

PROJECT NAME

ARES

BASED ON

Echoplex EP-3 Preamp

EFFECT TYPE

Preamp & tone enhancer

BUILD DIFFICULTY

■■■■■ Easy

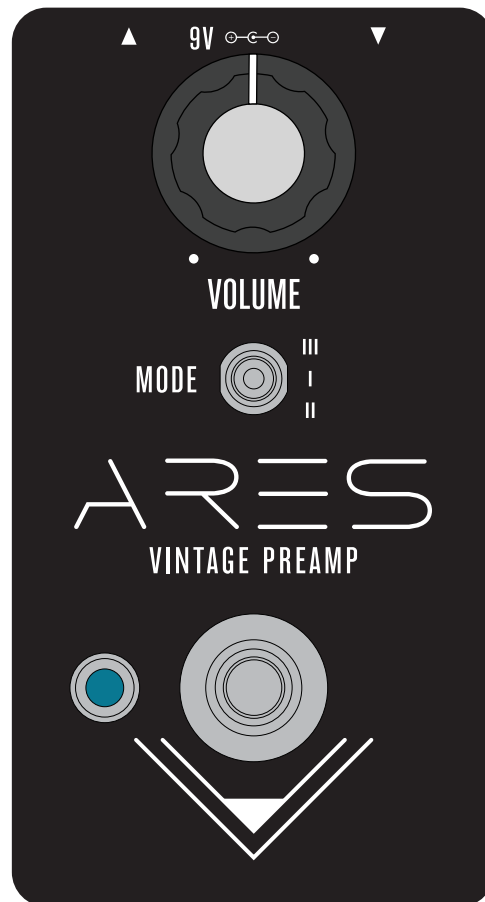
DOCUMENT VERSION

1.0.1 (2024-08-08)

qion
DIY GUITAR EFFECTS

PROJECT SUMMARY

A pedal adaptation of the preamp section of the EP-3 Echoplex tape echo unit. It captures the sound of the EP-3 in bypass mode, favored by many guitarists for its “sweetening” effect on the tone.



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
- 11 Diodes
- 12 Socket & IC
- 13 Transistors
- 14 Capacitors (Non-Polarized)
- 15 Wire Headers & Trimmer
- 16 Capacitors (Polarized)
- 17 Footswitch PCB
- 18 Input/Output PCB
- 19 Enclosure Layout: Panel Mounts
- 20 Enclosure Layout: Panel Mounts (Cont.)
- 21 Enclosure Layout: Main & Footswitch PCBs
- 22 Enclosure Layout: Input/Output PCB
- 23 Final Testing & Assembly
- 24 Usage
- 25 Schematic
- 26 Full Parts List
- 27 Troubleshooting Information
- 28 Support & Resale Terms
- 29 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 26.

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

Film Capacitors

NAME	QTY
22n (0.022)	3
100n (0.1 or “µ1J100”)	1
470n (0.47)	1

Electrolytic Capacitors

NAME	QTY
10uF	2
47uF	1
100uF	1

MLCC Capacitors

NAME	QTY
220pF (marked “221”)	2
100n (marked “104”)	2
470n (marked “474”)	2

Diodes

NAME	QTY
1N5817	3
1N4742A	1

Resistors

NAME	QTY
60R	1
3k3	1
10k	2
15k	1
22k	2
100k	3
220k	2
470k	1
1M	4
2M2	1

ICs

NAME	QTY
LT1054CP	1
8-pin socket	1

Transistors

NAME	QTY
2N5484	1
2N5485	1
PCB adapter pins	6

PACKING LIST (CONT.)

Potentiometers

NAME	QTY
500kB	1
Dust cover	1
Knob, large	1
Mounting nut, potentiometer, 0.44"	1
Lock washer, potentiometer, 0.5"	1
Outer washer, potentiometer, 0.475"	1

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
Mounting nut, toggle switch, 0.36"	1
Lock washer, toggle switch, 0.4"	1
Dress nut, toggle switch, 0.375"	1
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

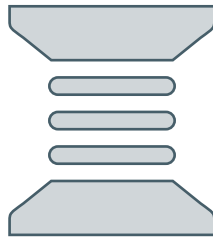
NAME	QTY
3-strand wire assembly, 70mm	2
4-strand wire assembly, 108mm	1
3-pin wire assembly header	2
4-pin wire assembly header	1

TOOLS NEEDED



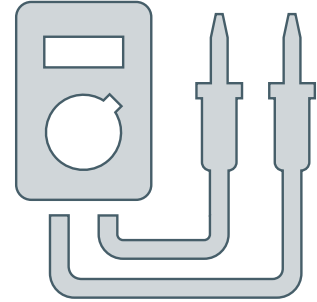
SOLDERING IRON

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.



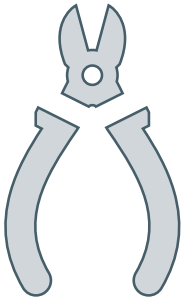
SOLDER

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.



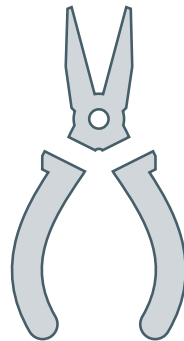
DIGITAL MULTIMETER (DMM)

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.



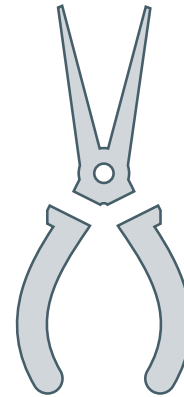
WIRE SNIPPERS

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



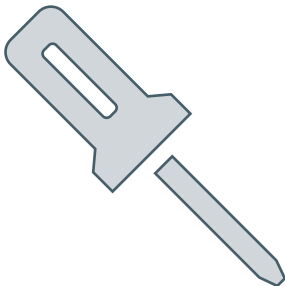
FLAT-NOSE PLIERS

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



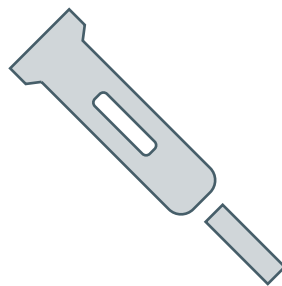
NEEDLE-NOSE PLIERS

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.



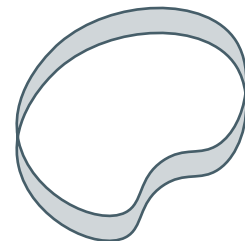
SCREWDRIVER (PHILLIPS)

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.



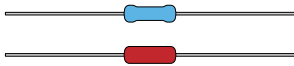
RUBBER BAND

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).

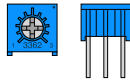
COMPONENT IDENTIFICATION

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.)

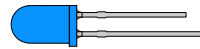
RESISTOR



TRIMMER POTENTIOMETER



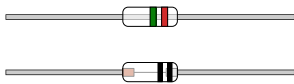
LED



SILICON DIODE



GERMANIUM DIODE



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

RECTIFIER DIODE



SCHOTTKY DIODE

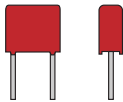


Some Schottky diodes also look like this.

ZENER DIODE



FILM CAPACITOR



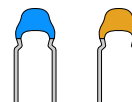
Not polarized. Color may vary by brand and type.

ELECTROLYTIC CAPACITOR



Polarized. The negative side is marked.

MLCC



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

TANTALUM CAPACITOR



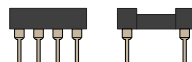
Polarized. The positive side is marked.

OP-AMP / IC

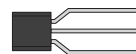


Charge pumps and delay chips also look like this. They may have more than 8 legs.

IC SOCKET



TRANSISTOR OR JFET



Some voltage regulators also look like this.

TRANSISTOR WITH ADAPTER

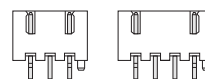


The pins will be soldered to the adapter during kit assembly.

