Building an Overdrive Guitar Pedal-

Replicating the Boss SD-1

Dustin Matthews

10/01/2023

1. Introduction

A guitar pedal is a type of consumer electronic device used by musicians to apply some sort of effect to their instrument’s signal, usually before the signal is passed through its final amplification stages. There are numerous types of effects, including, but by no means limited to, overdrive, distortion, reverb, delay, and chorus. Building guitar pedals is an introduction for many to begin exploring electronics and circuit design, and for some, it even opens the door to a career in electronics. It is popular among pedal builders, especially starting out, to search out existing circuit designs for commercially available guitar pedals and replicate them directly, or to make modifications to alter the circuit’s response to better suit their tastes. As a guitarist and electrical engineering student myself, this project provided an opportunity to gain more experience with tools such as LTSpice and Altium, as well as to end up with a device that can be utilized in my artistic pursuits. For this first guitar pedal build, I decided to replicate a Boss SD-1 overdrive pedal.

2. Objectives

The objectives of this project are to successfully replicate a modified Boss SD-1 overdrive guitar pedal and create a PCB using Altium Designer, as well as to use circuit analysis techniques to gain a better understanding of the circuit.

3. Preparation

3.1 Initial Research

Many guitar pedal circuits are readily available online, and this was the case with the Boss SD-1 circuit, which is available from hobby-hour.com/electronics. Initial research for this project revealed that Boss circuits utilize a special bypass circuit using transistors, which is essentially a flip-flop (Figure 1). The intention of this bypass circuit is to reduce the audible ‘pop’ that occurs when switching the circuit while connected to an amplifier.

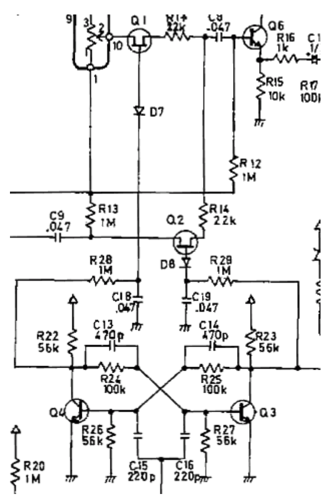


Figure 1 Boss Bypass Circuit

Rather than add these additional components, many builders opt for a ‘true bypass’ circuit, which is accomplished using a three-pole double-throw switch. The decision to pursue a true bypass build was made early on in this project, with the intent to make the circuit easier to troubleshoot in the event of issues, as well as to make the build process faster.

3.2 Schematic Capture

Having decided to opt for a true bypass version of this circuit, the schematic was recreated in LTSpice. The schematic found in the documentation of the PedalPCB Uberdrive (pedalpcb.com) was used as a reference during much of this project, which is itself a true bypass clone of the SD-1. The resulting Spice schematic is seen below in Figure 2.

