PROJECT NAME

GRAVITON

aion
DIY GUITAR EFFECTS

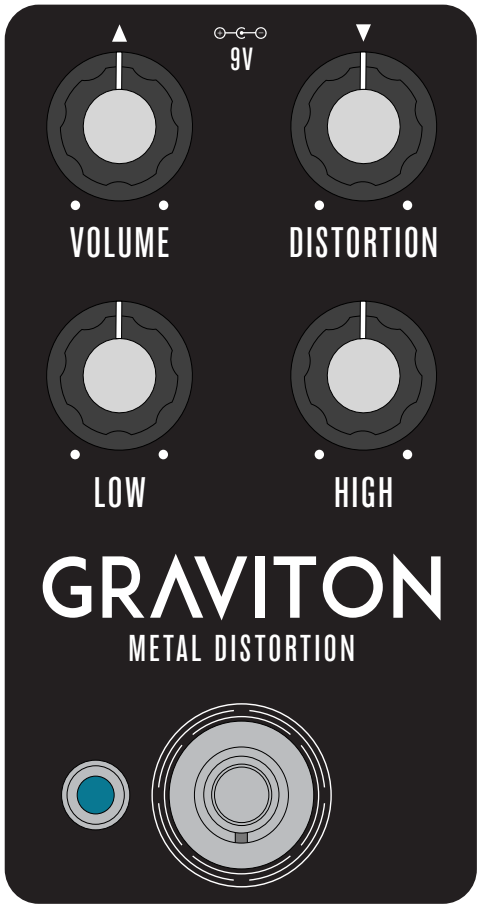
BASED ON
BOSS® HM-2 Heavy Metal

BUILD DIFFICULTY
Intermediate

EFFECT TYPE
Metal Distortion

DOCUMENT VERSION
1.0.5 (2024-11-21)

PROJECT SUMMARY
A unique distortion effect favored by extreme metal guitarists for its characteristic “buzzsaw” sound. It also works very well as a versatile high-gain effect in other styles of music.



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 Overview
10 Resistors
11 Diodes
12 Sockets & ICs
13 Transistors
14 Capacitors (Non-Polarized)
15 Wire Headers
16 Capacitors (Polarized)
17 Footswitch PCB
18 Input/Output PCB
19 Enclosure Layout: Panel Mounts
20 Enclosure Layout: Main & Footswitch PCBs
21 Enclosure Layout: Input/Output PCB
22 Final Testing & Assembly
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 24.

If you find that any parts are missing or damaged, please fill out the Missing Parts form.

Film Capacitors

NAME	QTY
1n	1
4n7	1
6n8	1
47n (0.047)	5
68n (0.068)	1
100n (0.1 or “µ1J100”)	1
150n (0.15)	1
1uF	3
1.5uF	1

Electrolytic Capacitors

NAME	QTY
10uF	2
47uF	2
100uF	1

MLCC Capacitors

NAME	QTY
100pF (marked “101”)	3
470pF (marked “471”)	1
100n (marked “104”)	1

Transistors

NAME	QTY
2N4303	1
2N3906	1
2N5088	2

Resistors

NAME	QTY
22R	1
120R	1
150R	1
330R	3
1k	2
3k3	2
10k	11
22k	2
47k	1
68k	2
82k	1
100k	5
220k	1
470k	3
1M	1
2M2	1

Diodes

NAME	QTY
1N5817	1
1N914	5
BAT46	2

ICs

NAME	QTY
JRC4558D	3
8-pin socket	3

PACKING LIST (CONT.)

Potentiometers

NAME	QTY
10kA	3
250kA	1
Dust cover	4
Knob	4
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
D9E germanium diode	2
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

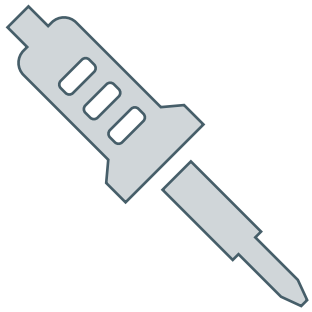
Switches

NAME	QTY
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

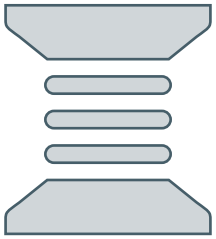
NAME	QTY
3-strand wire assembly, 70mm	2
4-strand wire assembly, 122mm	1
3-pin wire assembly header	2
4-pin wire assembly header	1

TOOLS NEEDED



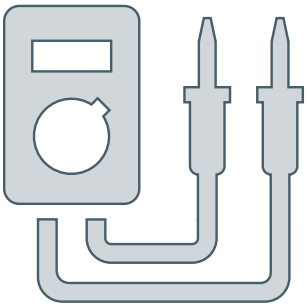
SOLDERING IRON

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.



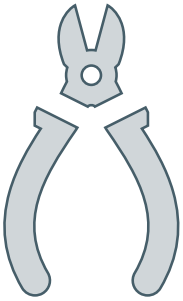
SOLDER

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.



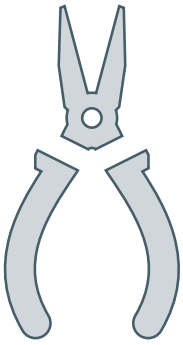
DIGITAL MULTIMETER (DMM)

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.



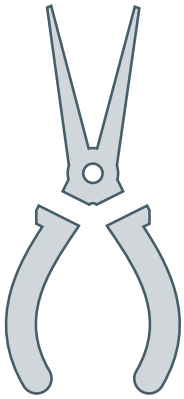
WIRE SNIPPERS

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



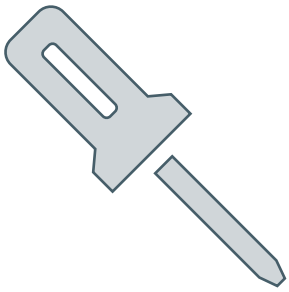
FLAT-NOSE PLIERS

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



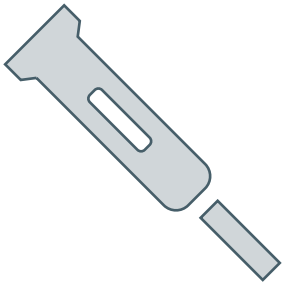
NEEDLE-NOSE PLIERS

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.



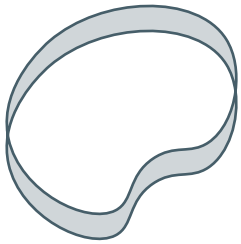
SCREWDRIVER (PHILLIPS)

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.



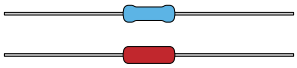
RUBBER BAND

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).

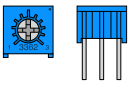
COMPONENT IDENTIFICATION

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.)

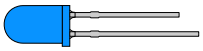
RESISTOR



TRIM POTENTIOMETER

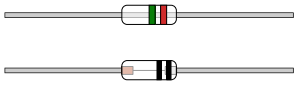


LED



SILICON DIODE

GERMANIUM DIODE



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

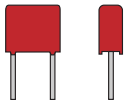
RECTIFIER DIODE

Some Schottky diodes also look like this.

SCHOTTKY DIODE

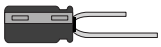
ZENER DIODE

FILM CAPACITOR



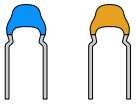
Not polarized. Color may vary by brand and type.

ELECTROLYTIC CAPACITOR



Polarized. The negative side is marked.

MLCC

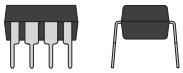


Not polarized. MLCC stands for "multi-layer ceramic capacitor."

TANTALUM CAPACITOR

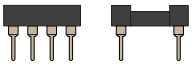
Polarized. The positive side is marked.

IC OR OP-AMP

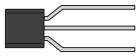


Charge pumps and delay chips also look like this. They may have more than 8 legs.

IC SOCKET



TRANSISTOR OR JFET



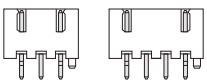
Some voltage regulators also look like this.

