PROJECT NAME

CERULEAN

aion DIY GUITAR EFFECTS

BASED ON
Marshall Bluesbreaker

BUILD DIFFICULTY
▮▮▯▯▯ Easy

EFFECT TYPE
Amp-Like Overdrive

DOCUMENT VERSION
1.1.0 (2020-05-18)

PROJECT SUMMARY
A classic overdrive effect that was designed to mimic the Marshall Bluesbreaker amplifier from the 1960s, the Bluesbreaker is also the source circuit for a number of high-dollar “boutique” guitar pedals.

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

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 40,000 people around the world with a passion for building homemade noise machines using obsolete electronics technologies, 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 25.

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

Film Capacitors

NAME	QTY
10n (0.01uF)	5
100n (0.1uF)	2

Electrolytic Capacitors

NAME	QTY
2.2uF	1
10uF	1
47uF	1
100uF	1

MLCC Capacitors

NAME	QTY
47pF (marked “470”)	1
100pF (marked “101”)	1
100n (marked “104”)	1

Diodes

NAME	QTY
1N5817	1
1N914	10
LED, red	2

Trimmers

NAME	QTY
50k (marked “503”)	1

Resistors

NAME	QTY
1k	1
3k3	1
4k7	2
6k8	2
10k	1
12k	2
22k	1
47k	2
68k	1
100k	1
220k	1
1M	2
2M2	1

ICs

NAME	QTY
TL072	1
8-pin socket	1

Transistors

NAME	QTY
2N4303 or 2N5457	1

PACKING LIST (CONT.)

Potentiometers

NAME	QTY
25kB	1
100kA	1
100kB	1
Dust cover	3
Knob	3
Mounting nut, potentiometer, 0.44"	3
Lock washer, potentiometer, 0.5"	3
Outer washer, potentiometer, 0.475"	3

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

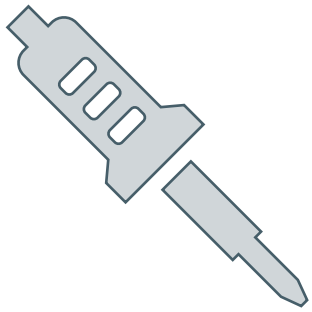
Switches

NAME	QTY
Toggle switch, SPDT on-off-on	2
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

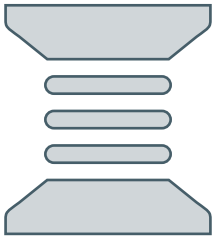
NAME	QTY
3-strand wire assembly, 70mm	2
4-strand wire assembly, 108mm	1
3-pin wire assembly header	2
4-pin wire assembly header	1

TOOLS NEEDED



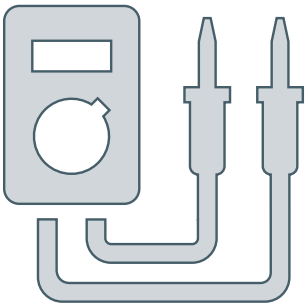
SOLDERING IRON

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.



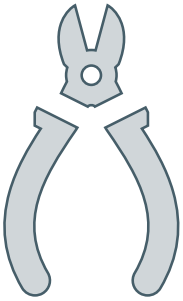
SOLDER

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.



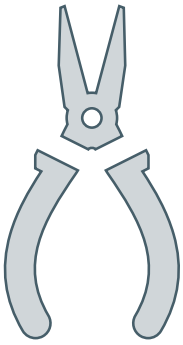
DIGITAL MULTIMETER (DMM)

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.



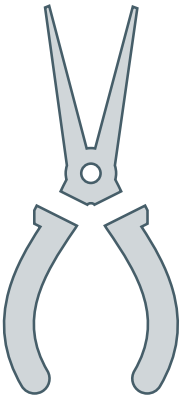
WIRE SNIPPERS

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



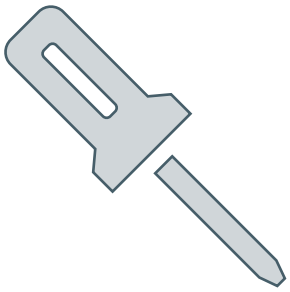
FLAT-NOSE PLIERS

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



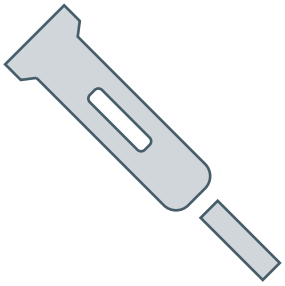
NEEDLE-NOSE PLIERS

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.

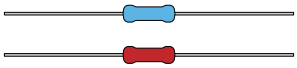
RUBBER BAND

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).

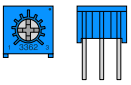
COMPONENT IDENTIFICATION

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.)

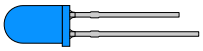
RESISTOR



TRIMMER POTENTIOMETER

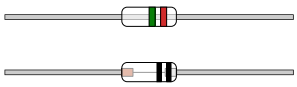


LED



SILICON DIODE

GERMANIUM DIODE



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

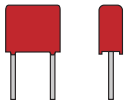
RECTIFIER DIODE

Some Schottky diodes also look like this.

SCHOTTKY DIODE

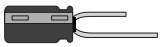
ZENER DIODE

FILM CAPACITOR



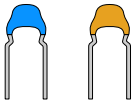
Not polarized. Color may vary by brand and type.

ELECTROLYTIC CAPACITOR



Polarized. The negative side is marked.

MLCC

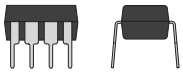


Not polarized. MLCC stands for "multi-layer ceramic capacitor."

TANTALUM CAPACITOR

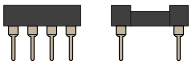
Polarized. The positive side is marked.

OP-AMP / IC

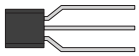


Charge pumps and delay chips also look like this. They may have more than 8 legs.

IC SOCKET



TRANSISTOR OR JFET



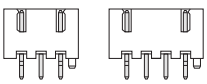
Some voltage regulators also look like this.

SNAP-APART SOCKET

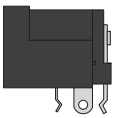
It's recommended to use a razor blade to separate these cleanly.

WIRE ASSEMBLY

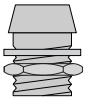
WIRE ASSEMBLY HEADER



DC JACK



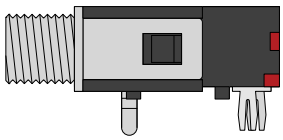
LED BEZEL



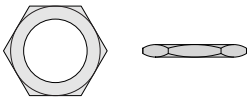
HARDWARE IDENTIFICATION

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

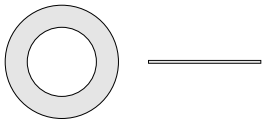


MOUNTING NUT



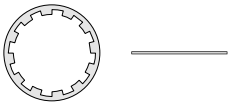
DIAMETER: 0.54" / 13.7mm

OUTER WASHER



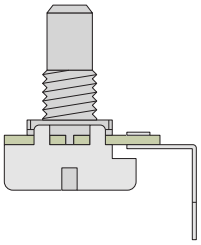
DIAMETER: 0.6" / 15.2mm

LOCK WASHER

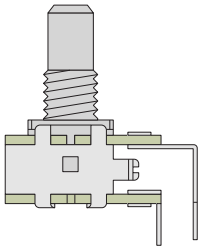


DIAMETER: 0.5" / 12.7mm

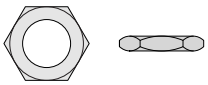
POTENTIOMETER (SINGLE)



POTENTIOMETER (DUAL)