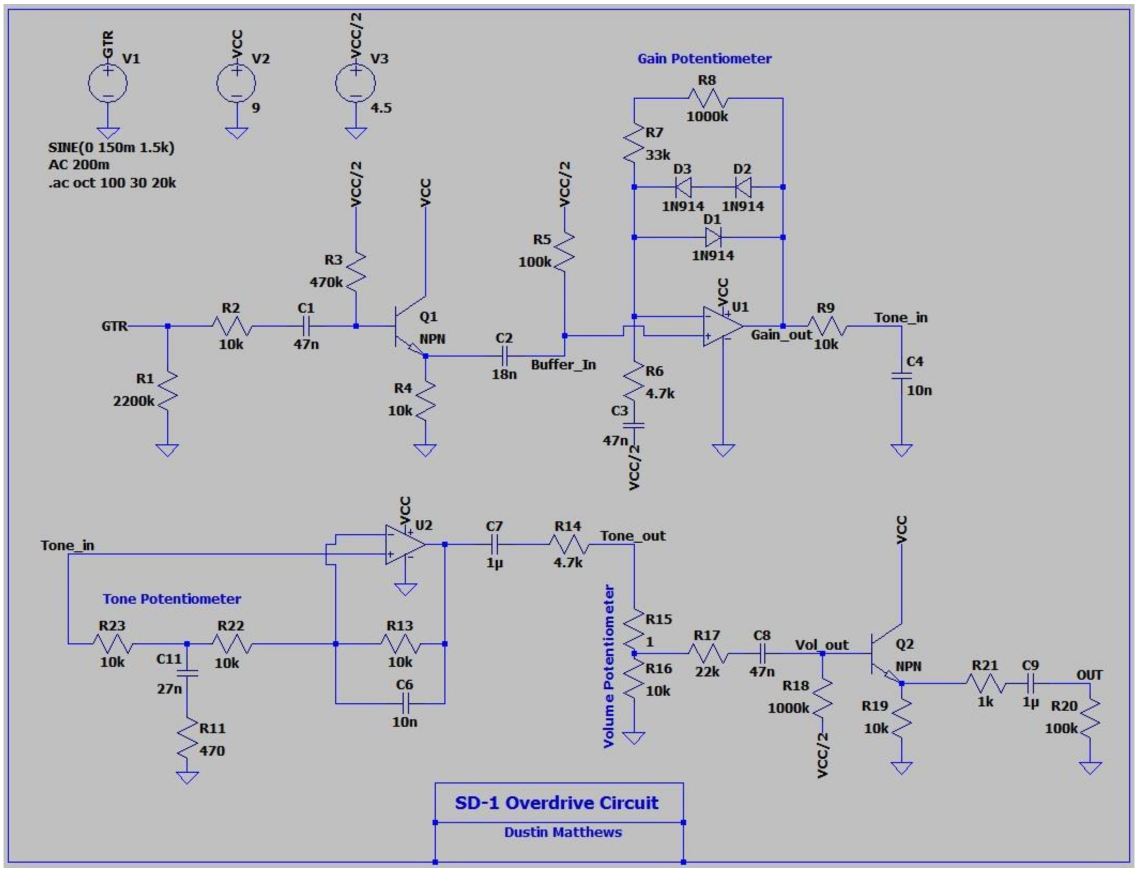


Figure 2 LTSpice Schematic

4. Analysis

4.1 Buffering

Many guitar pedals utilize BJTs as buffers at the input and output, which help to maintain the signal strength as it passes through one or more guitar pedals on its way to the amplifier. We can see in a transient analysis, for example, how the current is amplified by Q1:

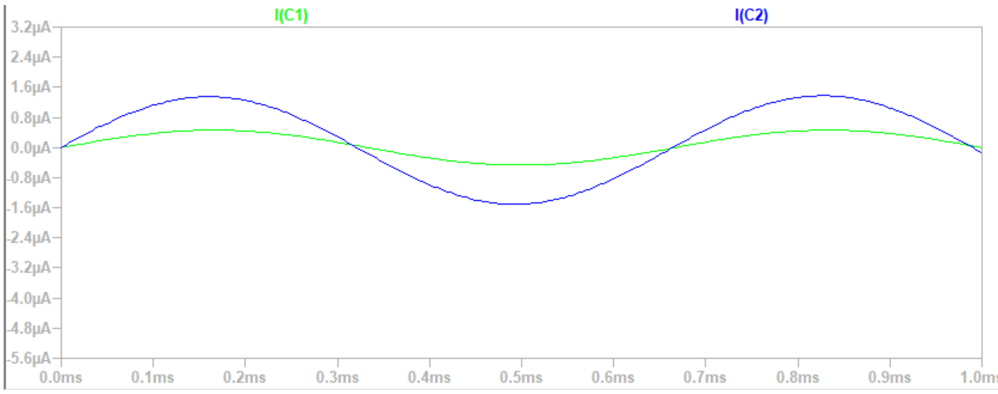


Figure 3 Q1 Current In = C1, Q1 Current Out = C2

The current is also similarly amplified on the output. The input buffering section of the circuit also serves to give the input signal a DC offset of 4.5V, which is needed to take advantage of the whole 0-9V range at the rails of the op-amps used in this circuit.

4.2 Overdrive

The next main stage of the circuit is the gain stage, achieved by the U1 op-amp circuit. A $1M\Omega$ potentiometer in the feedback loop controls the amount of gain applied to the signal. The feedback loop also contains three diodes arranged for what is called 'asymmetrical clipping', which is part of the characteristic 'overdriven' sound and is different from full-on distortion. An example of this clipping is shown in Figure 4, where the potentiometer is set for maximum gain on a sine wave. It is apparent that the clipping is not entirely symmetrical.

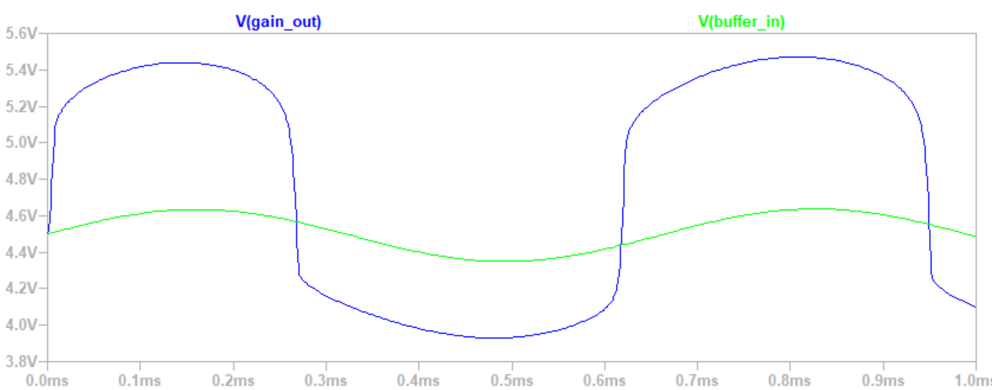


Figure 4 Asymmetrical Clipping

4.3 Tone Control

The tone control of the SD-1 consists of a passive low-pass filter, followed by an adjustable op-amp circuit, followed by a high-pass filtering section. The active filter section will be analyzed first, looking at the response when the potentiometer is in the middle position, the maximum position, and the minimum position.

