

PROJECT NAME

# REFRACTOR

BASED ON

Klon Centaur / KTR

EFFECT TYPE

Overdrive

BUILD DIFFICULTY

■■■■ Intermediate

DOCUMENT VERSION

1.0.5 (2023-11-23)

**qion**  
DIY GUITAR EFFECTS

## 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

---

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

# INTRODUCTION

---

If this is your first pedal, welcome to the hobby and thank you for choosing Aion FX. You've just joined a community of over 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 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	1
68n	2
82n	1
100n	2
390n	1
1uF	2

## Electrolytic Capacitors

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

## Tantalum Capacitors

NAME	QTY
1uF	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 (see note)	1
5k1	1
10k (see note)	3
12k	1
15k	2
22k	1
27k	3
47k	1
68k	2
100k	4
392k	1
422k	1
2M	2

**Note:** Kits purchased prior to November 2023 will have a 4k7 resistor instead of 10k for LEDR.

## ICs

NAME	QTY
TL072	2
LT1054 or TC1044S	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.



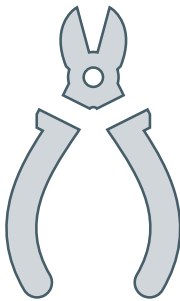
### SOLDER

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.



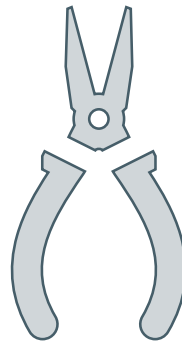
### DIGITAL MULTIMETER (DMM)

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.



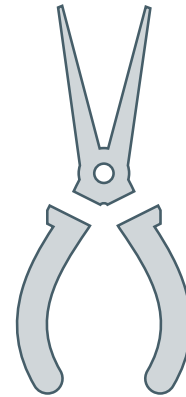
### WIRE SNIPPERS

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



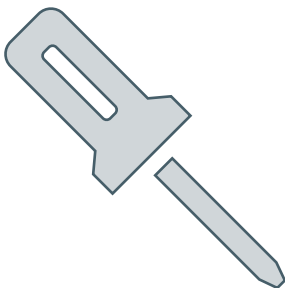
### FLAT-NOSE PLIERS

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



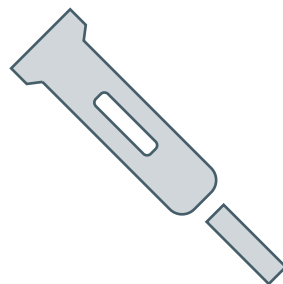
### NEEDLE-NOSE PLIERS

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.



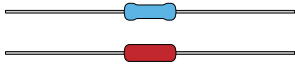
### RUBBER BAND

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).

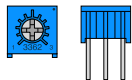
# COMPONENT IDENTIFICATION

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**



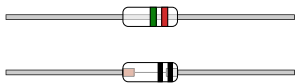
**LED**



**SILICON DIODE**



**GERMANIUM DIODE**



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

**RECTIFIER DIODE**



*Some Schottky diodes also look like this.*

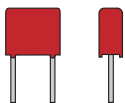
**SCHOTTKY DIODE**



**ZENER DIODE**



**FILM CAPACITOR**



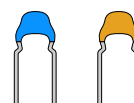
*Not polarized. Color may vary by brand and type.*

**ELECTROLYTIC CAPACITOR**



*Polarized. The negative side is marked.*

**MLCC**



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

**TANTALUM CAPACITOR**



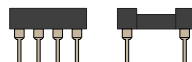
*Polarized. The positive side is marked.*

**IC OR OP-AMP**

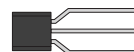


*Charge pumps and delay chips also look like this. They may have more than 8 legs.*

**IC SOCKET**

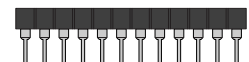


**TRANSISTOR OR JFET**



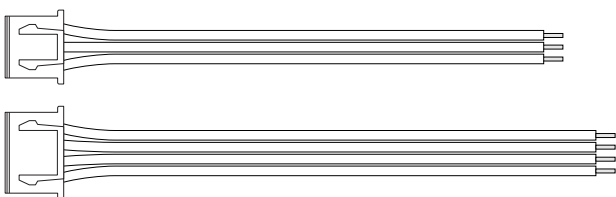
*Some voltage regulators also look like this.*