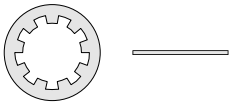


MOUNTING NUT



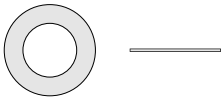
DIAMETER: 0.44" / 11.2mm

LOCK WASHER



DIAMETER: 0.5" / 12.7mm

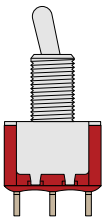
OUTER WASHER



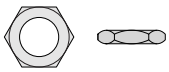
DIAMETER: 0.475" / 12mm

KNOB

TOGGLE SWITCH

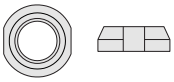


MOUNTING NUT



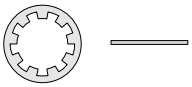
DIAMETER: 0.36" / 9.1mm

DRESS NUT



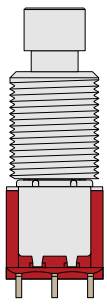
DIAMETER: 0.375" / 9.5mm

LOCK WASHER

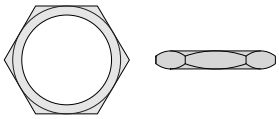


DIAMETER: 0.4" / 10.1mm

FOOTSWITCH

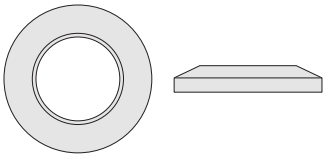


MOUNTING NUT



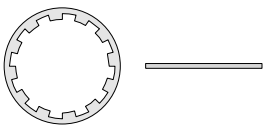
DIAMETER: 0.6" / 15.2mm

DRESS NUT



DIAMETER: 0.77" / 19.6mm

LOCK WASHER

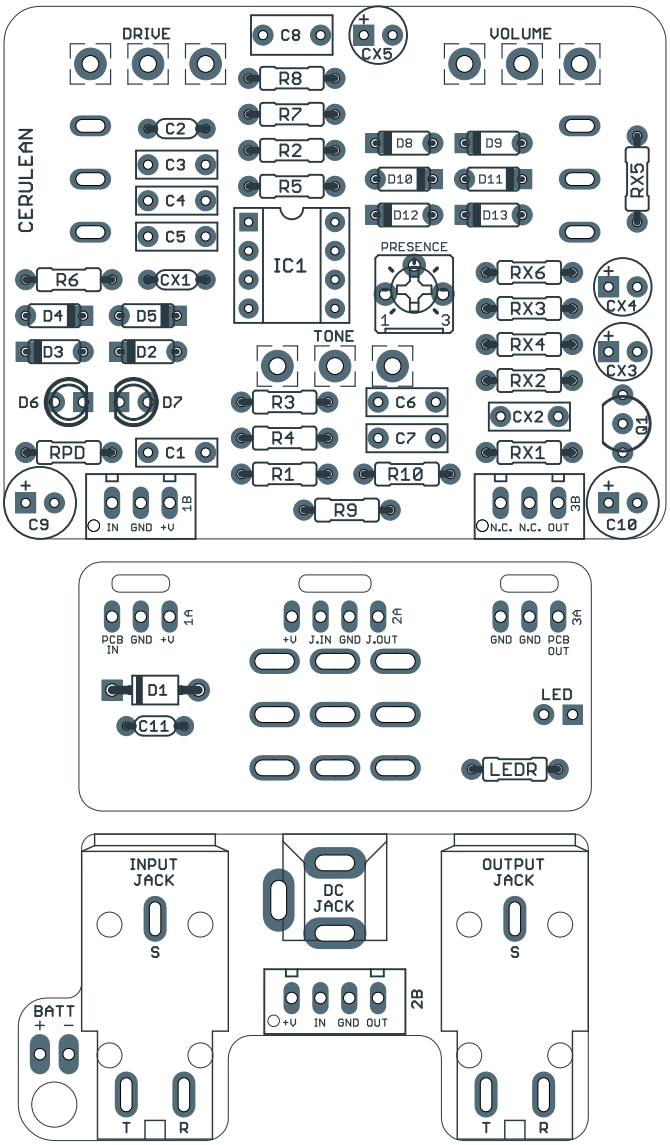


DIAMETER: 0.6" / 15.2mm

PCB ASSEMBLY OVERVIEW

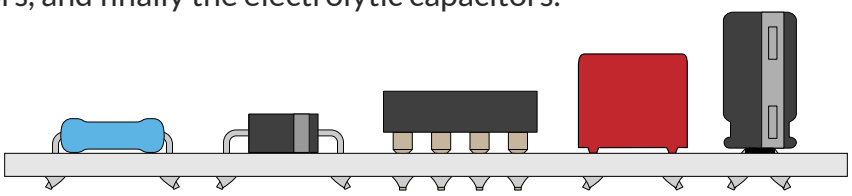
Now it's time to start building!

The first thing you need to do is snap apart the PCB into 3 separate boards (if needed) and break off the tabs from each using needle-nose or flat-head pliers. You should be left with this:



The general principle for PCB population is that you 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.

So, you will start by populating the resistors (the lowest-profile components), followed by the diodes, sockets, film capacitors, and finally the electrolytic capacitors.

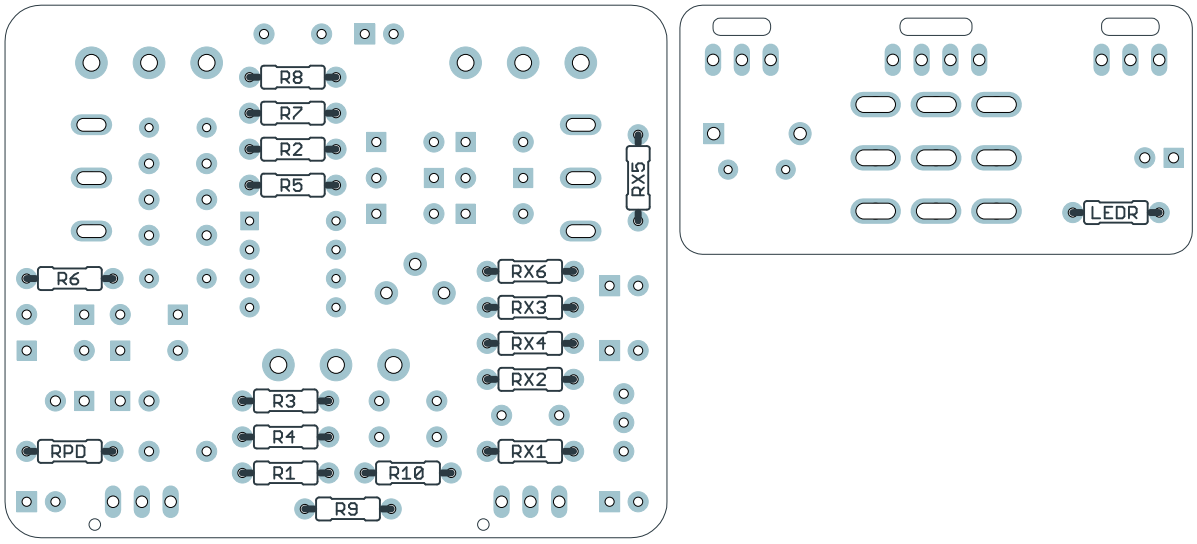


RESISTORS

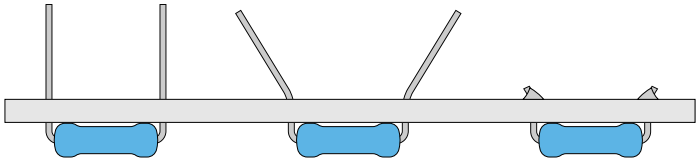
PART	VALUE
R1	1M
R2	3k3
R3	4k7
R4	10k
R5	220k
R6	6k8
R7	1k

PART	VALUE
R8	6k8
R9	47k
R10	47k
RX1	100k
RX2	68k
RX3	1M
RX4	22k

PART	VALUE
RX5	12k
RX6	12k
RPD	2M2
LEDR	4k7



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 15 to 20 resistors at a time or the bottom of the board will get too crowded.