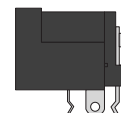
WIRE ASSEMBLY



WIRE ASSEMBLY HEADER



DC JACK

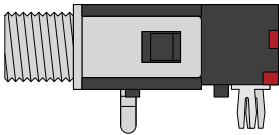
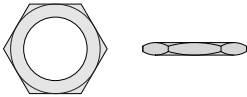
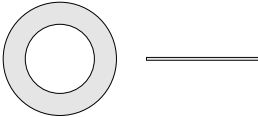
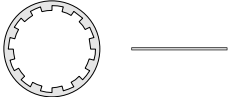


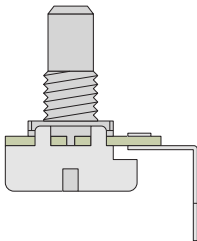
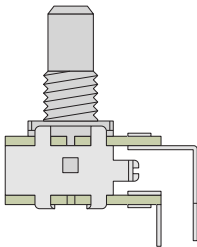

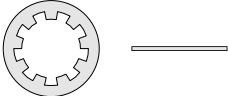
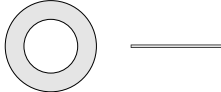

LED BEZEL

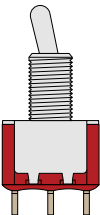
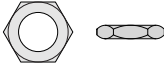
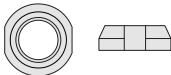
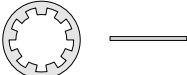


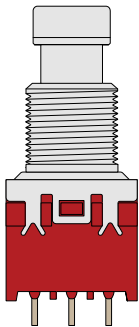
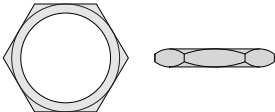
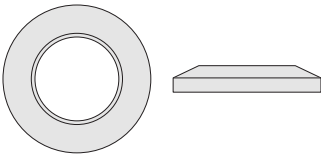
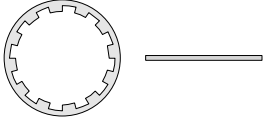
HARDWARE IDENTIFICATION

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.

<div>I/O JACK</div> 	<div>MOUNTING NUT</div>  <div>DIAMETER: 0.54" / 13.7mm</div>	<div>OUTER WASHER</div>  <div>DIAMETER: 0.6" / 15.2mm</div>	<div>LOCK WASHER</div>  <div>DIAMETER: 0.5" / 12.7mm</div>
---	---	---	---

<div>POTENTIOMETER (SINGLE)</div> 	<div>POTENTIOMETER (DUAL)</div> 	<div>MOUNTING NUT</div>  <div>DIAMETER: 0.44" / 11.2mm</div>	<div>LOCK WASHER</div>  <div>DIAMETER: 0.5" / 12.7mm</div>
		<div>OUTER WASHER</div>  <div>DIAMETER: 0.475" / 12mm</div>	<div>KNOB</div> 

<div>TOGGLE SWITCH</div> 	<div>MOUNTING NUT</div>  <div>DIAMETER: 0.36" / 9.1mm</div>	<div>DRESS NUT</div>  <div>DIAMETER: 0.375" / 9.5mm</div>	<div>LOCK WASHER</div>  <div>DIAMETER: 0.4" / 10.1mm</div>
--	--	---	---

<div>FOOTSWITCH</div> 	<div>MOUNTING NUT</div>  <div>DIAMETER: 0.6" / 15.2mm</div>	<div>DRESS NUT</div>  <div>DIAMETER: 0.77" / 19.6mm</div>	<div>LOCK WASHER</div>  <div>DIAMETER: 0.6" / 15.2mm</div>
---	--	---	---

PCB ASSEMBLY OVERVIEW

Now it's time to start building!

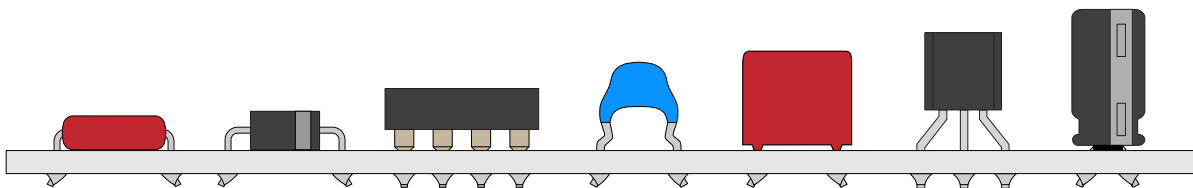
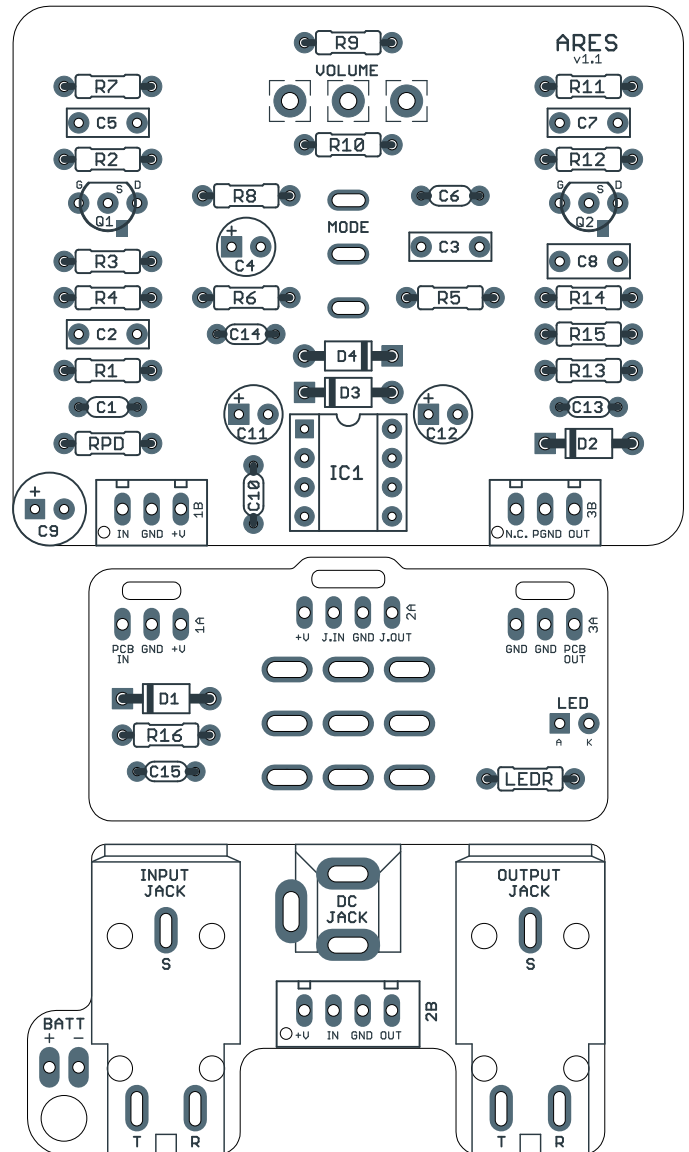
The first thing you need to do is separate the PCBs into 3 separate boards and break off the tabs from each using needle-nose or flat-head pliers. You should be left with the PCBs shown to the right.

The general principle for PCB population is that you want to work in layers from shortest components (i.e. lowest-profile) to tallest. This way, when you turn the PCB upside down, the components are held in place when soldering.

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

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

Not all of these component types are included in each kit, so skip them if they aren't applicable. Some types of film capacitors are taller than electrolytics, so those can be done last.

