PROJECT NAME

REFRACTOR

aion DIY GUITAR EFFECTS

BASED ON
Klon Centaur / KTR

BUILD DIFFICULTY
Intermediate

EFFECT TYPE
Overdrive

DOCUMENT VERSION
1.0.6 (2024-08-08)

PROJECT SUMMARY
A part-for-part replica of a mythical overdrive effect noted for its high-end tone and price. The Klon Centaur and its successor, the KTR, remain highly original designs in an industry full of clones and tweaks to existing circuits.

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 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 23.

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

Film Capacitors

NAME	QTY
2n2	1
3n9	1
27n (0.027)	1
68n (0.068)	2
82n (0.082)	1
100n (0.1 or "µ1J100")	2
390n (0.39)	1
1uF	2

Electrolytic Capacitors

NAME	QTY
4.7uF	2
10uF	4
47uF	1
100uF	1

Tantalum Capacitors

NAME	QTY
1uF (marked "105")	1

MLCC Capacitors

NAME	QTY
390pF (marked "391")	1
820pF (marked "821")	1

Diodes

NAME	QTY
1N4001	2
1N4742	1

Resistors

NAME	QTY
560R	2
1k	2
1k5	2
1k8	1
2k	1
4k7	1
5k1	1
10k	3
12k	1
15k	2
22k	1
27k	3
47k	1
68k	2
100k	4
392k	1
422k	1
2M	2

ICs

NAME	QTY
TL072	2
LT1054	1
8-pin socket	3

PACKING LIST (CONT.)

Potentiometers

NAME	QTY
10kB	2
100kB dual	1
Dust cover	2
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
D9E germanium diode	2
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

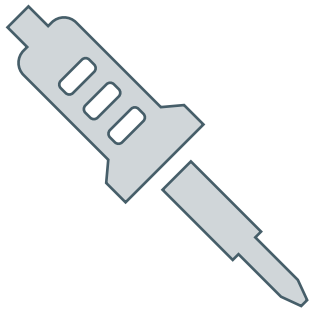
Switches

NAME	QTY
Slide switch, 4PDT	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

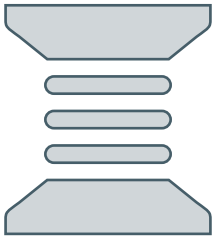
NAME	QTY
3-strand wire assembly, 70mm	2
4-strand wire assembly, 108mm	1
3-pin wire assembly header	2
4-pin wire assembly header	1

TOOLS NEEDED



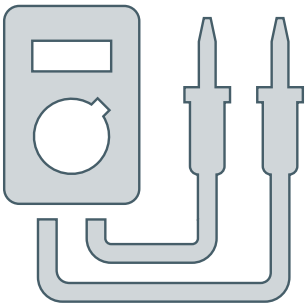
SOLDERING IRON

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.



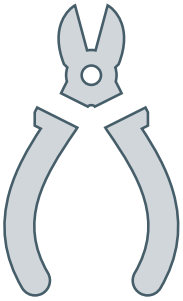
SOLDER

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.



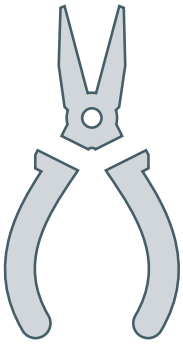
DIGITAL MULTIMETER (DMM)

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.



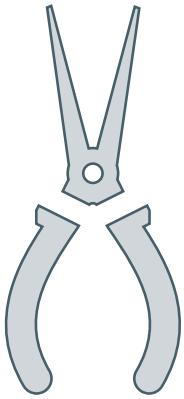
WIRE SNIPPERS

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



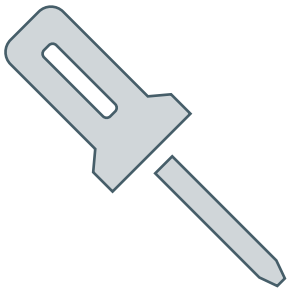
FLAT-NOSE PLIERS

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



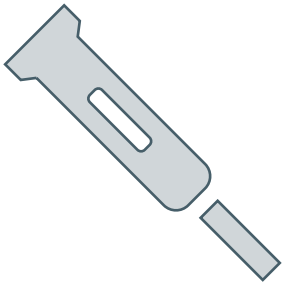
NEEDLE-NOSE PLIERS

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.



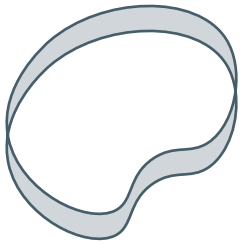
SCREWDRIVER (PHILLIPS)

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.



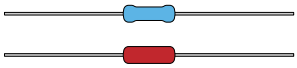
RUBBER BAND

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).

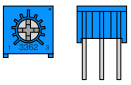
COMPONENT IDENTIFICATION

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.)

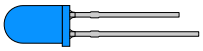
RESISTOR



TRIM POTENTIOMETER

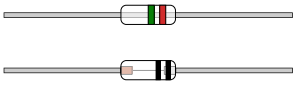


LED



SILICON DIODE

GERMANIUM DIODE



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

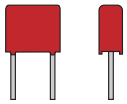
RECTIFIER DIODE

Some Schottky diodes also look like this.

SCHOTTKY DIODE

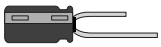
ZENER DIODE

FILM CAPACITOR



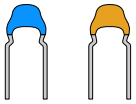
Not polarized. Color may vary by brand and type.

ELECTROLYTIC CAPACITOR



Polarized. The negative side is marked.

MLCC

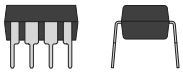


Not polarized. MLCC stands for "multi-layer ceramic capacitor."

TANTALUM CAPACITOR

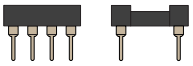
Polarized. The positive side is marked.

IC OR OP-AMP

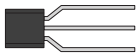


Charge pumps and delay chips also look like this. They may have more than 8 legs.

IC SOCKET



TRANSISTOR OR JFET



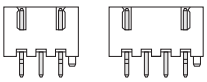
Some voltage regulators also look like this.

SNAP-APART SOCKET

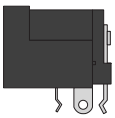
It's recommended to use a razor blade to separate these cleanly.

WIRE ASSEMBLY

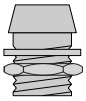
WIRE ASSEMBLY HEADER



DC JACK



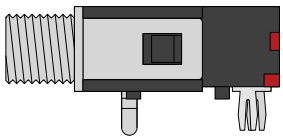
LED BEZEL



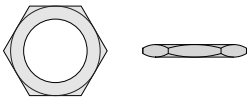
HARDWARE IDENTIFICATION

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

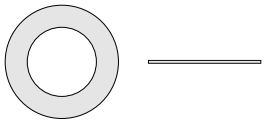


MOUNTING NUT



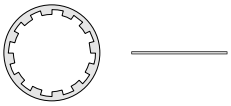
DIAMETER: 0.54" / 13.7mm

OUTER WASHER



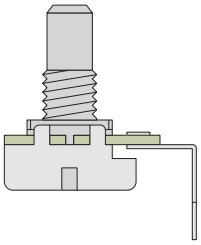
DIAMETER: 0.6" / 15.2mm

LOCK WASHER

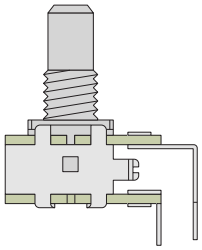


DIAMETER: 0.5" / 12.7mm

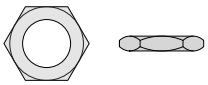
POTENTIOMETER (SINGLE)