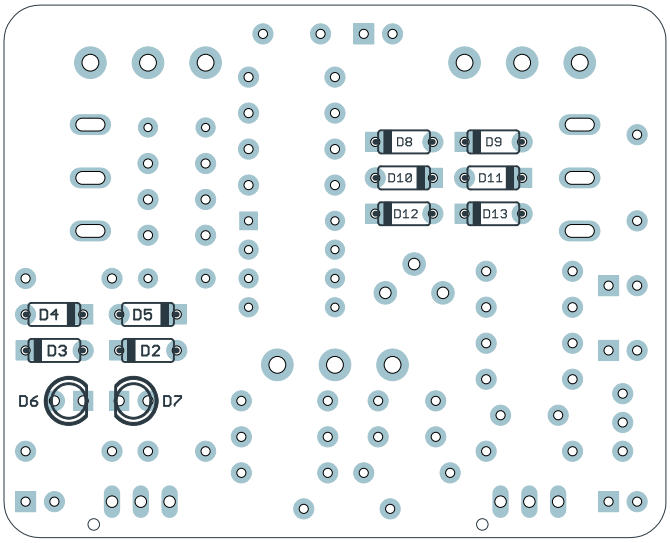
If this is your first time soldering, watch tutorial videos on YouTube and make sure you get it down before you begin. You don't want to practice or experiment on this board!

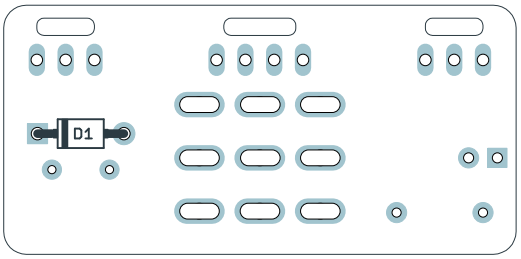
DIODES

PART	VALUE
D1	1N5817
D2	1N914
D3	1N914
D4	1N914
D5	1N914

PART	VALUE
D6	LED, red
D7	LED, red
D8	1N914
D9	1N914
D10	1N914

PART	VALUE
D11	1N914
D12	1N914
D13	1N914



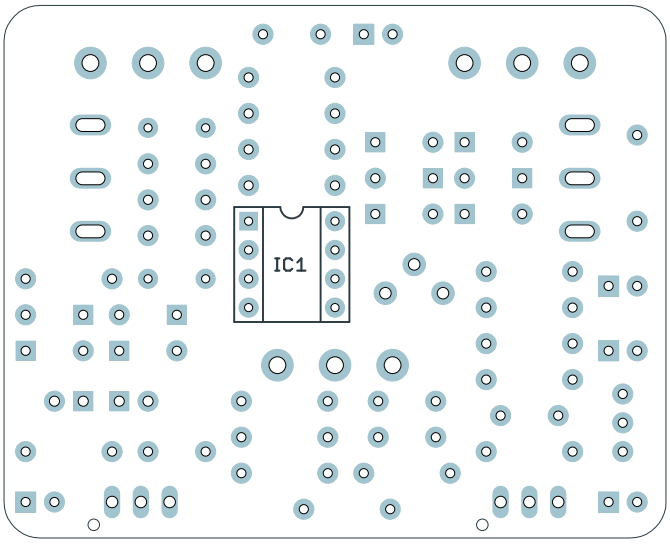


Next, you'll populate the diodes. Diodes are polarized, so make sure to identify the polarity band (which indicates the "cathode", or negative side) and match the band to the footprint on the PCB.

For LEDs, the flat edge indicates the cathode. Line this up with the PCB footprint on the top side of the PCB and solder them flush with the PCB. (These LEDs are used as clipping diodes. The indicator LED will be done in a later step.)

SOCKET & IC

PART	VALUE
IC1	TL072



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

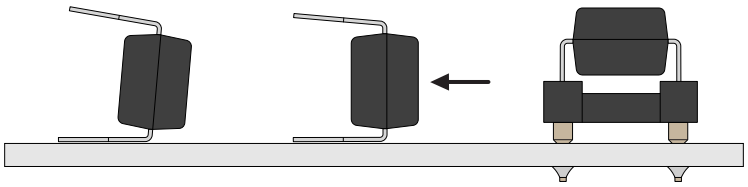
Again, it's much easier to do all of these at once with gravity holding it in place for you, so you'll want do this before you do any of the taller components.

Installing the IC

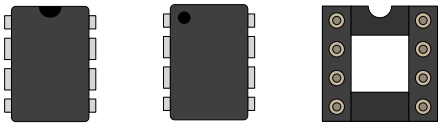
Don't insert the IC into the socket just yet. We will do this in a later step, after we've finished soldering the tallest components (the polarized capacitors). This information is just listed here for reference.

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.

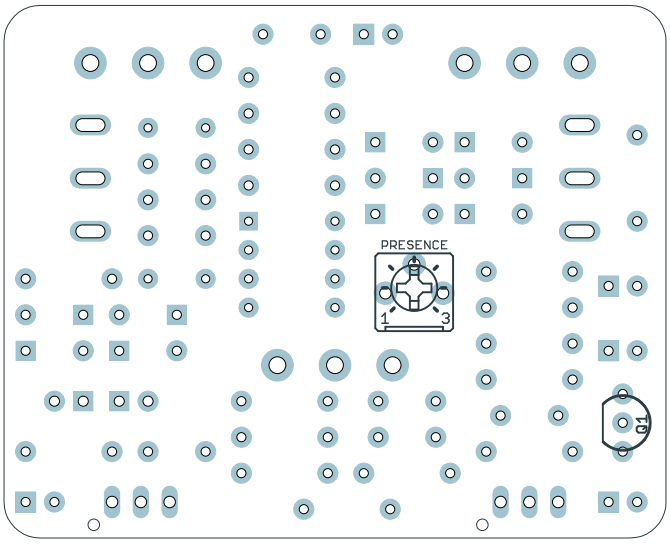


ICs may have two different orientation marks: either a dot in the upper-left or a half-circle notch in the middle of the top side. Some ICs have both marks. This shows which way the IC should be rotated when inserting it into a socket (the socket also has a half-circle notch).

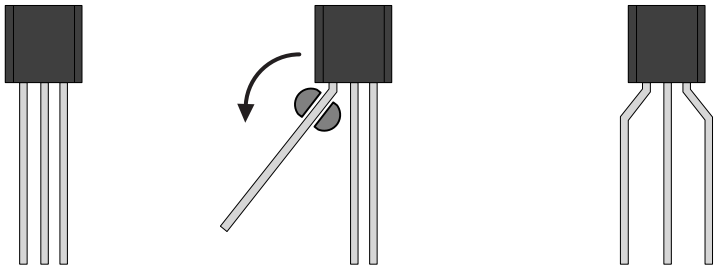


TRANSISTOR & TRIMMER

PART	VALUE
Q1	2N5457 or PN4303
PRES	50k trimmer

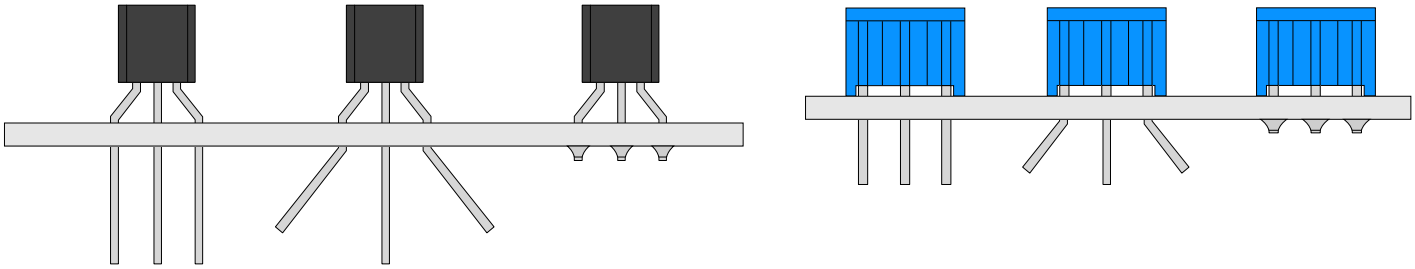


Now we'll do the JFET transistor and Presence trimmer. For the JFET, if the legs are not already bent into 0.1" spacing, use your needle-nose pliers to bend the outer two legs as shown.



