentiometer) - and too much.

In addition to basic noise levels, some of the higher gain transistors caused halfway rectified octave-up effects - "halfway rectified" meaning that the original pitch was still heard in conjunction with the added higher octave. A true, "fully rectified" octave-up effect would completely replace the original note. This can sometimes be a good or interesting quality in guitar pedals, but for our purposes, we wanted to avoid it because it only works properly for single-note inputs, meaning that it would cause chords to lose much of their clarity. This would in turn make our pedal much less versatile, and appeal to a smaller portion of our target audience.

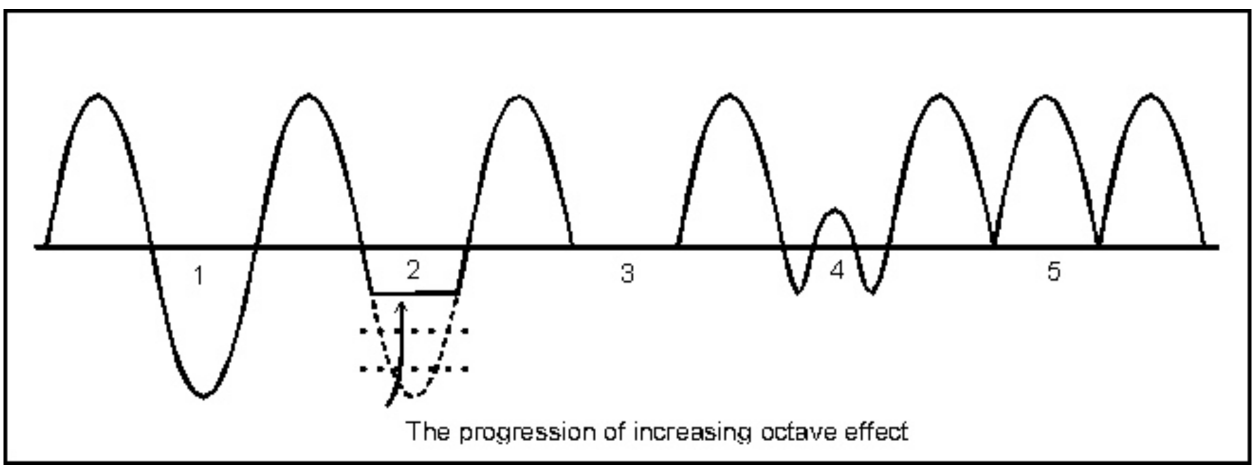

Figure 4: Octave effects - no octave at (1), halfway rectified at (3), fully rectified at (5)

We found that the transistors that we deemed to be best in terms of tone still caused quite a bit of noise in the circuit, so we focused our attention on reducing these noise levels. One way we accomplished this was by inserting a zener diode between the collector and base terminals of the two transistors. This helped to act as a noise gate and eliminate a significant portion of the oscillations that were taking place, particularly during times when the guitar was not being played and the only output was general background noise. Once we discovered this effect we tried using other diode types including normal diodes and a shottkey diode, but found that the zener diode still worked best for our purposes. We also rearranged our breadboard circuit such as to increase the distance between components and signal paths in order to reduce the noisy capacitive effects that are frequently caused by close wire proximity. Once these precautions successfully lowered our pedal's noise levels, it became more apparent that our circuit was also picking up certain radio frequencies that were compounding with its internal noise levels. However, this would hopefully not be an issue with our actual product, since the metal casings used on stompbox pedals are meant to effectively eliminate incoming radio transmissions by acting as a Faraday cage.

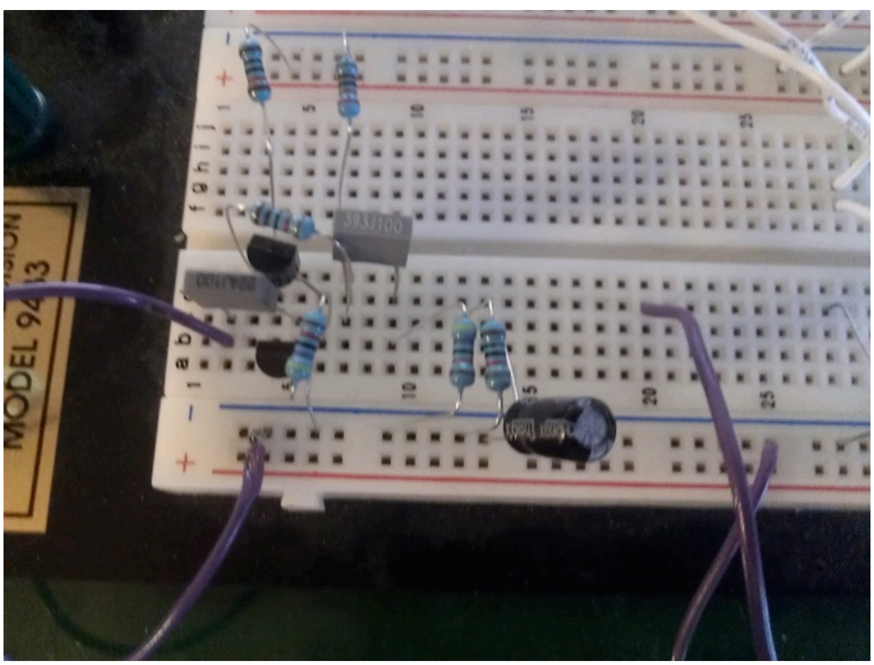

Figure 5: Breadboard circuit with increased spacing

Expanding on Our Design

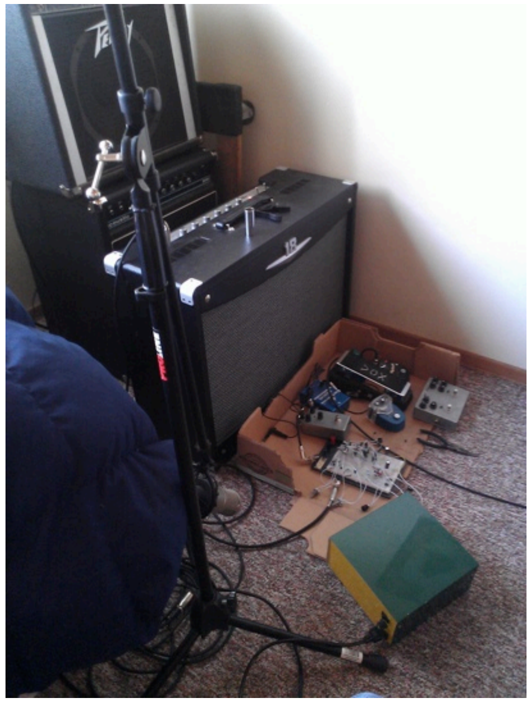

Figure 6: Michael's testing setup

Throughout our experimentation & design process, we had been testing our circuit with Michael's guitar and amp setup. Several other pedals, though not activated, were also included in the chain. The significance of this is that when we ran our guitar signal through our circuit alone, without the pedals, our tone suffered a huge decrease in sound quality and high-end treble response. We quickly realized that this was because the signal was being buffered by another pedal in the chain, and that our pedal relied on this buffering to produce a proper output. Fuzz pedals such as ours or the ToneBender are known to often have an unusually low input impedance, which can lead to undesired "tone-sucking" effects for certain input situations. We were happy with the tone we had achieved when using our pedal through Michael's setup, but it had become clear that we would have to add a buffer circuit at our pedal's input to allow for its independent use. We went about doing this with a simple buffer circuit diagram that we found online, using a JFET transistor. After building it, we found that we would have to use higher-valued resistors than those described in the diagram, because the default sizes allowed

for too high of an output which resulted in extreme levels of oscillating noise and feedback. Changing the resistors fixed the problem, creating an effective buffer that would ensure a solid, consistent tone regardless of the pedal's input setup.

