

---

# $\mu$ Blaster

Builder's Guide

Version 1  
January 5, 2022  
John W Snyder

# Revision History

Version	Changes
1	Initial Release (Jan 5 2022)
0	Draft Copy
-	

# Contents

<b>1 Introduction</b>	<b>2</b>
1.1 Background and Operation . . . . .	2
1.2 But it's Surface Mount! . . . . .	3
1.3 Before You Start . . . . .	4
1.4 Disclaimers and Licensing . . . . .	4
<b>2 Parts List</b>	<b>5</b>
<b>3 Wiring</b>	<b>7</b>
<b>4 Drill Template</b>	<b>9</b>
<b>5 Schematic</b>	<b>10</b>
<b>6 Circuit Description</b>	<b>11</b>
<b>7 Modifications</b>	<b>15</b>
7.1 Alternate JFETs . . . . .	15
7.2 New Tone Stack Configurations . . . . .	15
<b>A Generic Assembly Guide</b>	<b>16</b>

# 1 Introduction

This document contains a guide for assembling the  $\mu$  Blaster boost by Electronic Audio Experiments. This pedal is no longer in production, so we have released it as a DIY project for your continued enjoyment.

## 1.1 Background and Operation

The  $\mu$  (or Mu) Blaster is a powerful clean boost named for its cascading stages of JFET mu amps. It serves a single purpose: to slam the input of an amplifier. It features a high voltage  $\sim 27V$  power supply for extra headroom, and a unique Depth control for contouring the low frequency response of the boosted signal. In conjunction with its other controls, it serves as a useful extension of a single channel amp.

The controls are as follows:

**Volume** self explanatory, can deliver a massive output. Use judiciously, as high levels can damage certain input stages!

**Boost** adds additional saturation and presence. Counterclockwise is clean and relatively dark, while clockwise is brighter and moderately driven.

**Filter** simple low-pass filter at the output. Useful for taming harmonic content from high boost settings.

**Depth** active bass control. Lower settings produce rangemaster-like tones with lots of chime, while higher settings allow a huge amount of bass through. At max depth and boost settings, there is an additional frequency boost in the 50-100Hz range, right in the fundamental for heavier styles of music.

This pedal is one of the earliest models developed under the EAE brand—the first production units were sold in Spring 2016. It also has the distinction of being the first EAE pedal under the graphic design direction of Zach Weeks, who has been handling our layouts and fonts ever since. In about late 2015 I wanted a loud clean boost to go with a single-channel amp prototype I was working on at the time. My only real goals were that it was 1) LOUD and 2) worked nicely with high-output pickups. I think we achieved that goal! But production was always sporadic, and we decided to phase the model out in 2020.

I'm glad we finally got to give this one the open source treatment. While simple in its origins, it has some unique qualities due to a happy accident in the design process. In particular, the powerful Depth control owes its effectiveness to a breadboarding mistake. I do my best to explain it all in the Circuit Description section. We also decided to leave the PCB in its latest manufacturing revision, which uses a mix of through hole and surface mount parts, as there is a lack of DIY-friendly pedal PCBs using SMT.

## 1.2 But it's Surface Mount!

I have to emphasize that I am not an expert in surface mount soldering. That said, I've built a few dozen prototypes and can hopefully pass on some things that helped me get started. The major message I want to convey is that surface mount isn't hard—it's just different. We've collectively had decades to absorb through hole soldering as a concept. But there are plenty of annoying sides to it, as anyone who has tried to desolder a box cap with bent legs has probably observed. In comparison, plucking an 0805 component off a board is trivial. There are many good tutorials on the internet so I won't go into exhaustive detail. But, here are some pointers:

- Good tweezers are your friends. Practice with moving small objects around, and get used to using your left and right hands independently.
- Magnification can be a huge help. There are a variety of headband magnifiers and hands-free desktop magnifying glasses available for this purpose.
- Get a flux pen and a way to clean off that flux. Specific brands don't matter much, and 99% isopropyl alcohol and a cheap toothbrush will suffice for your cleaning needs.
- Don't be afraid to use a regular soldering tip. An ultra-fine tip can actually make things harder because you have less surface area to apply heat to a solder pad.

In general, when soldering components like 0805 passives (all of the resistors on this board) and SOT-23 semiconductors (such as JFETs), you can use the following procedure:

1. Heat one pad for the part you wish to solder and apply a small amount of solder.
2. While reheating the solder, bring the component into place and tack solder it down. Make sure it is properly aligned on all pads.
3. Solder the other leg(s) of the component one at a time.
4. Reflow the first solder point if need be. If it's not reflowing, add flux.

