

PROJECT NAME

# PENUMBRA

BASED ON

ZVEX Woolly Mammoth / Mastotron

BUILD DIFFICULTY

■■■■■ Easy

EFFECT TYPE

Bass fuzz

DOCUMENT VERSION

1.0.3 (2025-03-24)

**qion**  
DIY GUITAR EFFECTS

## PROJECT SUMMARY

A full-frequency silicon fuzz, originally designed for bass but just as suited for guitar. Famously used by Chris Wolstenholme of Muse.



## IMPORTANT NOTE

This documentation is for the **kit** version of the project. If you purchased the PCB by itself, please use the [PCB-only version](#) of the documentation instead. The circuit is the same, but the instructions are completely different due to the specialized parts and assembly methods used in the kit.

# TABLE OF CONTENTS

---

1	Project Overview
2	Table of Contents
3	Introduction
4	Packing List
5	Packing List (Cont.)
6	Tools Needed
7	Component Identification
8	Hardware Identification
9	PCB Assembly Overview
10	Resistors & Diode
11	Transistors
12	Capacitors (Non-Polarized)
13	Wire Headers
14	Capacitors (Polarized)
15	Transformer
16	Footswitch PCB
17	Input/Output PCB
18	Enclosure Layout: Panel Mounts
19	Enclosure Layout: Panel Mounts (Cont.)
20	Enclosure Layout: Main & Footswitch PCBs
21	Enclosure Layout: Input/Output PCB
22	Final Assembly & Testing
23	Schematic
24	Full Parts List
25	Troubleshooting Information
26	Support & Resale Terms
27	Legal Information & Document Revisions

# INTRODUCTION

---

If this is your first pedal, welcome to the hobby and thank you for choosing Aion FX. You've just joined a community of over 100,000 people around the world with a passion for building homemade noise machines using obsolete electronics technology, and we're glad to have you!

If you've done this before, it's great to see you again and we're confident you'll find this build experience an enjoyable one.

Aion FX kits are designed to empower anyone to build a high-quality pedal, no matter the skill level. The pedalbuilding hobby has traditionally had a steep learning curve, but don't be overwhelmed—we've done all the hard work for you. All you need to do is follow these instructions and you'll be on your way to transforming your tone.

There are a few things to go over before you get started.

- **You're going to have to get your hands dirty**—there's no way around it. Nothing here comes preassembled, and you'll have to learn the skills to put it all together. This document will walk you through everything you need, but be prepared to learn a few things along the way.
- **This will take time.** Plan on about two hours start to finish. It may take even longer if it's your first time building. Don't rush it. If you find yourself getting frustrated or overwhelmed, take a break and come back in a couple of hours or the next day.
- **No direct technical support is offered.** There are several DIY forums and Facebook groups with thousands of members who enjoy troubleshooting and teaching. But please be sensitive to the fact that the staff at Aion FX is minimal, and every minute spent helping individuals in private is time that can't be spent on new project development.
- **There is no implied guarantee of a final product.** Aion FX provides the ingredients and the recipe, but you are responsible for putting everything together to make it work. We've tried to make the process as clear and accessible as possible, but it must be expressly stated that purchasing the kit is not a guarantee that you will end up with a working pedal.

It's recommended to read through all of the instructions before you start, particularly if you've never built a pedal before. If you familiarize yourself with the entire process ahead of time and you know what the goal looks like, each step will make more sense.

Now, on to the fun stuff!

# PACKING LIST

---

This is a list of all the parts that are included with the kit, grouped by value. For a list of all the parts based on their PCB part numbers, please see page 30.

If you find that any parts are missing or damaged, please fill out the [Missing Parts](#) form.

## Film Capacitors

NAME	QTY
1n	1
10n (0.01)	2
33n (0.033)	1
68n (0.068)	1
220n (0.22)	2
1uF	1

## Electrolytic Capacitors

NAME	QTY
100uF	4

## MLCC Capacitors

NAME	QTY
100n (marked "104")	1

## Diodes

NAME	QTY
1N5817	1

## Resistors

NAME	QTY
100R	1
1k	1
2k2	1
5k1	1
10k	3
22k	1
56k	1
100k	1
1M	2
2M2	1

*Note: In January 2025, R6 changed from 1k to 100k and LEDR changed from 4k7 to 10k. Kits shipped after January 22 will use the new values.*

## Transistors

NAME	QTY
J112 or PN4303	1
2N3904	2

## Transformer

NAME	QTY
42TL019	1

## PACKING LIST (CONT.)

---

### Potentiometers

NAME	QTY
5kC	1
10kB	2
500kB	1
Dust cover	4
Knob, 3/4"	2
Knob, 1/2"	2
Mounting nut, potentiometer, 0.44"	4
Lock washer, potentiometer, 0.5"	4
Outer washer, potentiometer, 0.475"	4

### Other

NAME	QTY
LED bezel	1
LED, blue	1
9V battery snap	1
DC jack	1
Input/output jack	2
Mounting nut, jack, 0.54"	4
Outer washer, jack, 0.6"	2
Lock washer, jack, 0.5" (thin)	2
Enclosure	1
Enclosure screws	4
PCB, main circuit	1
PCB, footswitch	1
PCB, input/output/DC	1

### Switches

NAME	QTY
Toggle switch, SPDT on-off-on	1
Toggle switch, DPDT on-on	1
Mounting nut, toggle switch, 0.36"	2
Lock washer, toggle switch, 0.4"	2
Dress nut, toggle switch, 0.375"	2
Stomp switch, 3PDT	1
Mounting nut, stomp switch, 0.6"	2
Lock washer, stomp switch, 0.6"	1
Dress nut, stomp switch, 0.77"	1

### Wiring

NAME	QTY
3-strand wire assembly, 70mm	2
4-strand wire assembly, 108mm	1
3-pin wire assembly header	2
4-pin wire assembly header	1

## TOOLS NEEDED

---



### SOLDERING IRON

Temperature-adjustable is recommended. The optimum soldering temperature is 700-725° F (371-385° C) for leaded solder, or 750° F (400° C) for lead-free.



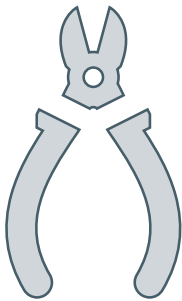
### SOLDER

Preferably 63/37 or 60/40 leaded solder. Lead-free is more difficult to use, so if that's the only type you can get, it's best to watch tutorials that are specific to lead-free solder.



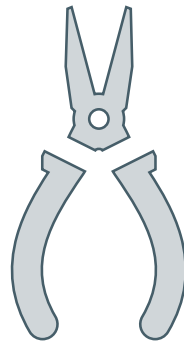
### DIGITAL MULTIMETER (DMM)

Most cheap ones in the \$10-30 range are fine for what we're doing. Make sure it has audible continuity testing (i.e. it beeps at the lowest resistance) and transistor hFE measurement.



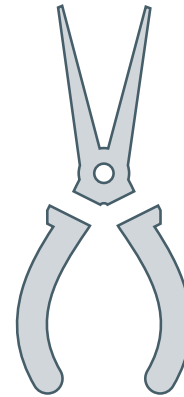
### WIRE SNIPPERS

Also called nippers or wire cutters. The Hakko CHP-170 is the best you can get for less than \$10.



### FLAT-NOSE PLIERS

Many general-purpose uses, but particularly tightening the nuts of pots, switches and jacks. Quicker than changing out sockets on a ratchet.



### NEEDLE-NOSE PLIERS

These are used for bending leads on components and other general uses. Use the smaller type with a tip that's approximately 0.05" (1.25mm) wide.



### SCREWDRIVER (PHILLIPS)

Used for the enclosure screws. Get a powered driver if you'll be building a lot of pedals!



### FLAT SCREWDRIVER (SMALL)

This is used for tightening the set screws on the knobs. The tip should be no more than 0.1" (2.5mm) wide.



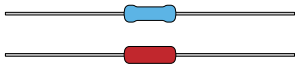
### RUBBER BAND

Yes, a plain old rubber band. This is used to tighten the dress nut to avoid scratching or denting it (which can happen with metal tools).

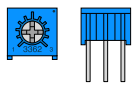
# COMPONENT IDENTIFICATION

If you've never built a pedal before, you'll need to know what all the components are. These are shown actual size. (Not all of these types of components may be part of this kit.)