Since this is just used as a simple gain stage, it doesn't need to be selected for any particular characteristics, so sockets are not necessary. You can just solder it directly to the board.

Bend the outer leads of both the JFET and trimmer to hold them in place on the board. Then, solder them and clip the leads.



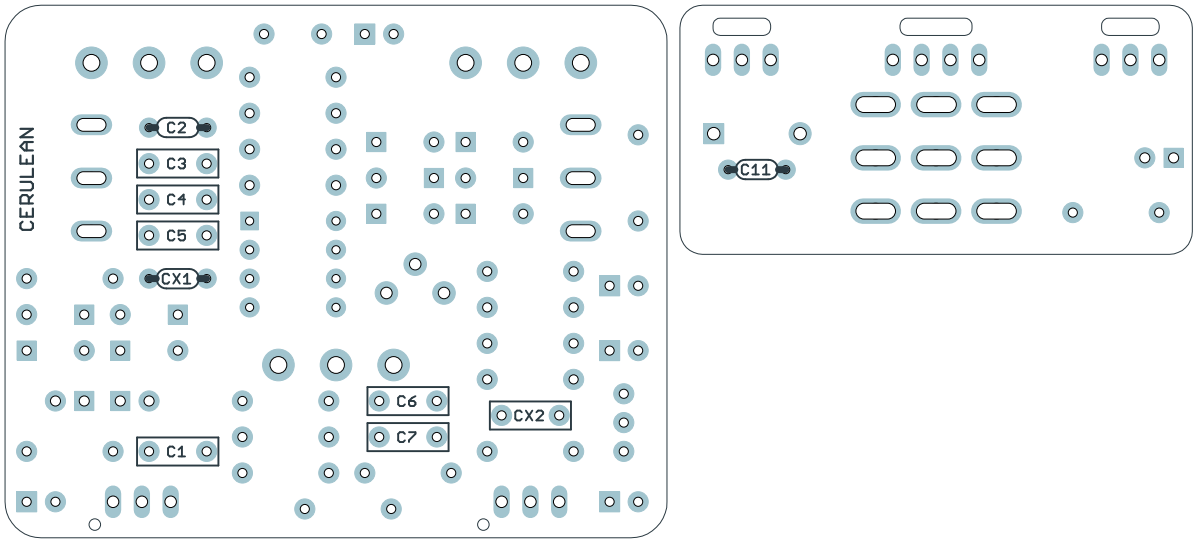
Setting the trimmer

The Presence trimmer allows you reduce the steep high-end cut that is normally applied after the tone pot. Start with it full up (clockwise), which is the stock Bluesbreaker circuit. From there, you can back it down as much as you want to increase the brightness. It's very interactive with the tone control, so test it with the tone control in several different positions to make sure you like the range.

CAPACITORS (NON-POLARIZED)

PART	VALUE
C1	10n (may also be marked “0.01uF” or “103”)
C2	47pF MLCC (may also be marked “471”)
C3	10n (may also be marked “0.01uF” or “103”)
C4	10n (may also be marked “0.01uF” or “103”)
C5	100n (may also be marked “0.1uF” or “104”)
C6	10n (may also be marked “0.01uF” or “103”)

PART	VALUE
C7	10n (may also be marked “0.01uF” or “103”)
C8	*leave empty*
C11	100n MLCC (may also be marked “104”)
CX1	100pF MLCC (may also be marked “101”)
CX2	100n (may also be marked “0.1uF” or “104”)

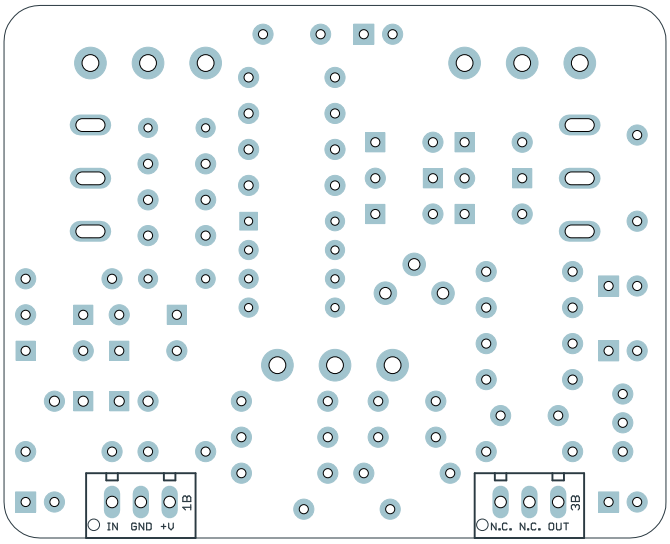


After the sockets come the box film and MLCC capacitors. These are all several different heights, but there aren’t as many, so just do them all at once. Bend the leads at an angle to hold them in place.

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.

Note: C8 is not used in this kit and will be empty on the PCB.

WIRE HEADERS

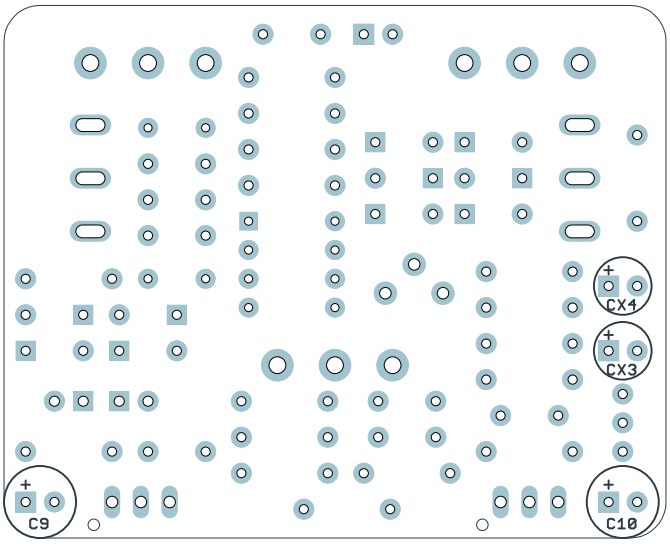


Install the two 3-pin headers (wire connectors) as shown above. These have a polarity pin, so as long as they are pressed all the way down, there's only one possible way to install them. They do fit pretty tightly in the holes, though, so press firmly.

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

CAPACITORS (POLARIZED)

PART	VALUE
C9	100uF
C10	47uF
CX3	10uF
CX4	2.2uF
CX5	*leave empty*



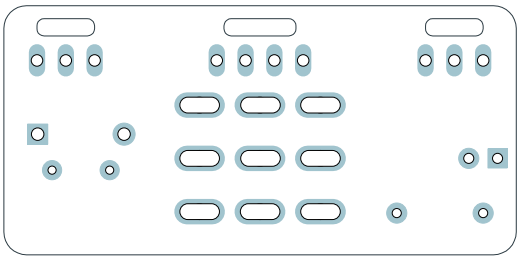
Populate the electrolytic capacitors. These are the tallest components so we save them for last. They are polarized (i.e. they will only work in one direction), so note the vertical mark that indicates the negative side. The longer leg is positive and fits in the square pad.

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.

Note: CX5 is not used in this kit and should be left empty on the PCB.

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 silkscreen facing up.

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

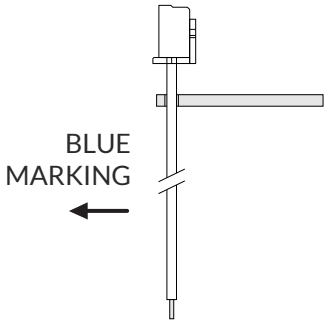
STEP 2

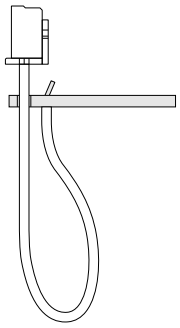
Next, bend the wires back upward and fit the ends of the wires into the solder pads.