## 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 bill of materials (BOM), schematic, and drill drawing. It also includes a broad analysis of the circuit and some suggested modifications. For convenience, you may also access an excel BOM. For generic assembly instructions, 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.



This document and all associated materials are released under the Creative Commons BY-SA License. You are free to share and adapt the material for any purpose, including commercially. However, you must provide credit and distribute your contributions under the same license. For more information visit <https://creativecommons.org/licenses/by-sa/3.0/>.

## 2 Parts List

For a spreadsheet with suggested part numbers, please see the Excel spreadsheet included with this document.

### Resistors, Potentiometers, Trimpots

Qty	Value	Ref #s	Description
1	100k $\Omega$ Audio	Volume	16mm potentiometer, PCB mount right angle
1	10k $\Omega$ Audio	Filter	16mm potentiometer, PCB mount right angle
1	10k $\Omega$ Rev. Audio	Boost	16mm potentiometer, PCB mount right angle
1	1M $\Omega$ Rev. Audio	Depth	16mm potentiometer, PCB mount right angle
4	10k $\Omega$ Audio	CLR, R2, R6, R10	Thick Film Resistor, 0805 1/8W 1%
7	1M $\Omega$ Audio	R1, R3, R4, R5, R7, R8, R9	Thick Film Resistor, 0805 1/8W 1%

### Capacitors

Qty	Value	Ref #s	Description
1	100nF	C20	Capacitor, SMT 0805 50V X7R
1	100pF	C3	Capacitor, SMT 0805 50V NP0
3	10 $\mu$ F	C10, C11, C21	Capacitor, SMT 1206 50V X5R or X7R
4	47 $\mu$ F	C7, C13, C14, C15	Capacitor, Electrolytic 35V
2	100nF	C2, C4	Capacitor, box film 5mm lead spacing
1	1nF	C5	Capacitor, box film 5mm lead spacing
3	1 $\mu$ F	C1, C6, C8	Capacitor, box film 5mm lead spacing
1	22nF	C9	Capacitor, box film 5mm lead spacing

## Semiconductors

Qty	Value	Ref #s	Description
5	SM5817	D1, D2, D3, D4, D5	Shottky Diode, SOD-123 package
1	LT1054	IC1	IC, Charge Pump (DIL8)
1	DIL8 Socket	IC1	Socket for IC1
4	MMBF5457	Q1, Q2, Q3, Q4	N-Channel JFET (see Modifications section for substitutes)
1	MMBT5088	Q5	NPN Transistor (any pin compatible part will work)
1	LED	LED	5MM LED

## Off-Board Parts

Qty	Value	Ref #s	Description
1	Enclosure	N/A	125B 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
4	Knob	N/A	1/4" shaft w/ set screw

### 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.

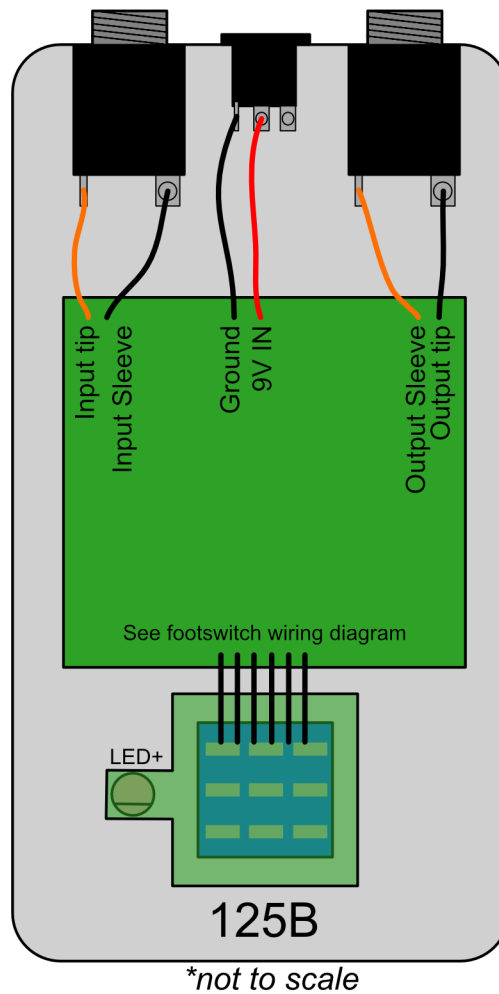


Figure 1: Wiring diagram. This view is looking into the enclosure from below. Note that this figure is for illustrative purposes only and is not drawn to scale. Tip/sleeve pads may be swapped on certain boards.