SNAP-APART SOCKET

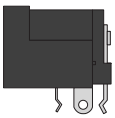
It's recommended to use a razor blade to separate these cleanly.

WIRE ASSEMBLY

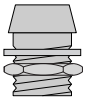
WIRE ASSEMBLY HEADER



DC JACK



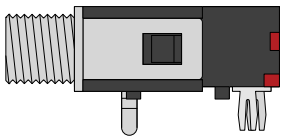
LED BEZEL



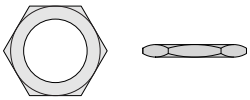
HARDWARE IDENTIFICATION

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.

I/O JACK

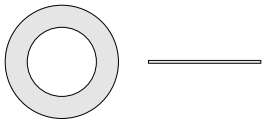


MOUNTING NUT



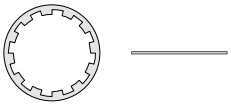
DIAMETER: 0.54" / 13.7mm

OUTER WASHER



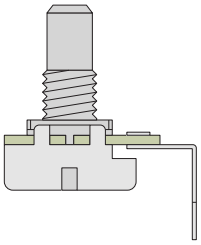
DIAMETER: 0.6" / 15.2mm

LOCK WASHER

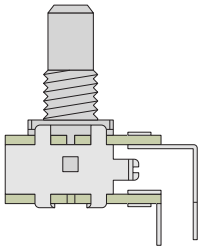


DIAMETER: 0.5" / 12.7mm

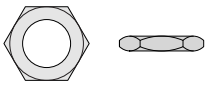
POTENTIOMETER (SINGLE)



POTENTIOMETER (DUAL)

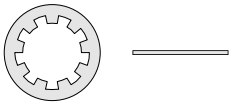


MOUNTING NUT



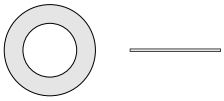
DIAMETER: 0.44" / 11.2mm

LOCK WASHER



DIAMETER: 0.5" / 12.7mm

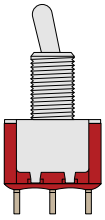
OUTER WASHER



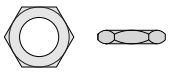
DIAMETER: 0.475" / 12mm

KNOB

TOGGLE SWITCH

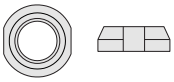


MOUNTING NUT



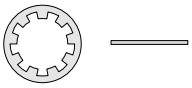
DIAMETER: 0.36" / 9.1mm

DRESS NUT



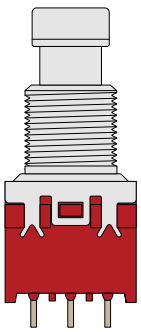
DIAMETER: 0.375" / 9.5mm

LOCK WASHER

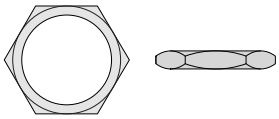


DIAMETER: 0.4" / 10.1mm

FOOTSWITCH

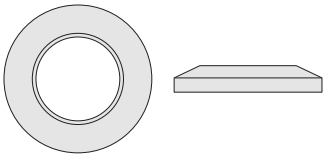


MOUNTING NUT



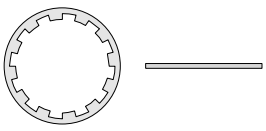
DIAMETER: 0.6" / 15.2mm

DRESS NUT



DIAMETER: 0.77" / 19.6mm

LOCK WASHER



DIAMETER: 0.6" / 15.2mm

PCB ASSEMBLY OVERVIEW

Now it's time to start building!

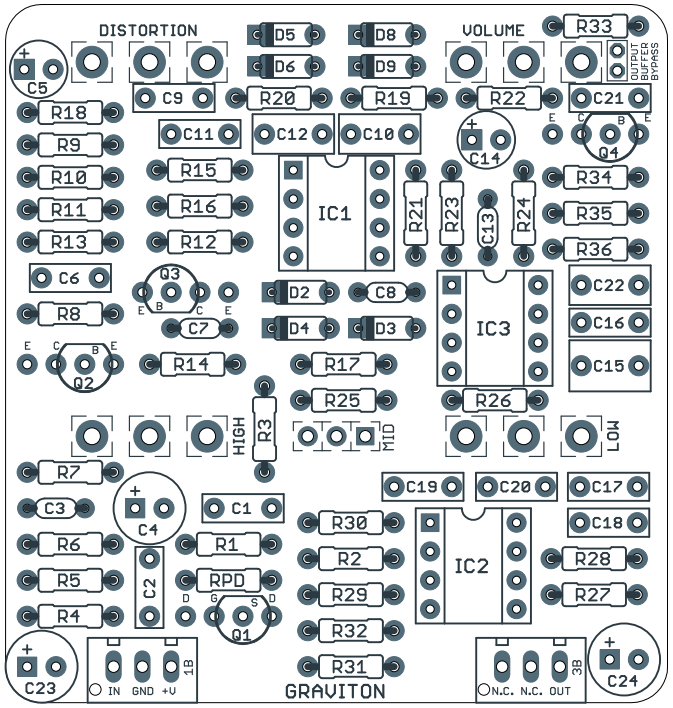
Before you begin, separate the PCBs into individual boards and break off the tabs from each using needle-nose or flat-head pliers. You should be left with the three PCBs shown to the right.

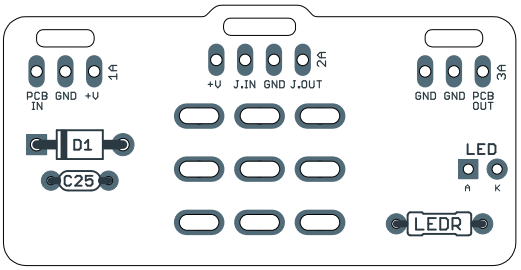
The general principle for PCB population is that you want to work in layers from shortest components (i.e. lowest-profile) to tallest so that when the PCB is upside-down, everything is making contact with the work surface and is held in place.

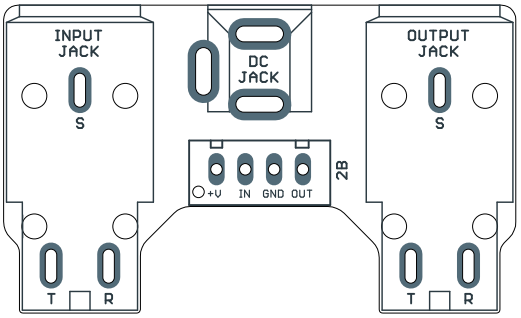
Generally speaking, you should populate the components in this order:

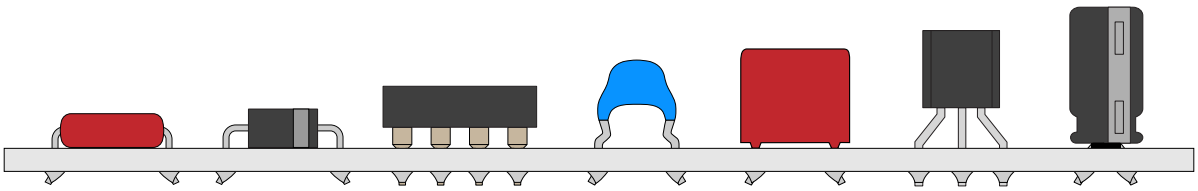
1. Resistors
2. Diodes
3. IC sockets
4. MLCC capacitors
5. Film capacitors
6. Transistors
7. Electrolytic capacitors

