EAE DIY

Fuzztain Clone

Builder's Guide

Version 1
July 23, 2024
John W Snyder

Revision History

Version	Changes
0	Draft Copy
-	

Contents

1 Introduction

This document contains a guide for assembling the Maestro Fuzztain Clone by Electronic Audio Experiments. This PCB is based on a schematic that we originally shared in 2017 as an open source project. It was originally shelved due to some parts issues (more on that later) but now we have finally made it available as a DIY project.

1.1 Background

The Maestro Fuzztain (aka MFZT-1) was originally released in the 1970s. At the time Maestro was owned by Norlin, who also owned Moog Music Inc and Gibson. This is somewhat of an infamous period for Gibson, but Bob Moog was quite prolific and had his hands in several products across Norlin's portfolio including various Maestro pedals and the brilliant Lab Series amplifiers. Under the Maestro Brand, Bob Moog developed iconic pedals such as the Stage Phaser and MPF-1 Parametric Filter. The Fuzztain is their relatively obscure sibling—not quite a fuzz, and not quite a compressor, I think nobody knew what to do with it.

So 40-ish years later, a friend of mine fixed an original Fuzztain and asked me to look into it. I laid out a PCB, some folks built them, and I posted the schematic on our website. But like many of my projects around that time, it was quickly abandoned for the next cool thing. Once Brad started working at EAE full time, he brought one in and we realized that it was actually a pretty fascinating design, absolutely worthy of inclusion in our DIY catalog.

1.2 Operation

The controls are as follows:

Sustain Input gain control, which changes the compressor behavior and interacts with the fuzz.

Fuzz Adjusts the gain of the fuzz circuit.

Volume Self-explanatory.

Mode Toggle Selects between clean compression and compression + fuzz.

The control set is quite simplistic, but hides a deceptively interesting and complex circuit design.

1.3 Before You Start

This board is recommended for moderate level to experienced builders. This guide assumes you have some familiarity with:

- Sourcing your own parts via Mouser, Digikey, etc. using a bill of materials
- Soldering a PCB from a parts list
- Drilling your own enclosure using a provided template
- Pedal assembly using PCB-mounted 16mm potentiometers
- Soldering jacks and switches with off-board wiring

This document includes a parts list, schematic, and drill template. It also includes a broad analysis of the circuit and some suggested modifications. For generic assembly tips, please refer to the Appendix.

1.4 Disclaimers and Licensing

⚠**Caution:** EAE is not responsible for the outcome of your build and cannot offer direct support or troubleshooting beyond this document.

2 Parts List

The following parts list assumes a build in the stock configuration. Parts marked with an asterisk (*) have additional information in Section 7, where you can also find suggested mods. We recommend reviewing possible modifications before you begin. For your convenience we have also provided a PCB BOM with manufacturer part numbers so that you can order parts from Mouser or Digikey.

Resistors and Trimpots

Qty	Value	Ref #s	Description
1	47R	R33	Resistor, Metal Film 1/8W
2	470R	R4, R26	Resistor, Metal Film 1/8W
1	1K	R6	Resistor, Metal Film 1/8W
1	1.5K	R11	Resistor, Metal Film 1/8W
7	10K	R1, R15, R16, R19, R20, R22, R31	Resistor, Metal Film 1/8W
1	15K	R10	Resistor, Metal Film 1/8W
1	22K	R21	Resistor, Metal Film 1/8W
2	47K	R9, R28	Resistor, Metal Film 1/8W
1	51K	R12	Resistor, Metal Film 1/8W
6	100K	R2, R5, R13, R14, R34, R35	Resistor, Metal Film 1/8W
5	200K	R3, R4, R7, R8, R32	Resistor, Metal Film 1/8W
1	470K	R23	Resistor, Metal Film 1/8W
2	510K	R17, R18	Resistor, Metal Film 1/8W

Capacitors

Qty	Value	Ref #s	Description
1	100p	C1	Capacitor, MLCC NP0 25V min 5mm LS
1	220p	C11	Capacitor, MLCC NP0 25V min 5mm LS
1	1n	C7	Capacitor, Box Film 5mm LS
1	3.3n	C10	Capacitor, Box Film 5mm LS
1	10n	C9	Capacitor, Box Film 5mm LS
2	100n	C2, C4	Capacitor, Box Film 5mm LS
1	220n	C3	Capacitor, Box Film 5mm LS
1	330n	C13	Capacitor, Box Film 5mm LS
2	1u	C22	Capacitor, Box Film 5mm LS
5	10u	C5, C6, C8, C12, C17	Capacitor, Elec 5mm dia 2.5mm LS 25V min
2	100u	C14, C15	Capacitor, Elec 6.3mm dia 2.5mm LS 25V min
4	100n MLCC	CB1, CB2, CB3, CB4	Capacitor, MLCC X7R 25V 0805