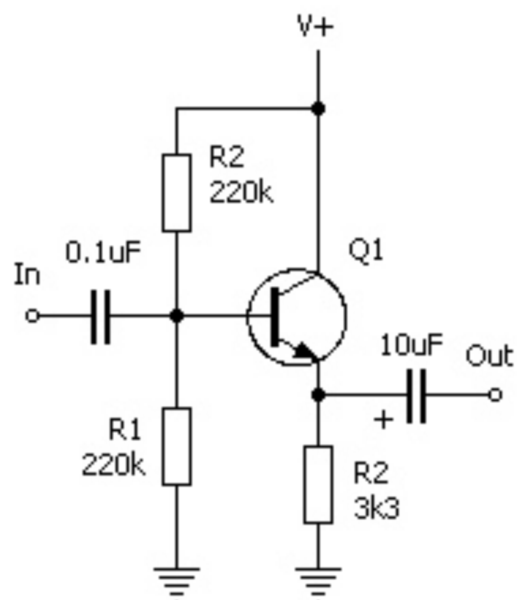

Figure 7: Buffer circuit schematic

During these testing sessions, we talked about the option of including other expanding features. We had the idea building a pedal that would have two footswitches to alternate between two different transistors with different gain ranges - one for a lighter sound with more clarity, and one for a much thicker and heavier distortion. We had produced tones from both of these categories, and we thought that some people might like to have both ends of the spectrum at their disposal with a single hit of a button. In the end though, we decided against this for a variety of reasons. The first was cost - adding a second footswitch would significantly increase the size of the enclosure, which would correspond to a large increase in pricing. Plus, the footswitches themselves were the second most expensive component in our design, so incorporating another one would further increase our prices. In addition to this, having two switches for different gain levels might not even prove to be necessary since our one existing gain control knob already allowed for a wide range of gain levels.

Our Fuzz Sound

At this point, we had succeeded in assembling a working fuzz pedal circuit with a tone that we were very happy with. It had a nice, full tone at both high and low gain levels, and although a certain level of background noise is unavoidable in highly distorted guitar signals, we felt that we had kept our noise level low enough for it to be negligible during playing. Our tone was well-balanced across the spectrum ranges, which is especially important at high levels of gain, because this effect can sometimes amplify the harmonic content unevenly and end up boosting too much of the high end at the cost of the lower bass range, resulting in a harsh or scratchy tone. This can be mostly attributed to our choice of capacitor sizes at the input, which have a large part in determining the circuit's level of bass response. It was also important to us that the low gain settings were mild enough to allow the subtleties of large, multi-phonic chord voicings to shine through without losing clarity.

In addition to these tonal balances, we were also pleased to find that our pedal featured very good touch response, meaning that the tone and distortion levels reacted well to how hard or loud the guitar was being played at any given time. This quality is commonly considered to be a defining characteristic of good fuzz pedals, and is something that separates fuzz from standard distortion effects. Picking a note more softly while playing through our pedal didn't just make it come out quieter, it caused a noticeable change in the output tone, which allowed us to utilize a range of distortion sounds without actually having to tweak any of the pedal parameters. A guitarist using our fuzz would be able to strum relatively quiet chords or melodies without creating any rough distortion, but would still have the ability to produce a thick, heavy sound by picking harder during appropriate times in the music. In addition to this, the pedal's tone and distortion level could be altered by using the volume knobs on the guitar itself, giving the player even more freedom in easily adapting to any desired sound in a live setting.

“The BoneTender”

Now that we had designed the bulk of our pedal, it was time to give it a name. Since we had based it on the Vox Tonebender, we decided to reference this fact by switching around a few letters and using a play on words to call our pedal the “BoneTender”. A friend of ours from the Audio Engineering Society, Ian Fetters, volunteered to provide an appropriately bone-themed graphic to print on the front of our pedals by using a template provided by pedalpartsplus.com - the site that we would use to manufacture our enclosures.