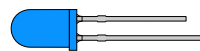
**RESISTOR**



**TRIMMER POTENTIOMETER**



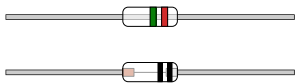
**LED**



**SILICON DIODE**



**GERMANIUM DIODE**



*These are very fragile, so be careful when handling them.*

**RECTIFIER DIODE**



**SCHOTTKY DIODE**

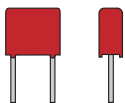


*Some Schottky diodes also look like this.*

**ZENER DIODE**



**FILM CAPACITOR**



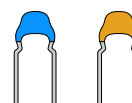
*Not polarized. Color may vary by brand and type.*

**ELECTROLYTIC CAPACITOR**



*Polarized. The negative side is marked.*

**MLCC**



*Not polarized. MLCC stands for "multi-layer ceramic capacitor."*

**TANTALUM CAPACITOR**



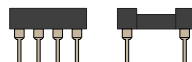
*Polarized. The positive side is marked.*

**OP-AMP / IC**

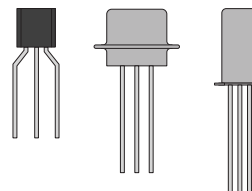


*Charge pumps and delay chips also look like this. They may have more than 8 legs.*

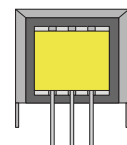
**IC SOCKET**



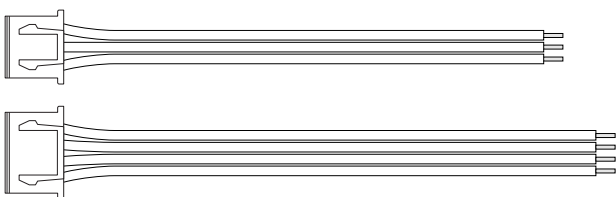
**TRANSISTOR OR JFET**



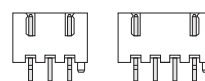
**TRANSFORMER**



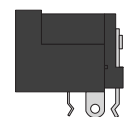
**WIRE ASSEMBLY**



**WIRE ASSEMBLY HEADER**



**DC JACK**

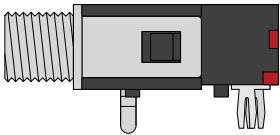
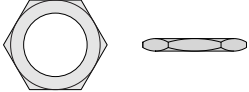
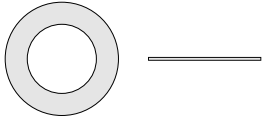
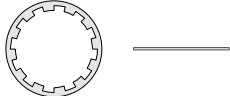
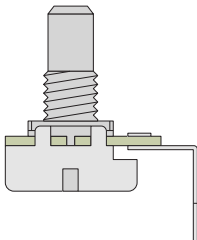
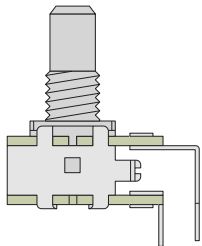
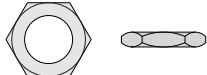
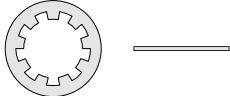
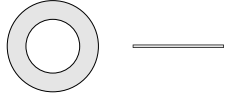

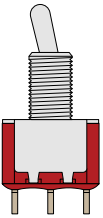
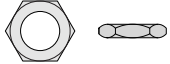
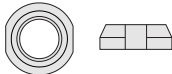
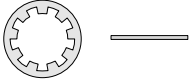
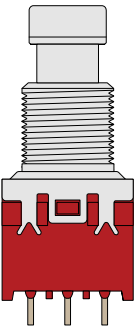
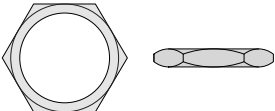
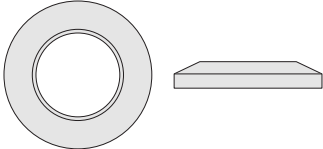
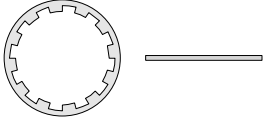


**LED BEZEL**



# HARDWARE IDENTIFICATION

The hardware comes unassembled, so you'll need to sort & identify each of the pieces. The diagrams below are actual size, so you can set them against the printed page to identify them if needed.

<b>I/O JACK</b> 	<b>MOUNTING NUT</b>  DIAMETER: 0.54" / 13.7mm	<b>OUTER WASHER</b>  DIAMETER: 0.6" / 15.2mm	<b>LOCK WASHER</b>  DIAMETER: 0.5" / 12.7mm
<b>POTENTIOMETER (SINGLE)</b> 	<b>POTENTIOMETER (DUAL)</b> 	<b>MOUNTING NUT</b>  DIAMETER: 0.44" / 11.2mm	<b>LOCK WASHER</b>  DIAMETER: 0.5" / 12.7mm
		<b>OUTER WASHER</b>  DIAMETER: 0.475" / 12mm	<b>KNOB</b> 
<b>TOGGLE SWITCH</b> 	<b>MOUNTING NUT</b>  DIAMETER: 0.36" / 9.1mm	<b>DRESS NUT</b>  DIAMETER: 0.375" / 9.5mm	<b>LOCK WASHER</b>  DIAMETER: 0.4" / 10.1mm
<b>FOOTSWITCH</b> 	<b>MOUNTING NUT</b>  DIAMETER: 0.6" / 15.2mm	<b>DRESS NUT</b>  DIAMETER: 0.77" / 19.6mm	<b>LOCK WASHER</b>  DIAMETER: 0.6" / 15.2mm



# PCB ASSEMBLY OVERVIEW

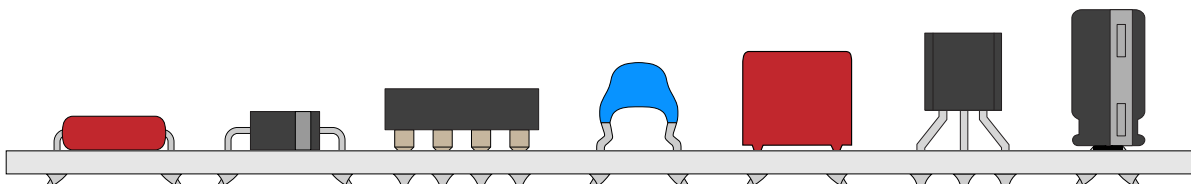
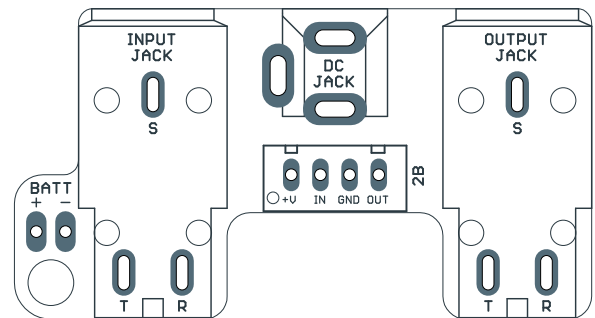
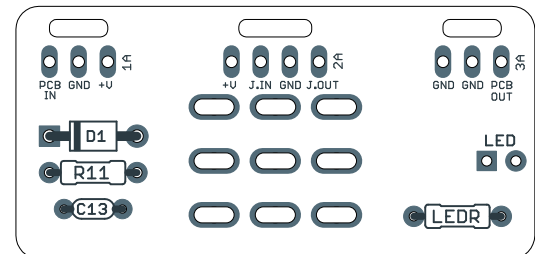
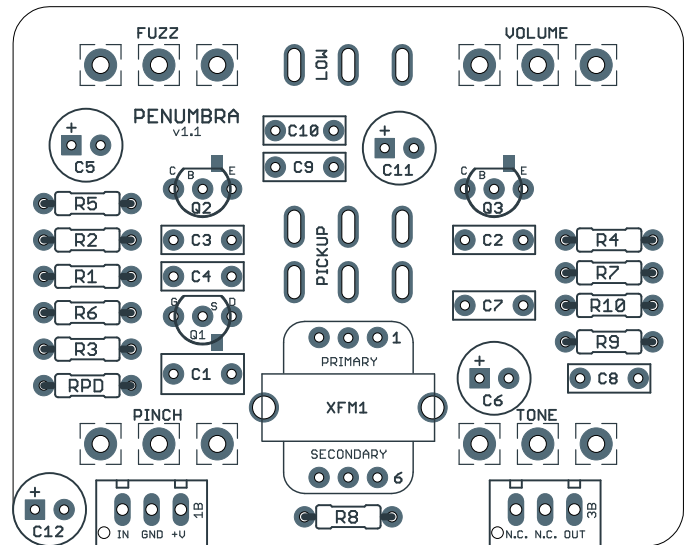
Now it's time to start building!