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

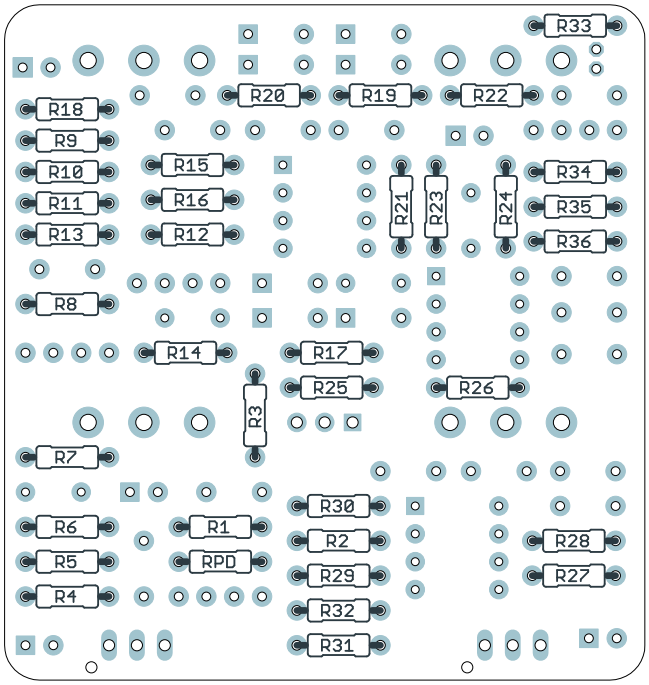
PART	VALUE
R1	10k
R2	1M
R3	10k
R4	22k
R5	100k
R6	470k
R7	10k
R8	22R
R9	150R
R10	22k

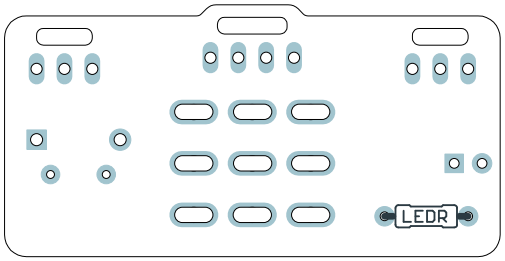
PART	VALUE
R11	100k
R12	470k
R13	120R
R14	1k
R15	10k
R16	68k
R17	220k
R18	47k
R19	10k
R20	10k

PART	VALUE
R21	68k
R22	3k3
R23	3k3
R24	10k
R25	100k
R26	330R
R27	82k
R28	330R
R29	100k
R30	330R

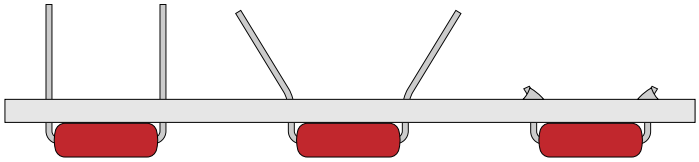
PART	VALUE
R31	10k
R32	10k
R33	470k
R34	10k
R35	1k
R36	100k
RPD	2M2
LEDR	10k

LEDR was changed from 4k7 to 10k in late 2024.





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.

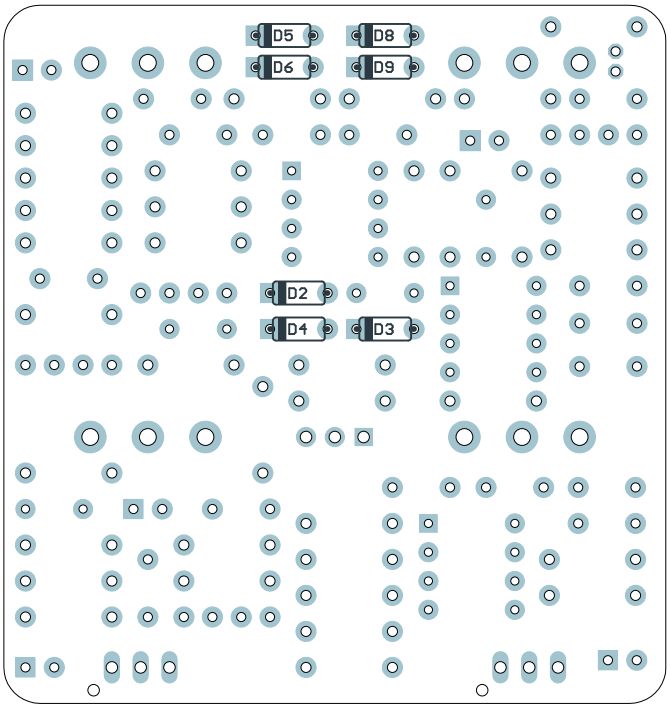


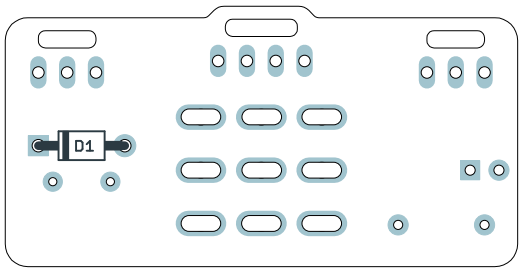
Don't try to do all of the resistors at once. You'll want to stop periodically flip the board and solder everything, then cut the leads using the wire snippers to make room for more.

DIODES

PART	VALUE
D1	1N5817
D2	1N914
D3	1N914
D4	1N914

PART	VALUE
D5	BAT46
D6	BAT46
D8	1N914
D9	1N914

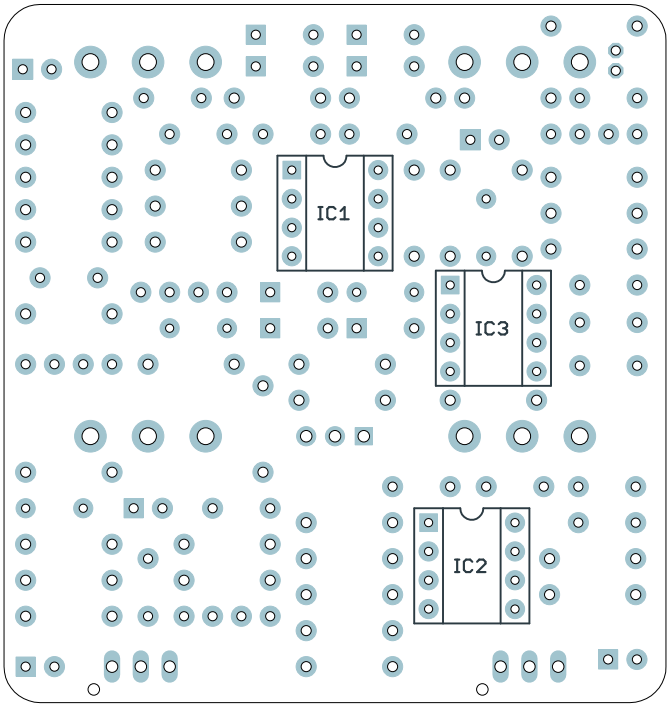




Next, you'll populate the diodes. Diodes are polarized, so make sure to identify the polarity band (which indicates the "cathode", or negative side) and match the band to the footprint on the PCB.

SOCKETS & ICS

PART	VALUE
IC1	JRC4558D
IC2	JRC4558D
IC3	JRC4558D

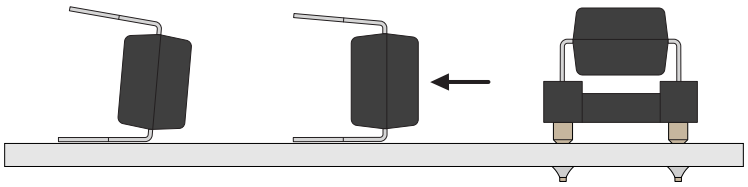


Next up are the sockets. You can't bend the leads of the sockets as with the other components, so they won't stay in on their own until they are soldered. Flip the PCB and use gravity to hold them in place.

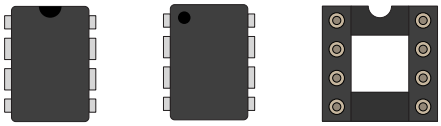
Installing the ICs

Don't insert the ICs into the sockets 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 each 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.