POTENTIOMETER (DUAL)

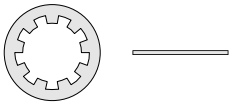


MOUNTING NUT



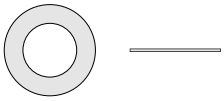
DIAMETER: 0.44" / 11.2mm

LOCK WASHER



DIAMETER: 0.5" / 12.7mm

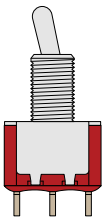
OUTER WASHER



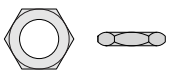
DIAMETER: 0.475" / 12mm

KNOB

TOGGLE SWITCH

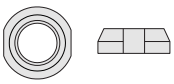


MOUNTING NUT



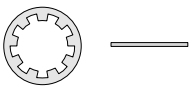
DIAMETER: 0.36" / 9.1mm

DRESS NUT



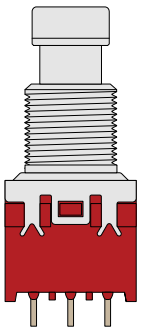
DIAMETER: 0.375" / 9.5mm

LOCK WASHER

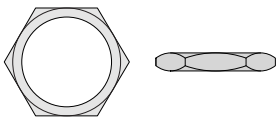


DIAMETER: 0.4" / 10.1mm

FOOTSWITCH

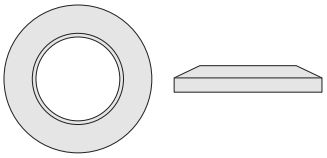


MOUNTING NUT



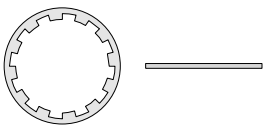
DIAMETER: 0.6" / 15.2mm

DRESS NUT



DIAMETER: 0.77" / 19.6mm

LOCK WASHER

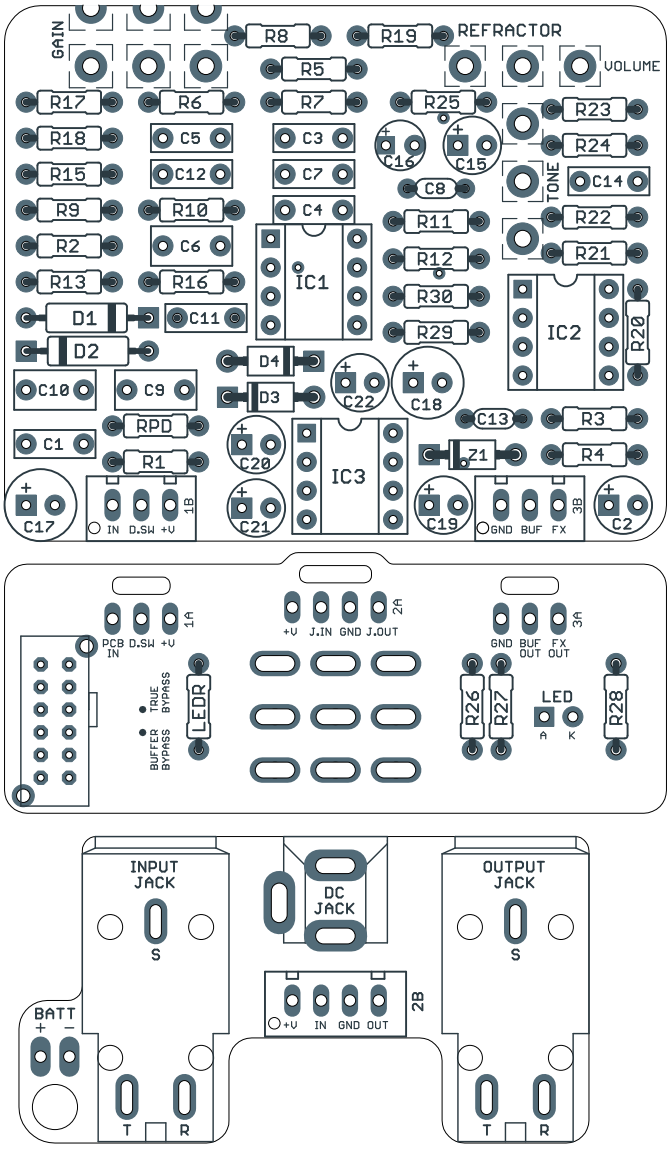


DIAMETER: 0.6" / 15.2mm

MAIN PCB: OVERVIEW

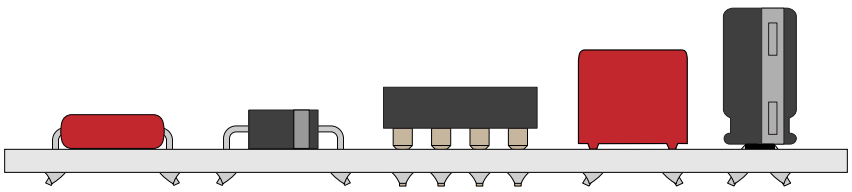
Now it's time to start building!

The first thing you need to do is snap apart the PCB into 3 separate boards 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.