Before you begin, separate the PCBs into individual boards and break off the tabs from each using needle-nose or flat-head pliers. You should be left with the three PCBs shown to the right.

The general principle for PCB population is that you want to work in layers from shortest components (i.e. lowest-profile) to tallest so that when the PCB is upside-down, everything is making contact with the work surface and is held in place.

Generally speaking, you should populate the components in this order:

1. Resistors
2. Diodes
3. MLCC capacitors
4. Film capacitors
5. Transistors
6. Electrolytic capacitors
7. Transformer



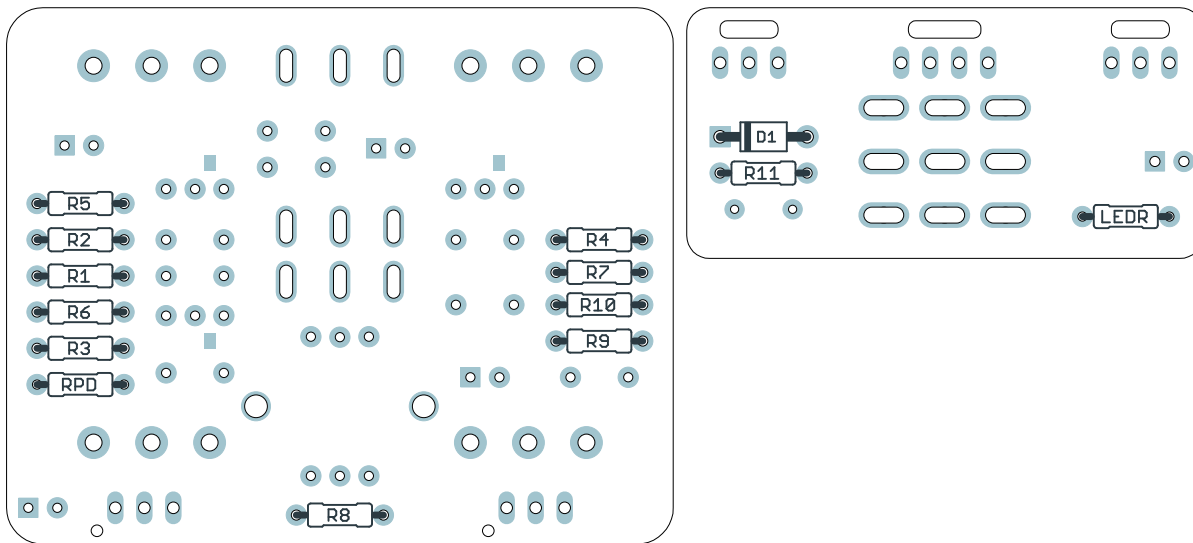
## RESISTORS & DIODE

PART	VALUE
R1	1M
R2	1M
R3	10k
R4	1k
R5	56k

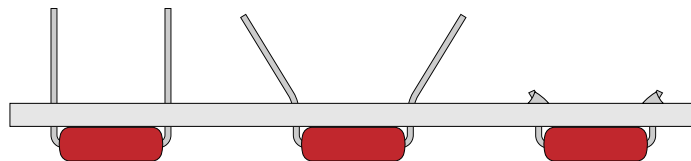
PART	VALUE
R6	100k
R7	2k2
R8	22k
R9	10k
R10	5k1

PART	VALUE
R11	100R
RPD	2M2
LED R	10k
D1	1N5817

*Note: In January 2025, R6 changed from 1k to 100k and LED R changed from 4k7 to 10k. Kits shipped after January 22 will use the new values.*



Using the parts list above, populate the resistors by pushing them through the holes and bending the leads outward at an angle to hold them in place. Resistors are not polarized, so they will work in any direction. Turn the board upside-down to keep the components held in place while you solder.



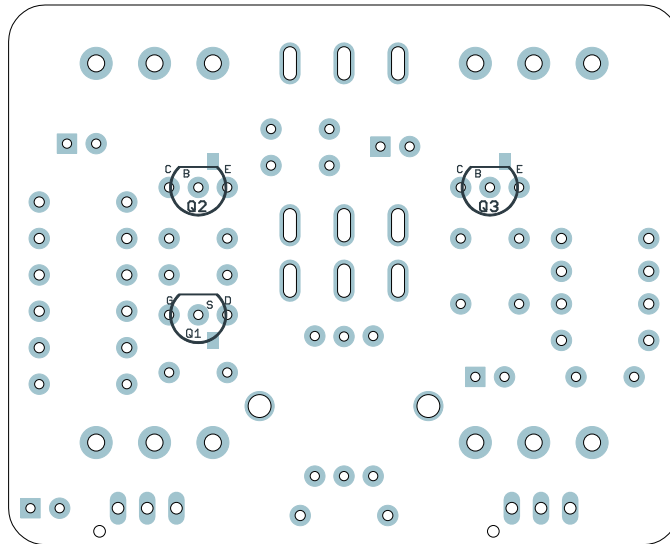
You'll use this same technique for most of the other components as well.

If it gets too crowded, just flip the board and solder everything you've done so far, then cut the leads using the wire snippers to make room for more.

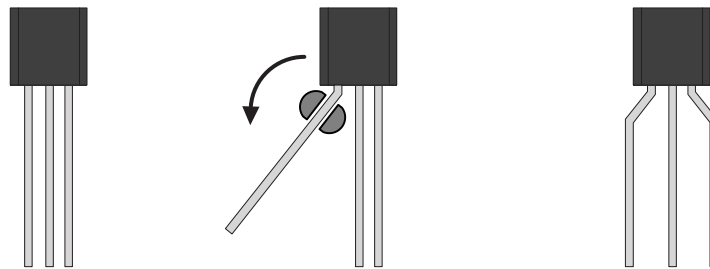
If this is your first time soldering, watch tutorial videos on YouTube and make sure you get it down before you begin. You don't want to practice or experiment on this board!

# TRANSISTORS

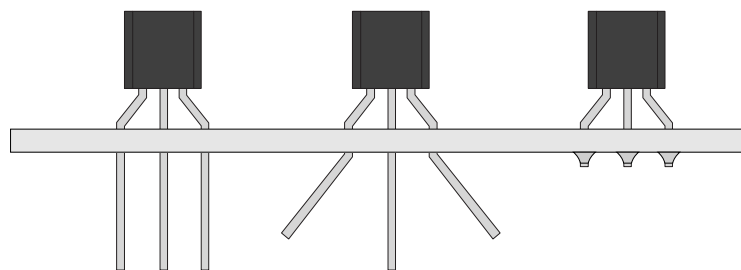
PART	VALUE
Q1	J112 or PN4303
Q2	2N3904
Q3	2N3904



Now we'll do the transistors. If the legs are not already bent into 0.1" spacing, use your needle-nose pliers to bend the outer two legs as shown.



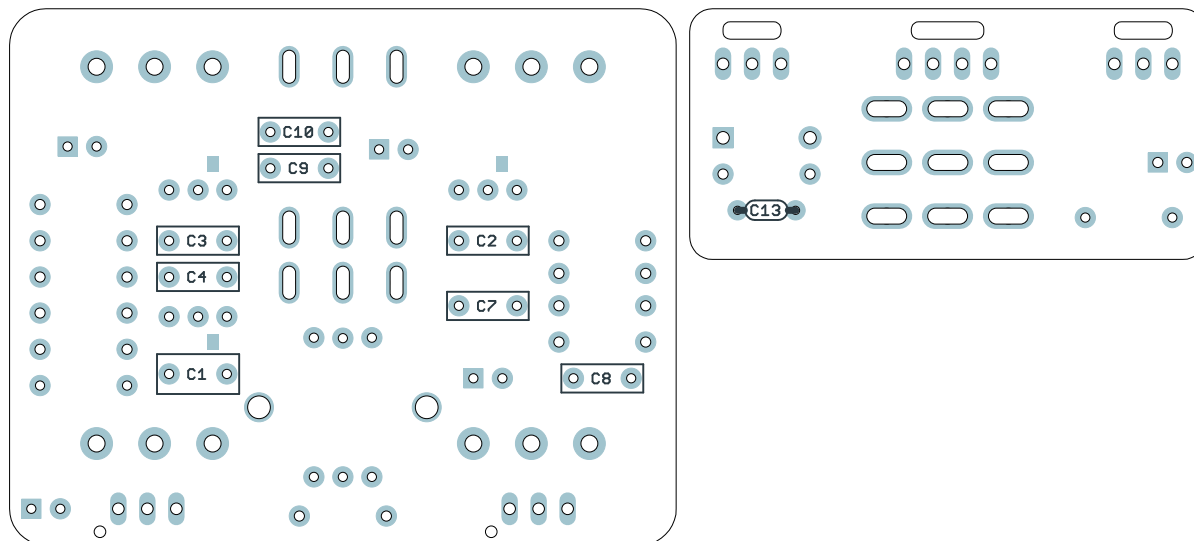
Bend the outer leads of the transistors to hold them in place on the board. Then, solder them and clip the leads. The two 2N3904 transistors are interchangeable and it does not matter which one is used for Q2 and which one is for Q3.