Discrete Semiconductors

Qty	Value	Ref #s	Description
1	1N5817	D3	Diode, Shottky
2	1N4148	D1, D2	Diode, General Purpose
3	2N3904	Q1, Q2, Q3	Transistor, BJT NPN General Purpose

Integrated Circuits

Qty	Value	Ref #s	Description
4	TL072	IC1, IC2, IC3, IC4	IC, Op Amp Dual
1	CD4007[†]	IC5	IC, MOSFET Array

Resistors, Potentiometers, Trimpots

Qty	Value	Ref #s	Description
1	100kΩ Audio	SUSTAIN	Potentiometer, 16mm PCB Mount Right Angle
1	25kΩ Audio	VOLUME	Potentiometer, 16mm PCB Mount Right Angle
1	1MΩ Reverse Audio	FUZZ	Potentiometer, 16mm PCB Mount Right Angle
1	SPDT	MODE	Switch, SPDT On/On Minuature (PCB Pins)

Off-Board Parts

Qty	Value	Ref #s	Description
1	Enclosure	N/A	1590BBM or equivalent
2	1/4” Mono Jack	IN, OUT	Switchcraft 111x or equivalent
1	DC Jack	9V	2.1mm barrel connector
1	3PDT	BYP	3PDT Footswitch, solder lug
3	Knob	N/A	1/4” shaft w/ set screw
1	3PDT Daughterboard	N/A	Included w/ PCB
1	LED	N/A	Any 3mm or 5mm LED
1	Ribbon Cable	N/A	6 conductor ribbon cable, prestripped

3 Wiring

The wiring diagram is shown in Figure 1. For overall assembly instructions please refer to the Appendix. Wire pads on the board are listed in Table 1. For convenience, pads make connections to hardware located in close proximity. Note that the LED is mounted on the reverse side of the footswitch PCB, or can be wired off-board using a bezel. If you are using a 3PDT daughterboard, a 6-pin ribbon cable may be used. If you are not using a daughterboard, refer to Figure 2. You may use another scheme provided the input is grounded in bypass.