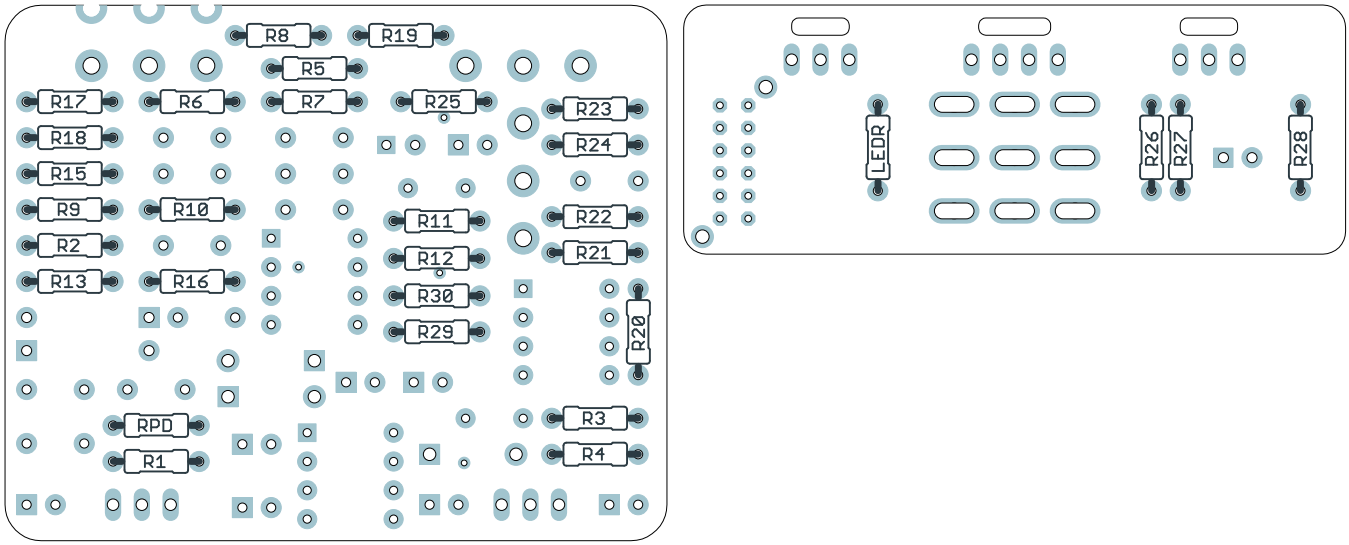
MAIN PCB: RESISTORS

PART	VALUE
R1	10k
R2	2M
R3	100k
R4	560R
R5	5k1
R6	10k
R7	1k5
R8	1k5

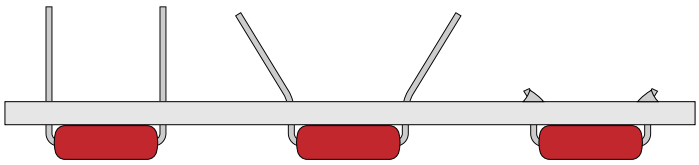
PART	VALUE
R9	1k
R10	2k
R11	15k
R12	422k
R13	1k
R15	22k
R16	47k
R17	27k

PART	VALUE
R18	12k
R19	15k
R20	392k
R21	1k8
R22	100k
R23	4k7
R24	100k
R25	560R

PART	VALUE
R26	68k
R27	68k
R28	100k
R29	27k
R30	27k
RPD	2M
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.



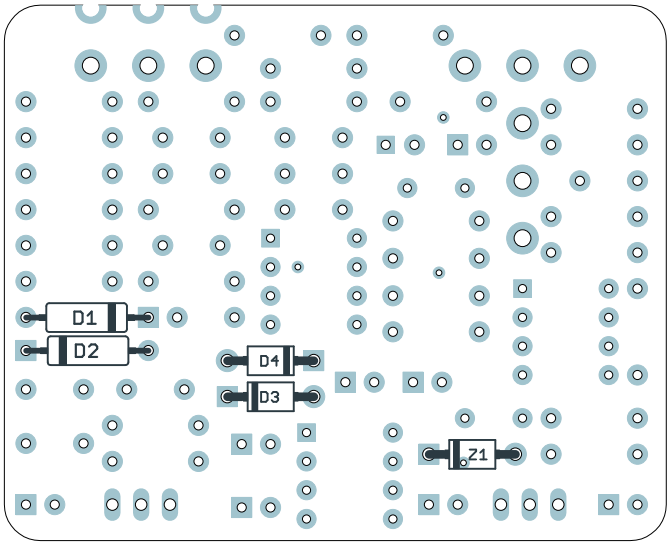
You'll use this same technique for most of the other components as well.

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 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!

MAIN PCB: DIODES

PART	VALUE
D1	Germanium
D2	Germanium
D3	1N4001
D4	1N4001
Z1	1N4742A



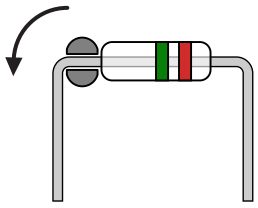
Next, you'll populate the diodes. Diodes are polarized, so make sure to identify the polarity band (which indicates the "cathode", or negative side) and match the band to the footprint on the PCB.

Germanium diodes will sometimes have more than one band. In these cases, the larger or wider band is the one that indicates the cathode side.

Precautions with germanium diodes

Germanium diodes are fragile and require more care than the other components. Make sure to observe the following precautions when working with them.

- To prevent stress on the glass body of the diode, when bending the leads, use needle-nose pliers or tweezers to clamp the lead as close to the body as possible while you bend it down. The bend should be about 0.05–0.08" from the body of the diode, so make sure to use pliers that are narrow enough.

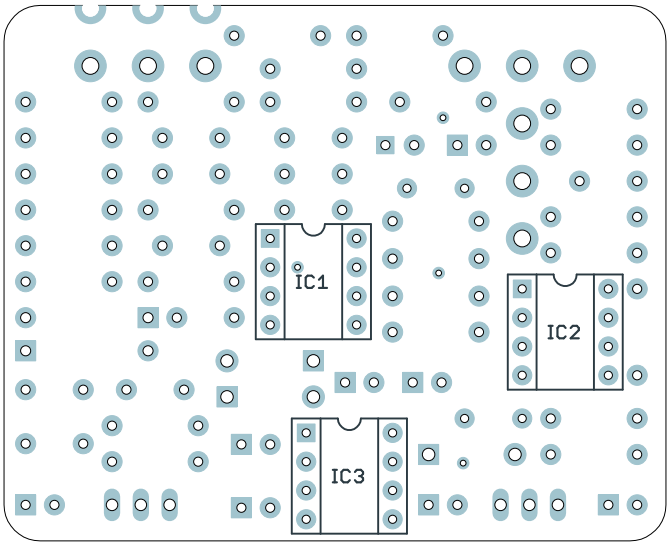


- Old-stock diodes can sometimes develop corrosion on the leads, making them difficult to solder. It's recommended to use sandpaper or a small file to gently rough up the leads where they will make contact with the solder. This will make adhesion much easier.
- Be quick when soldering. Germanium diodes can easily be damaged by overheating. Contact with the soldering iron should be limited to 1-2 seconds maximum. If you don't have a good solder joint, wait a minute or two for it to cool before trying again.

If you damage them during installation, send us a note via our missing parts form and we can send a replacement for the cost of shipping.

MAIN PCB: SOCKETS & ICS

PART	VALUE
IC1	TL072
IC2	TL072
IC3	LT1054



Next up are the sockets. 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.

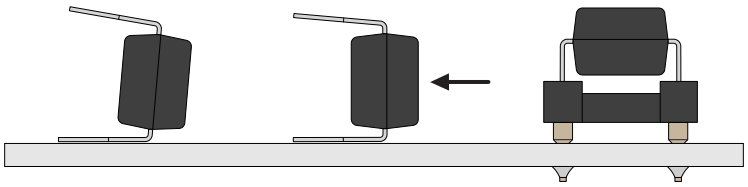
Again, it's much easier to do all of these at once with gravity holding them in place for you, so you'll want do them before you do any of the taller components.

Installing the ICs

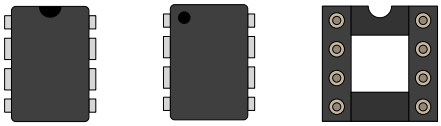
Don't insert the ICs into the sockets just yet. We will do this in a later step, after we've finished soldering the tallest components (the polarized capacitors). This information is just listed here for reference.

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.

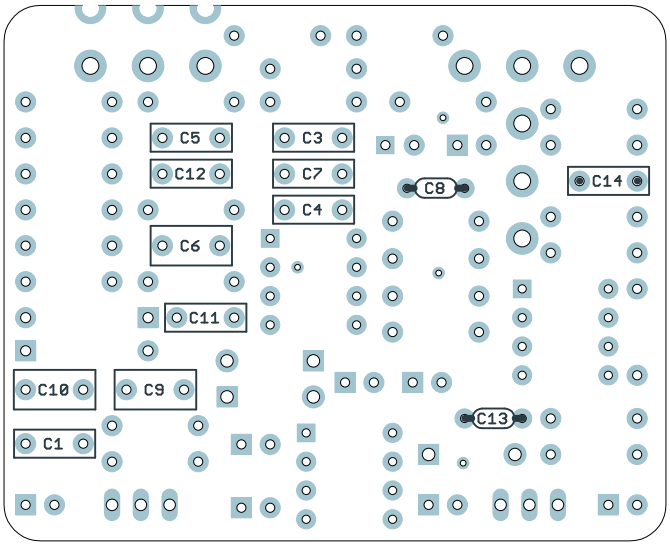


ICs may have two different orientation marks: either a dot in the upper-left or a half-circle notch in the middle of the top side. Some ICs have both marks. This shows which way the IC should be rotated when inserting it into a socket (the socket also has a half-circle notch).