## CAPACITORS (NON-POLARIZED)

PART	VALUE
C1	1uF
C2	1n
C3	220n (0.22)
C4	10n (0.01)
C7	10n (0.01)

PART	VALUE
C8	220n (0.22)
C9	33n (0.033)
C10	68n (0.068)
C13	100n MLCC



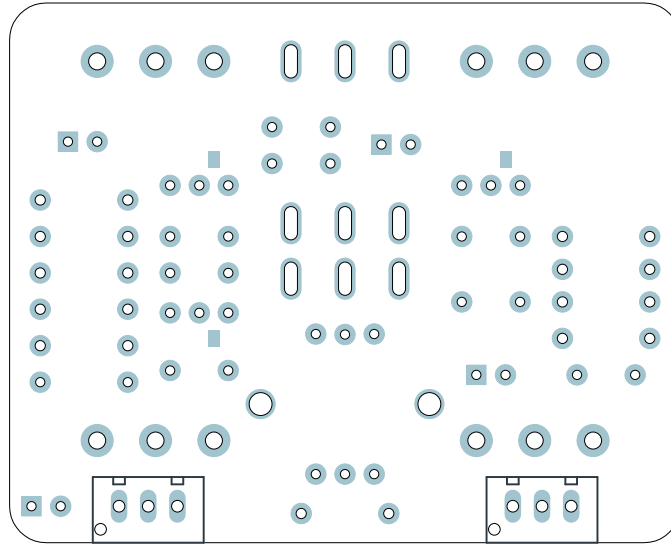
After the transistors come the box film and MLCC capacitors. These are all several different heights, but there aren't as many, so just do them all at once. Bend the leads at an angle to hold them in place.

MLCCs and box capacitors are not polarized and will work in any direction. To keep things neat, though, it's best to put them all facing the same way so the values can easily be read.

**Note:** The blue box film capacitors (C2 and usually C4 and C7) have the value printed on the top, while the red capacitors have the value on the side. The text on the side of the blue capacitors is not related to the value and can be ignored.

## WIRE HEADERS

---

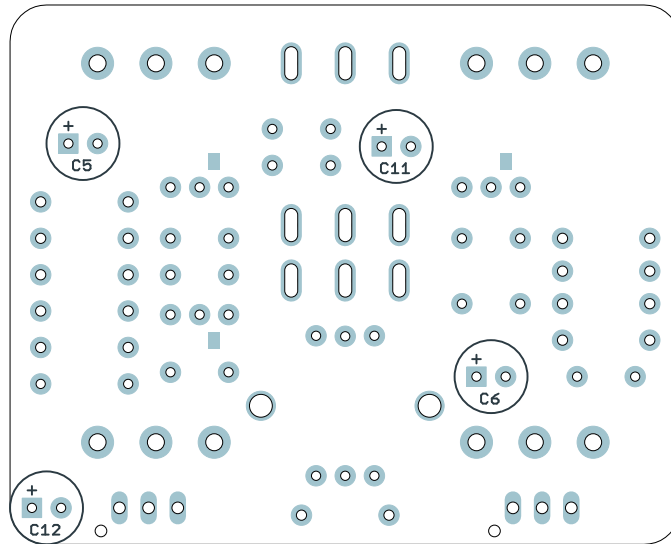


Install the two 3-pin headers (wire connectors) as shown above. These have a polarity pin, so as long as they are pressed all the way down, there's only one possible way to install them. They do fit pretty tightly in the holes, though, so press firmly.

There's also a 4-pin header on the I/O board that we will do in a later step.

## CAPACITORS (POLARIZED)

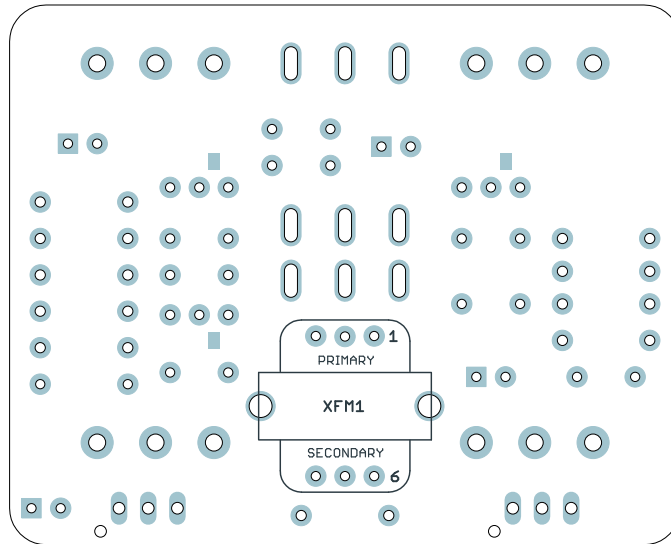
PART	VALUE
C5	100uF
C6	100uF
C11	100uF
C12	100uF



Populate the electrolytic capacitors. They are polarized (i.e. they will only work in one direction), so note the vertical mark that indicates the negative side. The longer leg is positive and fits in the square pad.

# TRANSFORMER

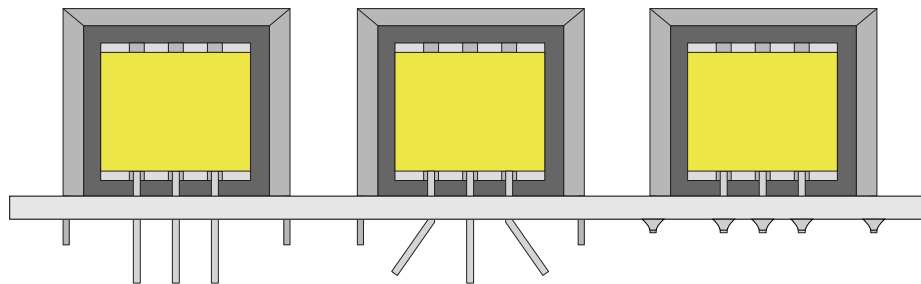
PART	VALUE
XFM1	42TL019



The transformer is the last of the on-board components. Before installing it to the PCB, scrape off any wax on the leads. You don't need to be terribly thorough since it will melt away with the heat of the soldering iron, but we don't want the wax getting in the way of a good solder joint.

The primary and secondary coils of the transformer have different properties, so the transformer must be facing the right direction or it won't work correctly. In this case, the primary is marked with a "P" and the top of the nylon bobbin is colored black on that side. The primary side should line up with the "PRIMARY" label on the silkscreen (in other words, facing toward the top of the PCB).

As with earlier components, first bend the six legs to hold it in place as shown in the diagram below. Solder the legs and the two outer tabs and then clip them short.

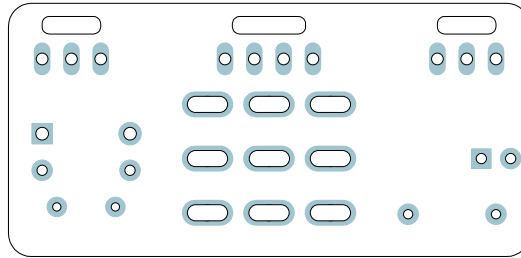


## FOOTSWITCH PCB

### PARTS

3-strand wire assembly (2)

4-strand wire assembly



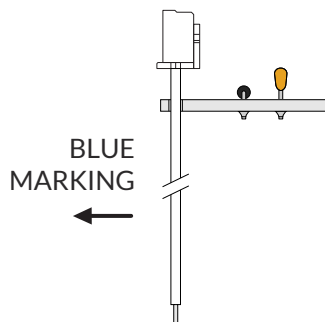
Next, it's time to finish up the footswitch board. You should have done most of the on-board components on this board in a previous step, but if not, go back and do those.

There will be one longer assembly with 4 wires and two shorter ones with 3 wires. The longer one goes in the middle and the shorter ones go on the left and right sides. The wire assemblies should then be soldered to the footswitch board as shown.

### STEP 1

First, thread the wire through the strain-relief slots, with the blue side facing outward and the PCB's previously-installed components facing up.