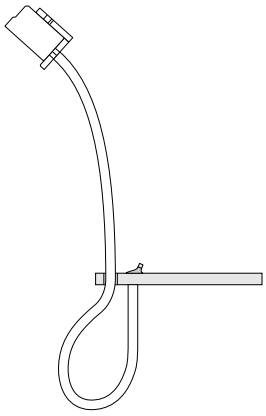
On the top side of the PCB, bend the exposed wires backward so it holds the wire in place. Pull the header back up through the slot partway.

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.





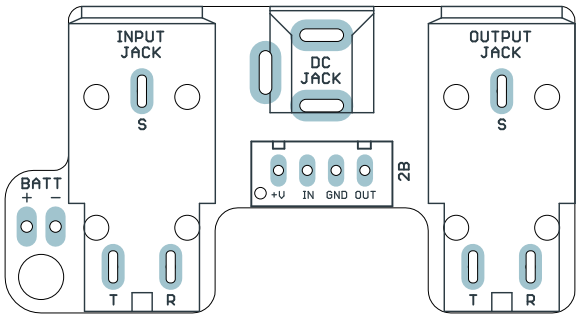


Once all three wire assemblies are soldered, set the footswitch PCB aside. We'll solder the actual footswitch and LED in a later step.

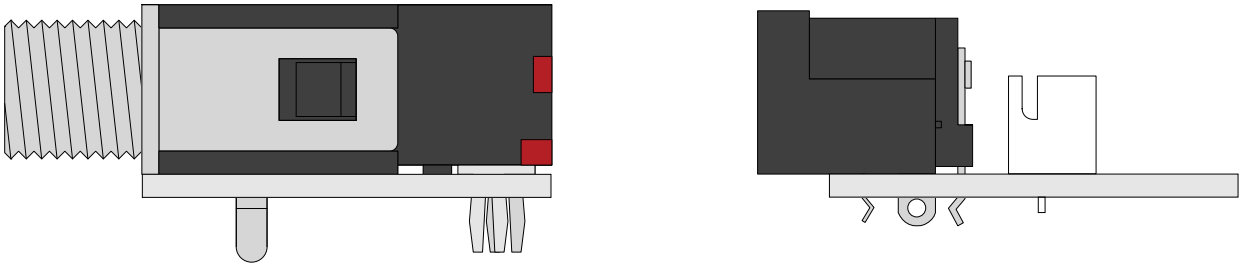
INPUT/OUTPUT PCB

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.

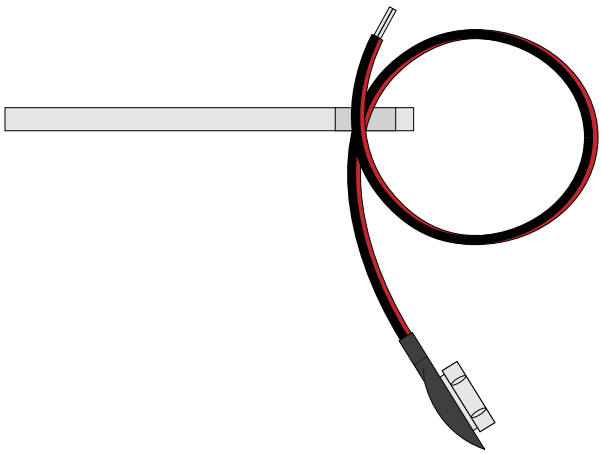


After you've soldered everything, make sure to **snip the leads on the I/O jacks as close as possible to the PCB**. There's not a lot of clearance between the bottom of this board and the top of the main PCB once everything is in place, and you don't want the pins to short against anything on accident.

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

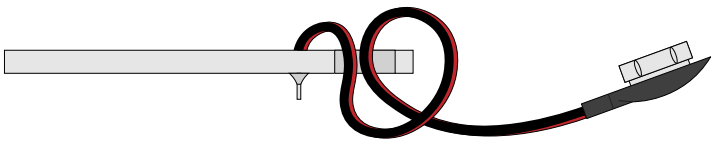
Thread the battery snap leads through the strain-relief hole twice so it forms a single loop.



STEP 2

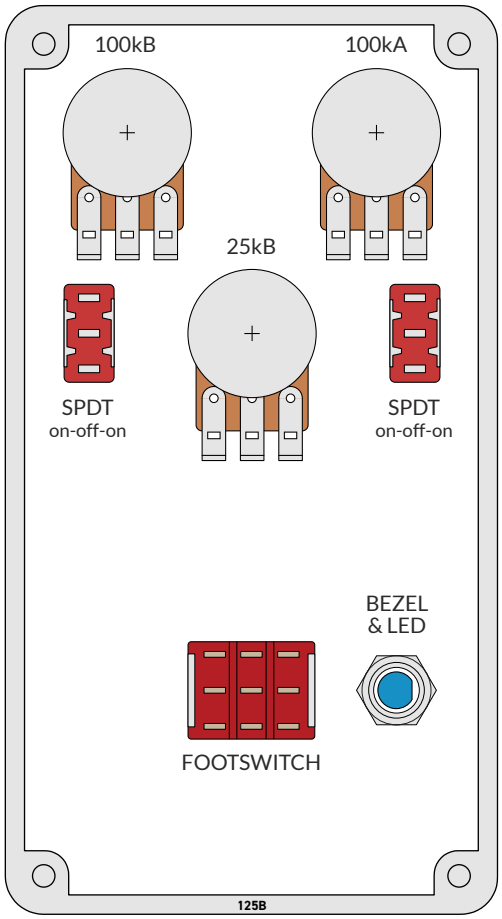
Bend the exposed wires back down and solder them into the pads. Red is positive (+), black is negative (-). After soldering, pull it tight.