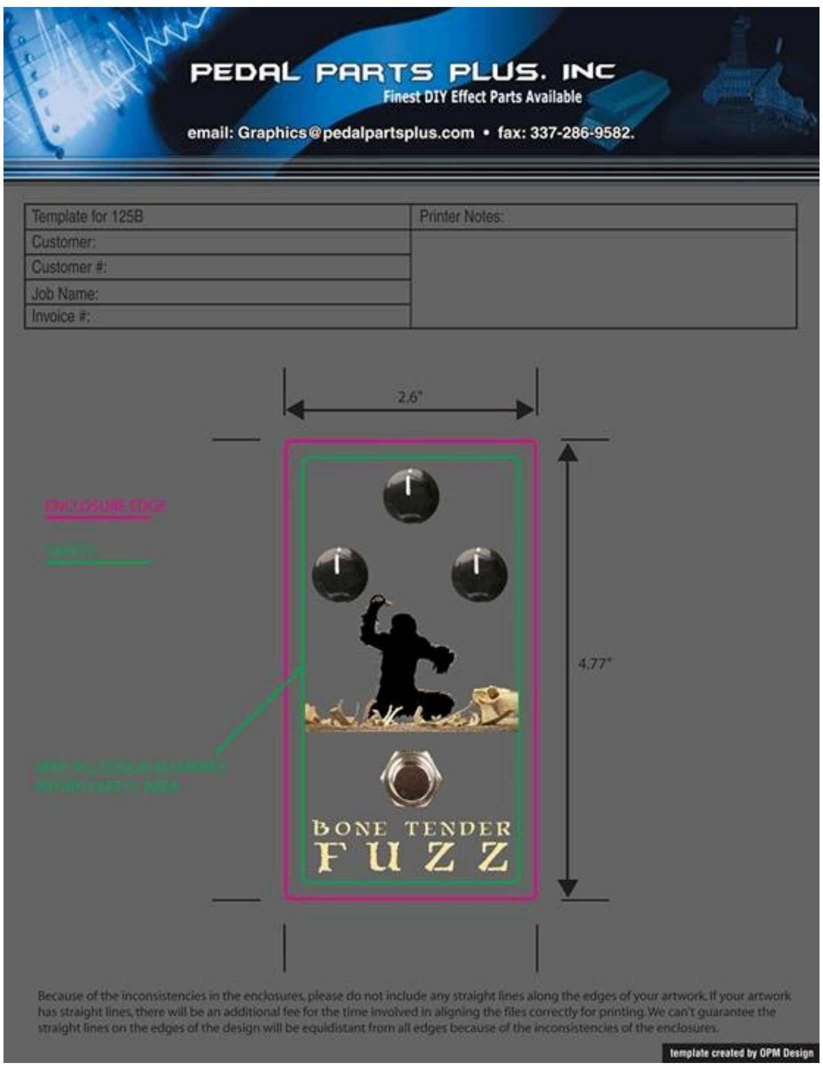

Figure 8: BoneTender pedal graphic

AES Design Meeting #1

With our plans properly outlined, we were ready to share and get some input from other AES members at an informal design meeting. At this meeting, we planned to learn about our potential customers' preferences regarding topics such as price range, desired features, enclosure type, and tonal characteristics. We brought our up-to-date breadboard circuit as well as a guitar and amp in order to let people play and hear it for themselves. We had a good turnout of 15-20 AES members, and received a very positive response overall to our prototype pedal circuit. It specifically received praise for its high quality tone, low noise level and good touch response, verifying my and Michael's impressions on the subject. We spent a few hours at the meeting, and everyone who wanted to play through the setup got a chance to do so.

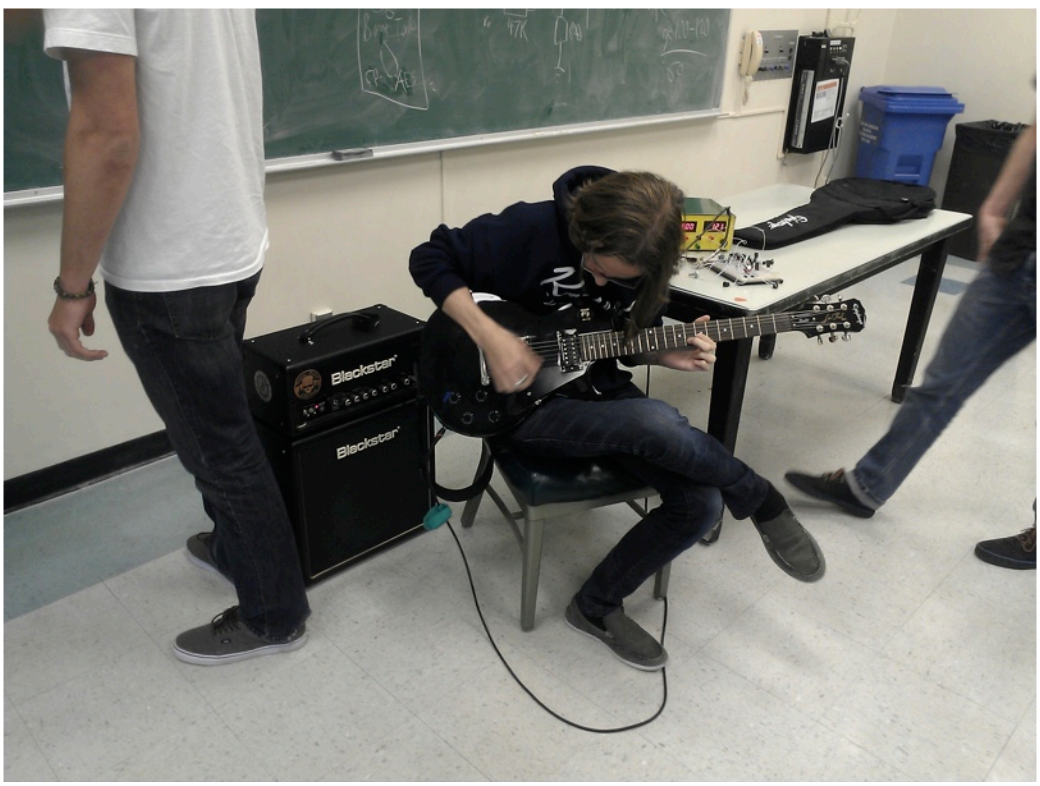

Figure 9: AES members at the first design meeting

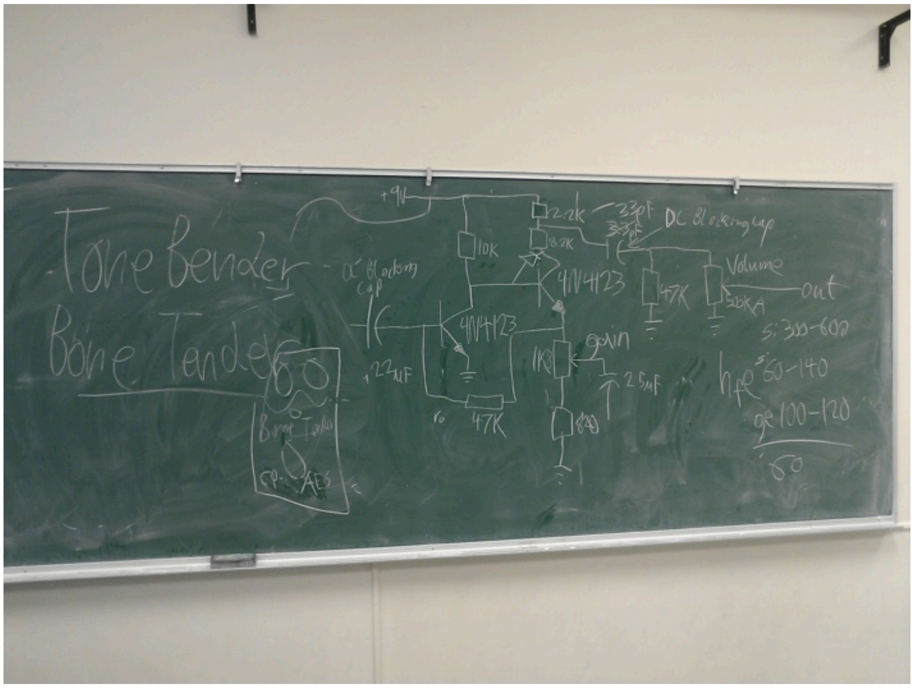

Figure 10: BoneTender chalkboard schematic

By the time of this meeting we had estimated that we could sell our fuzz pedal kits for around $50, which seemed to be an acceptable price range to most of our potential customers. Any additional features would significantly increase this cost, as well as the size and complexity of our circuit, and we found that the vast majority of those present at the meeting would prefer to have a straightforward pedal that stayed in this low price range than pay extra for something with additional functionality - especially since no one had anything in mind in particular that they would like to see added. Someone suggested that we could use their connections with the Mechanical Engineering department to create 3-D printed enclosures from the fabrication labs, which could potentially cost much less than ordering heavy duty metal enclosures from a third-party manufacturing site. After having a discussion on the topic, though, we decided against doing so, since the metal enclosures would play an important role in shielding the pedal from external radio noise, and creating our own enclosures would force us to spend time and effort drilling all of the necessary holes, which would increase the difficulty of building the kits.