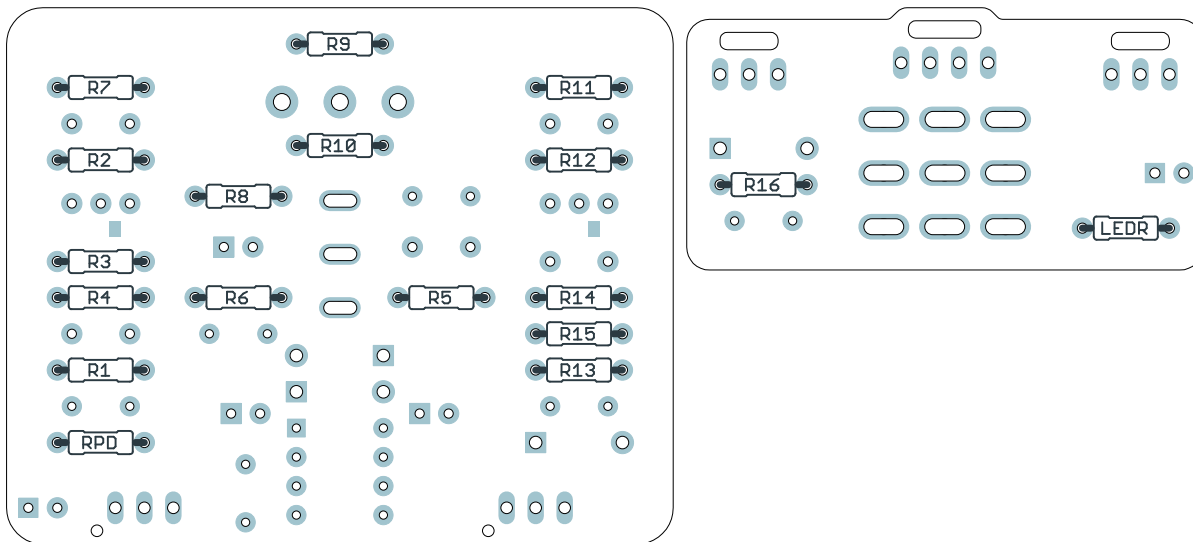


RESISTORS

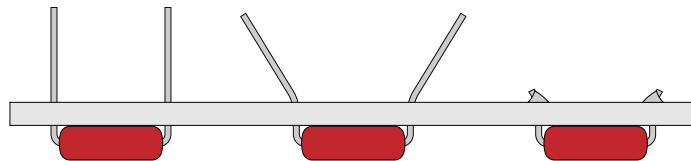
PART	VALUE
R1	100k
R2	1M
R3	3k3
R4	22k
R5	1M
R6	1M
R7	220k

PART	VALUE
R8	100k
R9	470k
R10	220k
R11	1M
R12	10k
R13	15k
R14	22k

PART	VALUE
R15	100k
R16	60R
RPD	2M2
LEDR	10k



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.



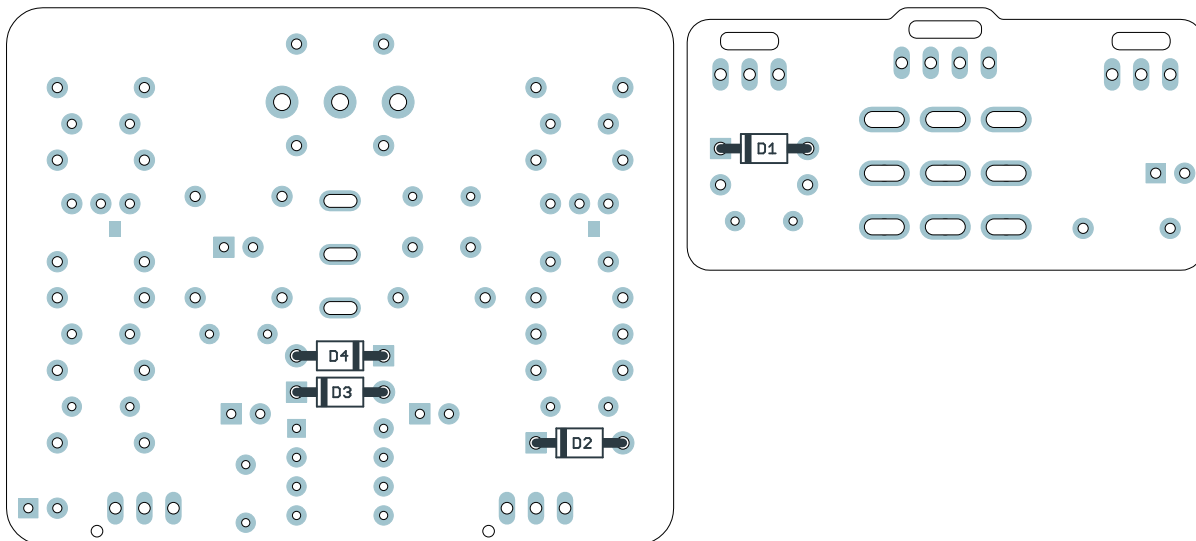
You'll use this same technique for most of the other components as well.

Don't try to do all of the resistors at once. You'll want to stop periodically flip the board and solder everything, then cut the leads using the wire snippers to make room for more. Generally you don't want to do more than 10 resistors at a time or the bottom of the board will get too crowded.

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!

DIODES

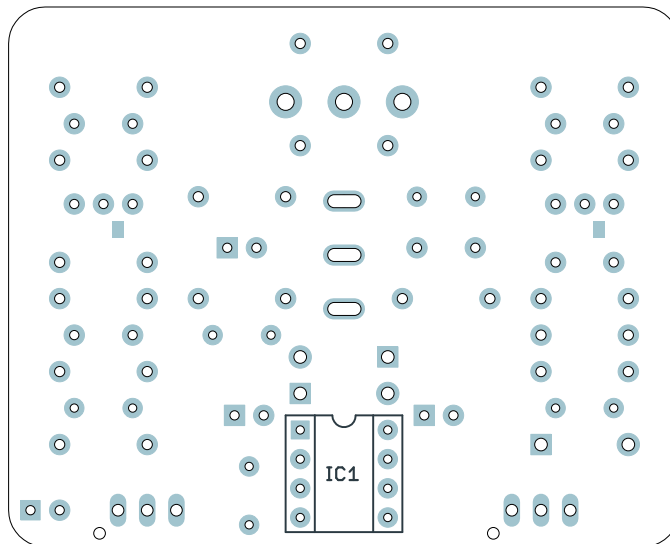
PART	VALUE
D1	1N5817
D2	1N4742A
D3	1N5817
D4	1N5817



Next, you'll populate the diodes. Diodes are polarized, so make sure to identify the polarity band (which indicates the "cathode", or negative side) and match the band to the footprint on the PCB.

SOCKET & IC

PART	VALUE
IC1	LT1054CP



Next up is the IC socket. You can't bend the leads of the socket like you can with the other components, so it won't stay in on its own until it is soldered.

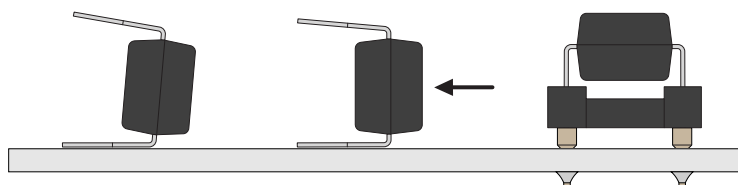
Again, it's much easier to do this with gravity holding it in place for you, so you'll want to install the socket before you do any of the taller components.

Installing the IC

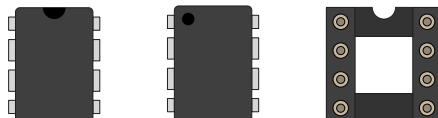
Don't insert the IC into the socket just yet. We will do this in a later step, after we've finished soldering the tallest components (the polarized capacitors). This information is just listed here for reference.

The legs of the IC are bent outward slightly during manufacturing, so they'll need to be bent back inward before they can be inserted into the sockets.

It's easiest to do this by laying the IC legs against the table and bending the body itself so all four legs on the side are straightened out at once. Then, flip it and do the other side.