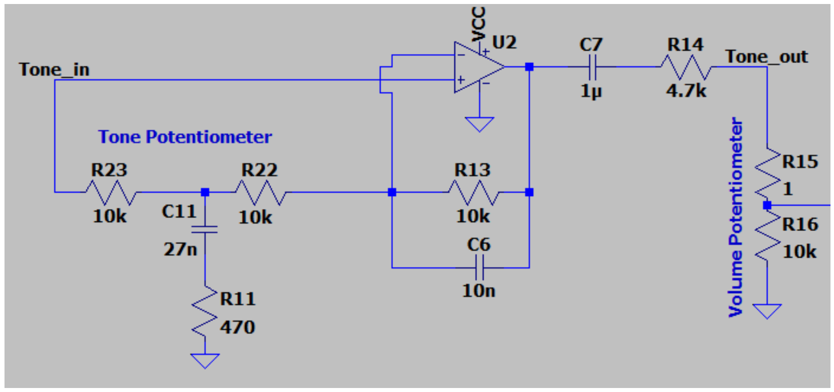


Figure 5 Tone Control Section

At a glance, however, we can surmise that the overall tone control section forms a bandpass filter with an adjustable response in the passband.

4.3.1 Potentiometer Middle Position

When the potentiometer is in the middle position, we have the circuit in Figure 6. To keep the calculations tidy, the two complex impedance combinations are referred to as Z_p and Z_f. Noting that the potentiometer resistances consisting of R23 and R22 are equal in this configuration, they will be referred to as R_p.

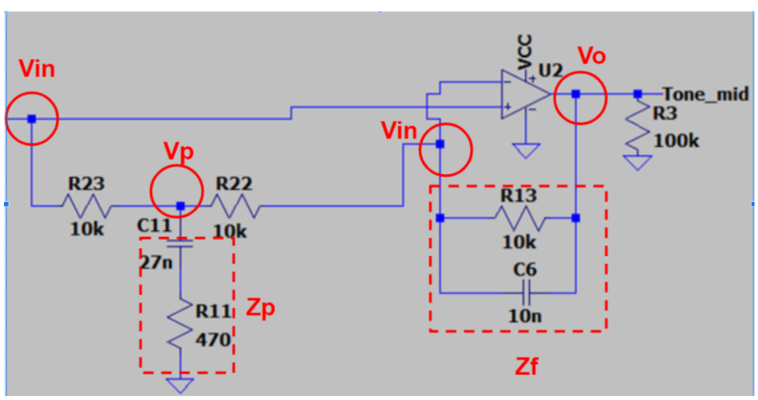


Figure 6 Active Tone Circuit, Middle Position

Nodal analysis can be used to derive the frequency response of this circuit. Using the notation established in the figure above, the equations can be set up as follows:

$$\frac{v_p - v_{in}}{R_p} + \frac{v_p}{Z_p} + \frac{v_p - v_{in}}{R_p} = 0 \qquad (1)$$

$$\frac{v_{in} - v_p}{R_p} + \frac{v_{in} - v_o}{Z_f} = 0 \qquad (2)$$

Equation (1) is rearranged to solve for v_p:

$$v_p = v_{in} \frac{2Z_p}{2Z_p + R_p} \tag{3}$$

This term is then substituted into (2), which, after rearranging, yields the transfer function:

$$\frac{v_o}{v_{in}} = Z_f \left(\frac{1}{R_p} + \frac{1}{Z_f} - \frac{2Z_p}{R_p^2 + 2Z_p R_p} \right) \tag{4}$$

This transfer function was plotted in Octave, a tool similar to MATLAB. The Octave script was used to approximate the properties of the filter, which are given in Table 1. We can observe from the plot that the filter has a peak just above 1kHz, which is considered the 'midrange' of the guitar's frequencies.