<b>Pad</b>	<b>Function</b>
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.

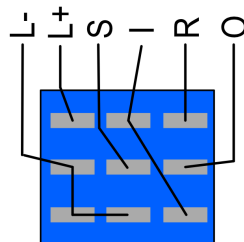
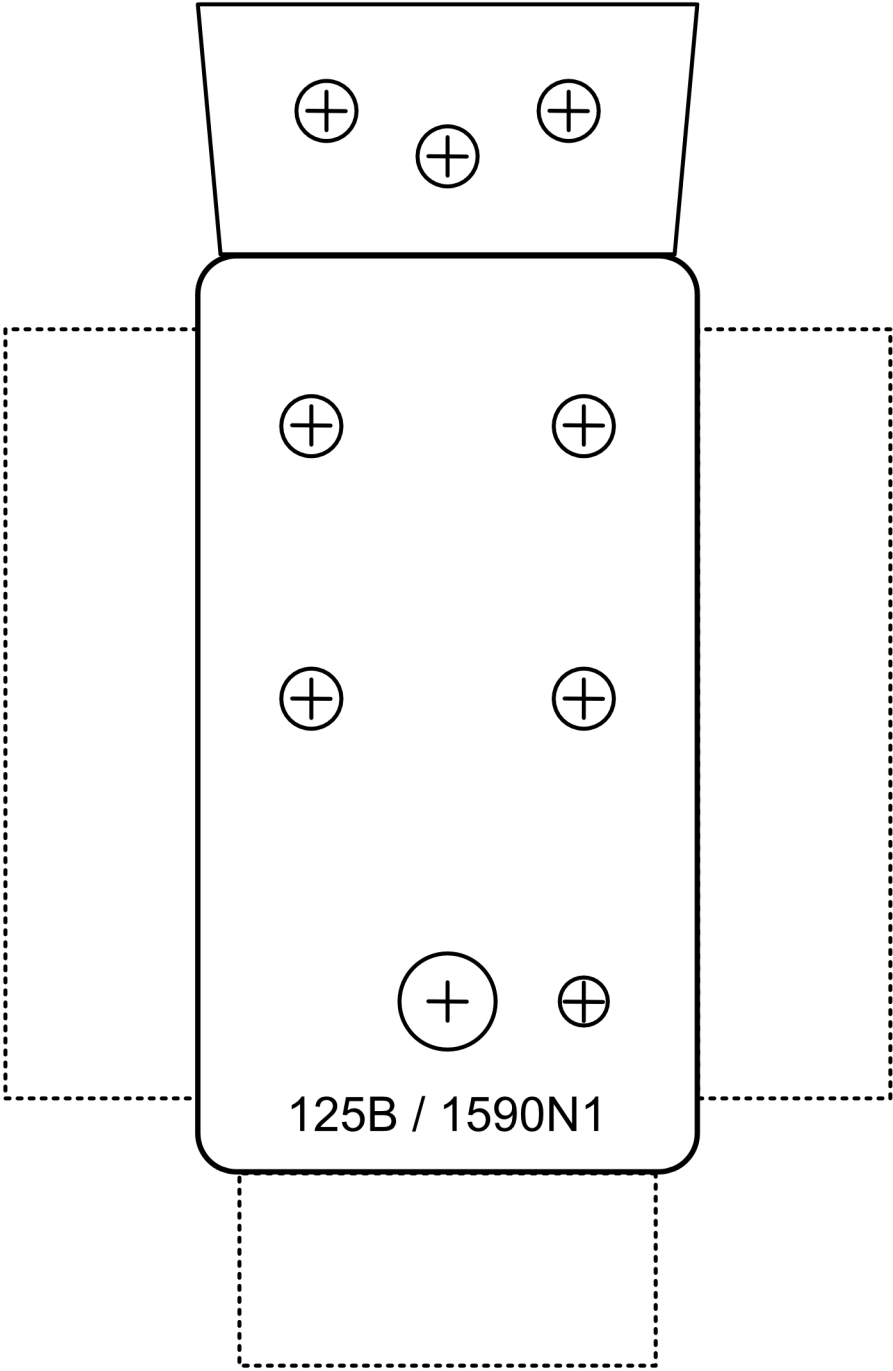