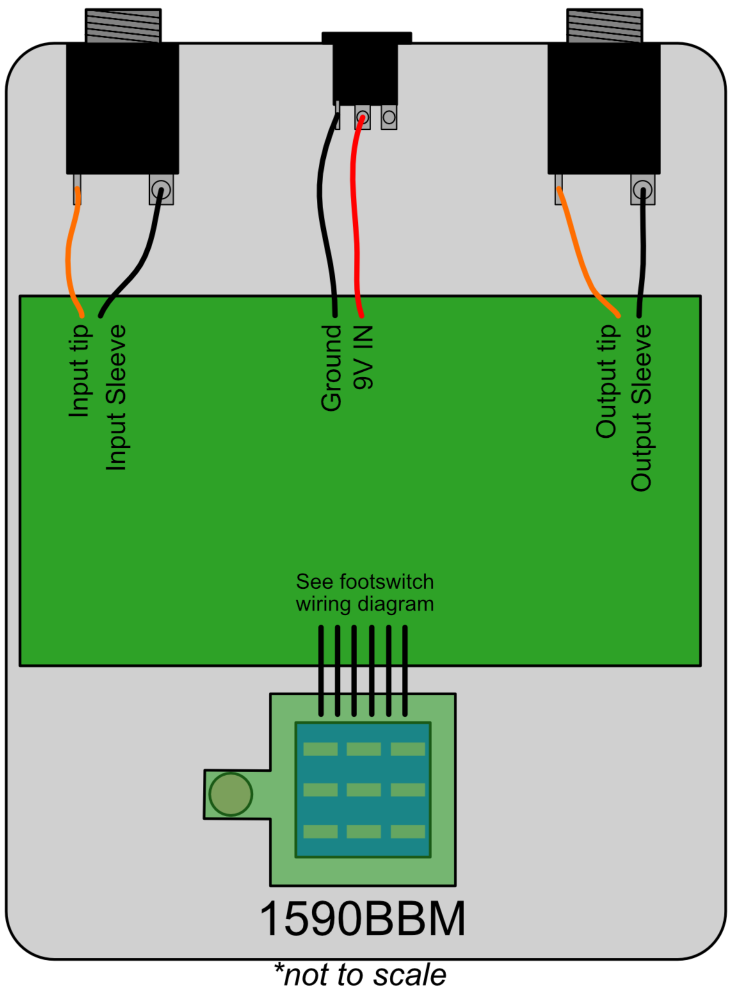


Figure 1: Wiring diagram. This view is looking into the enclosure from below. Note that this figure is for illustrative purposes only and is not drawn to scale. Tip/sleeve pads may be swapped on certain boards, so be sure to follow the markings on the board.

Pad	Function
I	Input Jack Tip
O	Output Jack Tip
G1, G2	Jack Sleeve Connections (Ground)
[+]	9V input (DC jack)
[-]	DC jack ground
JI	Bypass footswitch connection to input jack
L+	LED + to footswitch
L-	LED/Footswitch ground
S	Send/Input to effect
R	Return from effect
JO	Bypass footswitch connection to output jack

Table 1: List of wire pads and functions.

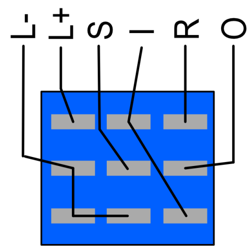
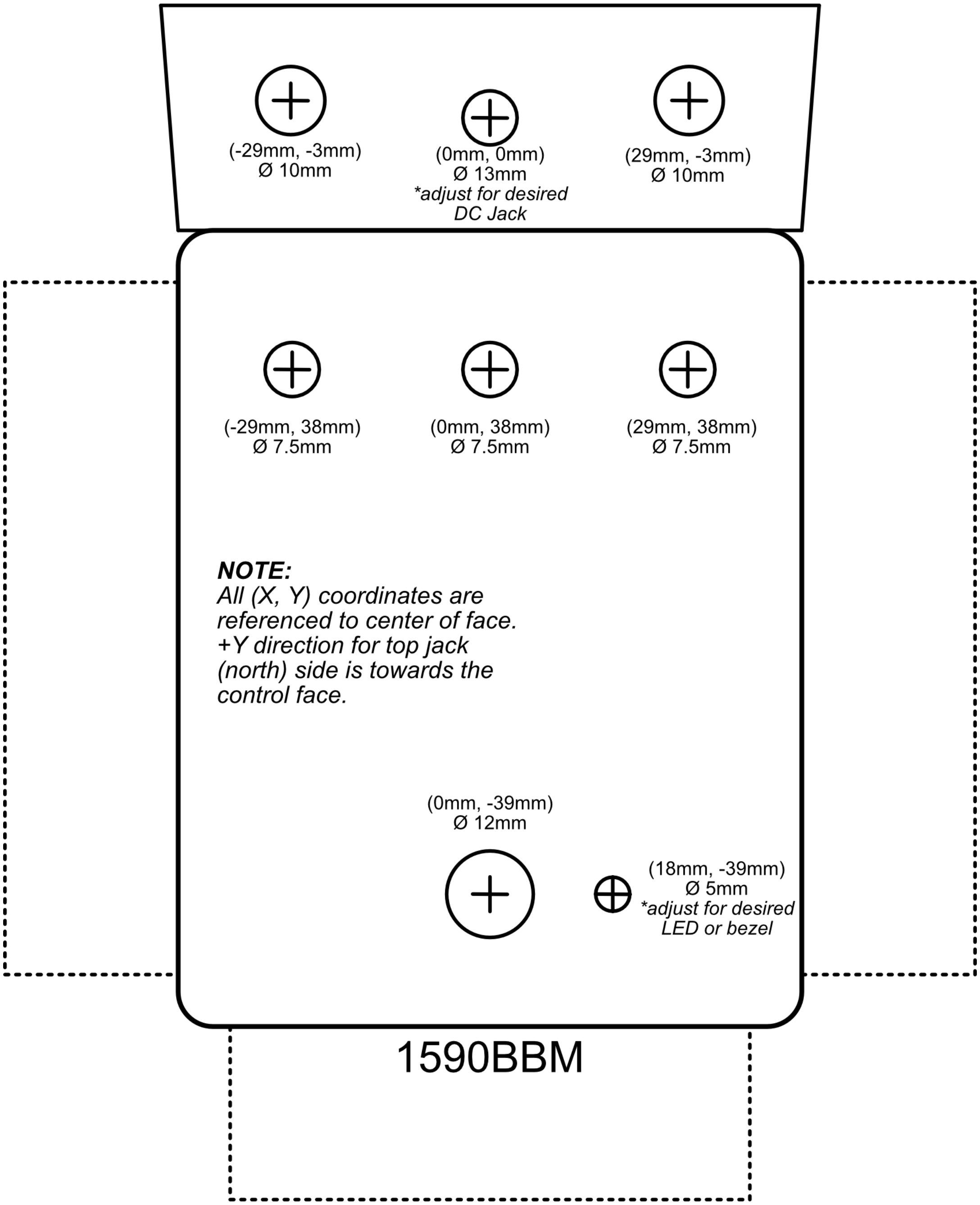