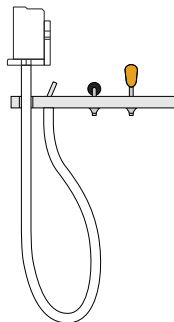
For now, pull it through as far as it can go.



### STEP 2

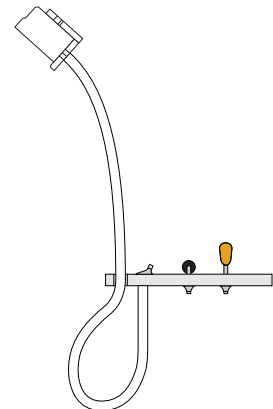
Next, bend the wires back upward and fit the ends of the wires into the solder pads.

On the top side of the PCB, bend the exposed wires backward so it holds the wire in place. Pull the header back up through the slot partway.



### STEP 3

Then, solder the wires from the top. This is the trickiest part of the whole build. You want to solder the pads without touching the iron to the wires themselves and risking burning through the insulation. It helps to use a sharp or narrow tip on the soldering iron.



Once all three wire assemblies are soldered, set the footswitch PCB aside. We'll solder the actual footswitch and LED in a later step.



# INPUT/OUTPUT PCB

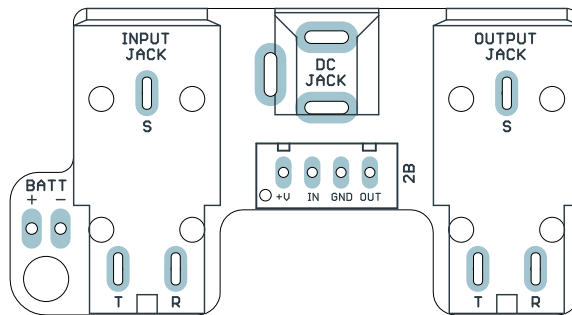
## PARTS

Input & output jacks

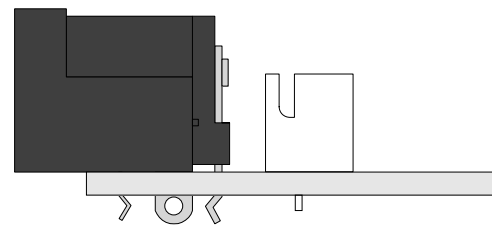
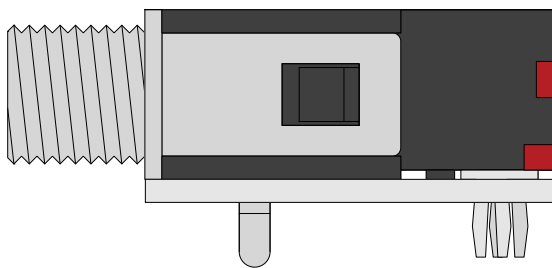
DC jack

Wire header

9V battery snap



Almost done! Get the two input/output jacks, the DC jack and the wire header and snap them in place. The PCB is designed for them to fit securely, so you can do them all at once before flipping and soldering.

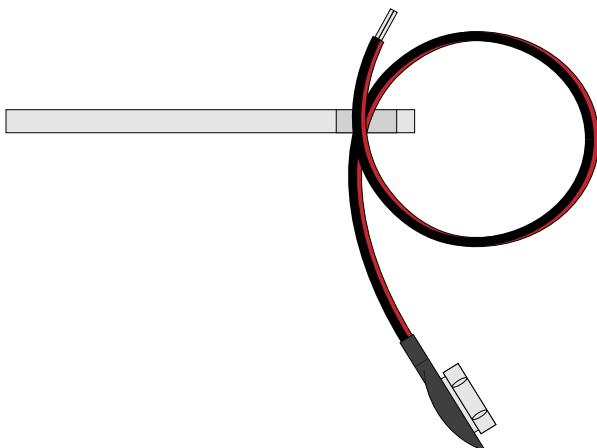


After you've soldered everything, make sure to **snip the leads on the I/O jacks as close as possible to the PCB**. There's not a lot of clearance between the bottom of this board and the top of the main PCB once everything is in place, and you don't want the pins to short against anything on accident.

Next, we'll hook up the 9V battery connector. **This is optional.** Not everyone uses batteries. But, if you do, this pedal should last a long time on a single 9V so you won't need to change it very often.

## STEP 1

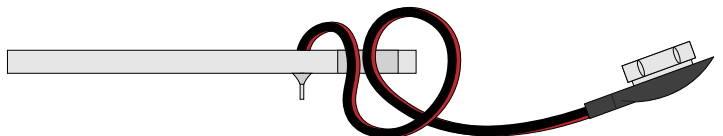
Thread the battery snap leads through the strain-relief hole twice so it forms a single loop.



## STEP 2

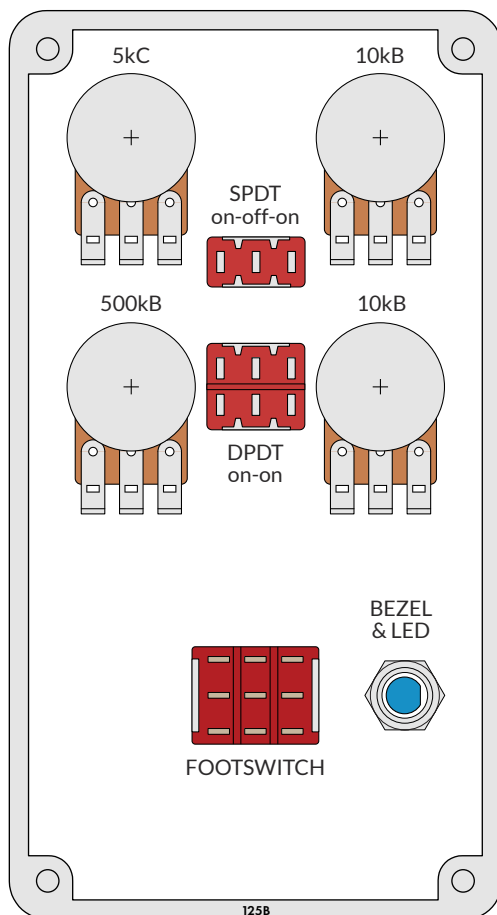
Bend the exposed wires back down and solder them into the pads. Red is positive (+), black is negative (-). After soldering, pull it tight.

For even more strain relief, you can thread the snap through the loop to form a knot. (not shown)

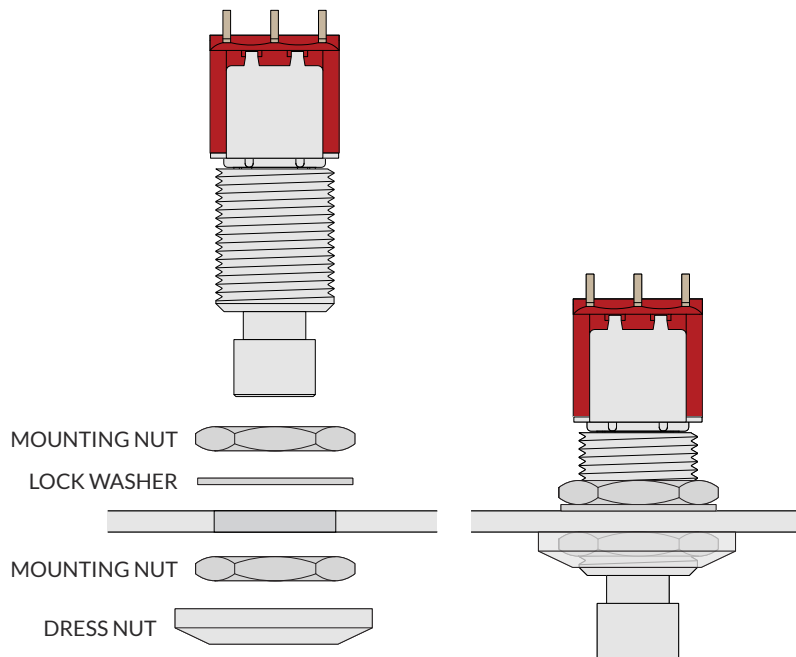


# ENCLOSURE LAYOUT: PANEL MOUNTS

Attach the hardware to the enclosure as shown. (The I/O board is done in a later step.)



## FOOTSWITCH



The dress nut fits over the top of the mounting nut and is for aesthetic purposes only. Wrap a rubber band around it to use as a grip when tightening. Do not use metal tools on it or you run the risk of scratching or denting it.