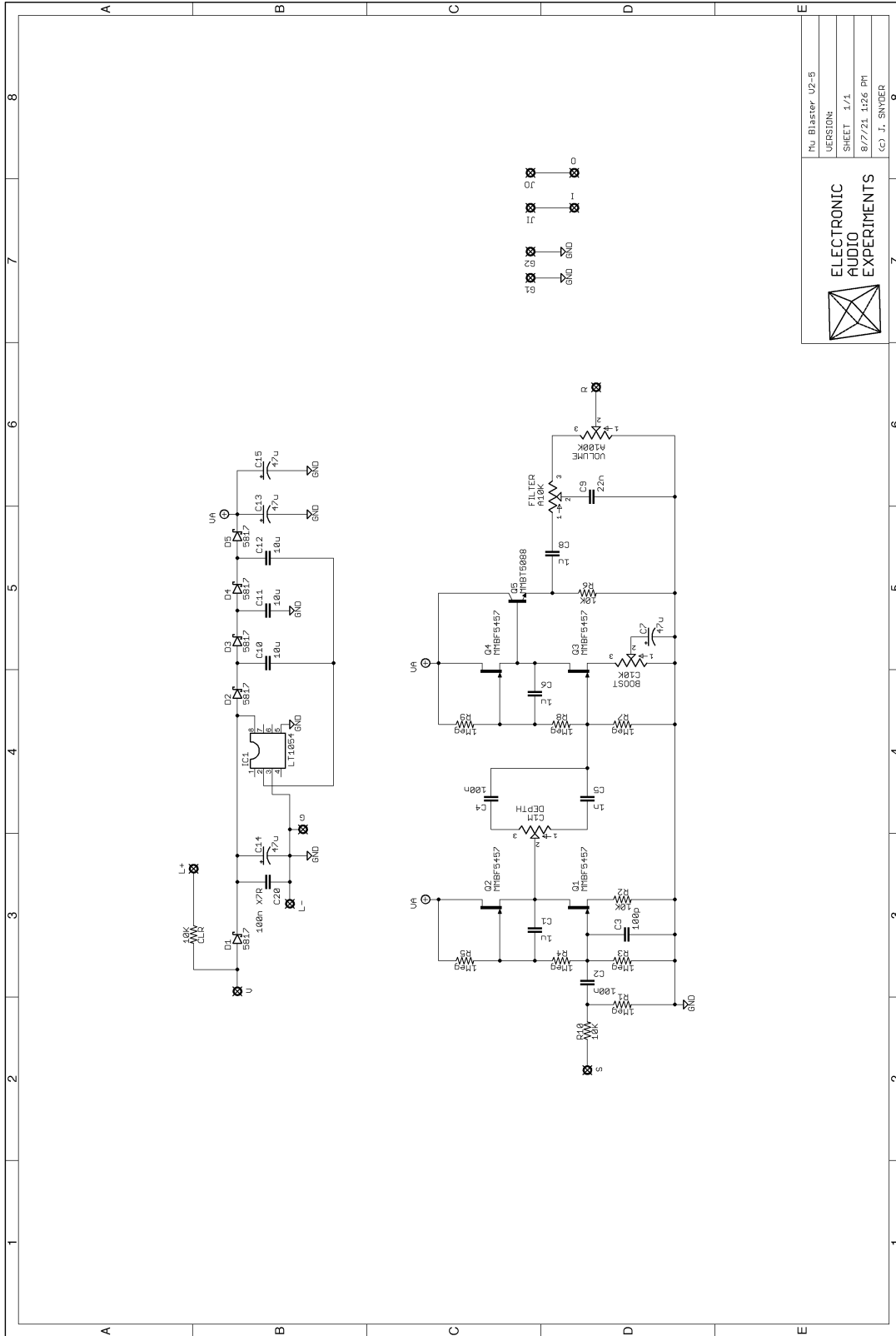
Figure 2: 3PDT Wiring Diagram. Note horizontal orientation of solder lugs.

# 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



## 6 Circuit Description

The Mu Blaster is a straightforward design consisting of a fixed gain stage, an adjustable gain stage, some tone controls, and a master volume. But for such a relatively simple circuit there is, in my opinion, a surprising amount of complexity deceptively baked in.