We also shared our proposed name, “The BoneTender”, and had it approved by the group. After posting an image of Ian’s graphic design to the AES facebook group, however, someone pointed out that the name “BoneTender” was already taken by several other pedals. We were surprised by this, since we thought the name was unusual enough that we hadn’t even thought to check if it was already taken. Since we already had bone-themed artwork that we liked, Ian suggested changing the name to “The Skeletone”, which we all agreed was a suitable alternative. Once he adjusted the artwork to account for the new name, our frontal enclosure design was complete.

Printed Circuit Board (PCB) Design

The next step at this point was to transfer our circuit schematic onto the actual layout that would be printed on our physical boards. I put myself in charge of this task, and since I had no previous experience with doing printed circuit board (PCB) design, I set off on educating myself on the process. There are many different software options on the market made for PCB design, and we decided to use a CAD program called "Eagle" for our project, due to its relative simplicity and ease of access. Eagle offers a full version with extensive features for about $800 as well as a "light" version free of charge, and we determined that the light version would have everything we needed for our relatively simple PCB requirements. Some more complex boards require many dense layers of circuitry, but ours would only need printing on the front and back of one flat board. Michael's initial size estimate was for us to use a 3-inch by 4-inch board, which was more than enough room for our circuit.

At this point in the design process, we had reached the end of Fall quarter. Michael and I met up to input our circuit schematic into Eagle, leaving me with the intention of finishing the PCB design by the end of Winter break. I first taught myself the basics of Eagle with the help of several online forums and tutorial videos. I also consulted with my neighbor Daniel Firu, a professional Electrical Engineer with experience on the topic of printed circuit boards. He verified that the most important aspects to keep in mind during the design would be avoiding unnecessarily close wire proximity that would create parasitic capacitance between components, and generally keeping the signal, ground and power paths as straight and separate as possible within our size constraints. My initial PCB design is shown below (Figure 11), with the power terminals along the top, and the ground wire along the bottom and right sides of the board.

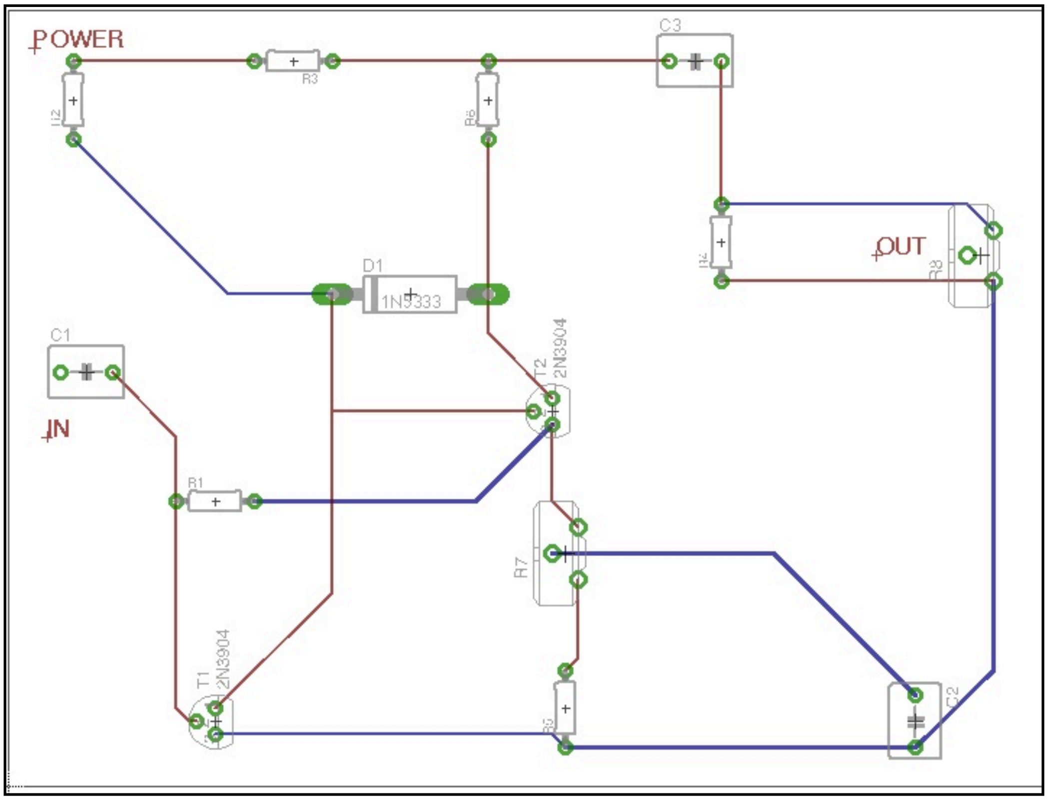

Figure 11: BoneTender PCB v1.0

I tried to keep any parallel-running wires as far away from each other as possible, and to only allow wires to cross in situations where they were running perpendicular on opposite sides of the board. Red wires in Eagle are placed on the front side of the board, and blue wires on the back. The 4”x3” board seemed extremely spacious, and I was already suspicious at this point that we might want to reduce the size of our board. This was confirmed when Michael contacted me suggesting that we reduce our board size to 2” x 2”, since this would allow it to fit within a smaller, cheaper and more efficient size of enclosure than we had previously planned on. At this time, we also realized that we had made a huge oversight in forgetting to include the buffer circuit in our board. So, then, my new task would be to fit my PCB design into roughly half of the allotted space while adding several more components to the design. This proved to be much more challenging than the first design process, but by this point I had already become somewhat familiar with Eagle and I felt ready to take on the task.

PCB Design (Part 2)

My second design ended up looking completely different from the first, and no longer resembled our schematic at all in terms of physical layout. It also forced me to use the space constraints much more efficiently, and to be more careful with avoiding interference between nearby wires. I still maintained the general placement of the power, signal and ground paths though, with power along the top, signal through the middle, and ground along the bottom edge of the board. I sent it off to a few other friends from AES with experience on the subject, who approved of my design and reassured me that I had left enough space between components.