MAIN PCB: CAPACITORS (NON-POLARIZED)

PART	VALUE
C1	100n (0.1)
C3	100n (0.1)
C4	68n (0.068)
C5	68n (0.068)
C6	390n (0.39)
C7	82n (0.082)
C8	390pF MLCC
C9	1uF
C10	1uF
C11	2n2
C12	27n (0.027)
C13	820pF MLCC
C14	3n9



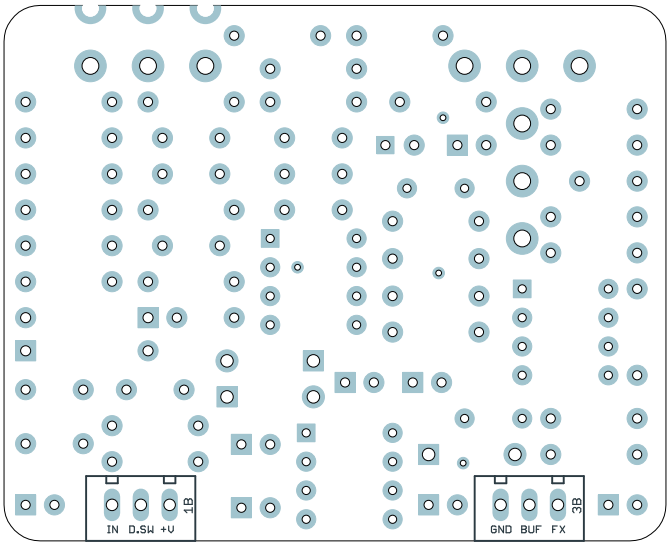
After the sockets come the box film and MLCC capacitors. These are all several different heights, but there aren't as many, so just do them all at once. Bend the leads at an angle to hold them in place.

MLCCs and box capacitors are not polarized, so they will work in any direction, but to keep things neat, it's recommended to put them all facing the same way.

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.

C8 and C13 are blue MLCCs taped to cardboard. The value is hard to read on the capacitor itself, but will always be written on the cardboard.

MAIN PCB: WIRE HEADERS

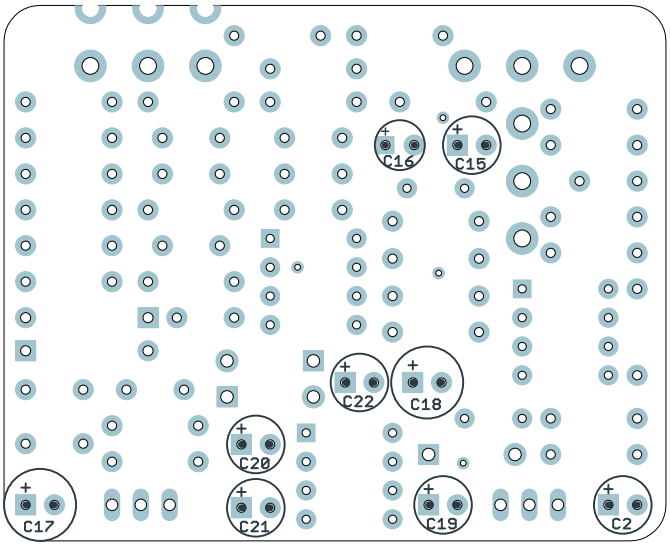


Install the two 3-pin headers (wire connectors) as shown above. These have a polarity pin, so as long as they are pressed all the way down, there's only one possible way to install them. They do fit pretty tightly in the holes, though, so press firmly.

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

MAIN PCB: CAPACITORS (POLARIZED)

PART	VALUE
C2	4.7uF
C15	4.7uF
C16	1uF tantalum
C17	100uF
C18	47uF
C19	10uF
C20	10uF
C21	10uF
C22	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.

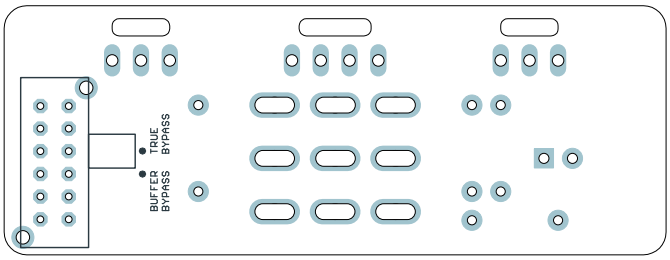
Next, populate the yellow tantalum capacitor (C16). Be very careful with this: unlike electrolytic capacitors, tantalum capacitors have the "+" (positive) side marked instead of the negative. Since tantalum capacitors aren't commonplace in guitar pedals, many people instinctively reverse them, and as a result, their build has issues. Like electrolytics, the longer leg still goes 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

- Slide switch, 4PDT
- 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.

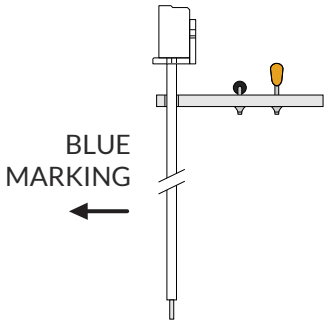
It's easiest to start with the slide switch. Fit it in as shown in the diagram above, with the slide lever facing to the right. Be careful—the pads are small. Make sure you don't accidentally "bridge" two pads together when soldering or you will have issues with the bypass.

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.

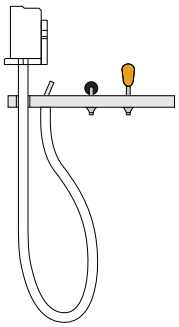
For now, pull it through as far as it can go.



STEP 2

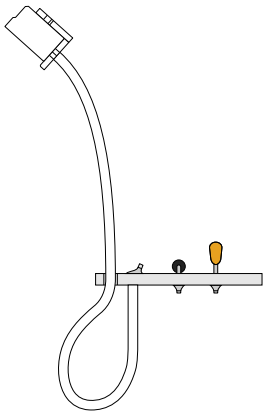
Next, bend the wires back upward and fit the ends of the wires into the solder pads.

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

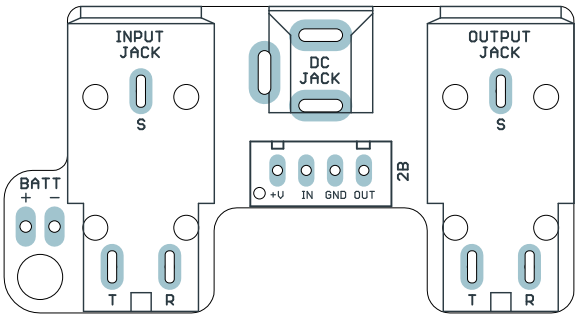
Then, solder the wires from the top. This is the trickiest part of the whole build. You want to solder the pads without touching the iron to the wires themselves and risking burning through the insulation. It helps to use a sharp or narrow tip on the soldering iron.