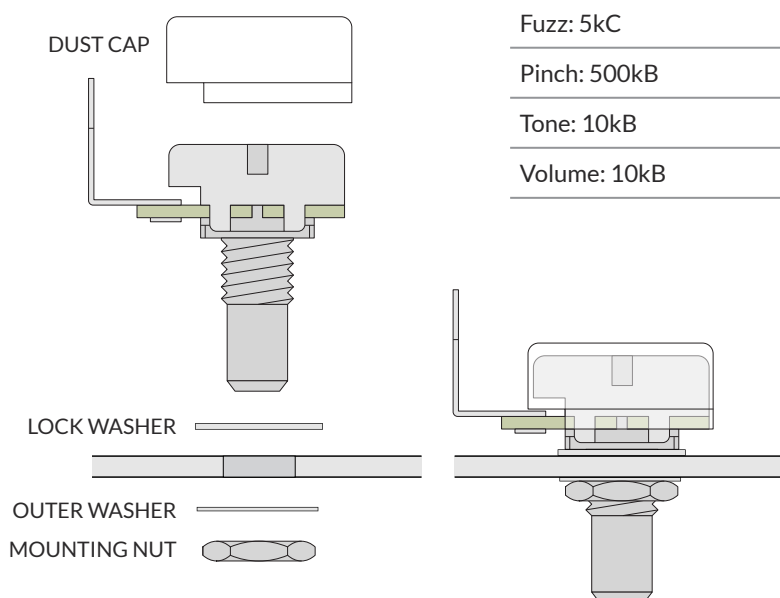
## POTENTIOMETERS

Fuzz: 5kC

Pinch: 500kB

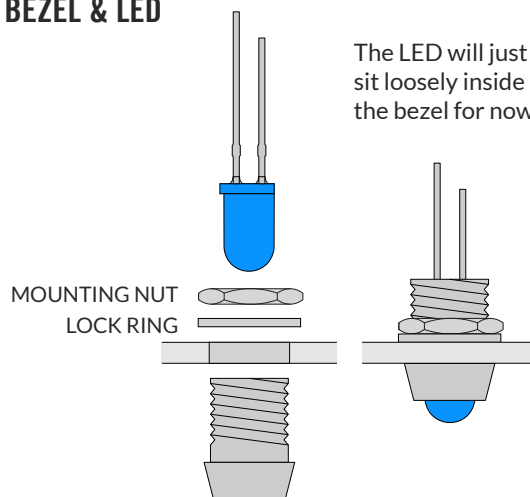
Tone: 10kB

Volume: 10kB



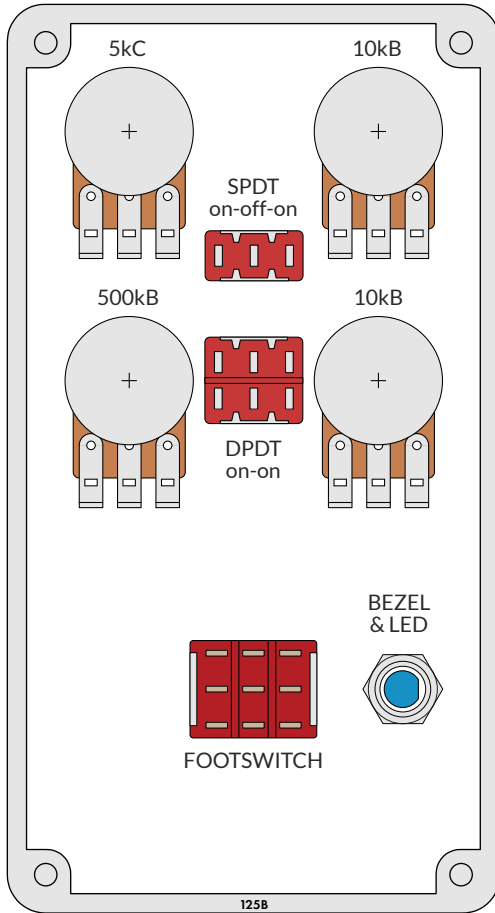
## BEZEL & LED

The LED will just sit loosely inside the bezel for now.

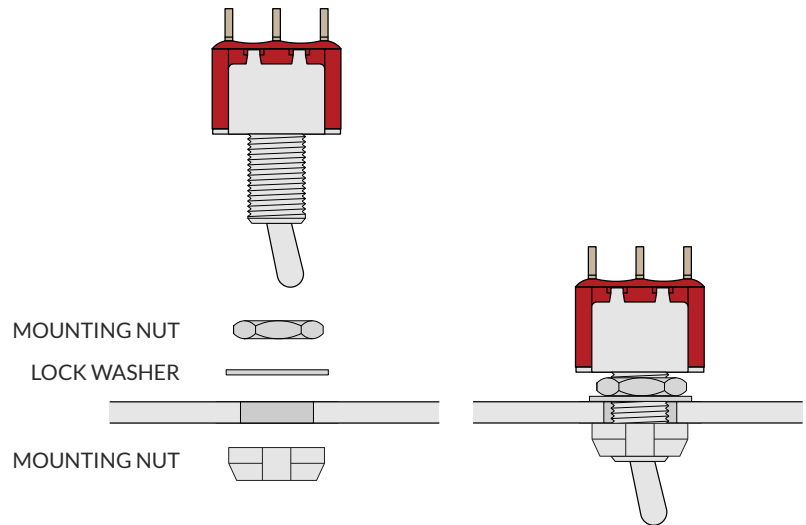


You'll need to hold the bezel in place when tightening the nut. The top of the bezel is fairly sharp, so try using a rubber band for grip instead of pressing your finger against the bottom.

## ENCLOSURE LAYOUT: PANEL MOUNTS (CONT.)

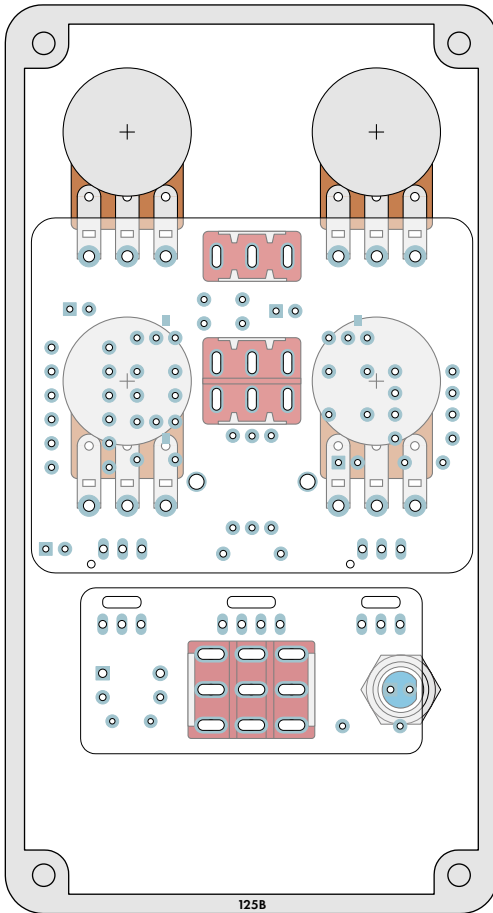


### TOGGLE SWITCHES



The dress nut acts as a mounting nut, unlike the footswitch dress nut. Use flat-nose pliers on the flat sides of the nut to tighten securely.

## ENCLOSURE LAYOUT: MAIN & FOOTSWITCH PCBs



After all the components are affixed to the enclosure as shown on the previous page, place the main PCB on top of the potentiometers and toggle switches as in the diagram to the left.

You may need to adjust the position of the potentiometers and toggle slightly if they are not aligned straight.

Once all of the pins are through their holes and the PCB is laying flat, solder each of the pins from the top. Be careful not to touch any of the surrounding components with the soldering iron.

After you've finished soldering the pots, **clip the leads as close as you can to the main PCB**. This is more important with the two uppermost pots because the input/output PCB overlaps them and you need to avoid any of the components shorting. (The toggle switch lugs do not need to be clipped.)

Next, move to the footswitch board and solder the 3PDT switch.

The LED is last. Before soldering the LED, double-check to **make sure the flat side of the LED is facing to the right**, as shown in the diagram, and that the short leg is coming through the pad on the right. It won't work if it's turned the other way. Then, clip the leads of the LED.