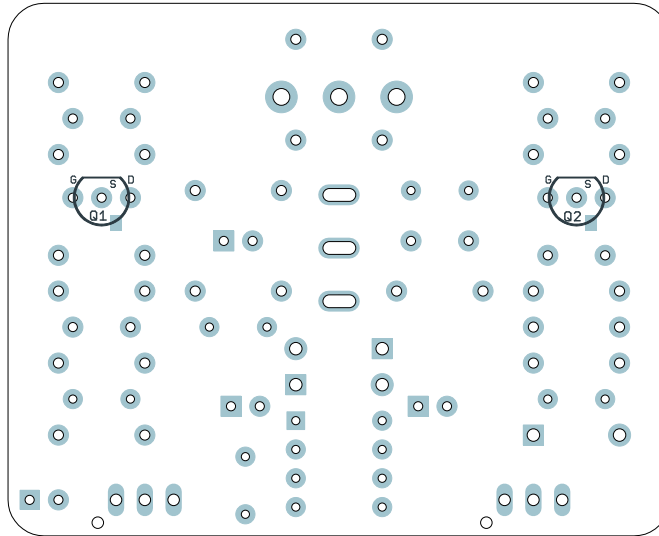


ICs may have two different orientation marks: either a dot in the upper-left or a half-circle notch in the middle of the top side. Some ICs have both marks. This shows which way the IC should be rotated when inserting it into a socket (the socket also has a half-circle notch).



TRANSISTORS

PART	VALUE
Q1	2N5484
Q2	2N5485

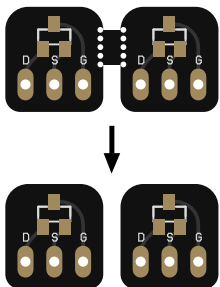


Now we'll do the two JFET transistors. The PCB outline shows through-hole TO-92 outlines, but the through-hole versions of these parts have been discontinued for many years and most of what is available today is either fake or outside the ideal specification.

We use surface-mount JFETs which perform identically to the type used in the Echoplex circuit. These are mounted to adapter boards so they can be used in through-hole designs such as this one. However, this does require some extra work since the legs must be soldered to the adapter.

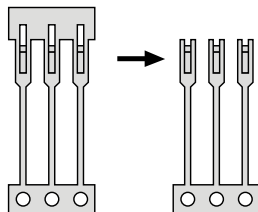
STEP 1

Break apart the adapter PCBs (you can just use your fingers for this) and then use pliers to snap off the tabs.



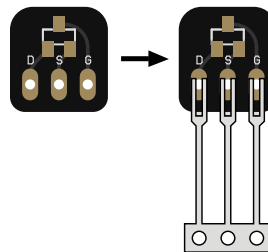
STEP 2

Using wire cutters, cut the pin strip into two sets of three. Separate the top panel from the pin strip by bending it back.



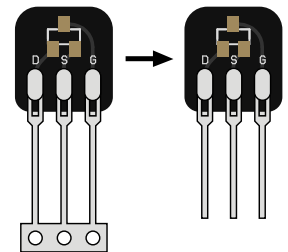
STEP 3

Clip the pins to the adapter, making sure they're centered on the pads and pushed up all the way.



STEP 4

Solder the pins (including the reverse side, if needed). Then, using wire cutters, clip the bottom strip from the pins.



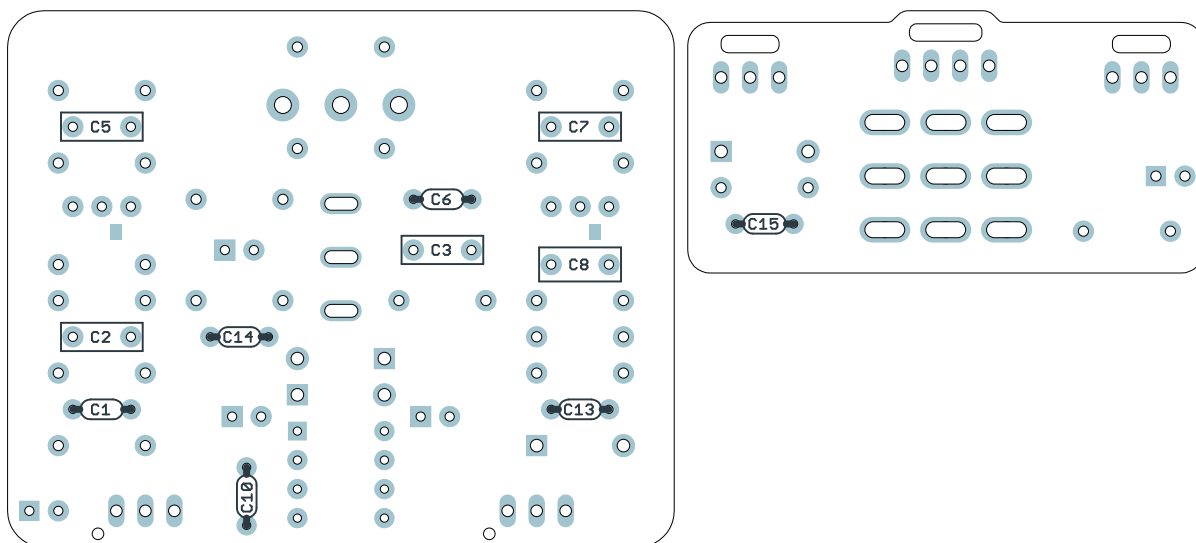
Once the pins are installed, the JFETs can be soldered to the PCB like any other normal component. Note the part number printed on the reverse side. One is labeled "5484" (used for Q1) and the other is labeled "5485" (used for Q2).

Be very careful to line up the D-S-G labels with the markings on the PCB or the effect will not work. The component side of the PCB adapter should face upwards toward the potentiometer.

CAPACITORS (NON-POLARIZED)

PART	VALUE
C1	220pF MLCC (221)
C2	22n film (0.022)
C3	22n film (0.022)
C5	100n film (0.1 or μ 1)
C6	220pF MLCC (221)
C7	22n film (0.022)

PART	VALUE
C8	470n film (0.47)
C10	470n MLCC (474)
C13	470n MLCC (474)
C14	100n MLCC (104)
C15	100n MLCC (104)



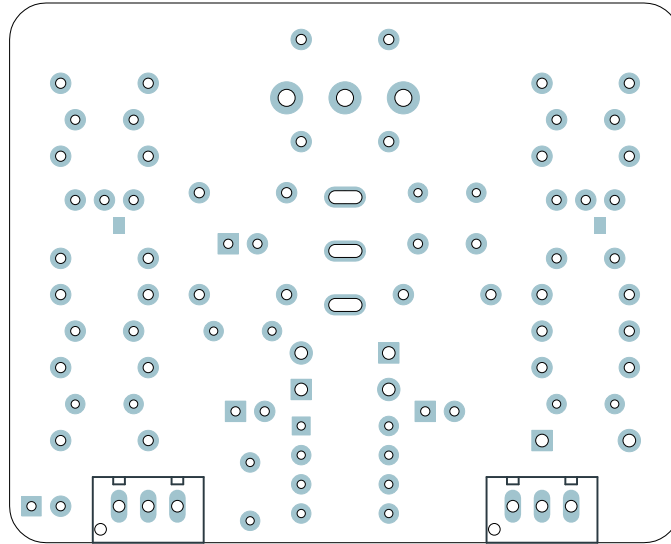
After the sockets come the box film and MLCC capacitors. These are all several different heights, but there aren't as many, so just do them all at once. Bend the leads at an angle to hold them in place.

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.

Capacitor identification:

- C2, C3 and C7 are red and will read "0.022" on the side.
- C5 is red and will either read "0.1" on the side or " μ 1J100" on the top.
- C8 is red and will either read "0.47" on the side or " μ 47J50" on the top.
- C1, C6, C10 and C13 are blue MLCCs and taped to cardboard. The value will be written on the cardboard in order to distinguish them from each other.
- C14 and C15 (100n MLCCs) are always yellow and can be identified by color since the value printed on the side can be hard to read.