The JFET mu amp is a classic building block of pedal circuits—look at enough schematics and you’ll start to see them regularly. A good explanation of how they work can be found in the “Foolin with FETs” article by the inimitable R.G. Keen<sup>1</sup>. A similar circuit topology<sup>2</sup> first appeared in vacuum tube circuits, and a JFET version was introduced in National Semiconductor Application Note 32, “FET Circuit Applications”<sup>3</sup>. In the context of pedal circuits, it was originally popularized in the DIY pedal community by (the also inimitable) Jack Orman of AMZ<sup>4</sup> in the early 2000s. The concept quickly gained popularity among boutique and DIY pedal makers because it provides a way to get consistent, high-gain amplification out of JFETs, which are otherwise known for having a wide spread of parameters.

The mu blaster circuit consists of two such gain stages. At first glance, it looks similar to a couple of classic designs based on this building block such as Jack Orman’s Booster 2.5<sup>5</sup> and the “Brown Sound in a Box” (aka BSIAB)<sup>6</sup>. Circa winter 2015-2016, I wanted to hear what these two-stage mu amp circuits sounded like at higher supply voltages, so I built one up on a breadboard—or at least, I tried to. (Recreational breadboarding after a long day of work and classes is, perhaps, not the most rigorous discipline.) Upon closer inspection of the mu blaster schematic, one can see that my mu amps are drawn incorrectly. Instead of connecting the lower gate resistors on the upper JFETs to ground, I accidentally connected them to the gate of the upper JFET. Trying the mu amps the “right” way spoiled the Depth control, which was my favorite part of the design, so I decided to proceed with my lucky mistake. However it was not until a couple years later that I actually came up with an explanation for what was happening. This modified mu amp (if you can still call it one—is it actually some kind of more general cascode?) that I stumbled upon has two primary side effects: i) it has an uncommon biasing method which actually *reduces* its headroom compared to the standard case, and ii) it changes the gain structure through the addition of negative feedback. The mu blaster contains two permutations of this circuit, but I think many others might be possible.

Let’s look at an example of this modified mu amp in Figure 3. The resistor ladder formed by R1, R2, and R3 changes the biasing scheme significantly. JFETs in pedal circuits are most commonly operated in a “self-biasing” configuration with a resistor placed from the gate to ground, typically  $\sim 1M\Omega$ . In the case of Figure 3, this would

---

<sup>1</sup>Available at [http://www.geofex.com/article\\_folders/foolwfets/foolwfets.htm](http://www.geofex.com/article_folders/foolwfets/foolwfets.htm)

<sup>2</sup>I realize I use this term often. Colloquially, it refers to the connections of a circuit at an abstract level. If you look for patterns in electronics, you’ll find them everywhere.

<sup>3</sup>Available at <https://www.ti.com/lit/an/snoa620/snoa620.pdf>.

<sup>4</sup>Available at <http://www.muzique.com/amz/mini.htm>.

<sup>5</sup>Available at <http://www.muzique.com/tech/Booster25.jpg>.

<sup>6</sup>Available at <http://www.muzique.com/tech/images/BSIAB2.jpg>.