**SNAP-APART SOCKET**

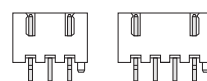


*It's recommended to use a razor blade to separate these cleanly.*

**WIRE ASSEMBLY**



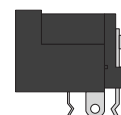
**WIRE ASSEMBLY HEADER**



**LED BEZEL**

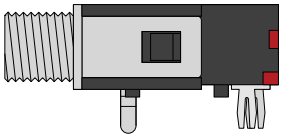
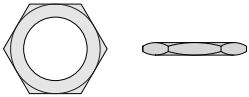
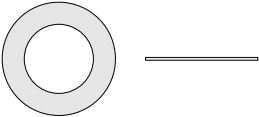
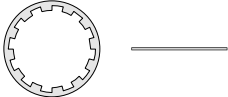


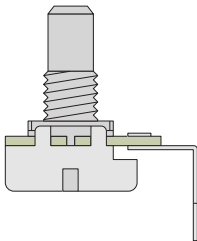
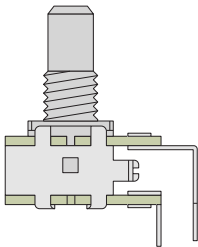
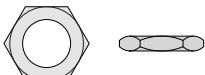
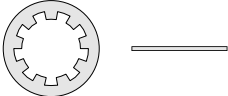
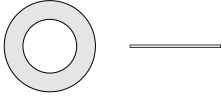

**DC JACK**

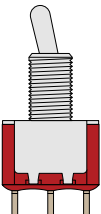
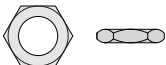
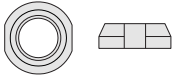
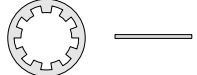


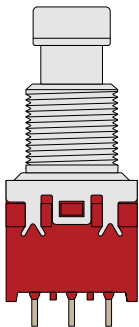
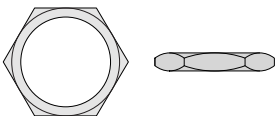
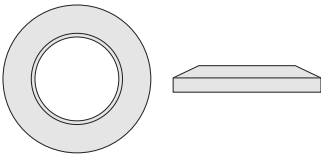
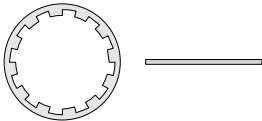
# HARDWARE IDENTIFICATION

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

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

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

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

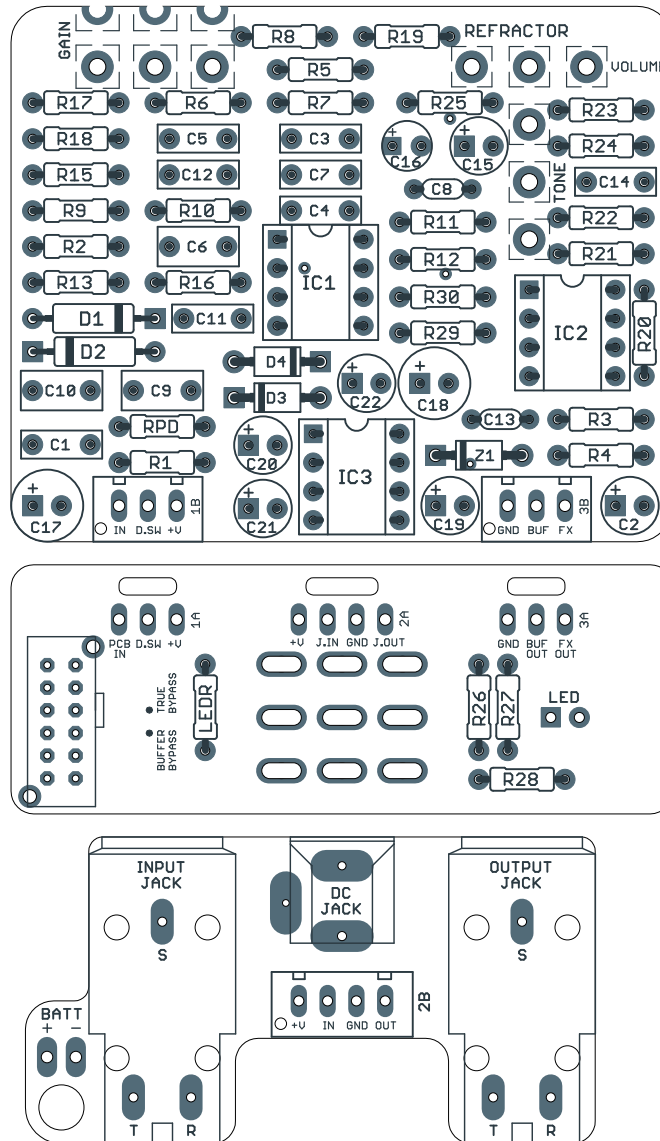
<b>FOOTSWITCH</b> 	<b>MOUNTING NUT</b>  DIAMETER: 0.6" / 15.2mm	<b>DRESS NUT</b>  DIAMETER: 0.77" / 19.6mm	<b>LOCK WASHER</b>  DIAMETER: 0.6" / 15.2mm
--	---	--	--