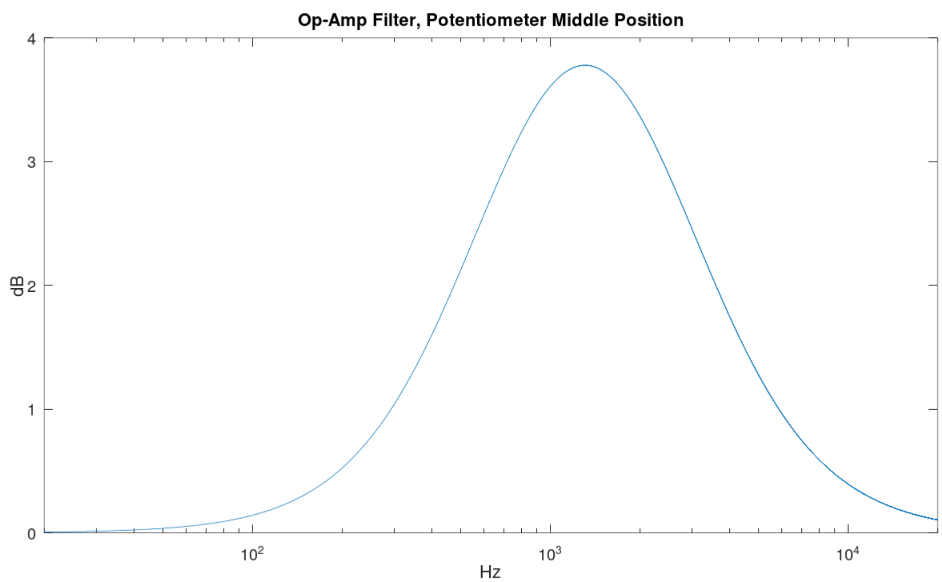


Figure 7 Octave Simulation of Filter in Middle Position

To verify the validity of these calculations, the circuit was simulated in LTSpice, resulting in the following plot. At a glance, the LTSpice AC Analysis appears to match the calculated frequency response very well.

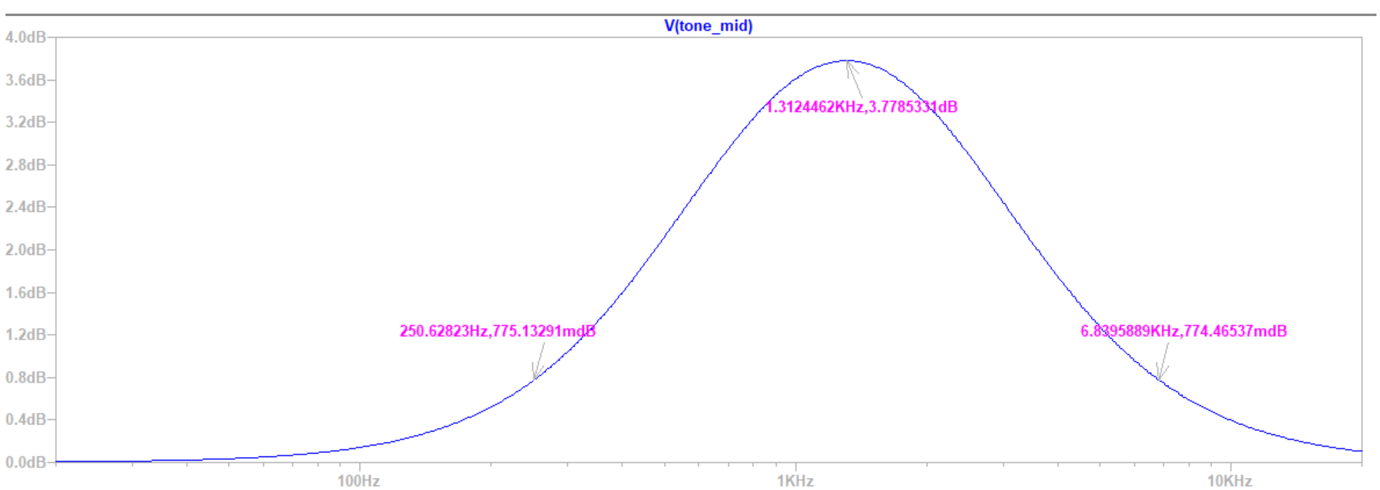


Figure 8 LTSpice Simulation: Potentiometer Middle

4.3.2 Potentiometer Max Position

Following the same approach as above, we can analyze this section of the circuit with the potentiometer adjusted to its maximum position, as shown in Figure 9. In this case, resistor R_p has a value of $20k\Omega$, representing the entire impedance of the potentiometer.

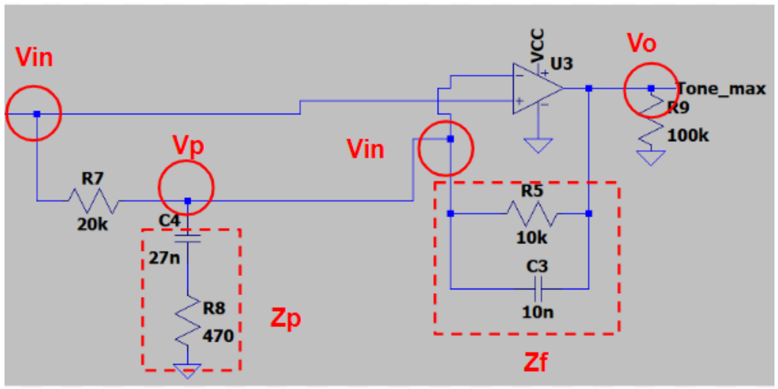


Figure 9 Active Tone Circuit, Max Position

We can see that in this configuration, the voltage at the output of the potentiometer is equal to the input voltage at the non-inverting input of the op-amp. This allows the simplification to be made that $v_p = v_{in}$. Incorporating this simplification, the calculations are performed as follows:

$$\frac{v_{in}}{Z_p} + \frac{v_{in} - v_o}{Z_f} = 0 \tag{5}$$

$$v_{in}\left(\frac{1}{Z_p} + \frac{1}{Z_f}\right) = \frac{v_o}{Z_f} \tag{6}$$

Which is rearranged to give the transfer function:

$$1 + \frac{Z_f}{Z_p} = \frac{v_o}{v_{in}} \tag{7}$$

This equation was plotted in Octave, yielding the following plot:

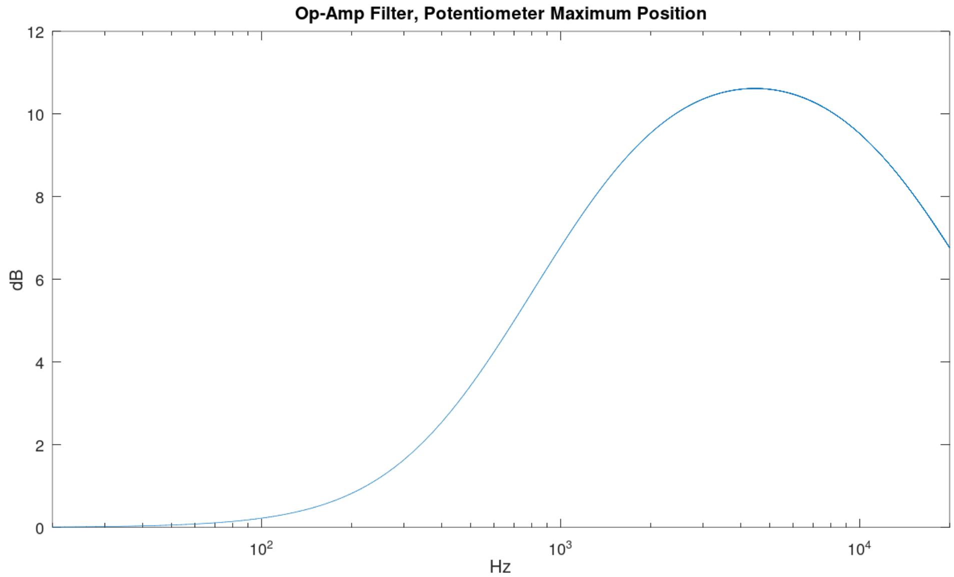


Figure 10 Octave Simulation of Filter in Max Position

The filter properties derived from this mathematical simulation are given in Table 1. Again, LTSpice was used to check the validity of these calculations, as seen below:

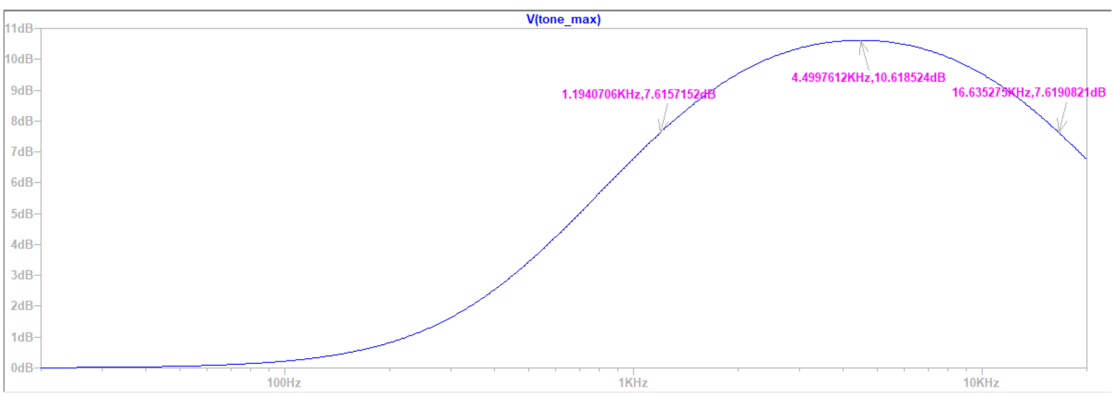


Figure 11 LTSpice Simulation: Potentiometer Max

The Spice simulation appears once again to verify that the calculations were indeed correct.