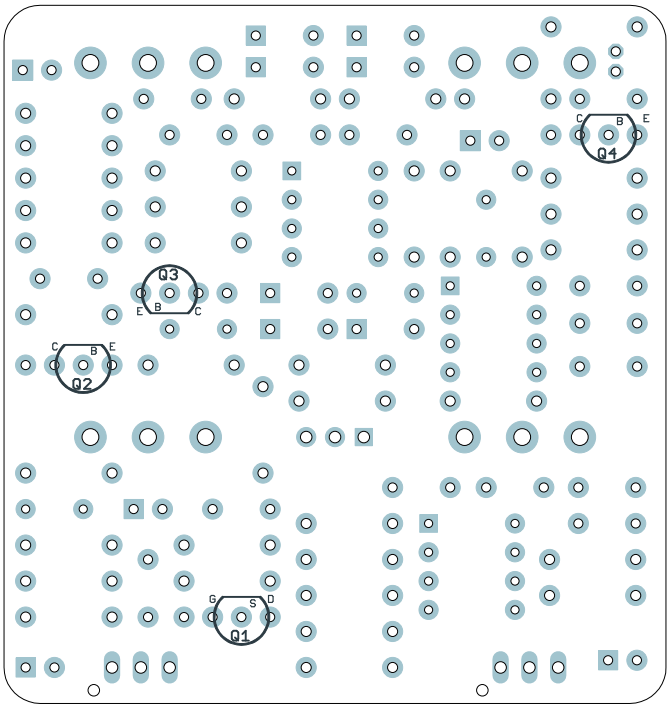


ICs may have two different orientation marks: either a dot in the upper-left or a half-circle notch in the middle of the top side. Some ICs have both marks. This shows which way the IC should be rotated when inserting it into a socket (the socket also has a half-circle notch).

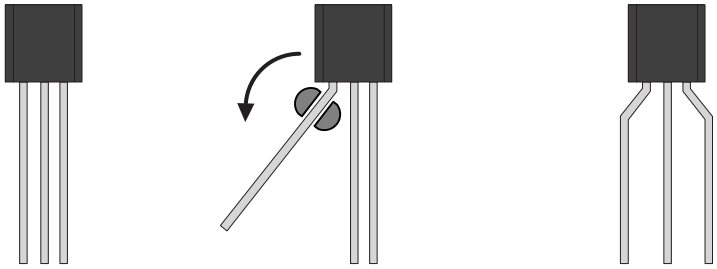


TRANSISTORS

PART	VALUE
Q1	2N4303
Q2	2N5088
Q3	2N3906
Q4	2N5088

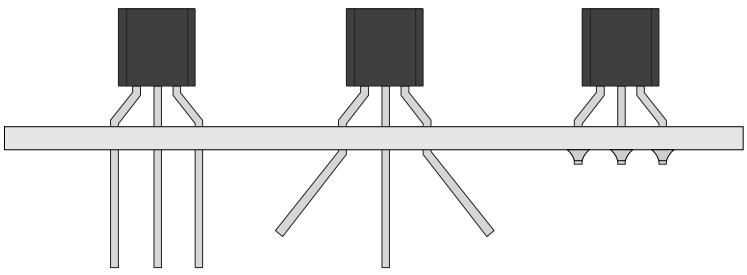


Now we'll do the transistors and JFET. If the legs are not already bent into 0.1" spacing, use your needle-nose pliers to bend the outer two legs as shown.



Since these are just used as buffers and don't need to be selected for gain, sockets are not necessary. You can just solder them directly to the board.

Bend the outer leads to hold it in place on the board. Then, solder them and clip the leads.



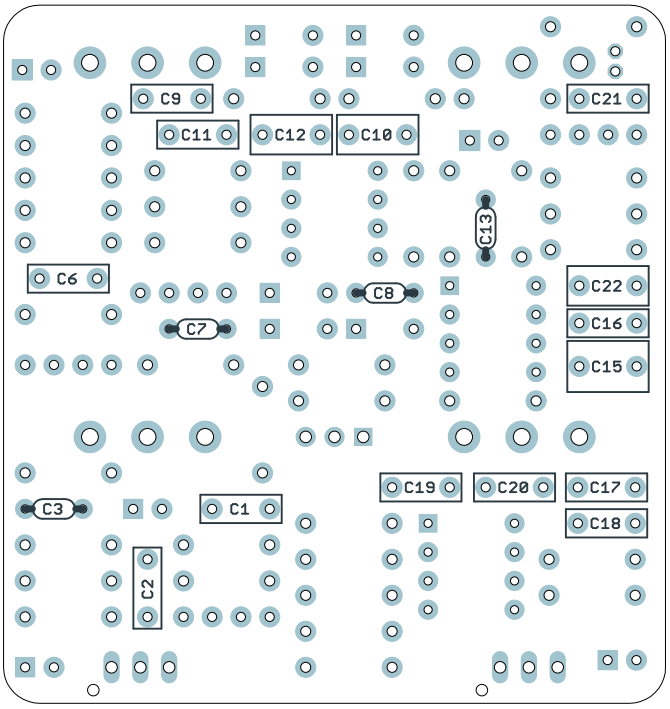
CAPACITORS (NON-POLARIZED)

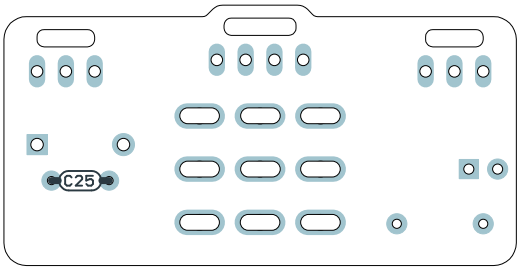
PART	VALUE
C1	47n (0.047)
C2	47n (0.047)
C3	100pF MLCC
C6	47n (0.047)
C7	100pF MLCC

PART	VALUE
C8	100pF MLCC
C9	47n (0.047)
C10	1uF
C11	1n
C12	1uF

PART	VALUE
C13	470pF MLCC
C15	1.5uF
C16	68n (0.068)
C17	150n (0.15)
C18	6n8

PART	VALUE
C19	100n (0.1)
C20	4n7
C21	47n (0.047)
C22	1uF
C25	100n MLCC





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, so they will work in any direction, but to keep things neat, it's best to put them all facing the same way.

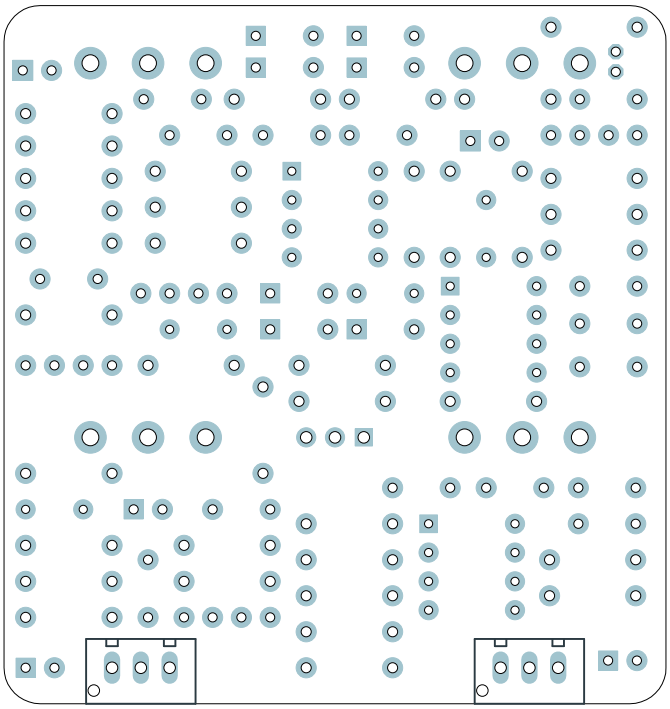
There is another MLCC capacitor located on the footswitch PCB which is not shown in the image above. This is C25, the yellow 100n capacitor.

Note: Depending on the type, the box film capacitors may have their value printed on either the top or the side. Usually the red ones have it printed on the side while the blue or gray ones have it on the top.

C19 is usually red, but may read "µ1J100" on the top rather than the side.

The blue MLCCs are taped to cardboard. The value is hard to read on the capacitor itself, but will always be written on the cardboard. C25 (100n MLCC) is always yellow.