WIRE HEADERS

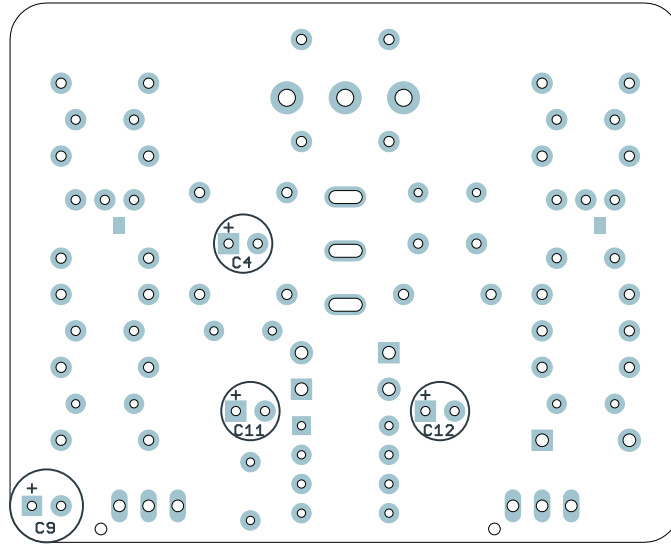


Install the two 3-pin headers (wire connectors) as shown above. These have a polarity pin, so as long as they are pressed all the way down, there's only one possible way to install them. They do fit pretty tightly in the holes, though, so press firmly.

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

CAPACITORS (POLARIZED)

PART	VALUE
C4	47uF
C9	100uF
C11	10uF
C12	10uF



Populate the electrolytic capacitors. These are the tallest components so we save them for last. 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.

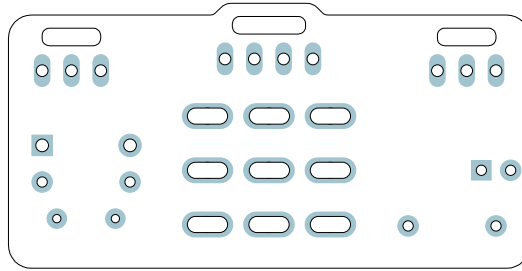
These are the last of the on-board components. Now is the time to go back to page 12 and insert the IC into the socket.

FOOTSWITCH PCB

PARTS

3-strand wire assembly (2)

4-strand wire assembly



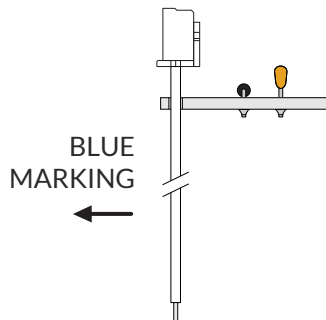
Next, it's time to finish up the footswitch board. You should have done most of the on-board components on this board in a previous step, but if not, go back and do those.

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.

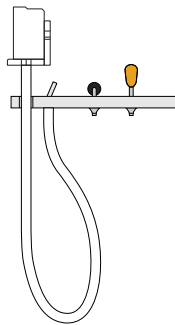
For now, pull it through as far as it can go.



STEP 2

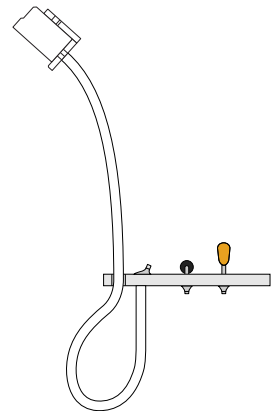
Next, bend the wires back upward and fit the ends of the wires into the solder pads.

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

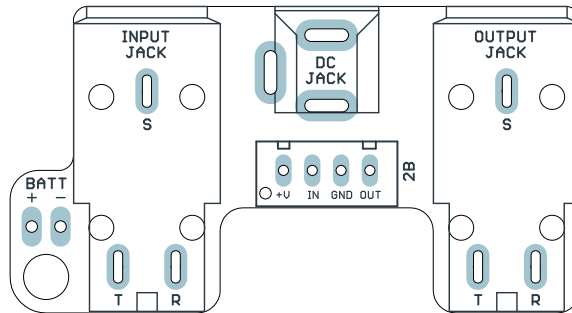
PARTS

Input & output jacks

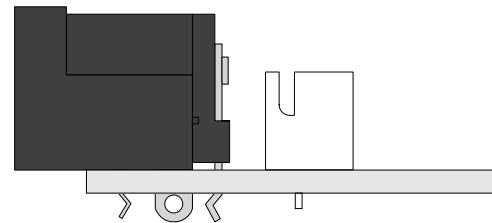
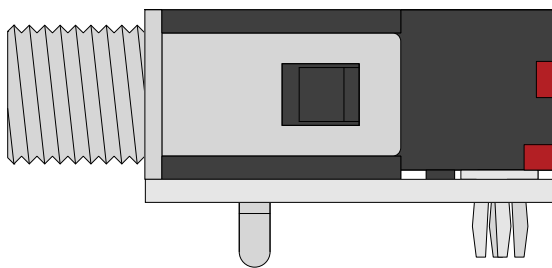
DC jack

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.

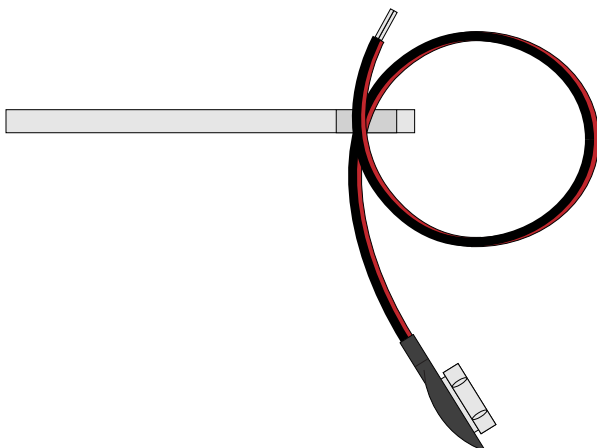


After you've soldered everything, make sure to **snip the leads on the I/O jacks as close as possible to the PCB**. There's not a lot of clearance between the bottom of this board and the top of the main PCB once everything is in place, and you don't want the pins to short against anything on accident.

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

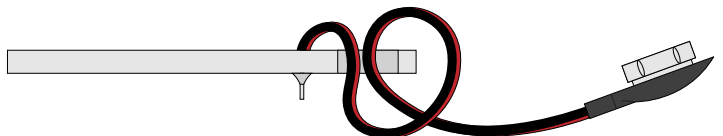
Thread the battery snap leads through the strain-relief hole twice so it forms a single loop.



STEP 2

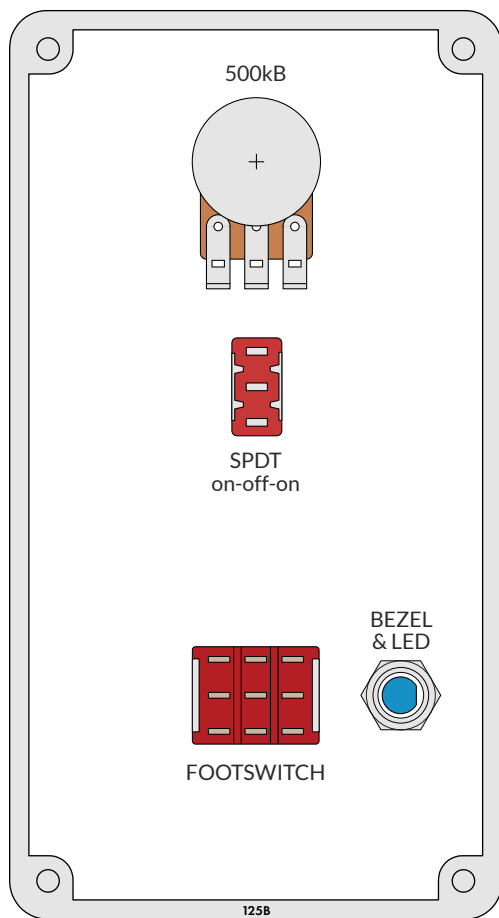
Bend the exposed wires back down and solder them into the pads. Red is positive (+), black is negative (-). After soldering, pull it tight.