4.3.4 Potentiometer Minimum Position

Setting the potentiometer to its minimum position results in the following configuration:

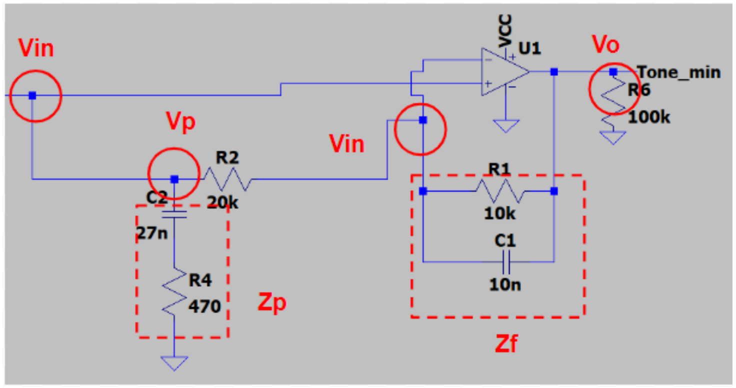


Figure 12 Active Tone Circuit; Min Position

By inspection, it is clear that node v_p is once again equal to v_{in}. This means that there is no voltage drop across the potentiometer. Nodal analysis then shows that:

$$\frac{v_{in}-v_o}{Z\_f} = 0 \tag{8}$$

which means that:

$$v_{in} = v_o \tag{9}$$

Octave was not used in this instance, but this configuration was simulated in LTSpice, which shows only a $600\mu dB$ change in gain through the audible frequency spectrum and is essentially equivalent to the result of (9).

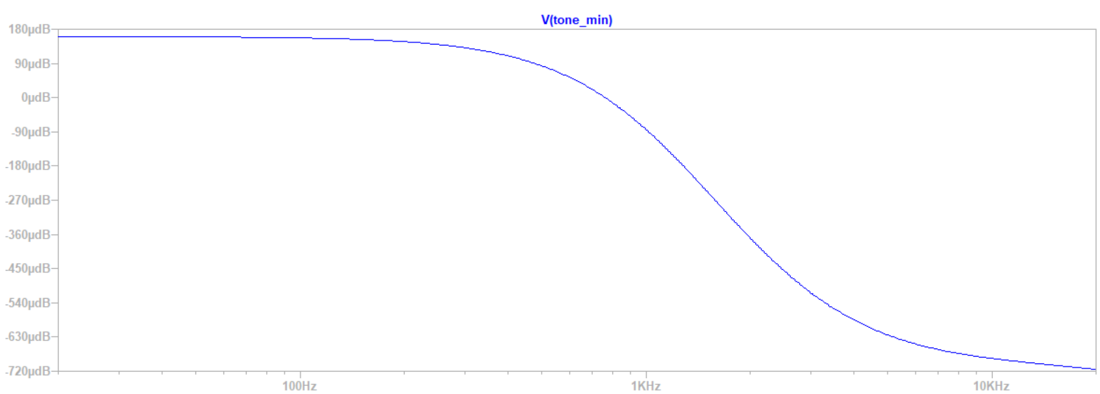


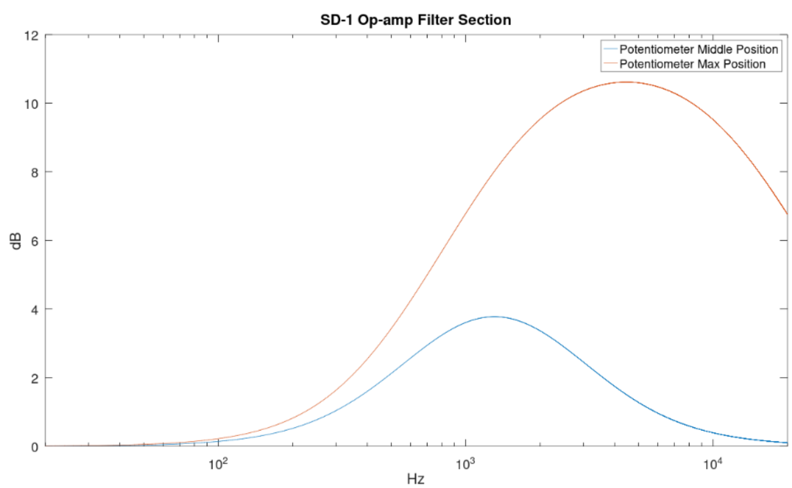
Figure 13 LTSpice Simulation: Potentiometer Min

This means that in the minimum position, the response of the pedal's filter section is dictated by the two passive filters that bookend the active op-amp filter section. These will be analyzed after a brief consideration of the results from the above calculations and simulations.