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

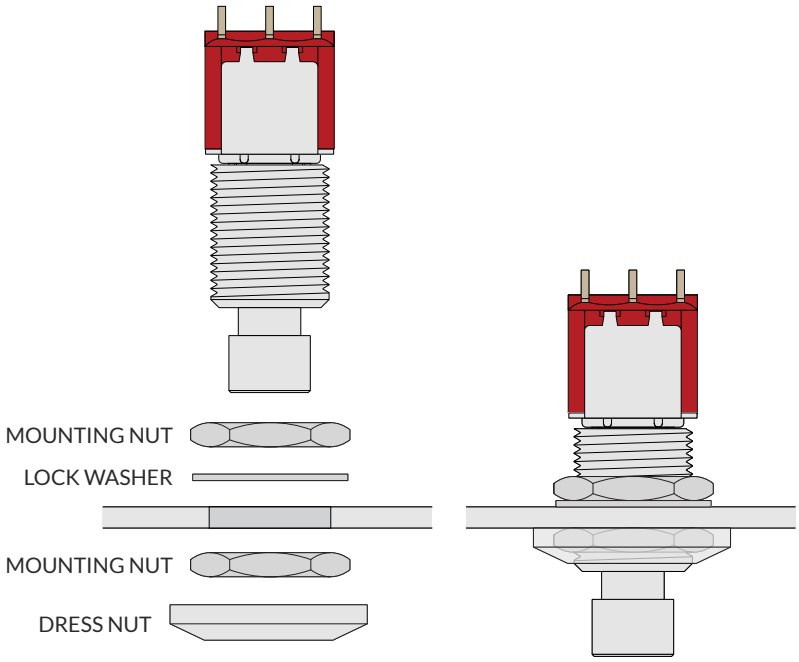


ENCLOSURE LAYOUT: PANEL MOUNTS

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



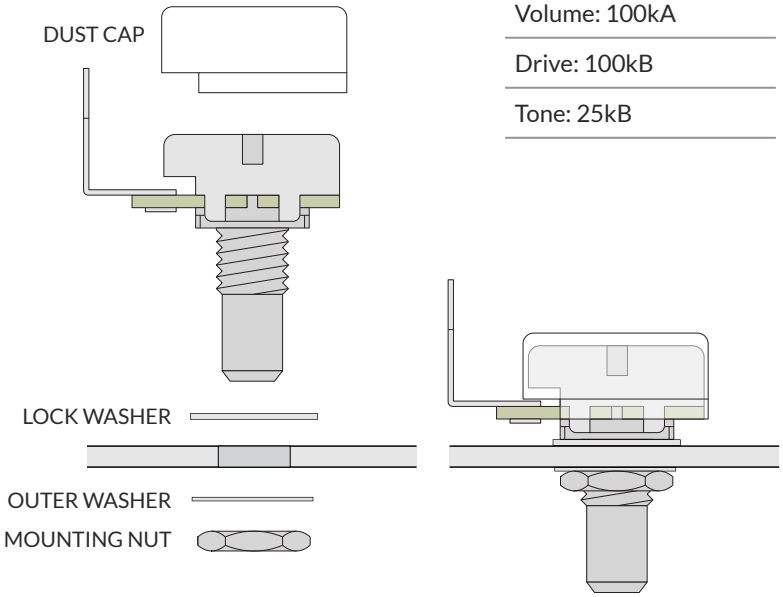
FOOTSWITCH



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.

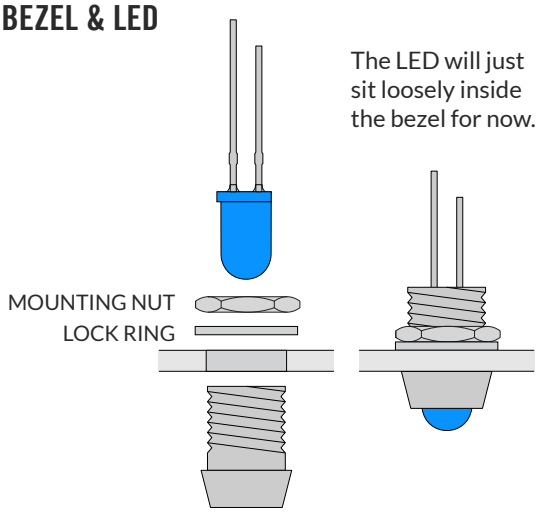
POTENTIOMETERS

Volume: 100kA
Drive: 100kB
Tone: 25kB



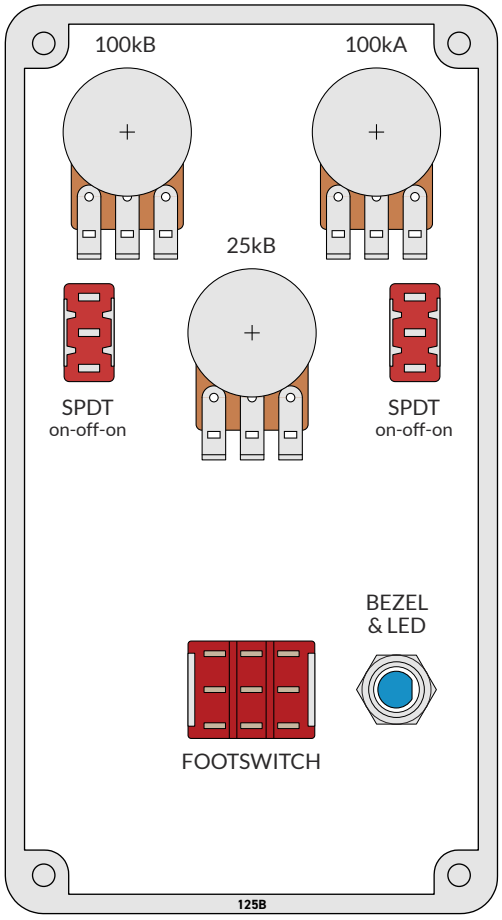
BEZEL & LED

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

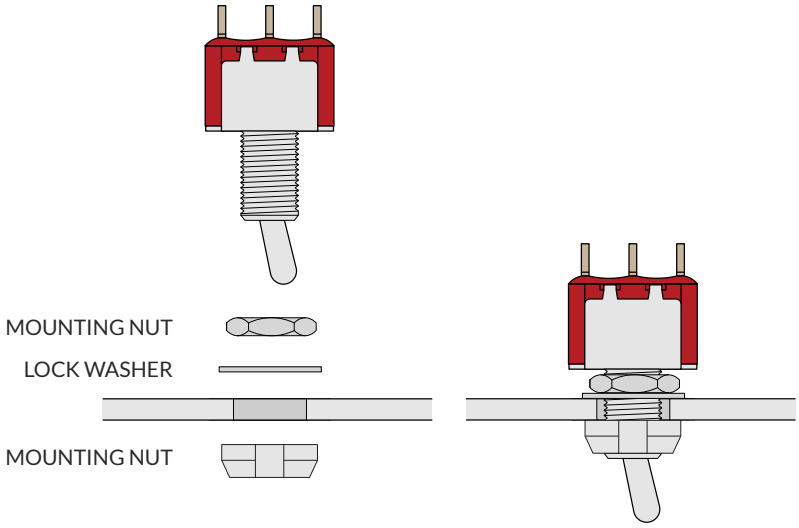


You'll need to hold the bezel in place when tightening the nut. The top of the bezel is fairly sharp, so try using a rubber band for grip instead of pressing your finger against the bottom.

ENCLOSURE LAYOUT: PANEL MOUNTS (CONT.)

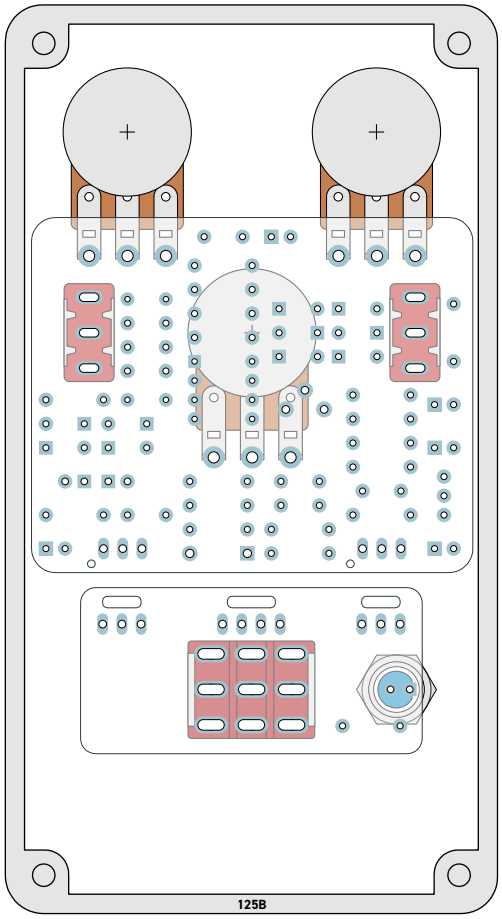


TOGGLE SWITCHES



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 toggles 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?