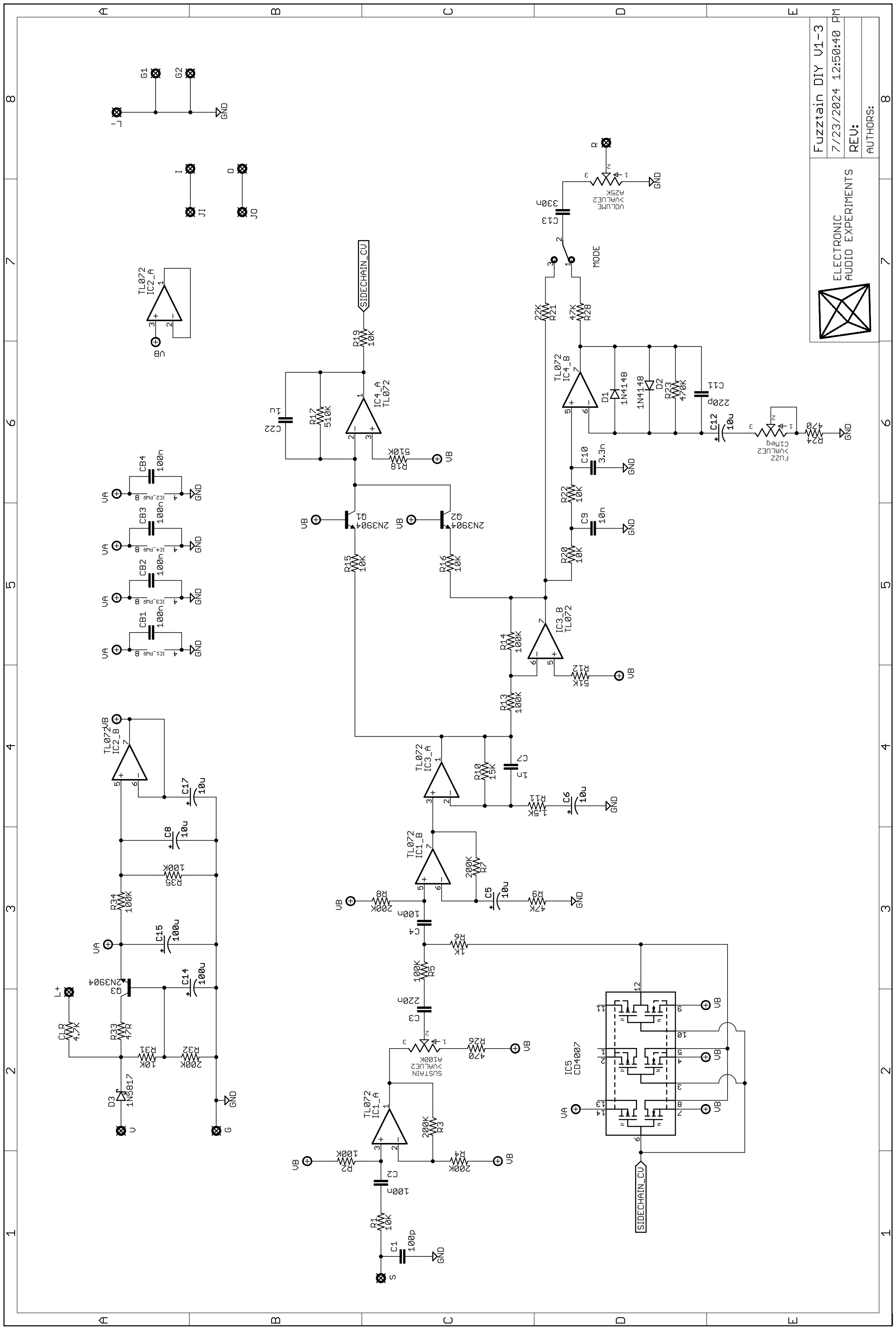
Figure 2: 3PDT Wiring Diagram. Note horizontal orientation of solder lugs.