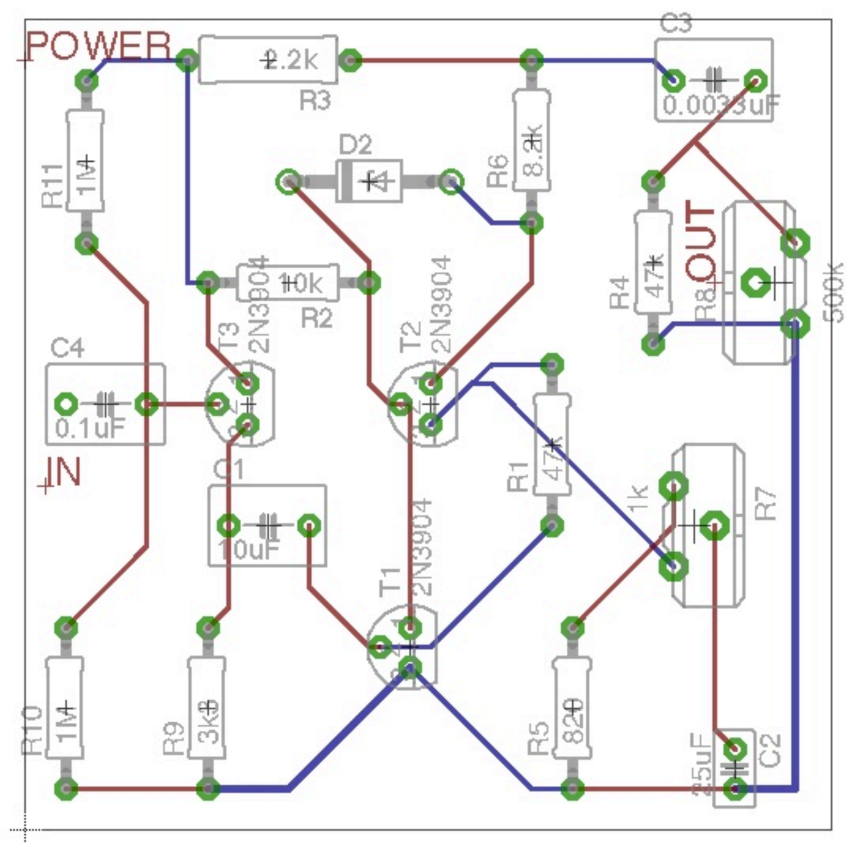

Figure 12: BoneTender PCB v2.0

However, we still had a few features to include in the board layout. Specifically, we would need to add connections for the pedal's LED, designate locations for two mounting peg holes, and add the through-hole vias that would be reserved for wiring to

the input and output jacks. This required me to adjust the placement of some components and change some of the wire routing from opposite sides of the board. Once I worked these features into our board, we finally felt that we had a good prototype board and were ready to place our first small-run order.

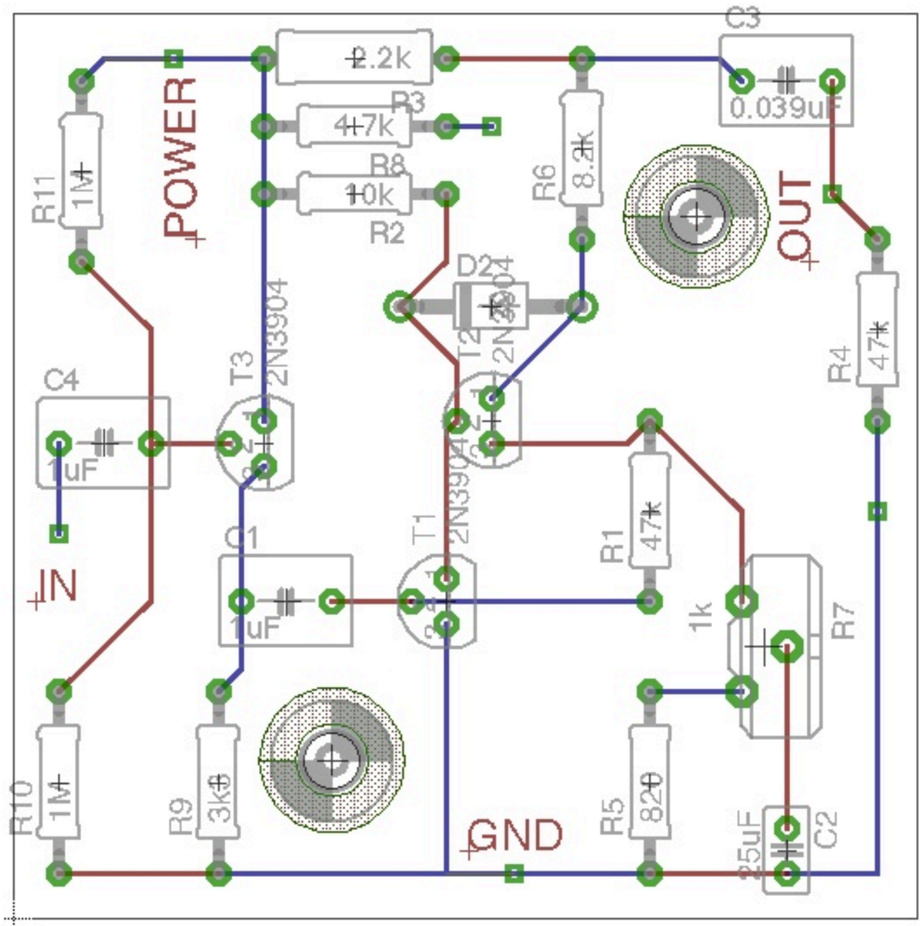

Figure 13: BoneTender PCB v2.1

Ordering Our Circuit Boards

After looking into several PCB-printing companies online, we settled on Oshpark.com due to their reasonable prices, good reviews and ease of use. They simply have you upload your Eagle file, and then manufacture your boards and send them with free shipping. The smallest run of boards they offered was 3, for $20, so we decided to purchase 3 boards and build prototype pedals for Michael, Ian and myself. Michael proceeded to order a set of 3 enclosures, and 3 sets of all required parts. The enclosures could be ordered from pedalpartsplus.com with our design printed on the front, and all drilling done in-house. Many different options were available in terms of color and finish on the metal, so we decided to order 3 different choices for our prototype run that we felt might look good with our design - namely “black hammertone”, “ghost black”, and a plain bare metal finish.

Figure 14: “Black Hammertone”

Figure 15: “Ghost Black”