WIRE HEADERS

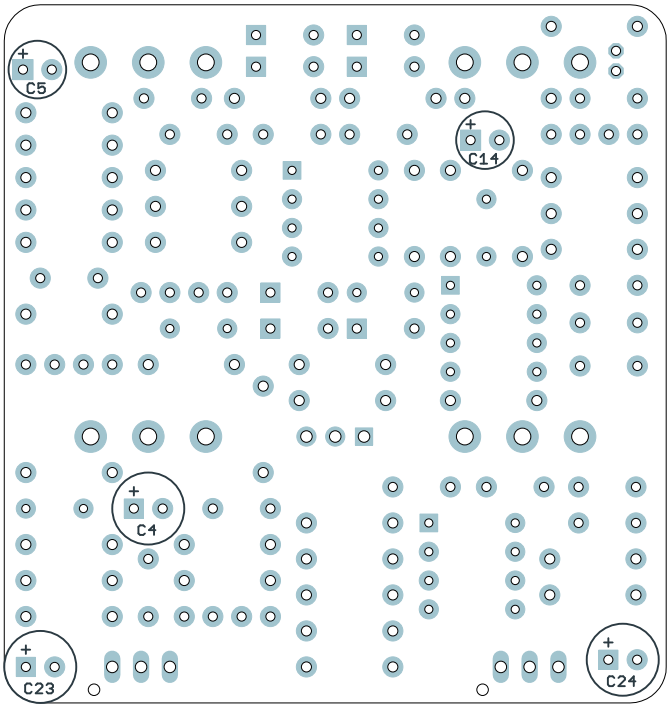


Install the two 3-pin headers (wire connectors) as shown above. These have a polarity pin, so as long as they are pressed all the way down, there's only one possible way to install them. They do fit pretty tightly in the holes, though, so press firmly.

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
C5	10uF
C14	10uF
C23	100uF
C24	47uF

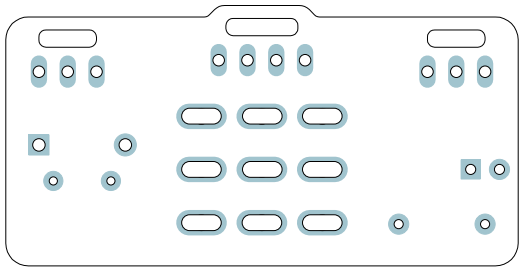


Populate the electrolytic capacitors. These are the tallest components so we save them for last. They are polarized (i.e. they will only work in one direction), so note the vertical mark that indicates the negative side. The longer leg is positive and fits in the square pad.

These are the last of the on-board components. Now is the time to go back to page 13 and insert the ICs into the sockets.

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.

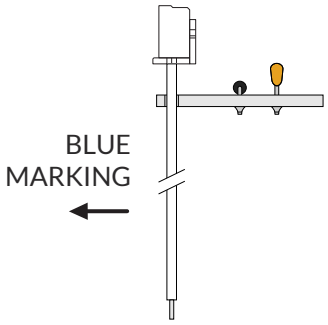
It's easiest to start with the slide switch. Fit it in as shown in the diagram above, with the slide lever facing to the right. Be careful—the pads are small. Make sure you don't accidentally "bridge" two pads together when soldering or you will have issues with the bypass.

The wires are next. 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.

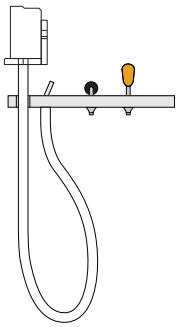
For now, pull it through as far as it can go.



STEP 2

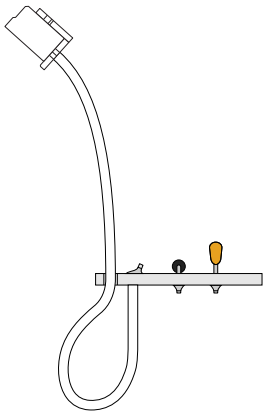
Next, bend the wires back upward and fit the ends of the wires into the solder pads.

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

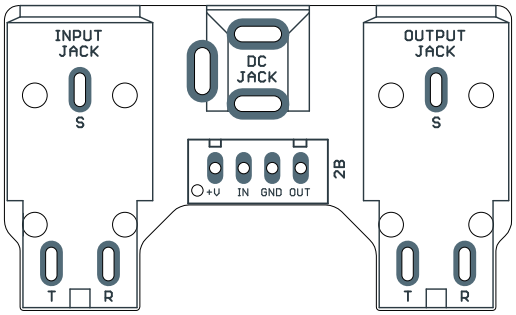
Then, solder the wires from the top. This is the trickiest part of the whole build. You want to solder the pads without touching the iron to the wires themselves and risking burning through the insulation. It helps to use a sharp or narrow tip on the soldering iron.



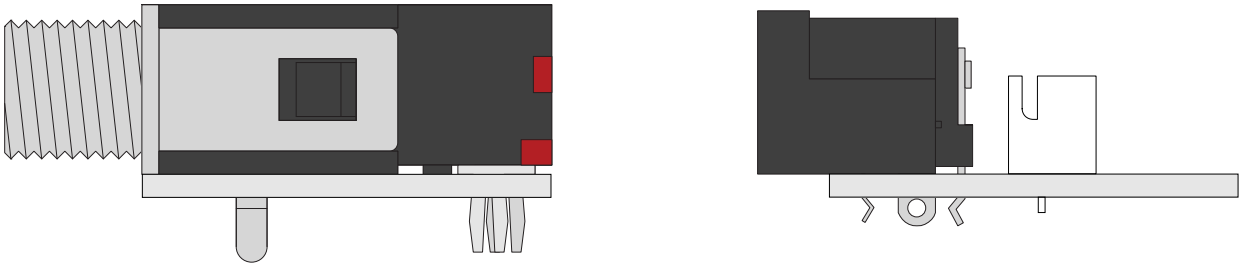
Once all three wire assemblies are soldered, set the footswitch PCB aside. We'll solder the actual footswitch and LED in a later step.

INPUT/OUTPUT PCB

PARTS
Input & output jacks
DC jack
Wire header



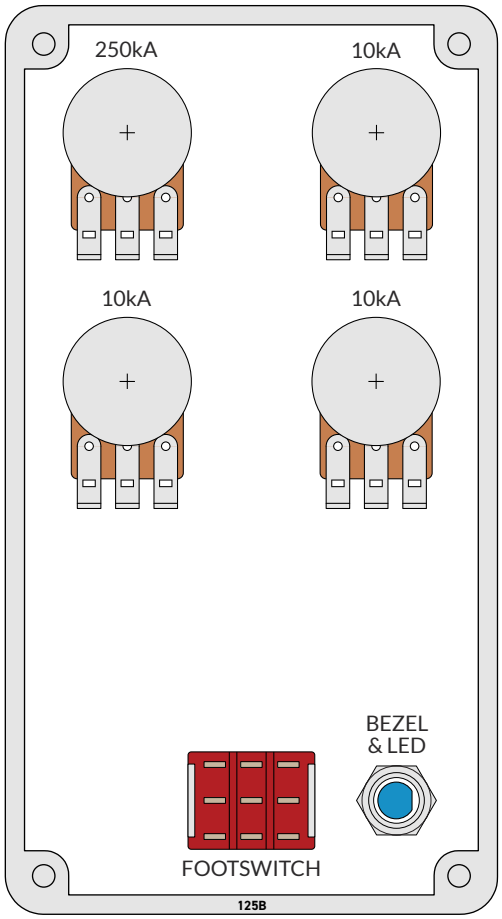
Almost done! Get the two input/output jacks, the DC jack and the wire header and snap them in place. The PCB is designed for them to fit securely, so you can do them all at once before flipping and soldering.