### Why solder everything inside the enclosure before testing it?

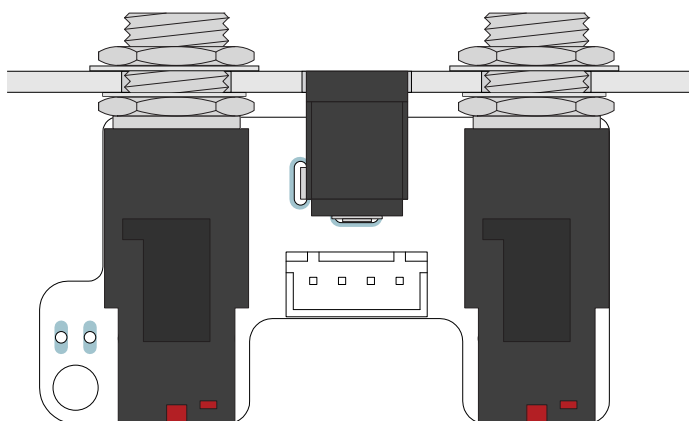
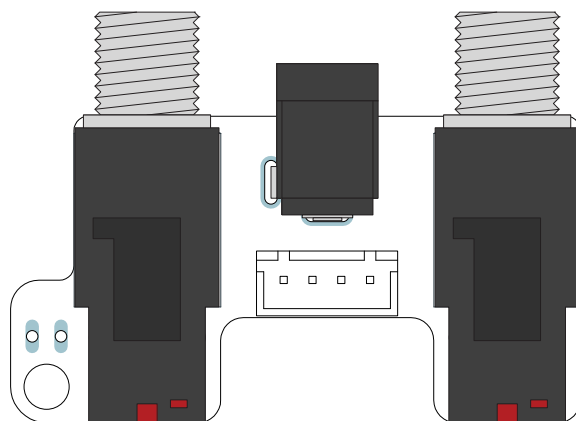
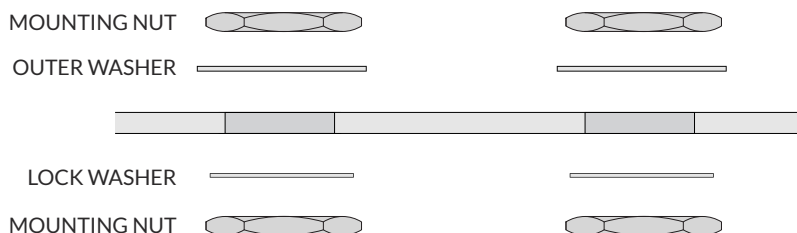
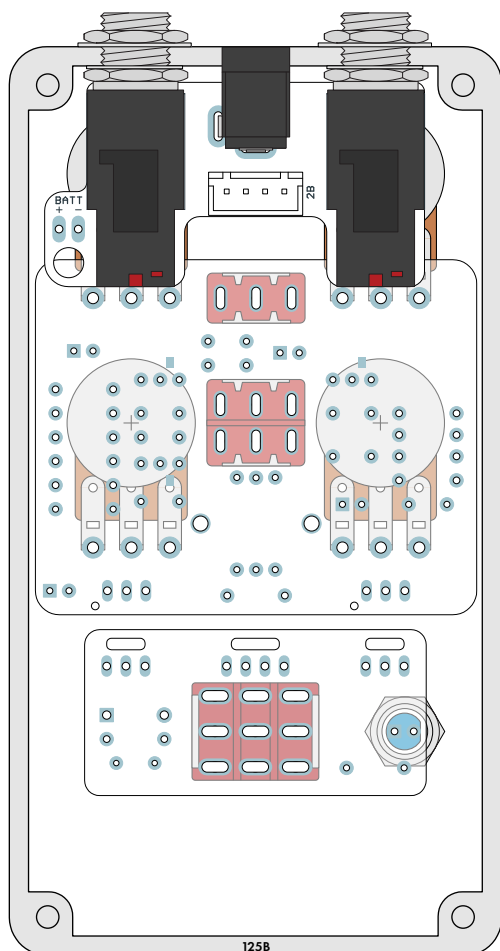
"Rock it before you box it" is conventional wisdom in pedalbuilding, and you'll often hear it recommended that builders should test the circuit before putting everything inside the enclosure. However, Aion FX projects are designed to be extremely easy to remove from the enclosure for troubleshooting, with no desoldering required—so with these kits, it's actually much easier to "box it before you rock it".

If you've read the documentation carefully and followed all the instructions, there's a good chance you will get it right the first time!

## ENCLOSURE LAYOUT: INPUT/OUTPUT PCB

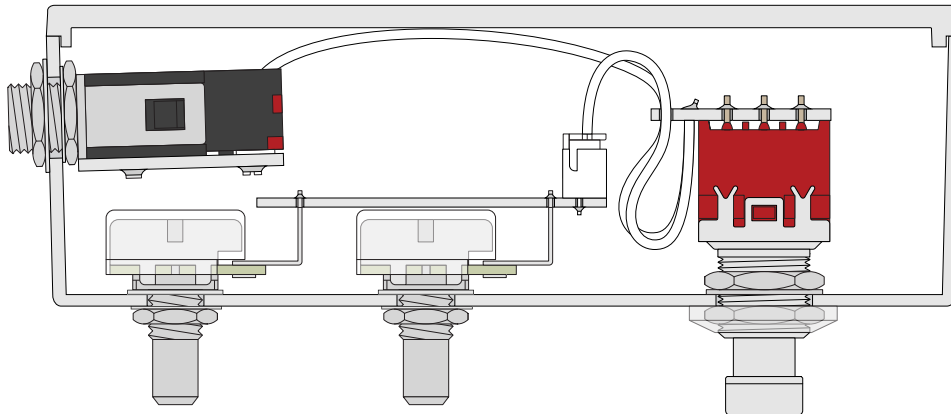
Affix the input/output PCB to the north-facing panel of the enclosure as shown.

Note the use of two mounting nuts on each of the jacks, one inside and one outside. The inner nut acts as a spacer to set the DC jack flush with the outside of the enclosure. The inner nuts should be threaded as far down as they can go.



## FINAL ASSEMBLY & BIASING

After everything is in place, just plug the 3 wire assemblies into their respective headers and make sure they're secure. Here is a cross-section of the inside of the completed pedal.



At this point, you have completed the full circuit as far as the electrons are concerned. Plug in a 9-volt supply and test it out with a guitar and an amplifier.

Test the bypass switch a few times, then start turning the knobs and see if everything sounds OK. If it works, great! If not, don't be discouraged. See page 25 for troubleshooting info.

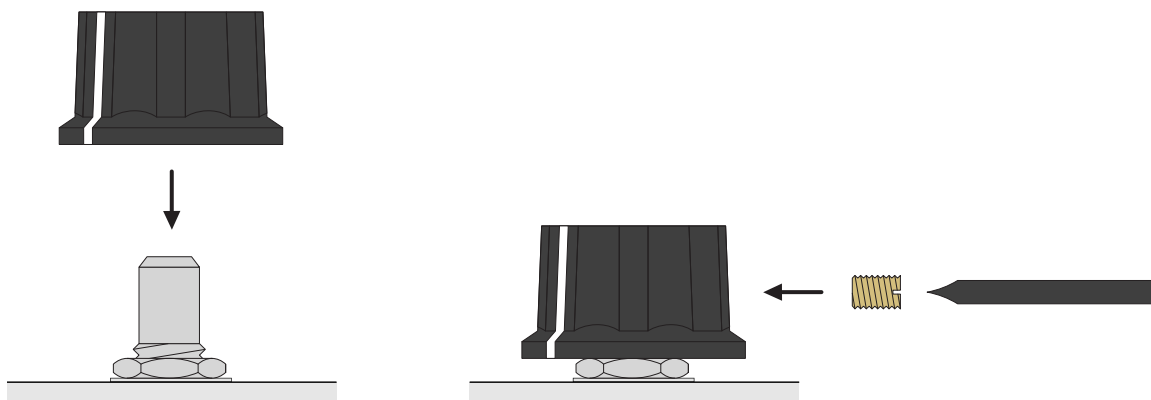
### Finishing touches

Now, just a couple of things for the final assembly. Turn the shafts all fully counter-clockwise, then put on the knob and rotate until the indicator line is aligned with the dot on the enclosure that shows the zero point. Affix the knobs to each of the potentiometer shafts as shown in the diagram below.

Note that the larger knobs are intended for the top row while the smaller 1/2" knobs are for the second row. This allows more room to adjust the toggle switches.