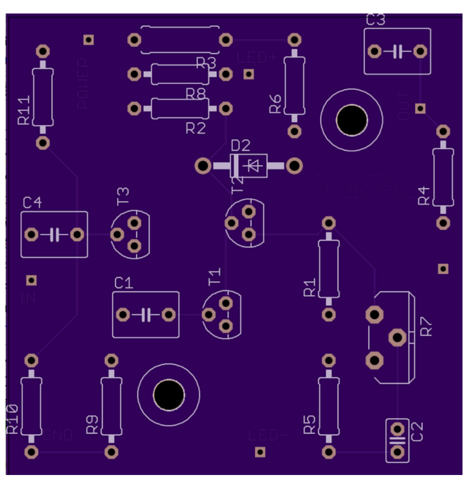

Figure 16: Oshpark board preview

Build Day #1

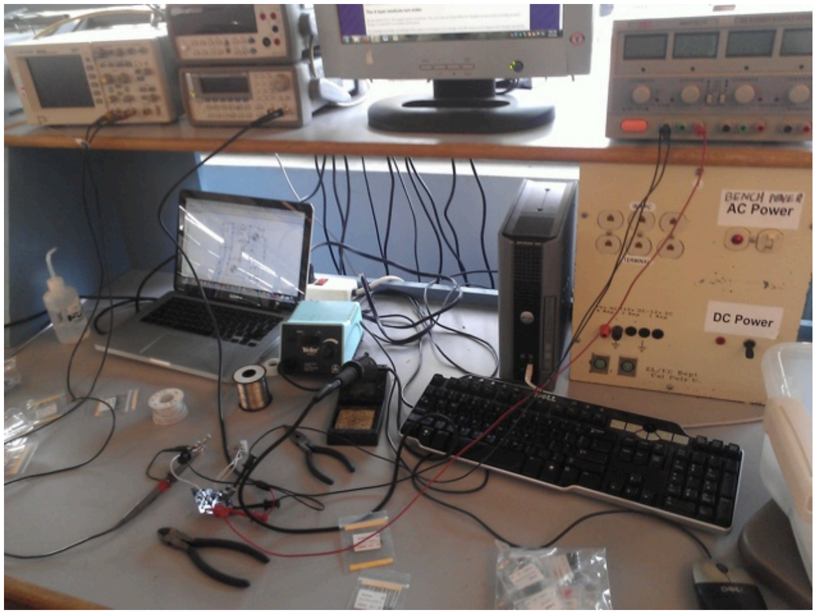

Figure 17: The IEEE lounge

Once our boards arrived, we were ready to start assembling our first pedal. At this time we were still waiting for our enclosures, but we wanted to get a head start on soldering the various components into board. I had never done any soldering before so this also served as a tutorial for me. We met in the IEEE lounge in building 20, since they provided access to all of the equipment we would need. With Michael's help, I was able to get the hang of the process enough to attach many of the necessary components to our board. We included removable sockets for a few of the components that we were still interested in testing and possibly switching out. This included the input capacitors, the LED's resistor, the zener diode, and the buffer transistor.

During this process, we discovered some flaws in our circuit board design. The via holes were a bit smaller than the rest of the component holes on the board, making it very difficult to fit in the necessary wires. We managed to make it work, but the sizing was still less than ideal. Also, we found that the component names were printed on the board (R1, C2, etc), but not their respective values. We were able to refer to my Eagle PCB blueprints during assembly but we wouldn't want our future customers to have to go through this trouble. In addition to this, our mounting peg holes were too small for

the pegs themselves. We were able to file them out to a larger size without much trouble, but again, we would want the assembly of our kits to be as straightforward and painless as possible. Finally, we decided to increase the copper pad sizes around each hole on our next run of boards in order to make soldering a bit easier.

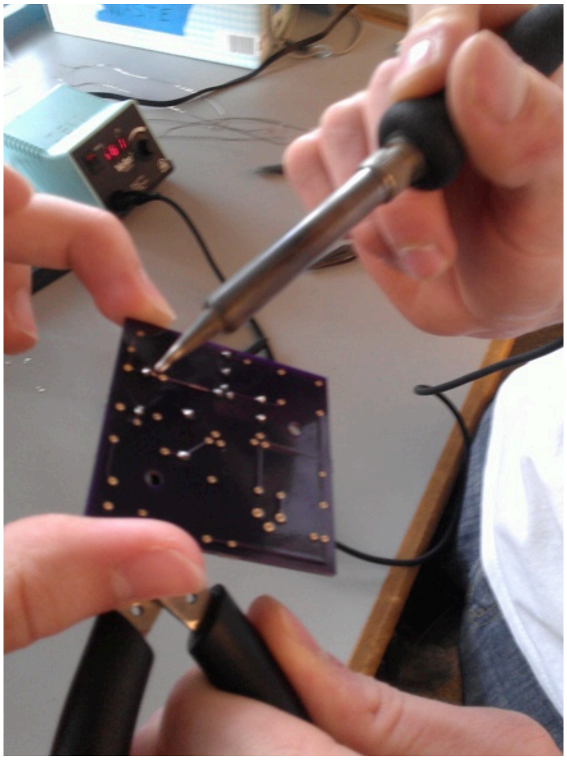

Figure 18: Soldering the board

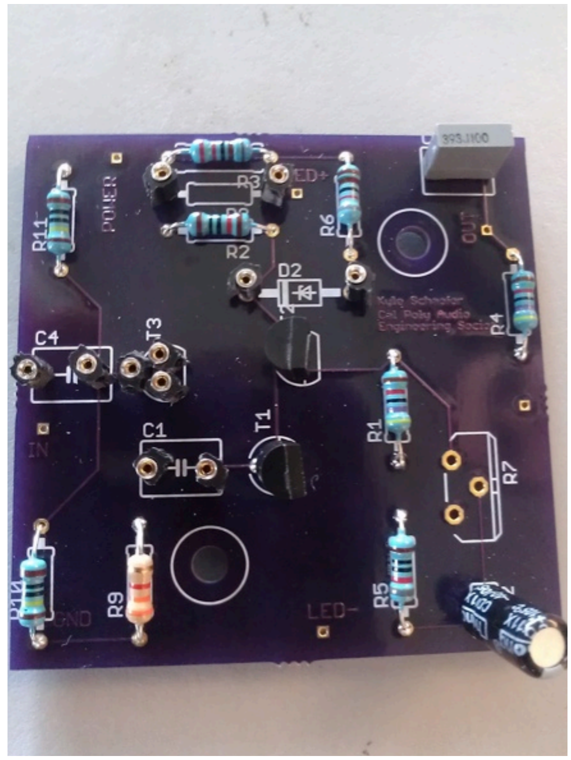

Figure 19: Board with components and sockets

At the end of this first build day, we hooked our circuit up to the IEEE lounge's oscilloscope in order to get some graphical data regarding the Skeletone's frequency response. We tested four different frequencies (80, 500, 1.6k and 4k Hz) at three different input voltages each (10, 30 and 80 mV). These oscilloscope images are included in Appendix A. We generally observed more clipping at the lower frequencies, and more symmetrical distortion at the higher frequencies. These images also show that the clipping levels at any one frequency increase with increasing input voltage. This serves as a demonstration our pedal's touch response, since the higher input voltages would correspond with playing more loudly on the guitar.

Build Day #2

Our first order of enclosures arrived about a week after the first build day, allowing us to finally complete the first of our Skeletone fuzz pedals. We finished soldering the components into the board, placed it in on the mounting pegs, and wired in the rest of the parts. This included the input, output & DC power jacks, the gain & volume control potentiometers, the LED, and the footswitch. Assembly went smoothly for the most part, except for when we accidentally soldered in a capacitor backwards, causing it to explode. We knew that the polarity on this one particular capacitor was important but switching it was clearly still an easy mistake to make, so we replaced it and decided to notate the correct polarity on future runs of the board. We also discovered that the noise-reducing zener diode was no longer necessary now that our circuit was properly shielded within the enclosure. We decided to leave its designated holes on our boards though, since the diode's absence wouldn't affect any signal paths and doing so would leave people the option of including it if they wished to modify their pedal with any higher gain transistors.

Figure 20: Inside the Skeletone

AES Pedal Showcase Meeting

Once our first prototype pedal was up and running, we were able to pass it around at the next general AES meeting for members to view. We also started taking orders and deposits at this time. We hoped that having an actual product to show would help to persuade more members to get interested in purchasing a kit. We also took a vote on which enclosure color/finish everyone preferred, between the rough & textured “black hammertone” and the smoother, shinier “ghost black”. The group was much more in favor of the ghost black, and since this was also preferred by both Michael and I, the decision was officially made. Apart from the showcase at the meeting, we advertised our pedal kits through the AES mailing list and Facebook page and set up a Google form for people to sign up and place their orders. I also recorded, produced and posted a short 1-minute demo song using the Skeletone to show off its sound to our potential customers. This recording can be found at: https://soundcloud.com/archaeologistmetal/skeletone-fuzz-pedal-demo.