## MAIN PCB: OVERVIEW

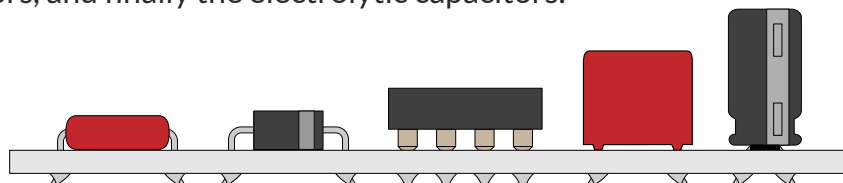
Now it's time to start building!

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

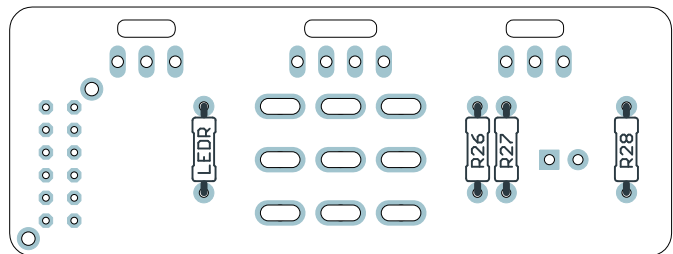
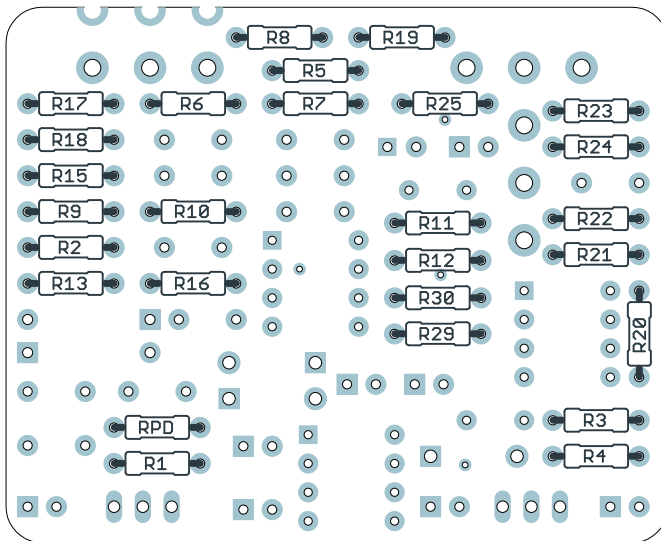
PART	VALUE
R1	10k
R2	2M
R3	100k
R4	560R
R5	5k1
R6	10k
R7	1k5
R8	1k5

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

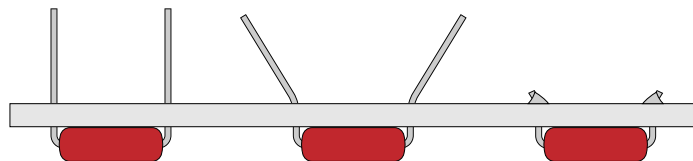
PART	VALUE
R18	12k
R19	15k
R20	392k
R21	1k8
R22	100k
R23	4k7
R24	100k
R25	560R

PART	VALUE
R26	68k
R27	68k
R28	100k
R29	27k
R30	27k
RPD	2M
LEDR	10k or 4k7

*LEDR was changed from 4k7 to 10k beginning in late 2023, which reduces the LED brightness.*



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.