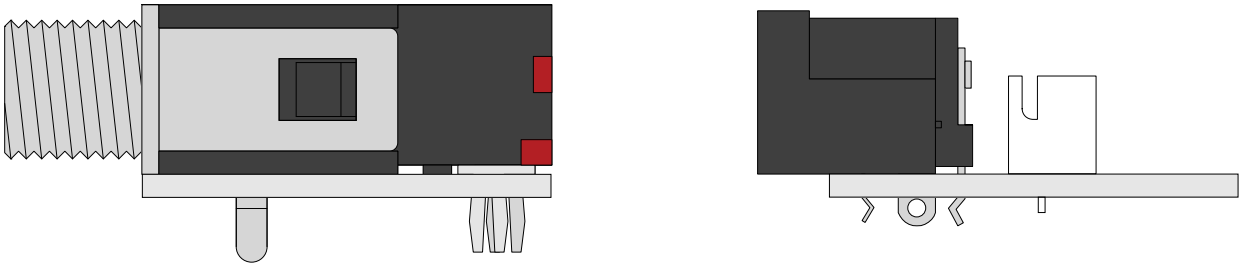
Once all three wire assemblies are soldered, set the footswitch PCB aside. We'll solder the actual footswitch and LED in a later step.

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.

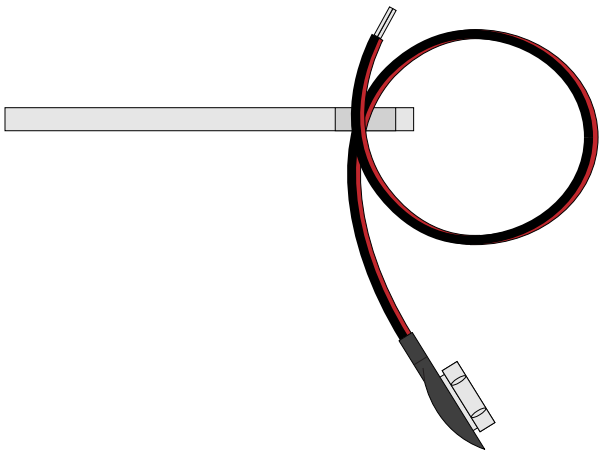


After you've soldered everything, make sure to **snip the leads on the I/O jacks as close as possible to the PCB**. There's not a lot of clearance between the bottom of this board and the top of the main PCB once everything is in place, and you don't want the pins to short against anything on accident.

Next, we'll hook up the 9V battery connector. **This is optional.** The first versions of the Centaur had battery snaps, but it was removed early on because the effect eats them pretty quickly. A battery snap has been provided with the kit, but it's still recommended to only power this with an external supply.

STEP 1

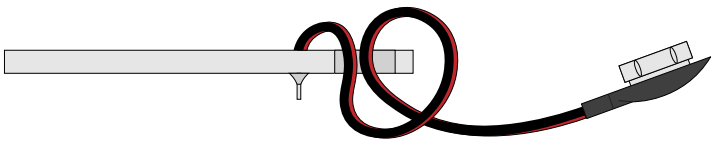
Thread the battery snap through the strain-relief hole twice so it forms a single loop.



STEP 2

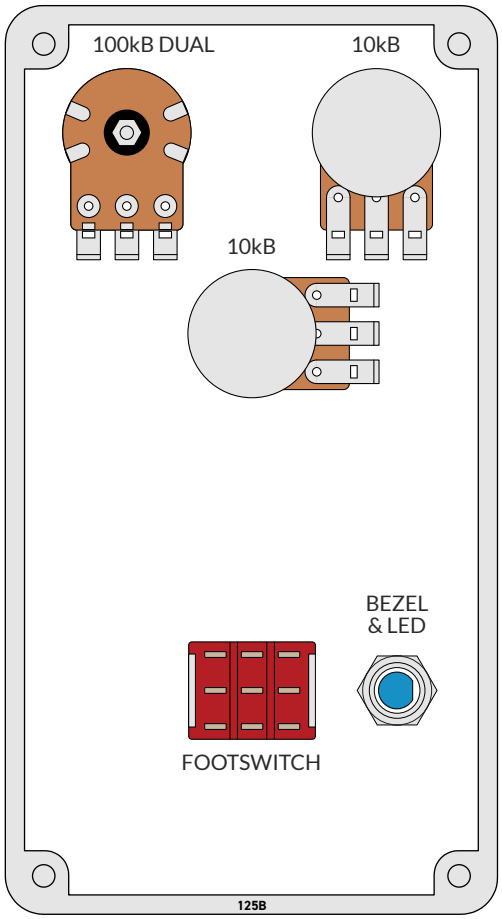
Bend the exposed wires back down and solder them into the pads. Red is positive (+), black is negative (-). After soldering, pull it tight.

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

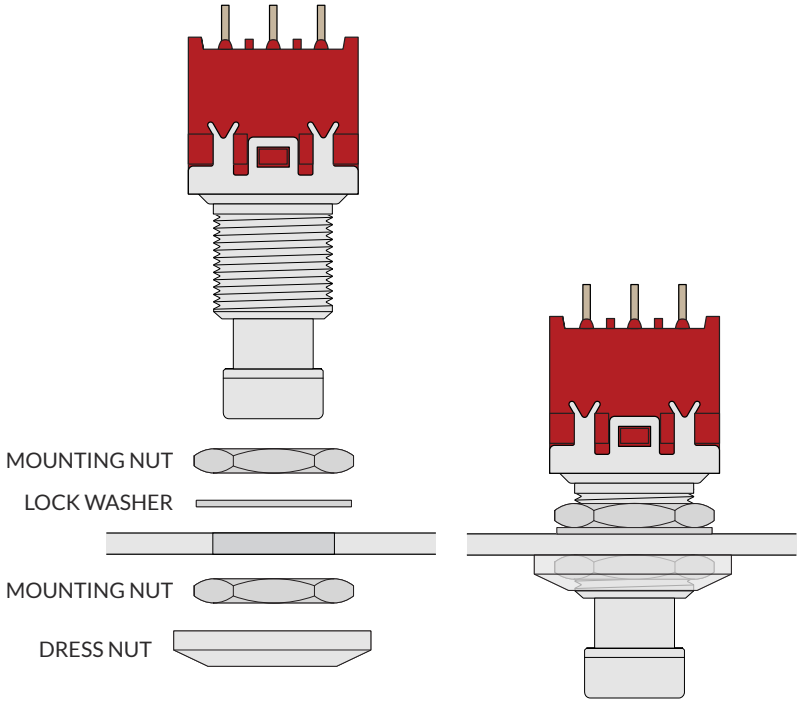


ENCLOSURE LAYOUT: PANEL MOUNTS

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



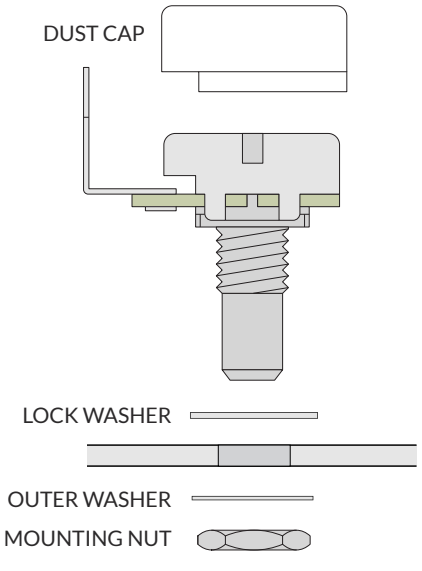
FOOTSWITCH



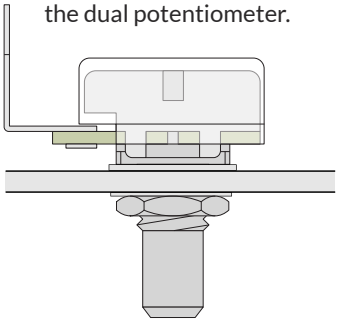
The dress nut fits over the top of the mounting nut and is for aesthetic purposes only. Wrap a rubber band around it to use as a grip when tightening. Avoid using metal tools on it or you run the risk of scratching or denting it.