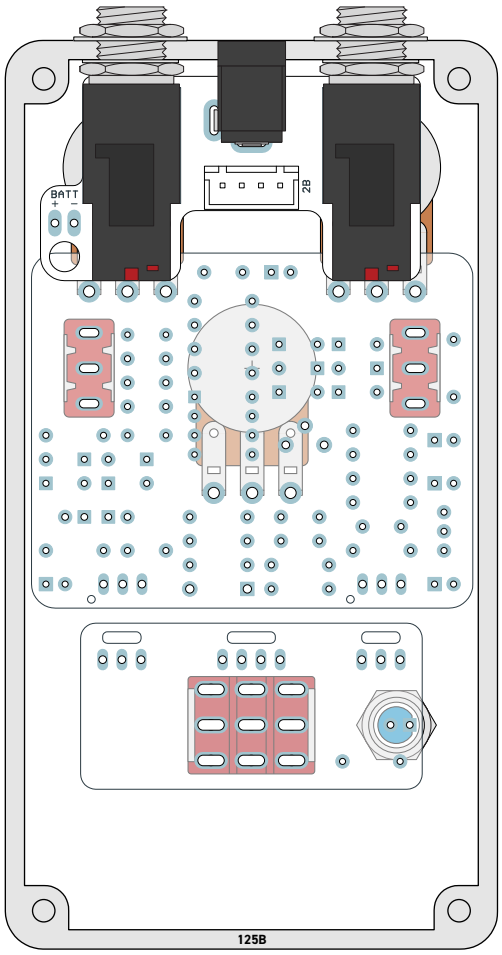
"Rock it before you box it" is conventional wisdom in pedalbuilding, and you'll often hear it recommended that builders should test the circuit before putting everything inside the enclosure. However, Aion FX projects are designed to be extremely easy to remove from the enclosure for troubleshooting,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

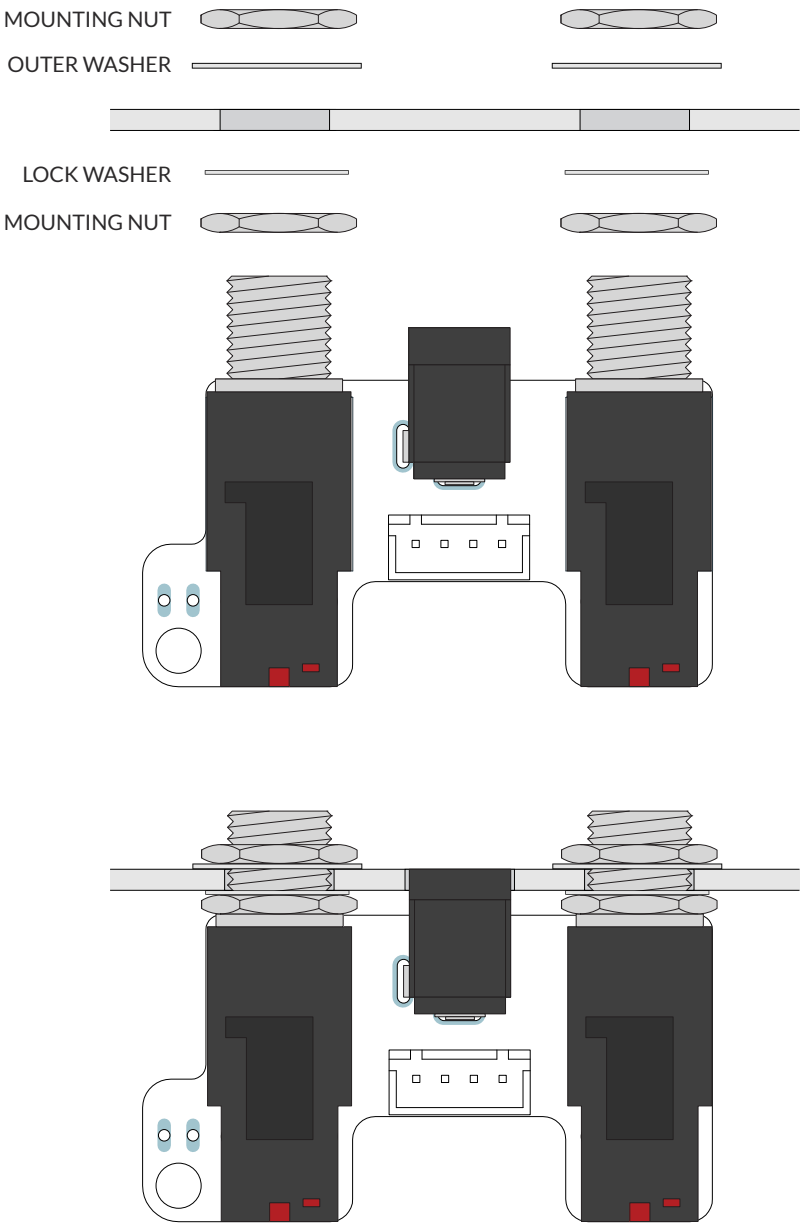
If you've read the documentation carefully and followed all the instructions, there's a good chance you will get it right the first time!

ENCLOSURE LAYOUT: INPUT/OUTPUT PCB

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

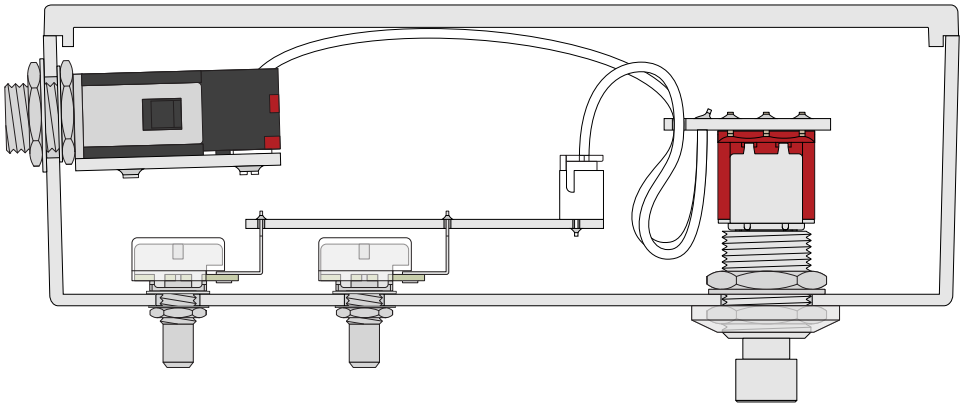
Note the use of two mounting nuts on each of the jacks, one inside and one outside. The inner nut acts as a spacer to set the DC jack flush with the outside of the enclosure. The inner nuts should be threaded as far down as they can go.





FINAL TESTING & ASSEMBLY

After everything is in place, just plug the 3 wire assemblies into their respective headers and make sure they're secure. That's it! 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.

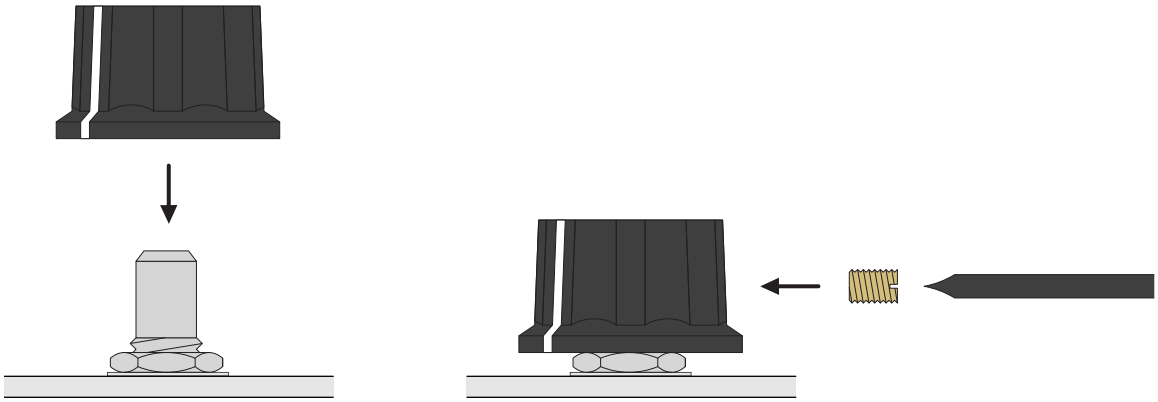
Test the bypass switch a few times, then start turning the knobs and see if everything sounds OK. If it works, great! If not, don't be discouraged. See page 26 for troubleshooting info.