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

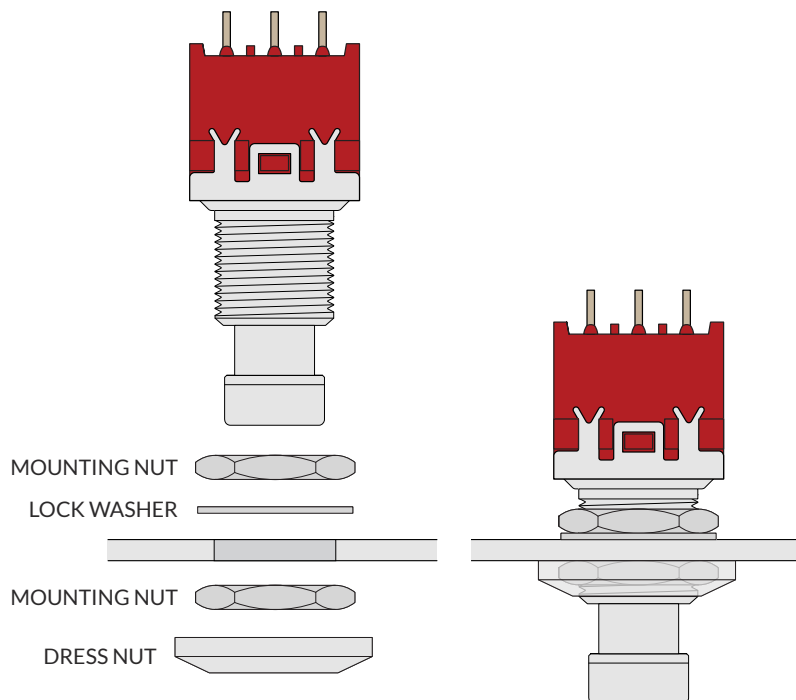


ENCLOSURE LAYOUT: PANEL MOUNTS

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



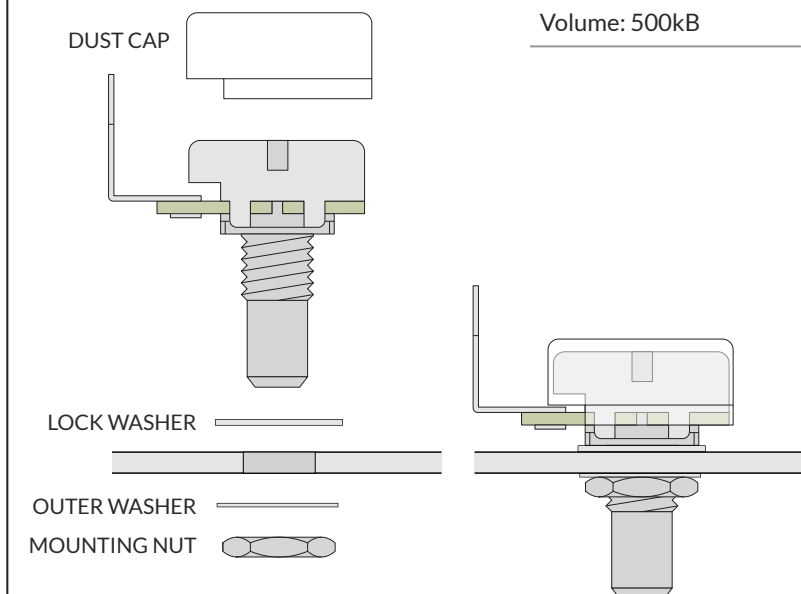
FOOTSWITCH



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.

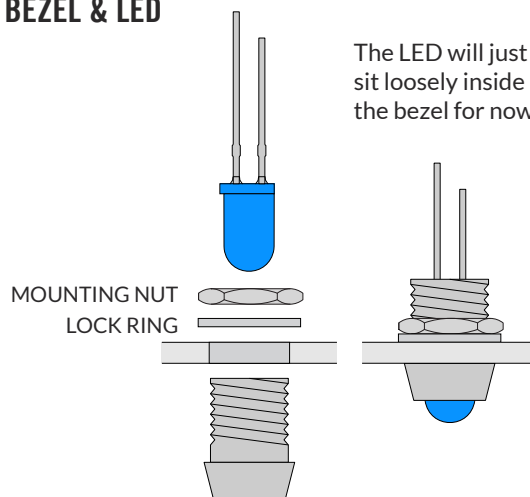
POTENTIOMETER

Volume: 500k



BEZEL & LED

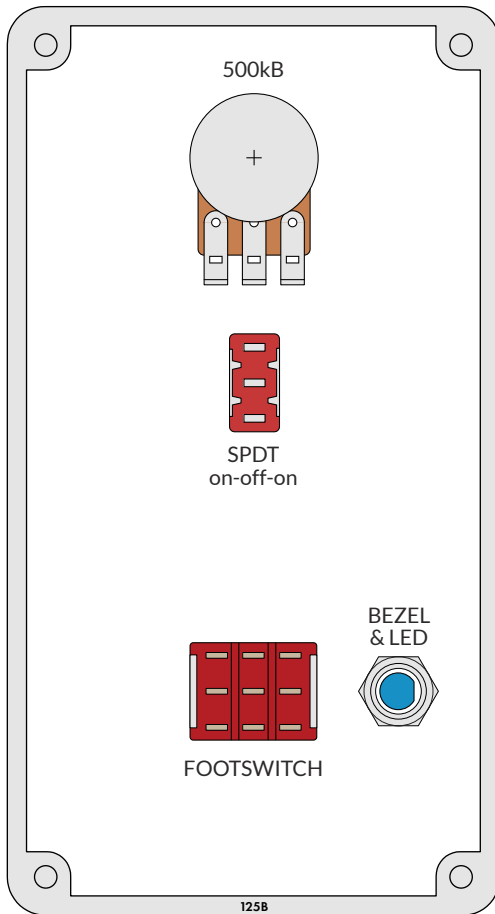
The LED will just sit loosely inside the bezel for now.



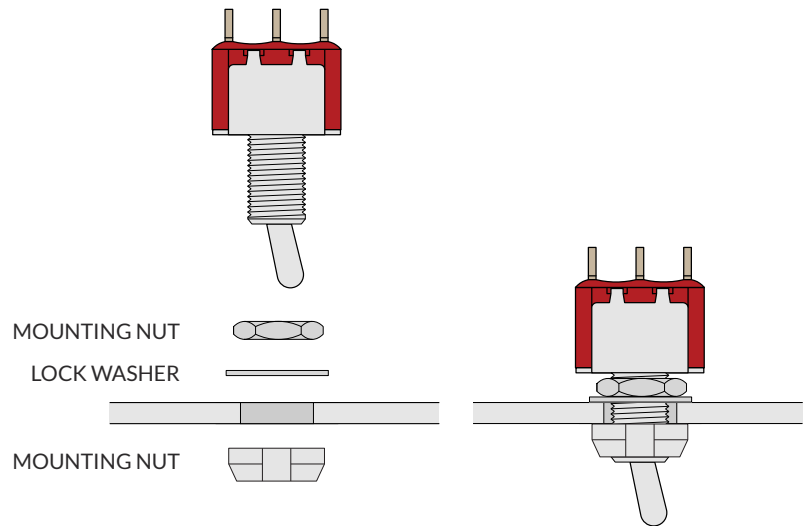
You'll need to hold the bezel in place when tightening the nut. The top of the bezel is fairly sharp, so try using a rubber band for grip instead of pressing your finger against the bottom.

ENCLOSURE LAYOUT: PANEL MOUNTS (CONT.)