POTENTIOMETERS

Volume: 10kB
Drive: 100kB dual
Tone: 10kB

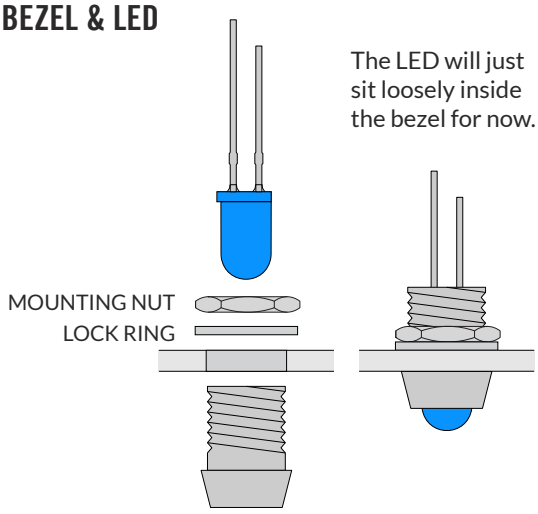


The dust cap is not used on the dual potentiometer.



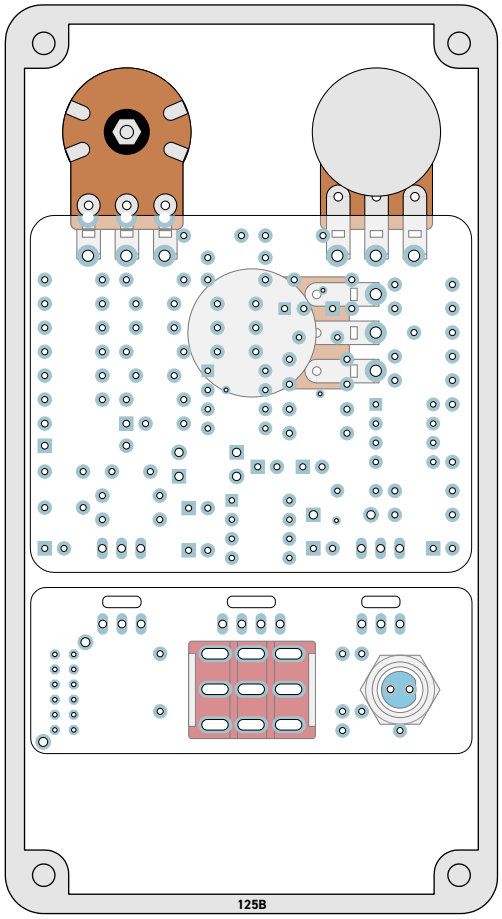
BEZEL & LED

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.

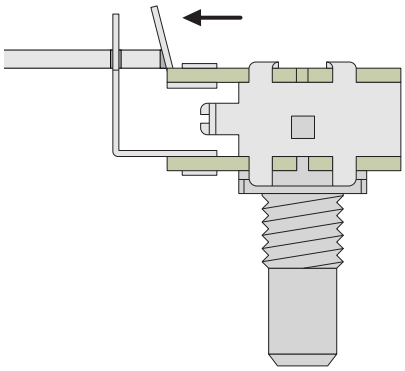
Next, do the same thing with the footswitch board—the 3PDT footswitch and the LED.

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.

A note about the dual-gang potentiometer

The upper pads for the dual-gang gain potentiometer appear to be cut in half. **This is intentional.** It's called a *plated half-hole* or *castellated hole*, and it's used so that the PCB can lay flat across the pots instead of angling upward for the dual pot.

Solder it like you would if they were normal pads, but bend the top pins of the pot forward slightly so they make contact with the inside edges of the half-holes, as shown in the diagram to the right.



Why solder everything inside the enclosure before testing it?

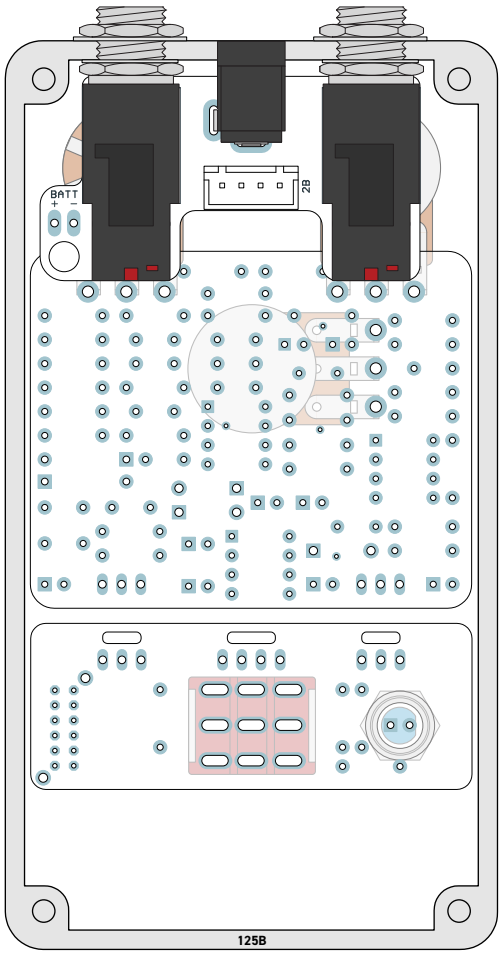
"Rock it before you box it" is conventional wisdom in pedalbuilding, and you'll often hear it recommended that builders should test the circuit before putting everything inside the enclosure. However, Aion FX projects are designed to be extremely easy to remove from the enclosure for troubleshooting,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