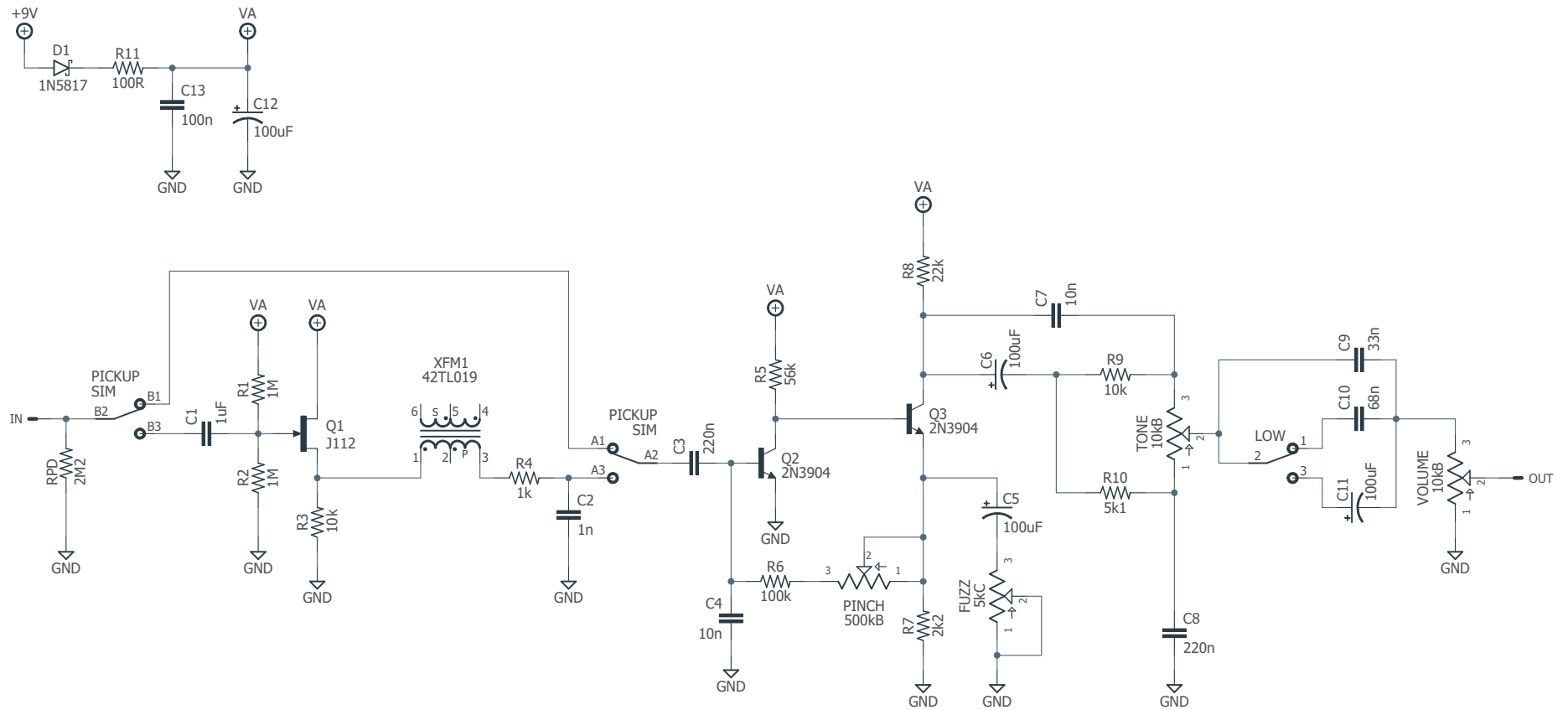
Using a small flat-head screwdriver (no more than 0.1" / 2.5mm in diameter), firmly tighten the set screw until it presses against the shaft of the potentiometer and holds the knob in place.

Be careful not to over-tighten or you may damage the set screw. But if it's not tight enough, the knob will be more likely to fall off or lose its alignment with the markings on the enclosure.



Last, just close the panel on the back using the four screws. That's it!

# SCHEMATIC



# FULL PARTS LIST

---

## Resistors

PART	VALUE
R1	1M
R2	1M
R3	10k
R4	1k
R5	56k

PART	VALUE
R6	100k
R7	2k2
R8	22k
R9	10k
R10	5k1

PART	VALUE
R11	100R
RPD	2M2
LEDR	10k

## Capacitors

PART	VALUE
C1	1uF
C2	1n (0.001)
C3	220n (0.22)
C4	10n (0.01)
C5	100uF electro
C6	100uF electro
C7	10n (0.01)

PART	VALUE
C8	220n (0.22)
C9	33n (0.033)
C10	68n (0.068)
C11	100uF electro
C12	100uF electro
C13	100n MLCC

## Transistors

PART	VALUE
Q1	J112 or PN4303
Q2	2N3904
Q3	2N3904

## Diodes

PART	VALUE
D1	1N5817

## Potentiometers

PART	VALUE
Fuzz	5kC
Pinch	500kB
Tone	10kB
Volume	10kB

## Switches

PART
SPDT on-off-on
DPDT on-on
3PDT stomp

## Transformer

PART
42TL019



# TROUBLESHOOTING INFORMATION

---

If you finish building the kit and find that it doesn't work right, we've written a separate in-depth [Troubleshooting Guide](#) that applies to all of our kits. The main troubleshooting process is covered there. Here you will find information specific to this kit that will help with that process.

## Voltages

The following voltages are taken from our prototype unit using a **9.60V** supply. Your measured voltages won't be exactly the same due to variance in power supplies and component tolerances. However, if you see anything more than +/-0.5V from the listed voltages, it's a good indicator of an issue, and the exact voltages can help narrow it down.

The JFET pins D-G-S and transistor pins E-B-C are labeled on the PCB, so use the silkscreen as a reference when measuring the voltages.

### Q1

PIN	VOLTAGE
D	9.28
S	7.16
G	4.44

### Q2

PIN	VOLTAGE
E	0
B	0.62
C	1.52

### Q3

PIN	VOLTAGE
E	0.70–0.90 <i>(see note)</i>
B	1.50
C	0.95

*Note: The Q3 emitter voltage varies depending on the position of the "Pinch" knob.*

## SUPPORT

---

**Aion FX does not offer direct support for these projects beyond the provided documentation.**

Replacements and refunds cannot be offered unless it can be shown that the circuit or documentation are in error or that the included components are non-functional.

### Where to get help

The three best places to ask for help are the [DIY Stompboxes forum](#), the [DIY Stompboxes Facebook group](#), and the [r/diypedals subreddit](#). These communities have more than 150,000 members between them and they are very accommodating to new builders.

When posting a troubleshooting request, always include the following:

1. A thorough description of the problem you are experiencing
2. A photo of the inside of the pedal
3. A list of all the measured voltages of each of the pins, described on the previous page

While we cannot offer direct, private support, you may send a link to your public troubleshooting thread to Aion FX using the contact form on the website. There is no guarantee that we will be able to join the discussion and help solve your problem, but this improves the chances.

It benefits the whole community if the troubleshooting process is public because then people who have the same issue in the future may come across it when searching. And if you do get help, remember to pay it forward! The best way to learn new skills is to help others. Even if you've only built one pedal, you have more experience than someone who is brand new, so you have something to offer.

## RESALE TERMS

---

These kits may be used for commercial endeavors in any quantity unless otherwise noted. It's okay to sell individual builds locally or online, or even to offer a service to build pedals based on these kits.

No direct attribution is necessary, though a link back is always greatly appreciated. The only usage restriction is that you cannot "goop" the PCB or otherwise obscure the source. In other words: you don't have to go out of your way to advertise the fact that you use Aion FX kits, but please don't go out of your way to hide it. The guitar effects industry needs more transparency, not less!

## LEGAL INFORMATION

---

Woolly Mammoth™ and Mastotron™ are trademarks of Zachary Vex d/b/a Z.Vex Effects.

Any use of trademarks is for comparative advertising purposes only under fair use. It is not an endorsement of this product by the trademark holders.

These kits are intended to be built by the customer. Aion FX is not responsible for language that may be used by the customer in the marketing or resale of the finished product.

All content and graphics in this document are original works and are copyrighted by Aion FX and may not be used without permission.

## DOCUMENT REVISIONS

---

### 1.0.3 (2025-03-24)

Added J112 as an alternate part for Q1.

### 1.0.2 (2025-01-23)

- Added reference voltages for troubleshooting to page 25.
- Updated R6 from 1k to 100k to shift the range of the Pinch control by ~20%.
- Updated LEDR (LED current-limiting resistor) to 10k to reduce LED brightness.

### 1.0.1 (2024-08-08)

Added link to troubleshooting guide on page 25.

### 1.0.0 (2022-01-21)

Initial release.