Finishing touches

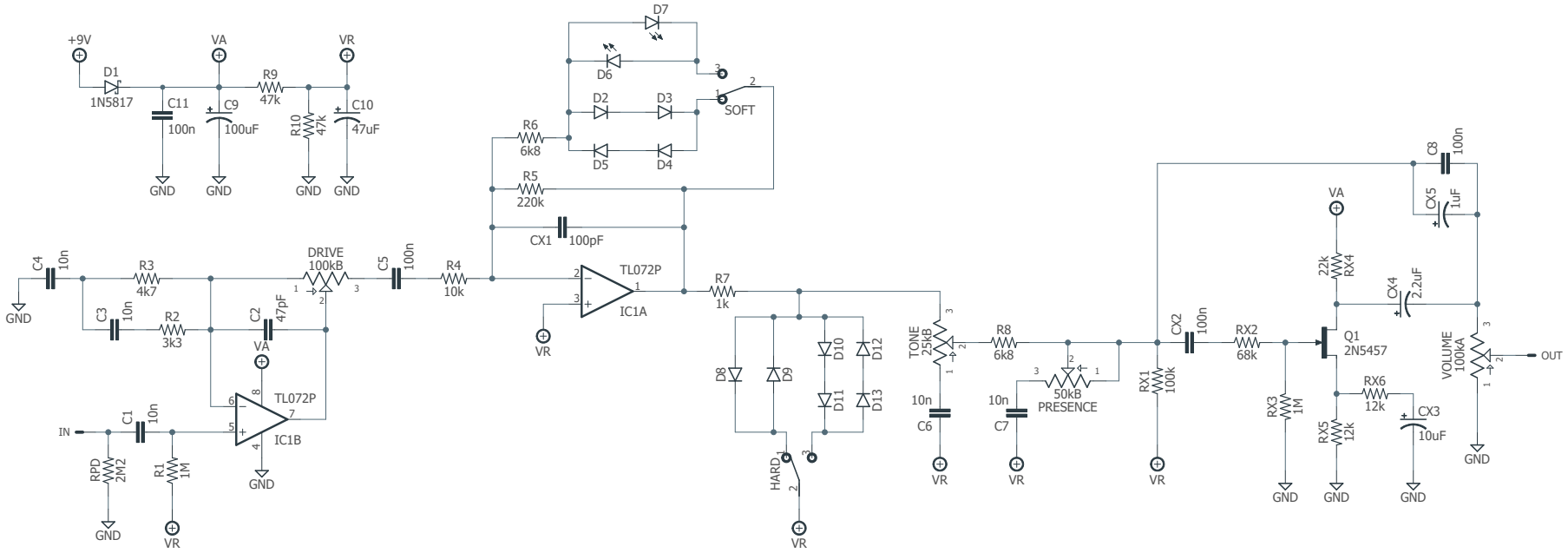
Now, just a couple of things for the final assembly. Turn the shafts all fully counter-clockwise, then put on the knob and rotate until the indicator line is aligned with the dot on the enclosure that shows the zero point. Affix the knobs to each of the potentiometer shafts as shown in the diagram below.

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

Don't over-tighten or you could damage the set screw. But on the other hand, if it's not tight enough then the knob will be prone to falling off or losing its alignment with the markings on the enclosure.



Last, just close the panel on the back using the four screws. Before that, though, grab a permanent marker and write your name and the completion date on the inside of the back panel. This is an accomplishment!


+9V
VA
VR
D1
1N5817
R9
47k
C11
100n
C9
100uF
R10
47k
C10
47uF
GND
D7
D6
D2
D3
SOFT
R6
6k8
D5
D4
R5
220k
CX1
100pF
C4
10n
R3
4k7
DRIVE
100kB
C5
100n
R4
10k
TL072P
IC1A
R7
1k
C3
10n
R2
3k3
C2
47pF
TL072P
IC1B
IN
C1
10n
RPD
2M2
R1
1M
D8
D9
D10
D11
D12
D13
HARD
TONE
25kB
10n
C6
R8
6k8
50kB
PRESENCE
10n
C7
RX1
100k
CX2
100n
RX2
68k
C8
100n
CX5
1uF
22k
RX4
CX4
2.2uF
Q1
2N5457
RX3
1M
RX6
12k
RX5
12k
CX3
10uF
VOLUME
100kA
OUT

FULL PARTS LIST

In this document, the parts list is spread out across several pages by step. For more experienced builders, though, it may be easier to have everything in one place.

Resistors

PART	VALUE
R1	1M
R2	3k3
R3	4k7
R4	10k
R5	220k
R6	6k8
R7	1k
R8	6k8

PART	VALUE
R9	47k
R10	47k
RX1	100k
RX2	68k
RX3	1M
RX4	22k
RX5	12k
RX6	12k

PART	VALUE
RPD	2M2
LEDR	4k7

Trimmers

PART	VALUE
PRES	50kB

Capacitors

PART	VALUE
C1	10n (0.01uF)
C2	47pF MLCC
C3	10n (0.01uF)
C4	10n (0.01uF)
C5	100n (0.1uF)
C6	10n (0.01uF)
C7	10n (0.01uf)
C9	100uF electro

PART	VALUE
C10	47uF electro
C11	100n MLCC
CX1	100pF MLCC
CX2	100n (0.1uF)
CX3	10uF electro
CX4	2.2uF electro

Diodes

PART	VALUE
D1	1N5817
D2	1N914
D3	1N914
D4	1N914
D5	1N914
D6	LED, red
D7	LED, red

PART	VALUE
D8	1N914
D9	1N914
D10	1N914
D11	1N914
D12	1N914
D13	1N914

Transistors

PART	VALUE
Q1	PN4303 or 2N5457

ICs

PART	VALUE
IC1	TL072

Potentiometers

PART	VALUE
Volume	100kA
Drive	100kB
Tone	25kB

Switches

PART
SPDT on-off-on (2)
3PDT stomp

TROUBLESHOOTING INFORMATION

What happens if you finish building the kit and find that it doesn't work right? Here are a few common problems people have with this pedal and how to solve them.

The LED doesn't light up.

First, does the pedal sound right? If you aren't getting any sound, you probably have a power issue with the whole circuit that is not specific to the LED, so you'll want to look elsewhere for the problem.

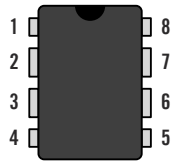
If it does pass a signal, it's probably just the LED itself. Is the flat side facing to the right (looking in the enclosure from the back)? If it's reversed, it won't work. You'll have to re-solder it the right way.

All other issues

For any other problems, the first course of action is to measure the voltages on each pin of the ICs using a digital multimeter. Set the multimeter to DC mode with a range of 20V or higher.

To start, touch the black lead to a ground point for the circuit. The easiest spot is inside a tapped screw hole in one of the corners of the enclosure. This way the probe stays in place without needing to use alligator clips. The circuit must of course be fully installed in the enclosure for this to work.

Then, touch the red probe to the first leg of IC1 and read the voltage. Note that IC pins are labeled counter-clockwise from the upper-left, as shown in the diagram to the right.



These baseline voltages are taken using a **9.87V** supply. Your measured voltages won't be exactly the same due to variance in power supplies and component tolerances. However, if you see anything that's more than around 0.5V higher or lower than these listed voltages, it's a good indicator of an issue and the voltages can help you or someone else narrow it down.

Q1

PIN	VOLTAGE
1	7.79V
2	1.00V
3	0V

IC1

PIN	VOLTAGE
1	4.84V
2	4.84V
3	4.82V
4	0V
5	4.38V
6	4.83V
7	4.83V
8	9.65V

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 two best places to ask for help are the DIY Stompboxes forum and the DIY Stompboxes Facebook group. Both communities have thousands upon thousands of members 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

Bluesbreaker® is a trademark of Marshall Amplification PLC.

All 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

1.1.0 (2020-05-18)

- Corrected part numbering of two electrolytic capacitors (C9 and C10).
- Slight PCB layout change, relocating D1 to footswitch board.
- Added C11 MLCC capacitor to footswitch board for improved power filtering & consistency with other projects.
- Added diagram showing installation of Presence trimmer.

1.0.0 (2018-11-22)
Initial release.