Attach the toggle switch to the enclosure as shown in the diagram.

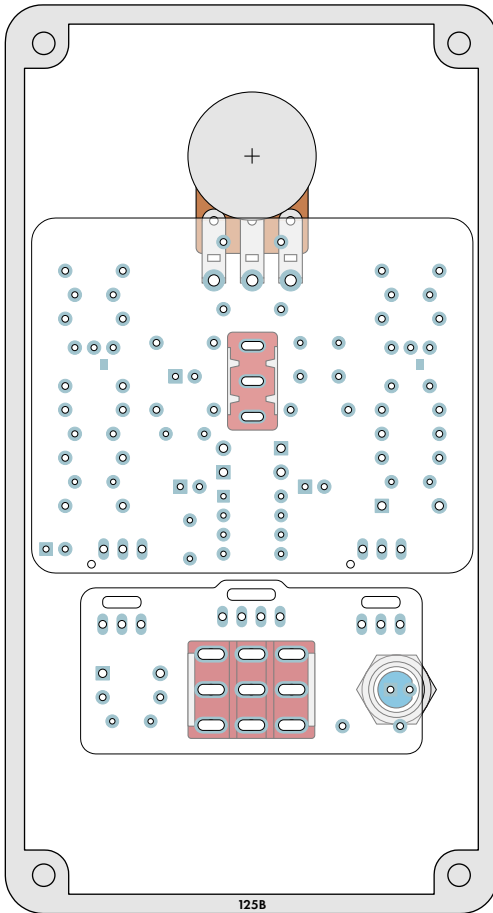


TOGGLE SWITCH



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 potentiometer and toggle switch as in the diagram to the left.

You may need to adjust the position of the potentiometer 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 potentiometer, **clip the leads as close as you can to the main PCB**. 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?

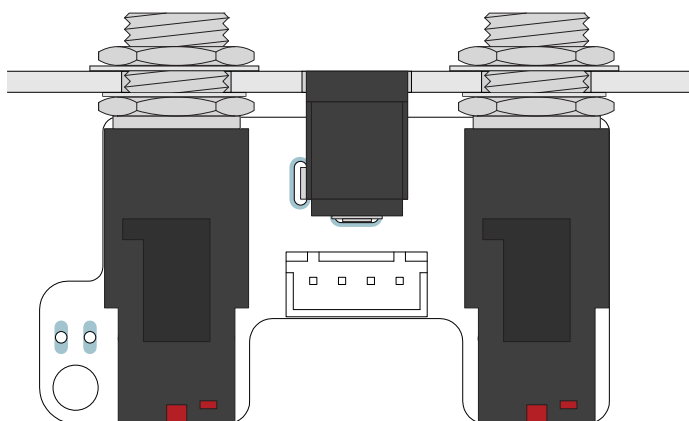
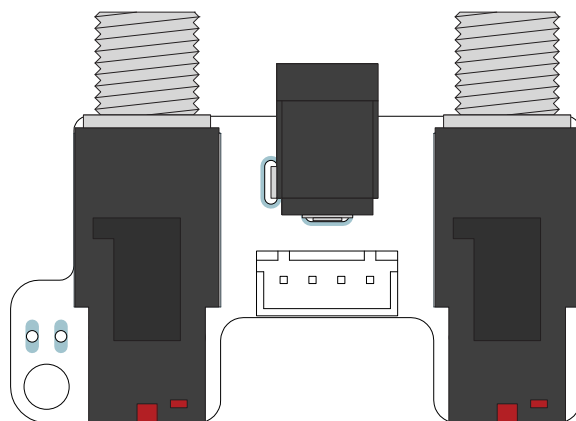
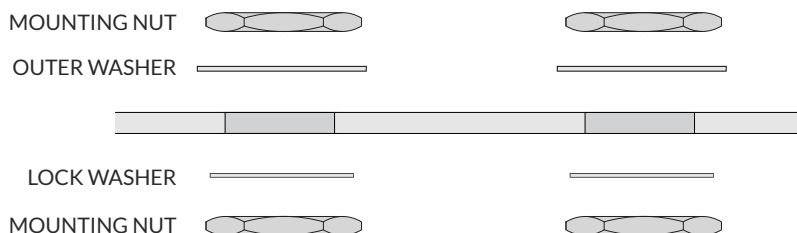
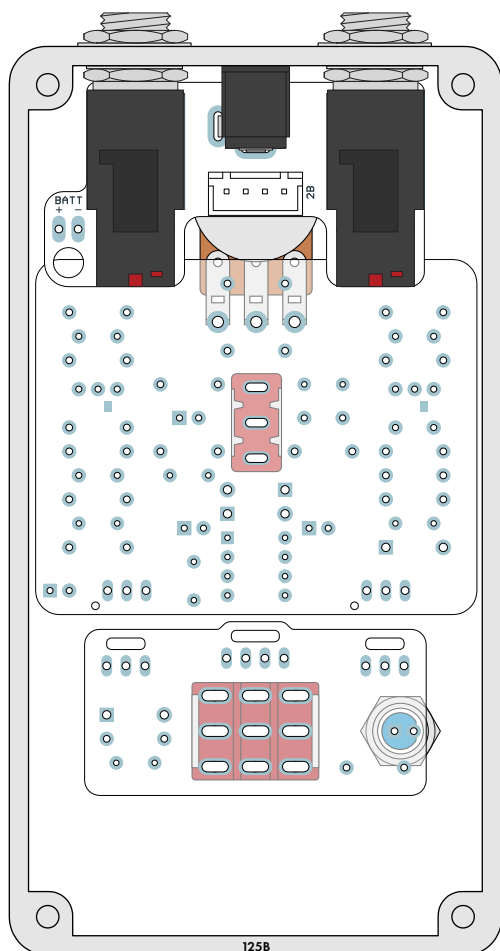
"Rock it before you box it" is conventional wisdom in pedalbuilding, and you'll often hear it recommended that builders should test the circuit before putting everything inside the enclosure. However, Aion FX projects are designed to be extremely easy to remove from the enclosure for troubleshooting, 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

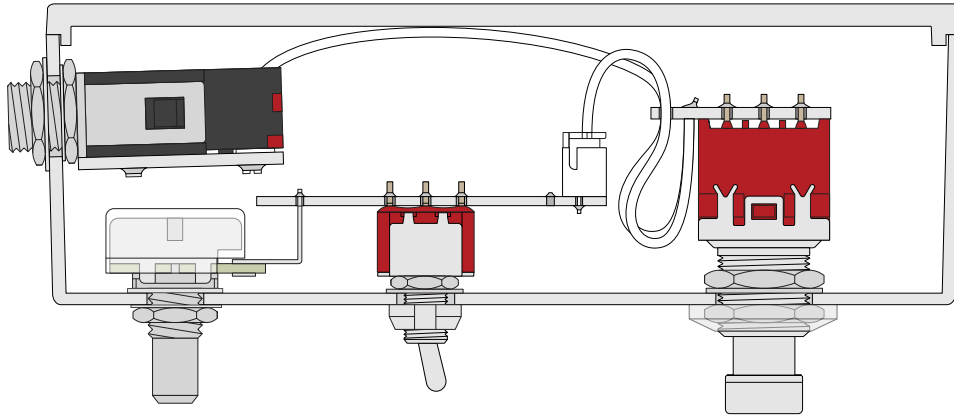
Affix the input/output PCB to the north-facing panel of the enclosure as shown.

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 TESTING & ASSEMBLY

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.