Table 1 Comparison Between Calculations and Simulations

	Potentiometer Middle Position			Potentiometer Max Position		
Filter Parameter	**Calculations and Numerical Simulation (Hz)**	**LTSpice Simulation (Hz)**	**% Difference**	**Calculations and Numerical Simulation (Hz)**	**LTSpice Simulation (Hz)**	**% Difference**
Center Frequency	1308.84	1312.446	0.28%	4467.99	4499.76	0.71%
Center Frequency Gain (dB)	3.77877	3.7785	0.01%	10.6192	10.6185	0.01%
3dB Frequency (lower)	251.791	250.628	0.46%	1194.94	1194.07	0.07%
3dB Frequency (upper)	6797.89	6839.59	0.61%	16683	16635.27	0.29%
Bandwidth	6546.1	6588.962	0.65%	15488	15441.2	0.30%

Granted that the values found in the numerical simulations were approximate and that the actual center and 3dB frequencies were not analytically calculated, the results from these exercises indicate that the calculations performed and upon which the numerical simulations were based were well within the acceptable range, within 1% of the simulated circuit response in LTSpice. We can also plot the responses of the two potentiometer positions side-by-side for a visual comparison:



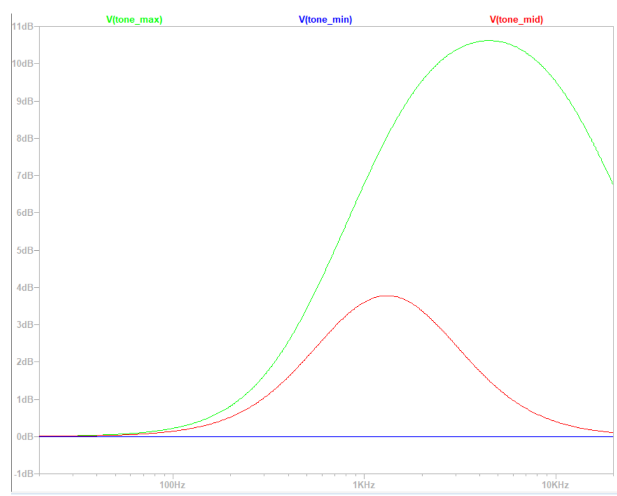


Figure 14 Numerical Simulations (Left) vs. LTSpice Simulations (Right)

4.3.4 Passive Filtering

The passive low-pass filter, which might be referred to as the 'pre-filter', is shown in Figure 15 below.

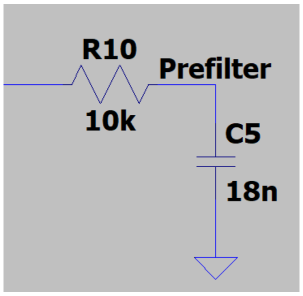


Figure 15 Pre-filter

This section can be analyzed using the basic equation below:

$$\omega_c = \frac{1}{RC} \tag{10}$$

which results in a corner frequency of 884.19Hz. This calculation is confirmed through a simulation in LTSpice, which gives a 3dB frequency of 883.08Hz:

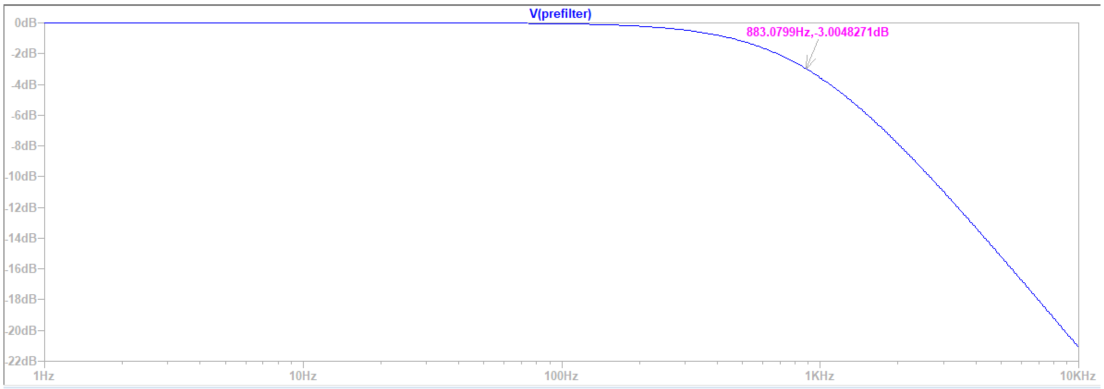


Figure 16 LTSpice Pre-filter Simulation

Looking at the section following the op-amp filter, which will be referred to here as the 'post-filter', it makes sense to add the impedance of the volume potentiometer (R12) to the 4.7$k\Omega$ resistor. This high-pass filter configuration is shown in Figure 17.

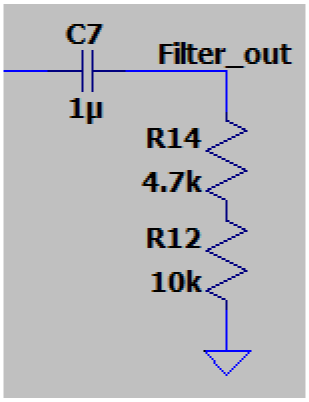


Figure 17 Passive High-Pass Post-filter

Using equation (10) with these values yields a theoretical corner frequency of 10.827Hz. Plotting the response of this filter in LTSpice confirms this calculation, giving a 3dB frequency of 10.839Hz.