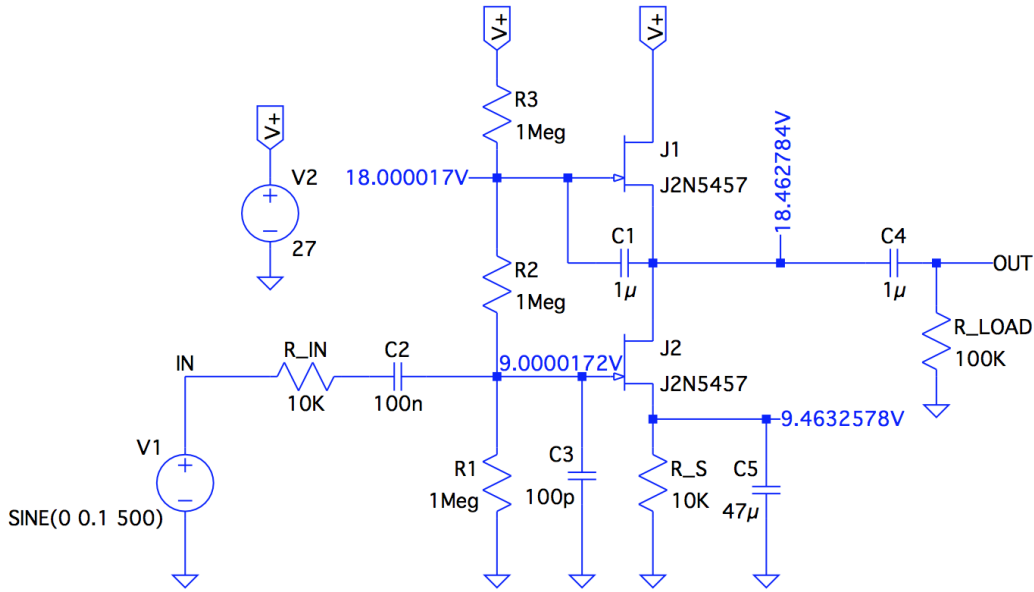


Figure 3: Example of modified mu amp stage with bias voltages indicated. Some values differ slightly from the mu blaster to better illustrate the circuit behavior.

mean J2 is biased solely through R1. Here we are instead biasing the gate of J2 above 0V using the same resistor ladder used for biasing J1. If we assume the drain current is well below  $I_{DSS}$  (a minimum of  $\sim 1\text{mA}$  for a 2N5457, which is a fairly safe assumption here) this configuration biases the gate of J2 at 1/3 the supply voltage ( $\sim 9\text{V}$ ) and J1 at 2/3 the supply voltage ( $\sim 18\text{V}$ ). Coincidentally, the overload behavior is still symmetrical despite the J2 drain bias not being half the supply voltage. With the 2N5457 model used in my simulation, the drain of J2 sits a little over 18V and can swing about 8V in either direction before clipping. Note that were it a traditional mu amp with symmetrical biasing, the output would probably swing around 11-12V in either direction. The capacitor C5 bypasses the source resistor R\_S to maximize the available gain for AC signals without disturbing the DC operating point.

More importantly, this configuration forms a path for negative feedback via R2. Negative feedback is the process of using the output of a system to moderate its behavior. In an electronic amplifier, we can introduce negative feedback by subtracting some portion of amplifier's output from its input. This is an extremely useful tool: an amplifier with negative feedback has reduced distortion, improved stability, lower output impedance, a wider bandwidth, and reduced sensitivity to component variations. In an ideal inverting amplifier with infinite *open loop* gain (i.e., the gain with no negative feedback) the *closed loop* gain depends on the amount of output signal fed back relative to the original input signal.

All single stage discrete voltage amplifiers invert the signal, which means negative feedback may be applied by connecting the output to the input via a resistor. In effect, we have a summing amp where the output signal is being subtracted. In

Figure 3,  $R_2$  serves this purpose. It is not directly connected to the output, but the output has a path through the  $1\mu\text{F}$  cap  $C_1$  so that the feedback does not impact the DC bias voltage. The magnitude of the gain in this case can be approximated as  $\sim R_2/R_{in}$ , or about 40dB, which happens to be roughly equivalent to the open loop gain of the amplifier under the same biasing conditions (and when using 2N5457s). By increasing  $R_{in}$  or decreasing  $R_2$ , we can decrease the gain.

In the first stage, formed by Q1/Q2, negative feedback actually doesn't matter very much. The lack of a bypass capacitor reduces the open loop gain by quite a bit, to about 15dB. So in this case the negative feedback has little impact on its behavior since there's no open loop gain for it to reduce. (It was accidental, after all—so nothing ventured, nothing gained.) The second stage formed by Q3 and Q4 is another story. The Boost pot is a variable bypass of the source resistance, which adjusts the open loop gain. The Depth pot changes the input impedance of the stage. If we extend our feedback approximation to include the impedance of the capacitors  $C_4$  and  $C_5$ , we can approximate the second stage gain  $A_2$  as:

$$A_2 \approx \frac{-R_8}{(C_4 + R_D d) \parallel (C_5 + R_D(1 - d)) + Z_{out,1}} \quad (1)$$

Where  $R_D$  is the resistance of the Depth pot,  $d$  is its rotation from 0 to 1, and  $Z_{out,1}$  is the output impedance of the first gain stage. The operation  $\parallel$  denotes the parallel combination  $(Z_1 Z_2)/(Z_1 + Z_2)$  for two impedances  $Z_1, Z_2$ . This is quite complicated to do by hand, especially since  $Z_{out,1}$  is not constant with frequency either. Thankfully, we have LTSpice to help. The frequency response at various Depth settings is plotted in Figure 4. Also, note that the gain is negative! This indicates that the phase is inverted. The action of the Depth control is my favorite part of this circuit. It morphs from a strong treble emphasis, to a flatter frequency response with more low end, to a prominent bass boost. The extra +3dB peak at about 70Hz appears to be a side effect of the bootstrap capacitor in the mu amp introducing positive feedback, though I have to confess I haven't been able to crack that one.