Figure 21: The Skeletone

The Future

As of now, the Skeletone fuzz pedal has been successfully designed and built, but there are several ways in which this project will continue in the future. Michael will continue to take and fulfill orders of our kit throughout next quarter, after I've graduated. He plans on putting together a thorough set of instructions to include with each of these kits. He will also be hosting a "build workshop" early next quarter, in which anyone who has ordered a kit can meet up to get help with building and soldering their pedals. We plan on ordering enough parts for about 40 kits, in the hopes that we will be able have some leftover for the Audio Engineering Society to sell to future members.

Appendix A: Frequency & Input Response

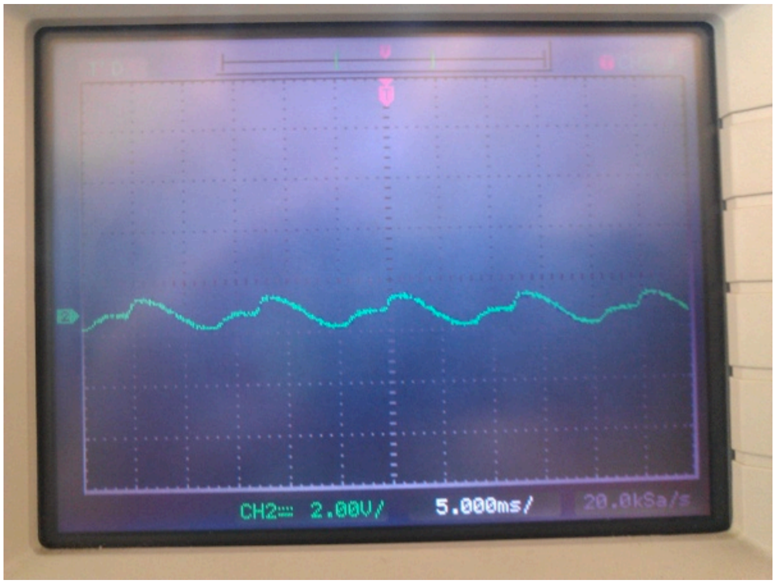

Figure A1: 80 Hz & 10mV

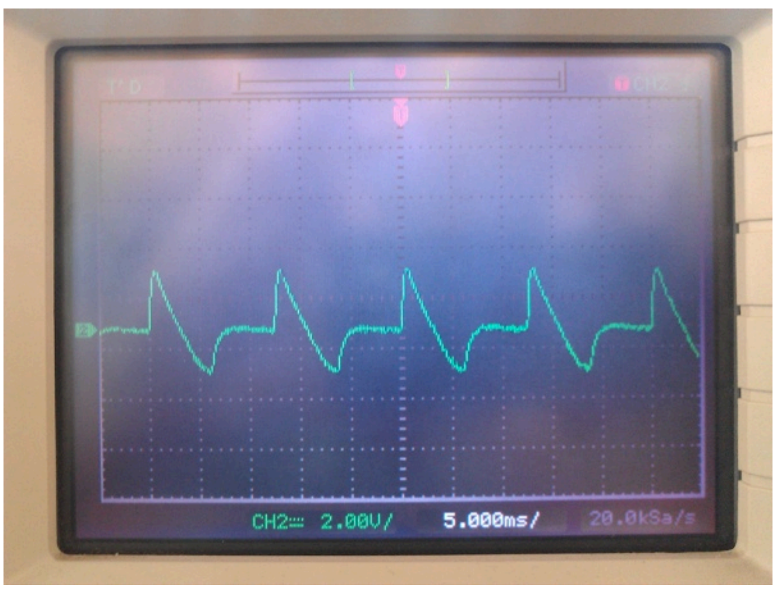

Figure A2: 80 Hz & 30mV

Figure A3: 80 Hz & 80mV

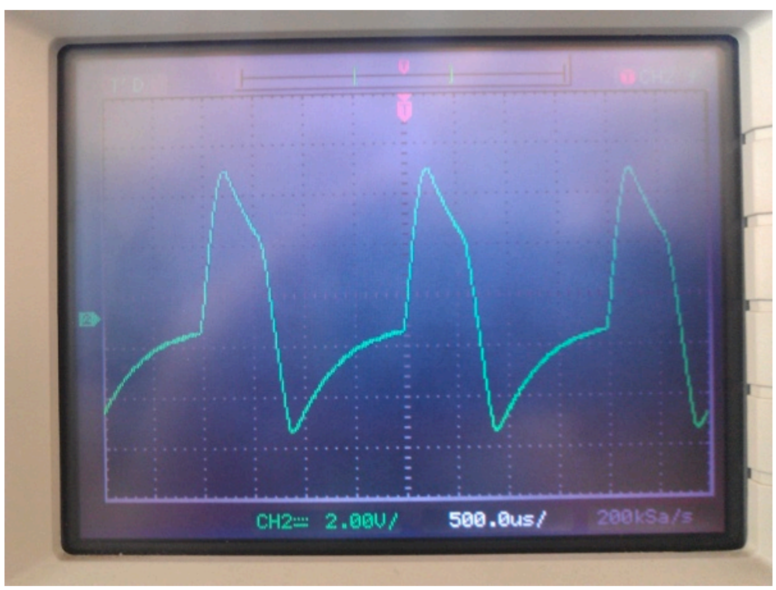

Figure A4: 500 Hz & 10mV