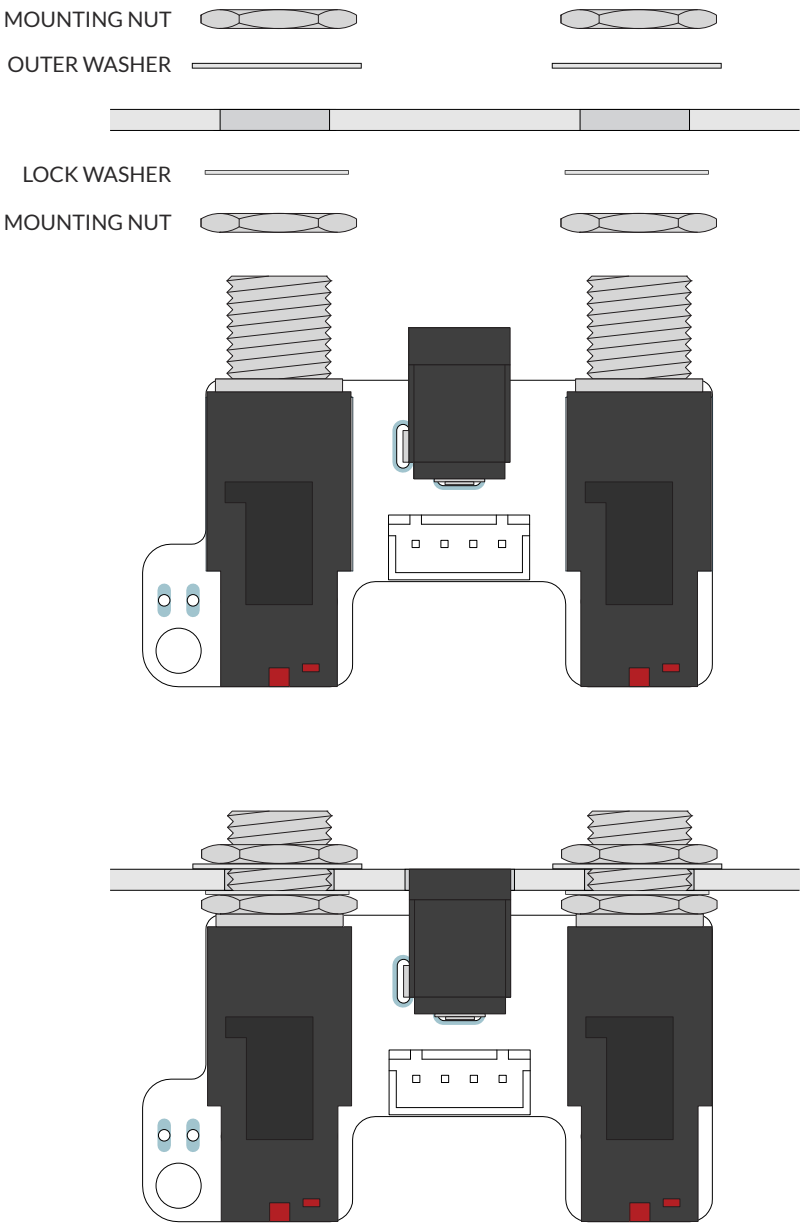
If you've read the documentation carefully and followed all the instructions, there's a good chance you will get it right the first time!

ENCLOSURE LAYOUT: INPUT/OUTPUT PCB

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

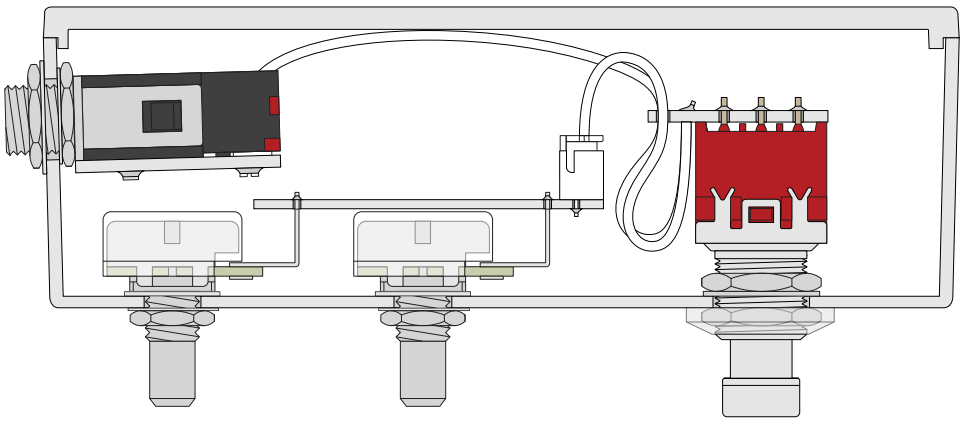
Note the use of two mounting nuts on each of the jacks, one inside and one outside. The inner nut acts as a spacer to set the DC jack flush with the outside of the enclosure. The inner nuts should be threaded as far down as they can go.





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.

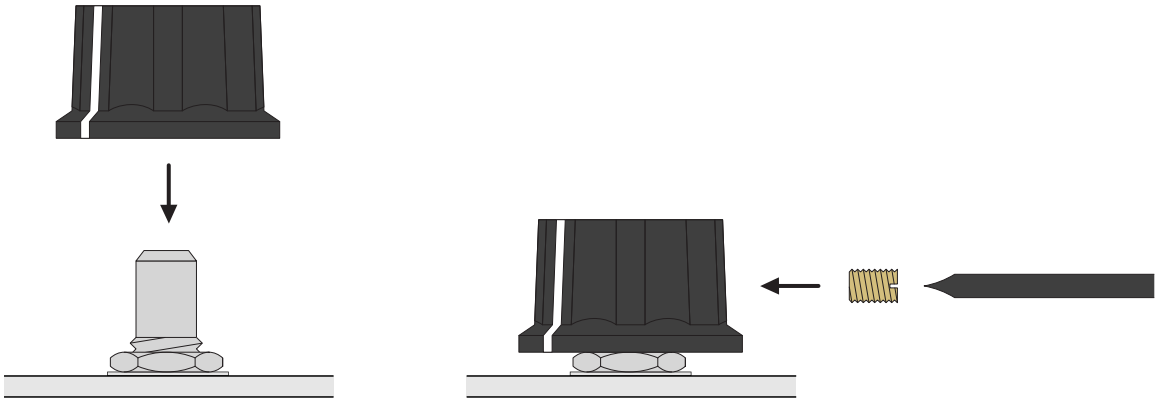
Test the bypass switch a few times, then start turning the knobs and see if everything sounds OK. If it works, great! If not, don't be discouraged. See page 24 for troubleshooting info.

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.