4 Drill Template

Print this page out using 1:1 settings on your printer and fold to form the drill template. We recommend drilling pilot holes first to check alignment.



5 Schematic


Fuzztain DIY V1-3
7/23/2024 12:50:40 PM
REV:
AUTHORS:
ELECTRONIC
AUDIO EXPERIMENTS

6 Circuit Analysis

6.1 Signal Path

The Fuzztain consists of two primary circuit blocks. First is the compressor, second is the clipping amplifier which serves as the fuzz stage. Some things jump out immediately: first, the compressor design is pretty unusual on multiple levels. Second, the fuzz circuit looks more like a stereotypical soft-clipping overdrive! The Fuzztain has some truly clever design tricks up its sleeve, which we'll dive into below.

To help make sense of this, let's first trace the signal flow. The first gain stage, formed by IC1A, is a basic non-inverting gain stage with a gain of 6dB (or 2x)[1]. Note that R2 is 100kΩ which is quite low for a guitar pedal. In many classic pedals you would see 1MΩ here. This lower input impedance will load down a guitar that is plugged in directly, attenuating the signal slightly and lowering the resonant peak of pickups.

This is immediately followed by the Sustain pot, which is a basic attenuator. From here the signal goes two places: first, to whatever the heck IC5 is doing, and second to another gain stage formed by IC1B, which has a gain of 14.5dB (about 5x). We'll come back to IC5 in a bit.

Immediately following IC1B, we have IC3A. This is another non-inverting gain stage with a gain of about 21dB (11x) and some band-pass filtering thnaks to C6 and C7. This feeds a few different things. We're going to also ignore the sidechain circuit formed by Q1, Q2, and IC4A for now. The output of IC3A goes to R13, which is the input of an inverting amplifier formed by IC3B, R13, and R14. Since R13=R14, the gain is -1, which means it is just flipping the phase.

The output of IC3B goes to two possible paths, as determined by the Mode toggle. One is simply through R21 to the output, for the clean compressor mode. The other is to the fuzz stage. First we have a simple passive low pass filter, formed by R20, R22, C9, and C10. Then we have IC4B, which should be somewhat familiar if you have looked at other pedal schematics. It's a fairly common type of clipping stage using back-to-back diodes in the feedback. The gain is the ratio of R23 divided by wherever the Fuzz pot is set. At minimum, the gain is about 1.5x. At maximum, the gain is about 1000x or 60dB—or at least it would be, if the diodes weren't limiting the output. Folks often associate feedback clipping gain stages such as this one with low-gain pedals like the Tubescreamer, but with all of the gain up front (somewhere around 40dB when no gain reduction is taking place), this sounds like something else entirely.

Now that we've traced the signal flow, let's make sense of the compressor portion of the circuit. The Fuzztain uses a CD4007 MOSFET array as a gain reduction element. How does this work? When a MOSFET is biased into its linear region, the current flowing from the drain to the source is a function of the voltage ap-

[1]Recall that the gain is $1 + R_3/R_4$, so if R3=R4 the gain is 2.

plied to the gate. In other words, a MOSFET acts like a voltage-controlled resistor with resistance R_{DS} (where DS stands for Drain to Source). In this case we have three MOSFETs contained within the CD4007 operating as the lower leg of a voltage divider. This divider is formed by R5 and R6 plus the parallel resistance of the MOSFETs, giving us:

$$V_{out} = V_{in} \frac{R_{DS} + R_6}{R5 + R6 + R_{DS}} \tag{1}$$

Where R_{DS} is the combined parallel resistance of all three MOSFETs[2]. If this looks intimidating you can look at the range where R_{DS} is very large and very small. If $R_{DS} >> R5$, then $V_{out} \approx V_{in}$. If R_{DS} is close to zero, then the gain is $R6/(R5+R6)$ which is pretty heavily attenuated.