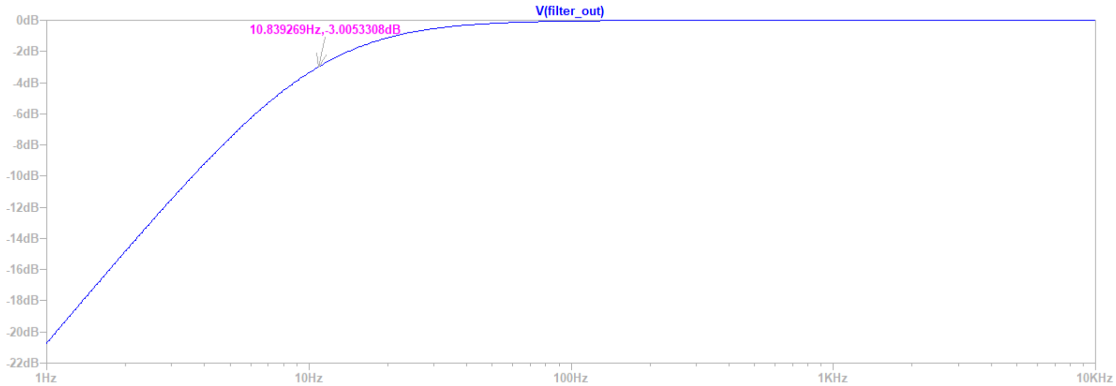


Figure 18 LTSpice Post-Filter Simulation

Combining the response of these two passive filters gives a good indication of what the overall response will be when the tone control potentiometer is in the minimum position, where the op-amp filter section has essentially a flat response.

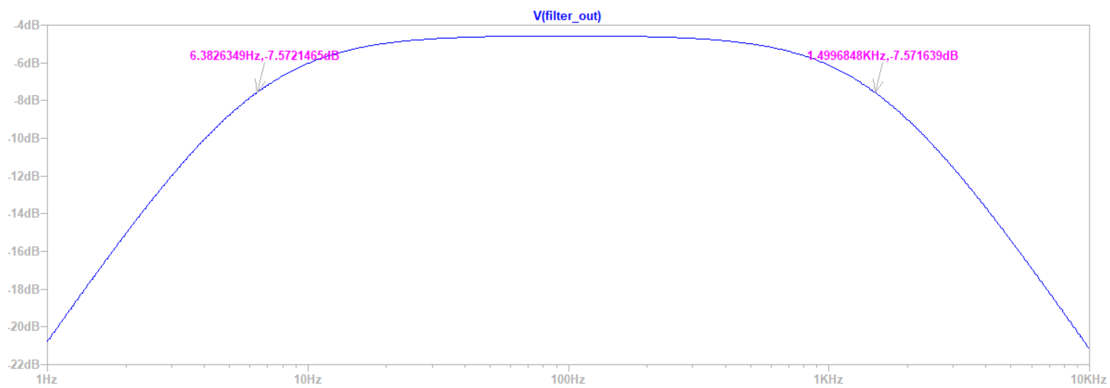


Figure 19 Combined Passive Filters

Adding in the other two previously analyzed filter sections would of course present variations of this response with the major differences found in the midrange.

5. Build and Test

The schematic was recreated in Altium Designer and the PCB was layout was completed.

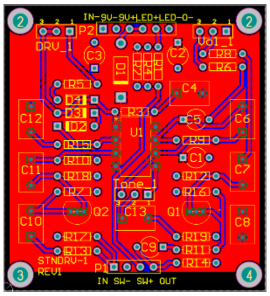


Figure 20 PCB Layout

Prior to ordering the PCB, the circuit was also recreated on a breadboard to verify that all components received were in good working order. The main issue found during this test was that the pinout of the BJTs received did not match the pinout shown in the datasheet of the online vendor. The schematic and PCB layout were therefore adjusted to accommodate this and, with the breadboard test seeming to function properly, the PCBs were ordered.

Final assembly proceeded largely without issue; the biggest lesson learned here was to have greater care in verifying the physical sizes of components used in Altium and to take greater care in part selection. When creating the schematic and PCB, only two capacitor models were used, one electrolytic and one non-polarized, and several of the non-polarized capacitors used in this build were physically larger than the footprint of the models used. This necessitated some of the legs to be bent to ensure a proper fit. Additionally, in future builds, different potentiometers will likely be used, as the ring-terminal potentiometers require additional wire jumpers between the board and their terminals, which creates more clutter within the enclosure and adds to the build time. Long-legged PCB-mounted potentiometers would also create a convenient way to secure the board within the enclosure.

6. Closing Remarks

The build was very successful; following the final assembly, the pedal functioned as desired without the need to troubleshoot any mistakes. The project overall was a great learning experience- it was rewarding to be able to apply techniques learned in various classes to analyze a circuit and be able to recreate it. The pedal has already joined the ranks of other pedals that I have purchased and will hopefully soon be joined by pedals of my own design.

Figure 21 Assembled PCB in Enclosure