Figure A5: 500 Hz & 30mV

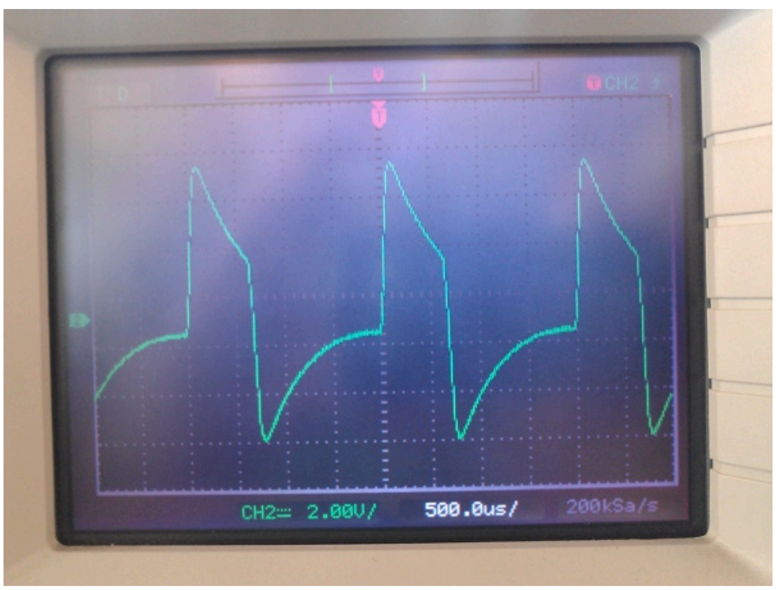

Figure A6: 500 Hz & 80mV

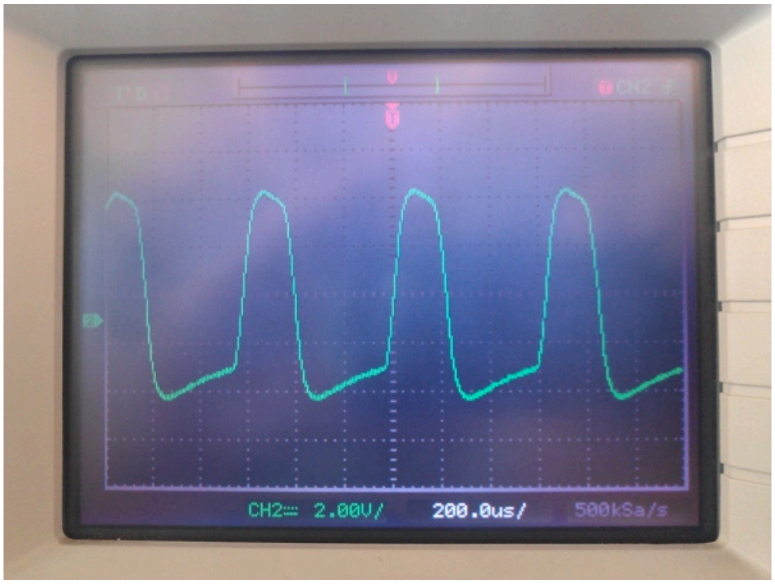

Figure A7: 1600 Hz & 10mV

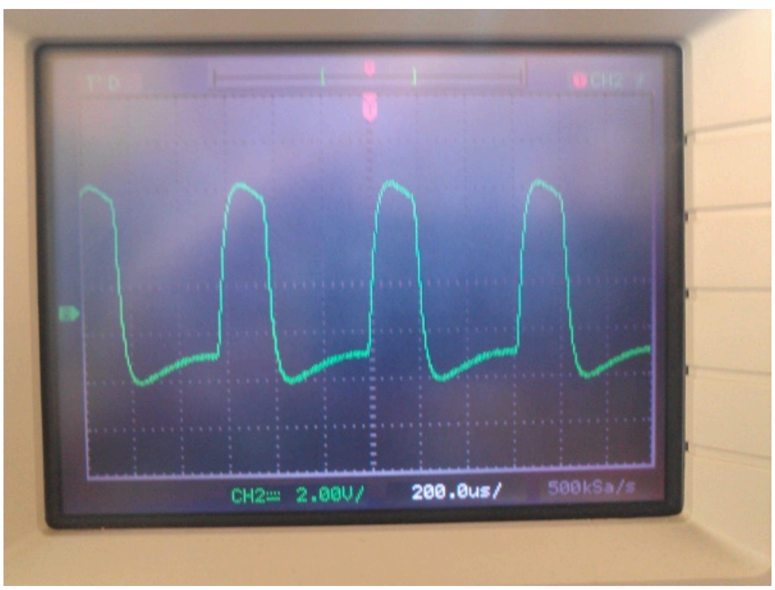

Figure A8: 1600 Hz & 30mV

Figure A9: 1600 Hz & 80mV

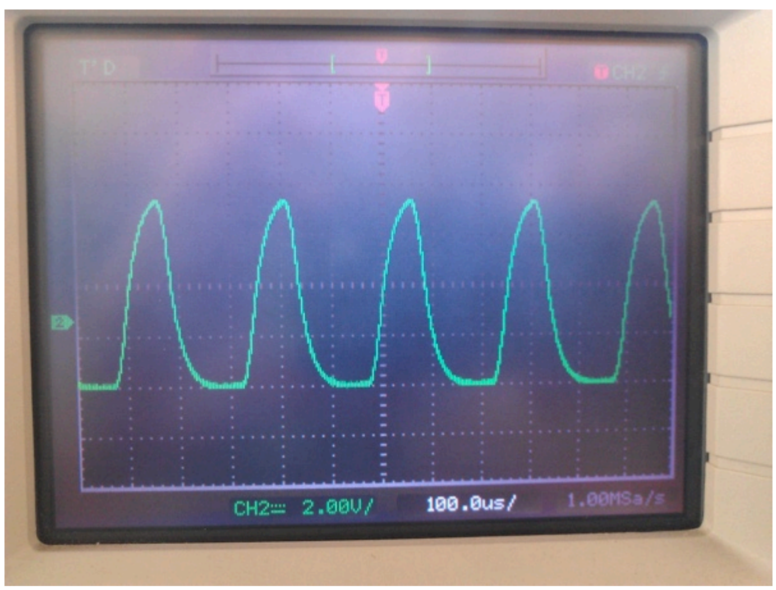

Figure A10: 4000 Hz & 10mV

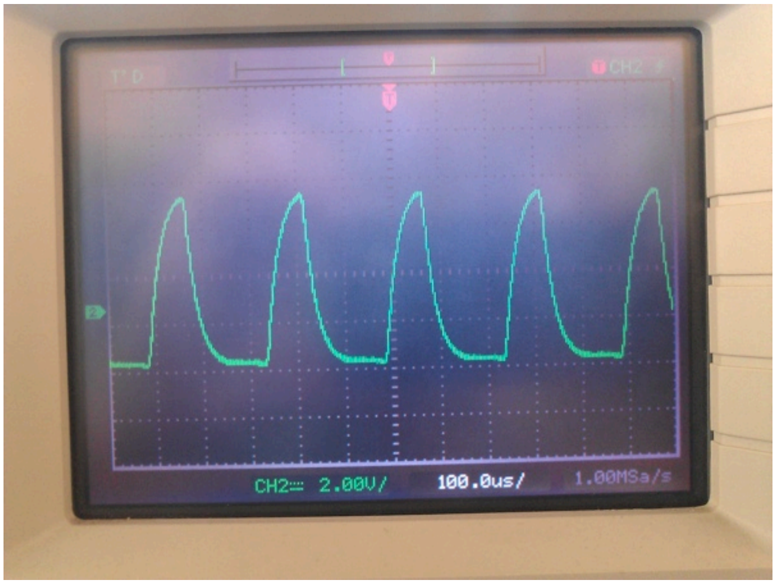

Figure A11: 4000 Hz & 30mV

Figure A12: 4000 Hz & 80mV

Appendix B: References

http://www.geofex.com/article_folders/fuzzface/fffram.htm

http://www2.gibson.com/News-Lifestyle/Features/en-us/effects-explained-overdrive-di.aspx

https://robertkeeley.com/2013/09/germanium-transistors/

http://www.engineersgarage.com/tutorials/diodes?page=2

http://www.wisegeek.org/what-is-a-faraday-cage.htm

http://www.muzique.com/lab/buffers.htm

http://screaminfx.com/tech/why-and-when-to-use-a-guitar-buffer-pedal.htm

http://www.geofex.com/effxfaq/distn101.htm

https://soundcloud.com/archaeologistmetal/skeletone-fuzz-pedal-demo