While this analysis glosses over the finer points of MOSFET behavior, intuitively we can see that the amount of signal coming out of this attenuator will depend on the gate voltage, which is the control voltage coming from the sidechain. The job of a sidechain is to produce an envelope output, or a slowly changing voltage representing the loudness of a signal. The louder the input, the larger the envelope voltage. That voltage is then applied to the gain reduction. Quieter signals will be amplified (mostly by (IC1B and IC3A), and louder signals will be attenuated. Thus, we have compression.

The sidechain circuit itself is formed by Q1, Q2, and IC4A. Q1 and Q1 are a less common configuration referred to as a *Common Base Amplifier*, where the signal is fed into the emitter of a transistor and the base is held at a fixed voltage. In this case, each transistor is biased at VB which is the power supply mid rail. Q1 is fed from IC3A, and Q2 is fed from IC3B, which simply inverts the signal. Q1 and Q2 each only conduct the positive half of the waveform, and these two halves are summed together by IC4A. So all this toegther forms a rectifier, or a circuit that converts AC (like an audio signal) into DC. The final piece of the puzzle is C22, which filters the ripply rectifier output into a smooth envelope. That output goes to IC5, controlling the gain reduction MOSFETs.

Finally, I want to briefly discuss the power supply. Note that the incoming voltage is passed through Q3. This circuit is called a *Capacitance Multiplier* and is essentially a primitive regulator[3]. In this circuit, the capacitor C14 is effectively multiplied by the gain of the transistor, which for a typical 2N3904 is around 100. Thus the capacitor value "seen" by the circuit is actually 0.1F—which is massive, and much cheaper (and more compact) than a real 0.1F capacitor. This circuit isn't perfect, however. Its ability to filter is mostly applicable to low frequency noise, like 60Hz buzz, rather than high frequency noise like BBD clock whine. Another downside is that the power supply voltage is reduced by the saturation voltage of Q3 (V_{CE}(sat)), which is around 1V. For a high-gain fuzz like this, the benefits greatly outweigh the drawbacks.

---

[2]Since they are all on the same die, I assume they are at least reasonably well-matched.

[3]This circuit is often seen in Boss pedals as well.

7 Modifications

⚠**Caution:** Modifications should not be attempted unless you have the means to troubleshoot changes beyond the base configuration.

7.1 Some modifications

This circuit really doesn't need much in my opinion, but here are some things you can try.

- Try socketing C22 to adjust the compressor release time. The attack time is more or less fixed unless you really want to mess with the sidechain behavior.
- Try LEDs in D1/D2. It gets loud!
- Try splicing in a tone control (such as the popular "Stupidly Wonderful Tone control") right before the volume pot.
- The low pass filter formed by R20, C9, R22, C10 is fairly dark. Try socketing C9 to audition some brighter, nastier sounds. We like a 4.7n cap in the C9 spot.
- Try something low noise like an NE5532 for IC1 and IC3.
- The Mode toggle can be replaced with a pot to give the fuzz an effective clean blend. Try a B50K pot to start and replace R28/R21 with 1K resistors to allow more range.

A Generic Assembly Guide

This is a fully general pedal assembly procedure that applies to all EAE open source projects. Note that minor details can vary between PCBs. But this will serve as a quick reference for those already familiar with this type of assembly.

1. Populate and solder the PCB per the parts list. Do not solder in enclosure-mounted components such as pots, indicator LEDs, and/or toggle switches. Ensure that all leads are trimmed to 0.1" or less.

2. Drill the enclosure per the template. Use a center punch first to mark holes for accuracy. If you have one, a stepper bit is very convenient here. If you are unsure about any hole sizes, check using the appropriate hardware.

3. Solder pots and toggle switches (if present) to the PCB using the enclosure as a jig to hold the hardware in place. We recommend using potentiometers with dust covers. If the pots are not insulated, ensure they are not making contact with the back of the PCB.

4. Install PCB mounted LED on the reverse side of the board. Gently bend the legs to hold it in place, but do not solder yet.

5. Install flying wires on the board. These go to the jacks and footswitch(es).

6. Insert PCB into enclosure and tighten all hardware. Fit LED(s) into place, solder, and trim the leads.

7. Solder flying wires to the appropriate pads. Refer to the wiring diagram or table for the specific pedal in question.

8. Pedal is ready for testing!