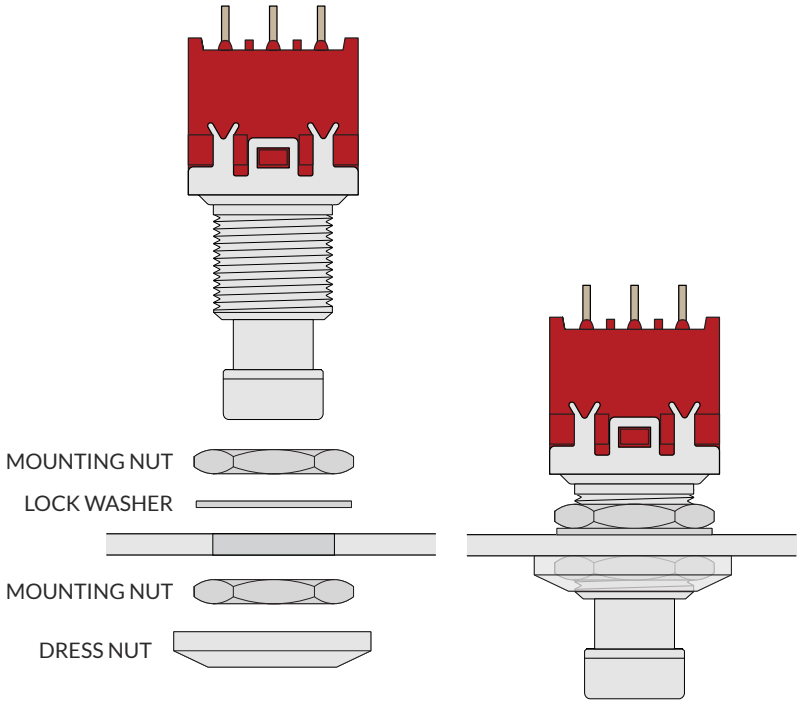
After you've soldered everything, make sure to **snip the leads on the I/O jacks as close as possible to the PCB**. There's not a lot of clearance between the bottom of this board and the top of the main PCB once everything is in place, and you don't want the pins to short against anything on accident.

ENCLOSURE LAYOUT: PANEL MOUNTS

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

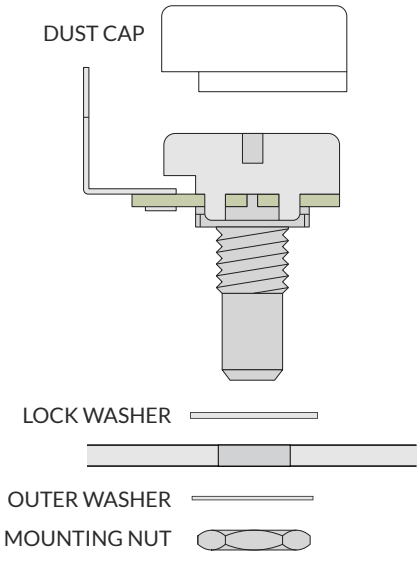


FOOTSWITCH



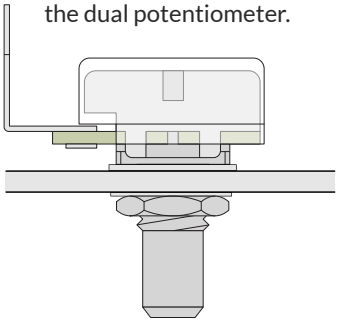
The dress nut fits over the top of the mounting nut and is for aesthetic purposes only. Wrap a rubber band around it to use as a grip when tightening. Avoid using metal tools on it or you run the risk of scratching or denting it.

POTENTIOMETERS



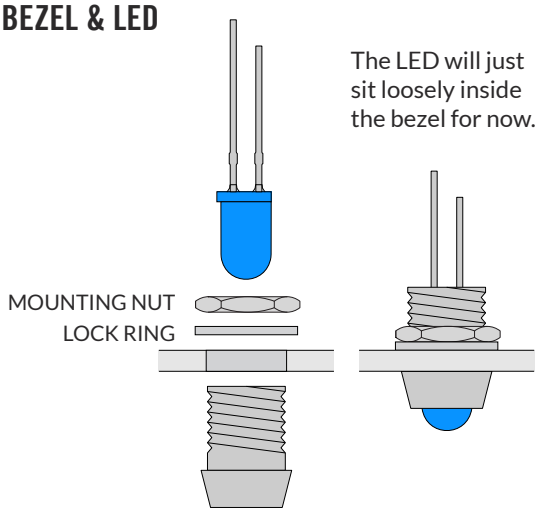
Low: 10kA
High: 10kA
Volume: 10kA
Distortion: 250kA

The dust cap is not used on the dual potentiometer.



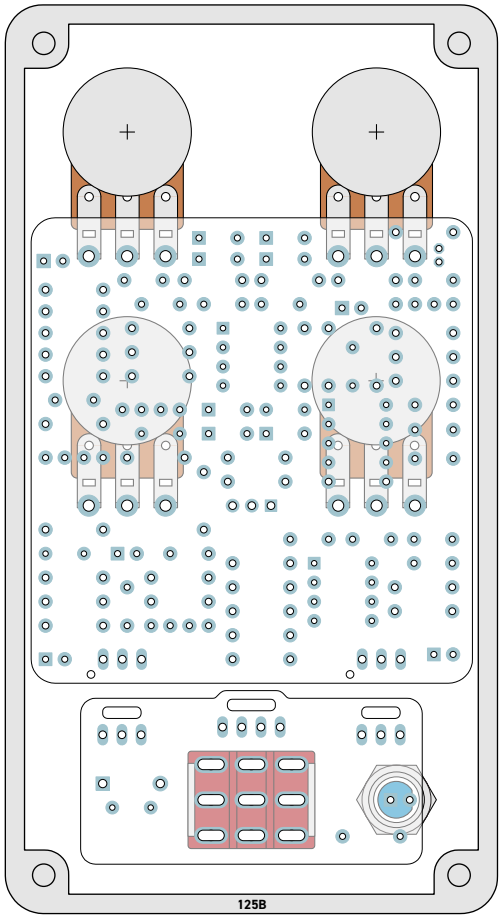
BEZEL & LED

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



You'll need to hold the bezel in place when tightening the nut. Be aware that the bezel is fairly sharp. Try using a rubber band for grip instead of just pressing your finger against the bottom.

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 as in the diagram to the left.

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

Once all of the pins are through and the PCB is laying flat, solder each of the pins from the top, being 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.

Next, do the same thing with the footswitch board—the 3PDT footswitch and the LED.

Before soldering, 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.

Why solder everything inside the enclosure before testing it?

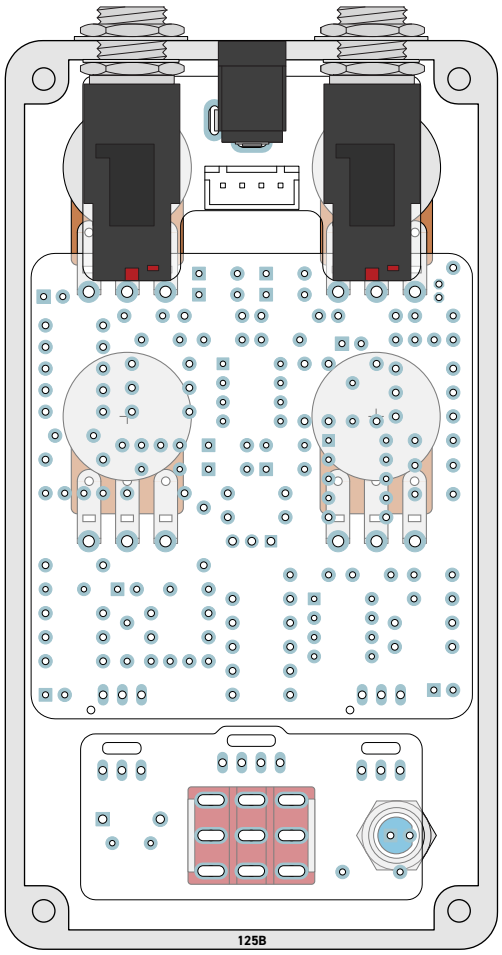
"Rock it before you box it" is conventional wisdom in pedalbuilding, and you'll often hear it recommended that builders should test the circuit before putting everything inside the enclosure. However, Aion FX projects are designed to be extremely easy to remove from the enclosure for troubleshooting,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