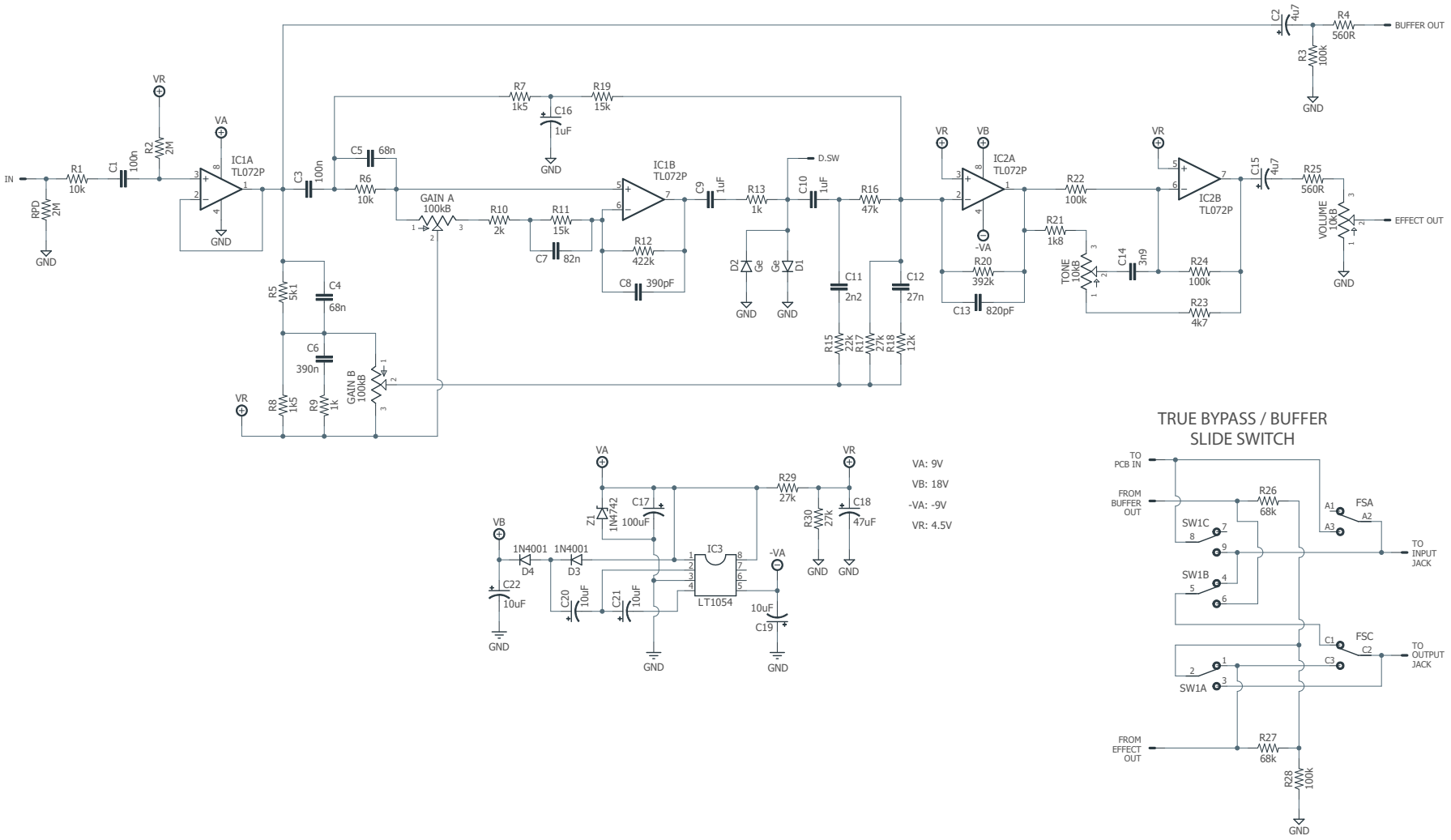
Using a small flat-head screwdriver (no more than 0.1" / 2.5mm in diameter), firmly tighten the set screw until it presses against the shaft of the potentiometer and holds the knob in place.

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


IN
RPD 2M
GND
R1 10k
C1 100n
VR
R2 2M
VA
IC1A TL072P
C3 100n
C5 68n
R6 10k
R7 1k5
R19 15k
C16 1uF
GAIN A 100kB
R10 2k
R11 15k
C7 82n
IC1B TL072P
R12 422k
C8 390pF
C9 1uF
R13 1k
D.SW
C10 1uF
D2 Ge
D1 Ge
C11 2n2
C12 27n
R15 22k
R17 27k
R18 12k
R16 47k
VB
IC2A TL072P
-VA
R20 392k
C13 820pF
R22 100k
R21 1k8
TONE 10kB
C14 3n9
IC2B TL072P
R24 100k
R23 4k7
C15 4u7
R25 560R
VOLUME 10kB
EFFECT OUT
C2 4u7
R4 560R
BUFFER OUT
R3 100k
R5 5k1
C4 68n
C6 390n
GAIN B 100kB
R8 1k5
R9 1k
Z1 1N4742
C17 100uF
R29 27k
R30 27k
C18 47uF
1N4001 D4
1N4001 D3
IC3 LT1054
C22 10uF
C20 10uF
C21 10uF
C19 10uF
VA: 9V
VB: 18V
-VA: -9V
VR: 4.5V
TRUE BYPASS / BUFFER SLIDE SWITCH
TO PCB IN
FROM BUFFER OUT
SW1C
SW1B
SW1A
R26 68k
FSA
FSC
TO INPUT JACK
TO OUTPUT JACK
FROM EFFECT OUT
R27 68k
R28 100k

FULL PARTS LIST

Resistors

PART	VALUE
R1	10k
R2	2M
R3	100k
R4	560R
R5	5k1
R6	10k
R7	1k5
R8	1k5

PART	VALUE
R9	1k
R10	2k
R11	15k
R12	422k
R13	1k
R15	22k
R16	47k
R17	27k

PART	VALUE
R18	12k
R19	15k
R20	392k
R21	1k8
R22	100k
R23	4k7
R24	100k
R25	560R

PART	VALUE
R26	68k
R27	68k
R28	100k
R29	27k
R30	27k
RPD	2M
LEDR	10k

Capacitors

PART	VALUE
C1	100n film
C2	4.7uF electro
C3	100n film
C4	68n film
C5	68n film
C6	390n film
C7	82n film
C8	390pF MLCC

PART	VALUE
C9	1uF film
C10	1uF film
C11	2n2 film
C12	27n film
C13	820pF MLCC
C14	3n9 film
C15	4.7uF electro
C16	1uF tantalum

PART	VALUE
C17	100uF electro
C18	47uF electro
C19	10uF electro
C20	10uF electro
C21	10uF electro
C22	10uF electro

Diodes

PART	VALUE
D1	Germanium
D2	Germanium
D3	1N4001
D4	1N4001
Z1	1N4742A

ICs

PART	VALUE
IC1	TL072
IC2	TL072
IC3	LT1054

Potentiometers

PART	VALUE
Volume	10kB
Drive	100kB dual
Tone	10kB

Switches

PART
4PDT slide
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.

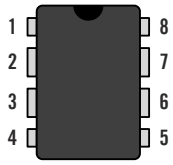
It works initially, but develops issues after about 30 seconds of being powered on.

Check that C16, the tantalum capacitor, is oriented correctly. Some people install this incorrectly due to the fact that tantalums have the positive leg marked instead of the negative leg like electrolytics.

Voltages

The following voltages are taken from our prototype unit using a **9.86V** 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.



IC1

PIN	VOLTAGE
1	4.92V
2	4.92V
3	3V–4V (drifts)
4	0V
5	4.90V
6	4.92V
7	4.97V
8	9.86V

IC2

PIN	VOLTAGE
1	4.96V
2	4.92V
3	4.91V
4	-9.49V
5	4.91V
6	4.92V
7	4.87V
8	17.99V

IC3

PIN	VOLTAGE
1	9.86V
2	5.02V
3	0V
4	-4.66V
5	-9.49V
6	4.95V
7	6.35V
8	9.86V

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

Klon®, Centaur® and KTR® are registered trademarks of Klon LLC. Any 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

1.0.6 (2024-08-08)
Added link to troubleshooting guide on page 24.

1.0.5 (2023-11-23)
Updated LEDR (LED current-limiting resistor) to 10k to reduce brightness.

1.0.4 (2022-07-01)
Changed IC3 to LT1054.

1.0.3 (2018-11-22)
Corrected an omission in the packing list on pg 4.

1.0.2 (2018-10-24)
Corrected a minor error in the schematic (not present in the PCB) and re-drew portions for clarity.

1.0.0 (2018-08-12)
Initial release.