All that aside, the rest of the circuit is straightforward. The output of the second gain stage is buffered by the BJT emitter follower Q5. Since mu amps tend to have very poor load driving capability, the follower lets us add a tone control and a passive volume control without compromising performance. The tone control is a variation on Mark Hammer's classic "Stupidly Wonderful Tone Control"<sup>7</sup> which is a great way to implement a passive low pass filter without also changing the overall volume.

Finally, I should briefly touch on the power supply. We use an LT1054 charge pump as a voltage multiplier with three stages. This converts the incoming 9V supply to about 27V, less the diode drops of each shottky diode. The Mu Blaster began my (as of yet unabated) enthusiasm for boosted rails in guitar circuits. If you like boosters that can swing excessively high peak voltages, this is certainly a way to do it.

---

<sup>7</sup>Available at <https://hammer.ampage.org/files/SWTC.gif>.

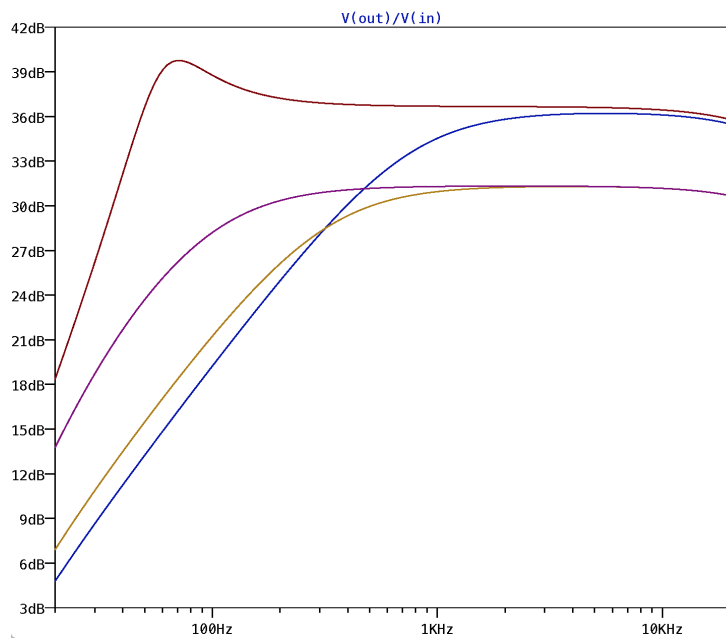


Figure 4: Mu Blaster depth pot response. Simulation was implemented with 2N5457 JFET models—spectrum may vary with other parts.

## 7 Modifications

<p><b>⚠ Caution:</b> Modifications should not be attempted unless you have the means to troubleshoot changes beyond the base configuration.</p>
---

### 7.1 Alternate JFETs

This circuit was originally designed with the 2N5457 (and later the MMBF5457, its surface mount counterpart) in mind. At the time of this document (late 2021) these are rather hard to come by. We have had luck with the MMBFJ201, MMBFJ112, and MMBF5103. The mu amp configuration is rather forgiving of device tolerances, so it is relatively safe to try different JFETs.

### 7.2 New Tone Stack Configurations

The tone control is an extremely simple low pass filter, and could be replaced with something more sophisticated. The Mu Blaster has enough output that some insertion loss from a passive tone stack is within acceptable design margins for a very loud booster. In addition, the second gain stage has a buffered output thanks to Q5. I think a passive James stack could work very nicely. An FMV tone stack would also work well if you want to pull back the mids a bit and go all in on an “amp-like” response. Either of these can be built on a small piece of vero board, or even point-to-point on solder lug pots.

To patch in a new tone stack, simply use Lug 1 and Lug 3 of the filter pot as the input and output, respectively. If you need a ground reference, short the pads for C9 and connect the ground to Filter Lug 2.

If you use an active tone control, be sure to add an attenuator before it that corresponds to the maximum boost range to prevent undesirable clipping. For example if you add an active Baxandall with a +12dB boost range, you should attenuate the whole signal by 12dB beforehand.



## 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!