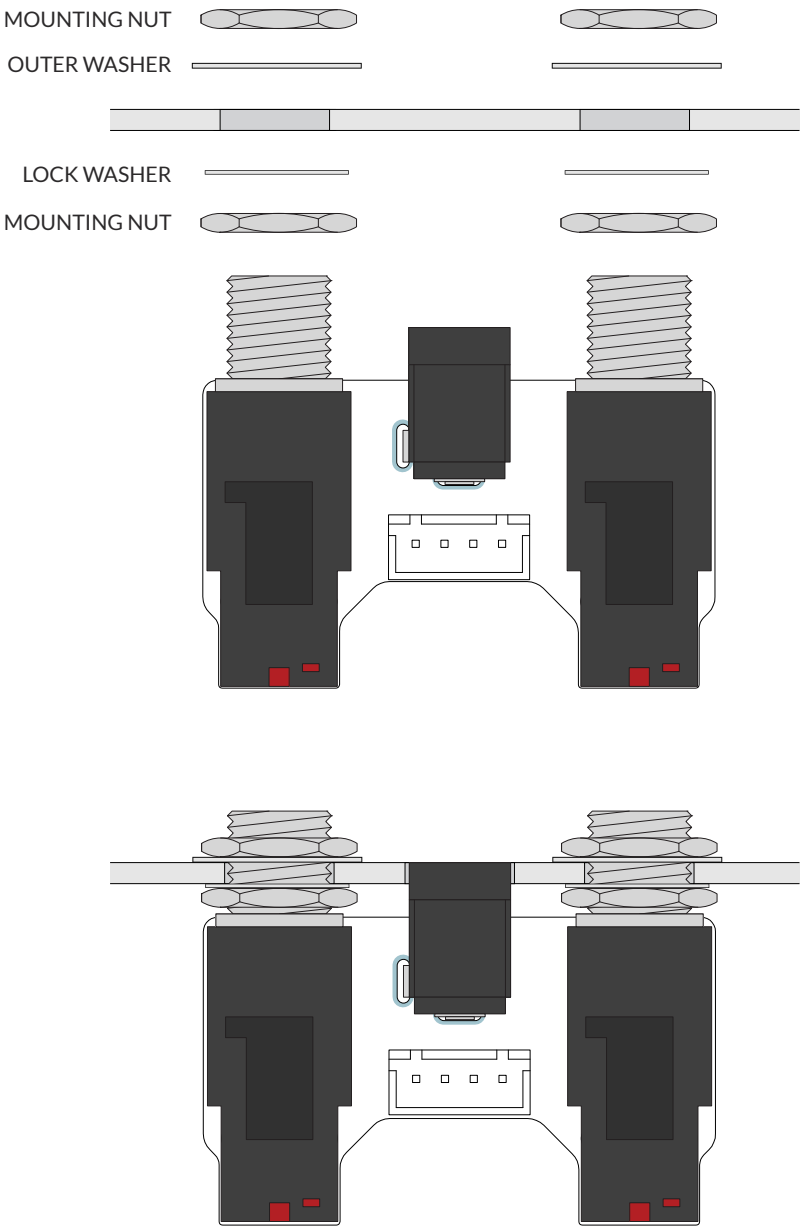
If you've read the documentation carefully and followed all the instructions, there's a good chance you will get it right the first time!

ENCLOSURE LAYOUT: INPUT/OUTPUT PCB

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

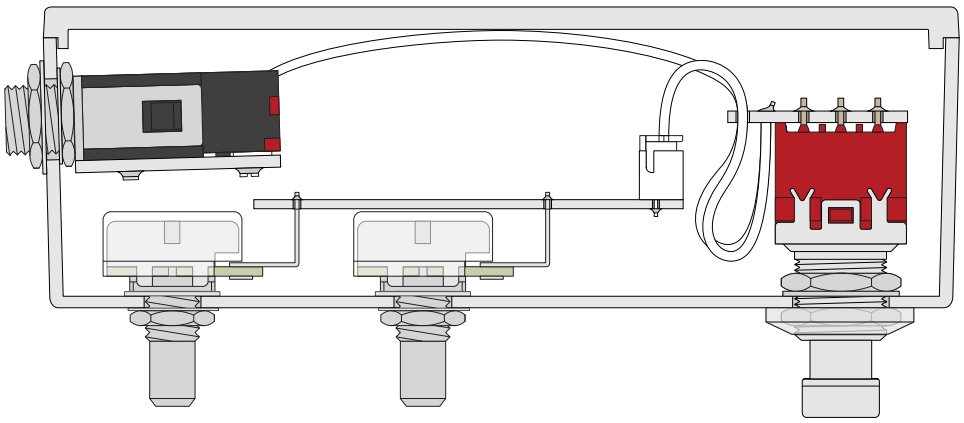
Note the use of two mounting nuts on each of the jacks, one inside and one outside. The inner nut acts as a spacer to set the DC jack flush with the outside of the enclosure. The inner nuts should be threaded as far down as they can go.





FINAL TESTING & ASSEMBLY

After everything is in place, 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.

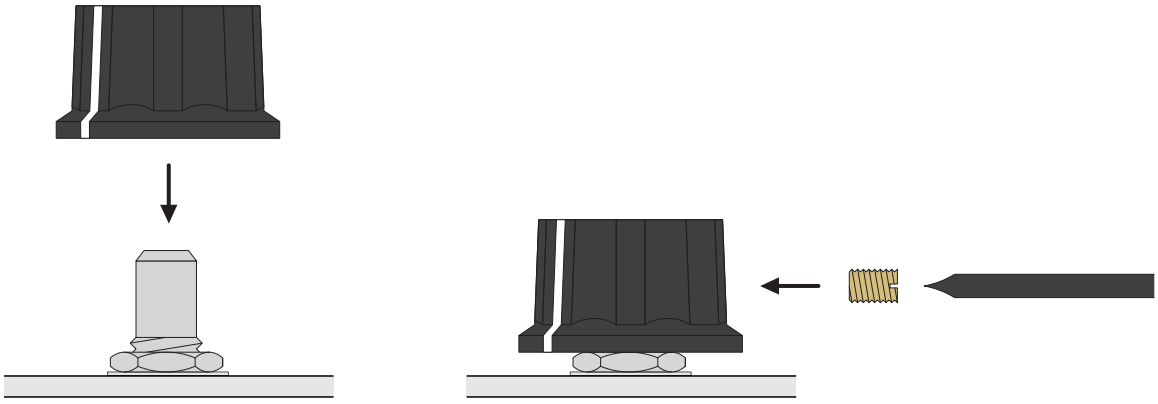
Test the bypass switch a few times, then start turning the knobs and see if everything sounds OK. If it works, great! If not, don't be discouraged. See page 25 for troubleshooting info.

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.