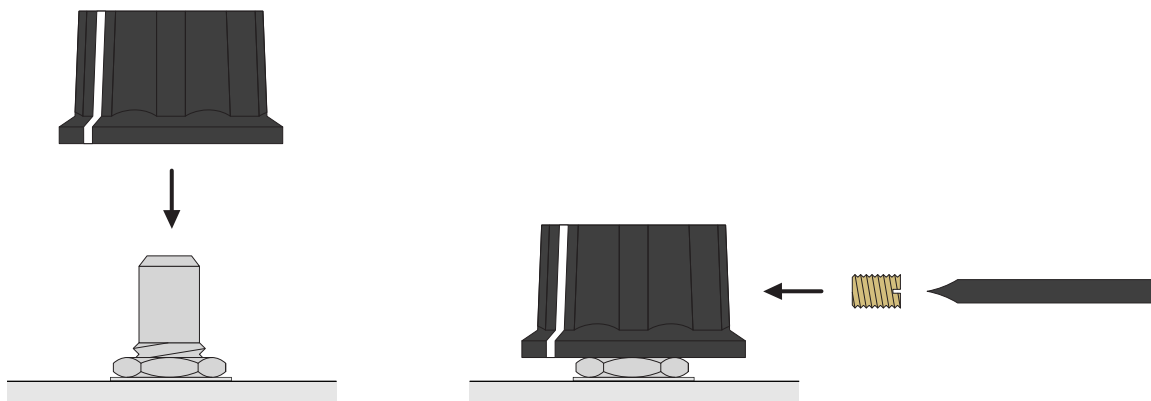
Test the bypass switch a few times, then start turning the knobs and see if everything sounds OK. If it works, great! If not, don't be discouraged. See page 27 for troubleshooting info.

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.

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!

USAGE

The Ares has the following controls:

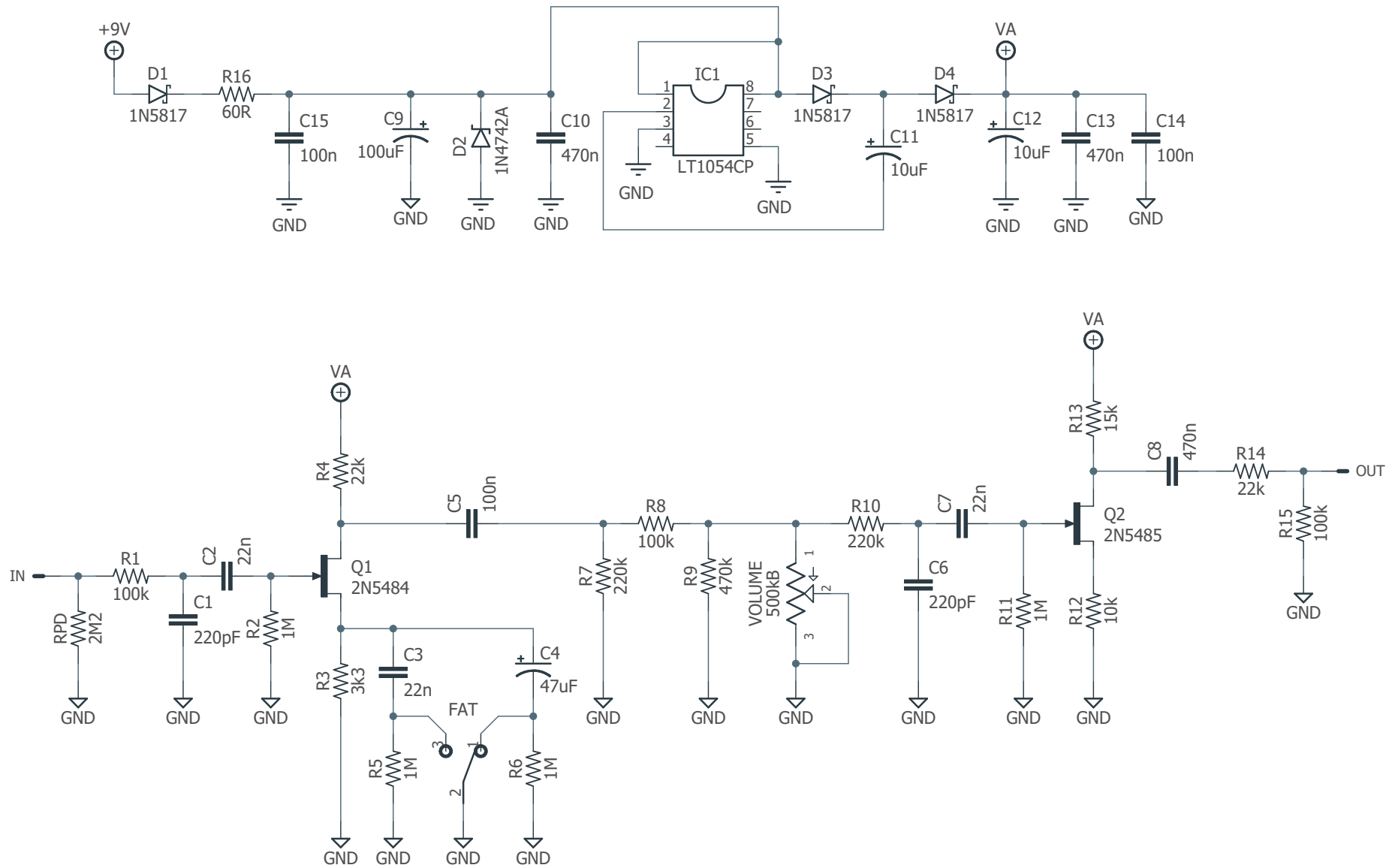
- **Volume** sets the overall volume level. It starts at zero, and the upper end of the range is dependent on the position of the Mode switch.
- **Mode** selects between early EP-3 (I), late EP-3 (II), and a third “fat” mode (III) that is more full-range than the other two. Modes II and III increase the available gain.

Note that with the Mode switch in the center position (I), the maximum volume is just slightly above unity. This volume is reached at around 2:00 on the rotation, although there is still a slight tonal shift from 2:00 to 5:00.

The maximum boost level is increased with the other two switch positions, but it’s still not a significant amount in either case, far less than a traditional clean boost pedal.

So, if you notice it doesn’t seem to have much volume on tap, be aware that this is by design and not an issue with your build. The EP-3 circuit best used as an always-on tone sweetener or buffer than something you turn on to push the amp for lead parts.

SCHEMATIC



FULL PARTS LIST

Resistors

PART	VALUE
R1	100k
R2	1M
R3	3k3
R4	22k
R5	1M
R6	1M
R7	220k

PART	VALUE
R8	100k
R9	470k
R10	220k
R11	1M
R12	10k
R13	15k
R14	22k

PART	VALUE
R15	100k
R16	60R
RPD	2M2
LEDR	10k

Capacitors

PART	VALUE
C1	220pF MLCC
C2	22n film (0.022)
C3	22n film (0.022)
C4	47uF electro
C5	100n film (0.1)
C6	220pF MLCC
C7	22n film (0.022)
C8	470n film (0.47)

PART	VALUE
C9	100uF electro
C10	470n MLCC
C11	10uF electro
C12	10uF electro
C13	470n MLCC
C14	100n MLCC
C15	100n MLCC

Diodes

PART	VALUE
D1	1N5817
D2	1N4742A
D3	1N5817
D4	1N5817

IC

PART	VALUE
IC1	LT1054CP

Transistors

PART	VALUE
Q1	2N5484
Q2	2N5485

Potentiometer

PART	VALUE
Volume	500kB

Switches

PART
SPDT on-off-on
3PDT stomp

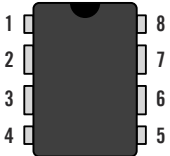
TROUBLESHOOTING INFORMATION

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.

Note that IC pins are labeled counter-clockwise from the upper-left, as shown in the diagram to the right. For the JFETs, the pins are labeled on the PCB.



IC1	
PIN	VOLTAGE
1	
2	
3	
4	
5	
6	
7	
8	

Q1	
PIN	VOLTAGE
D	
G	
S	

Q2	
PIN	VOLTAGE
D	
G	
S	

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

All trademarks are property of their respective owners.

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

1.0.1 (2024-08-08)

Added link to troubleshooting guide on page 27.

1.0.0 (2024-01-26)

Initial release.