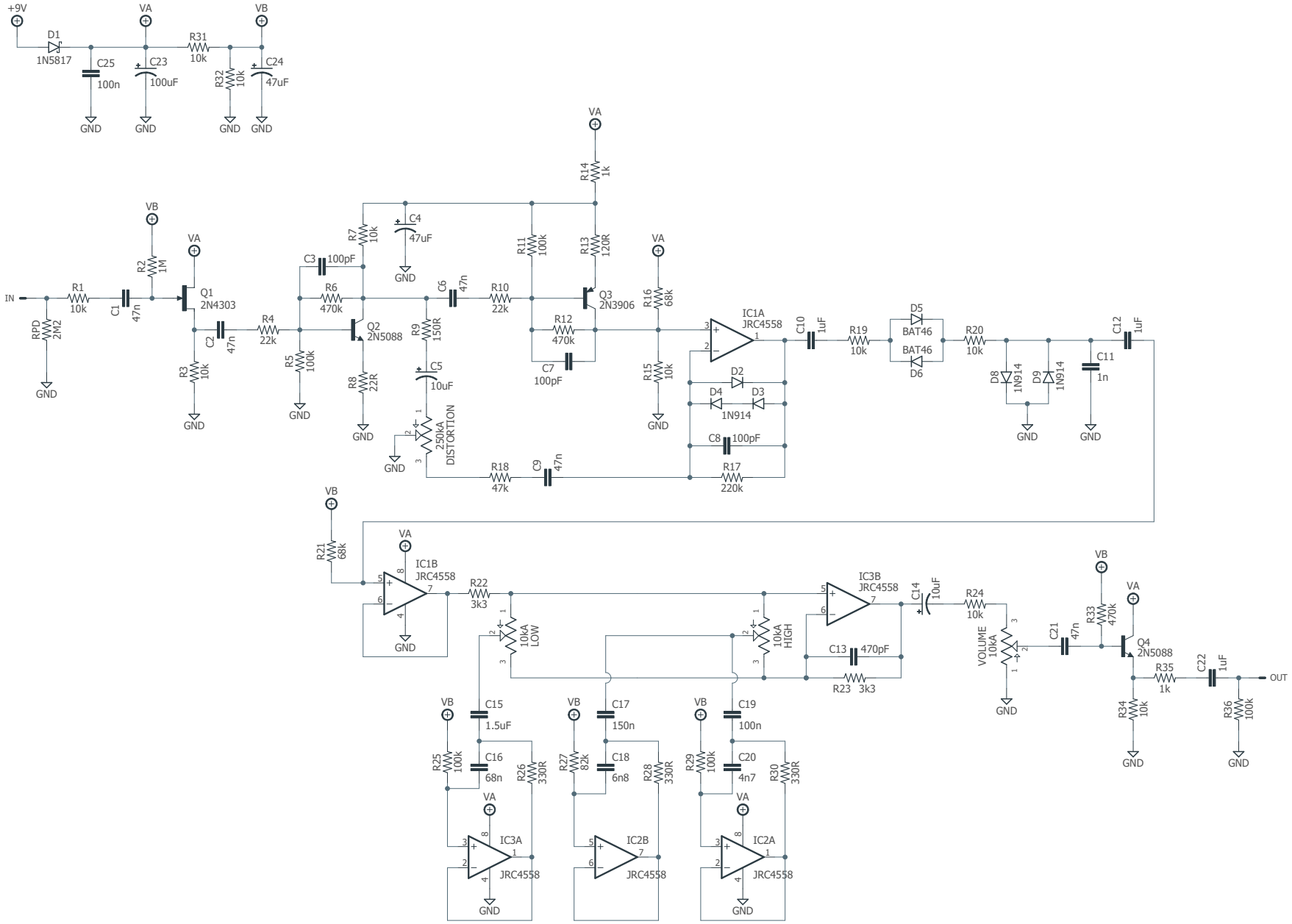
Using a small flat-head screwdriver (no more than 0.1" / 2.5mm in diameter), firmly tighten the set screw until it presses against the shaft of the potentiometer and holds the knob in place.

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


+9V
VA
VB
D1
1N5817
C25
100n
C23
100uF
R31
10k
R32
10k
C24
47uF
GND
R14
1k
C4
47uF
R7
10k
C3
100pF
R6
470k
C6
47n
R10
22k
R11
100k
R13
120R
R16
68k
IN
R1
10k
C1
47n
R2
1M
RPD
2M2
Q1
2N4303
C2
47n
R4
22k
R3
10k
R5
100k
R8
22R
R9
150R
C5
10uF
250kA
DISTORTION
Q2
2N5088
R12
470k
C7
100pF
Q3
2N3906
R15
10k
IC1A
JRC4558
C10
1uF
R19
10k
D5
BAT46
BAT46
D6
R20
10k
D8
1N914
D9
1N914
C12
1uF
C11
1n
D2
D4
D3
1N914
C8
100pF
R17
220k
R18
47k
C9
47n
R21
68k
IC1B
JRC4558
R22
3k3
10kA
LOW
10kA
HIGH
IC3B
JRC4558
C14
10uF
R24
10k
C13
470pF
R23
3k3
VOLUME
10kA
C21
47n
R33
470k
Q4
2N5088
R35
1k
C22
1uF
OUT
R34
10k
R36
100k
C15
1.5uF
C16
68n
R25
100k
R26
330R
C17
150n
C18
6n8
R27
82k
R28
330R
C19
100n
C20
4n7
R29
100k
R30
330R
IC3A
IC2B
IC2A
JRC4558

FULL PARTS LIST

Resistors

PART	VALUE
R1	10k
R2	1M
R3	10k
R4	22k
R5	100k
R6	470k
R7	10k
R8	22R
R9	150R
R10	22k

PART	VALUE
R11	100k
R12	470k
R13	120R
R14	1k
R15	10k
R16	68k
R17	220k
R18	47k
R19	10k
R20	10k

PART	VALUE
R21	68k
R22	3k3
R23	3k3
R24	10k
R25	100k
R26	330R
R27	82k
R28	330R
R29	100k
R30	330R

PART	VALUE
R31	10k
R32	10k
R33	470k
R34	10k
R35	1k
R36	100k
RPD	2M2
LEDR	10k

Capacitors

PART	VALUE
C1	47n film
C2	47n film
C3	100pF MLCC
C4	47uF electro
C5	10uF electro
C6	47n film
C7	100pF MLCC
C8	100pF MLCC
C9	47n film

PART	VALUE
C10	1uF film
C11	1n film
C12	1uF film
C13	470pF MLCC
C14	10uF electro
C15	1.5uF film
C16	68n film
C17	150n film
C18	6n8 film

PART	VALUE
C19	100n film
C20	4n7 film
C21	47n film
C22	1uF film
C23	100uF electro
C24	47uF electro
C25	100n MLCC

Diodes

PART	VALUE
D1	1N5817
D2	1N914
D3	1N914
D4	1N914
D5	BAT46
D6	BAT46
D8	1N914
D9	1N914

ICs

PART	VALUE
IC1	JRC4558D
IC2	JRC4558D
IC3	JRC4558D

Transistors

PART	VALUE
Q1	2N4303
Q2	2N5088
Q3	2N3906
Q4	2N5088

Potentiometers

PART	VALUE
Low	10kA
High	10kA
Volume	10kA
Dist.	250kA

Switches

PART
3PDT stomp

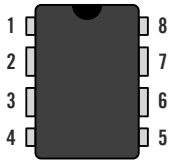
TROUBLESHOOTING INFORMATION

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 **9.88V** 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 transistors and JFETs, the legs will be marked on the PCB.



IC1

PIN	VOLTAGE
1	4.63V
2	4.51V
3	4.51V
4	0V
5	4.77V
6	4.80V
7	4.80V
8	9.60V

IC2

PIN	VOLTAGE
1	4.81V
2	4.81V
3	4.75V
4	0V
5	4.76V
6	4.80V
7	4.80V
8	9.60V

IC3

PIN	VOLTAGE
1	4.81V
2	4.81V
3	4.75V
4	0V
5	4.80V
6	4.80V
7	4.80V
8	9.60V

Q1

PIN	VOLTAGE
D	9.60V
S	7.49V
G	4.36V

Q2

PIN	VOLTAGE
E	0.01V
B	0.62V
C	4.03V

Q3

PIN	VOLTAGE
E	8.69V
B	8.04V
C	4.54V

Q4

PIN	VOLTAGE
E	3.80V
B	4.22V
C	9.60V

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

BOSS® is a registered trademark of Roland Corporation. Other 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.

DOCUMENT REVISIONS

1.0.5 (2024-11-21)
Changed LEDR from 4k7 to 10k to reduce LED brightness slightly.

1.0.4 (2024-08-08)
Added link to troubleshooting guide on page 25.

1.0.3 (2022-07-01)
Added note about blue box film capacitors on page 14.

1.0.2 (2021-05-10)
Revised some diagrams for clarity.

1.0.1 (2019-08-06)
Added C25 (100n ceramic capacitor) for extra power filtering to make it consistent with other Aion FX PCBs. Slight layout tweak to move the D1 polarity-protection diode to the footswitch PCB.

1.0.0 (2018-11-22)
Initial release.