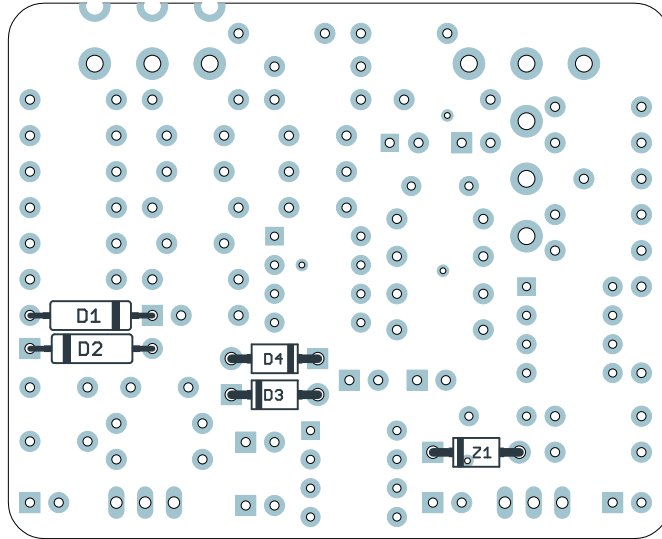
You'll use this same technique for most of the other components as well.

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	D9E
D2	D9E
D3	1N4001
D4	1N4001
Z1	1N4742A



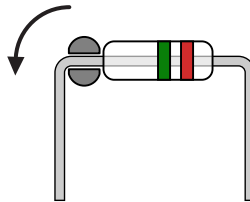
Next, you'll populate the diodes. Diodes are polarized, so make sure to identify the polarity band (which indicates the "cathode", or negative side) and match the band to the footprint on the PCB.

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.

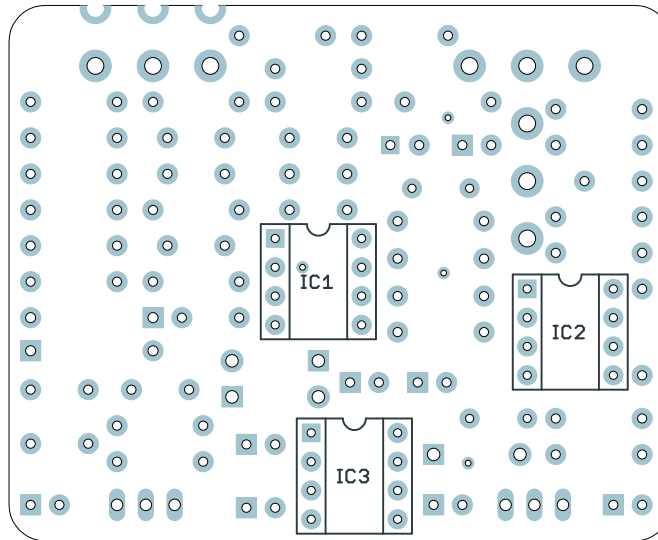


- Old-stock diodes can sometimes develop corrosion on the leads, making them difficult to solder. It's recommended to use sandpaper or a small file to gently rough up the leads where they will make contact with the solder. This will make adhesion much easier.
- Be quick when soldering. Germanium diodes 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.

The germanium diodes included with this kit have been individually tested and verified as working. Unless they arrive damaged, free replacements will not be offered, so please be careful!

## MAIN PCB: SOCKETS & ICs

PART	VALUE
IC1	TL072
IC2	TL072
IC3	TC1044S or 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.

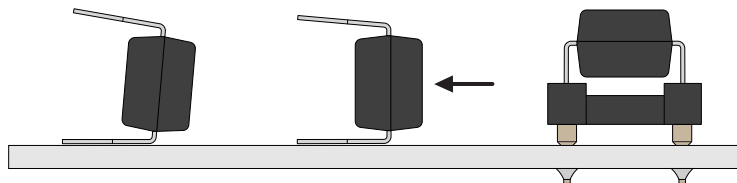
Again, it's much easier to do all of these at once with gravity holding them in place for you, so you'll want to do them before you do any of the taller components.

### Installing the ICs

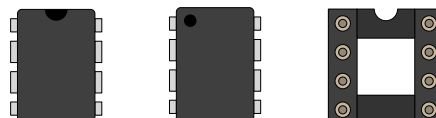
Don't insert the ICs into the sockets just yet. We will do this in a later step, after we've finished soldering the tallest components (the polarized capacitors). This information is just listed here for reference.

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.

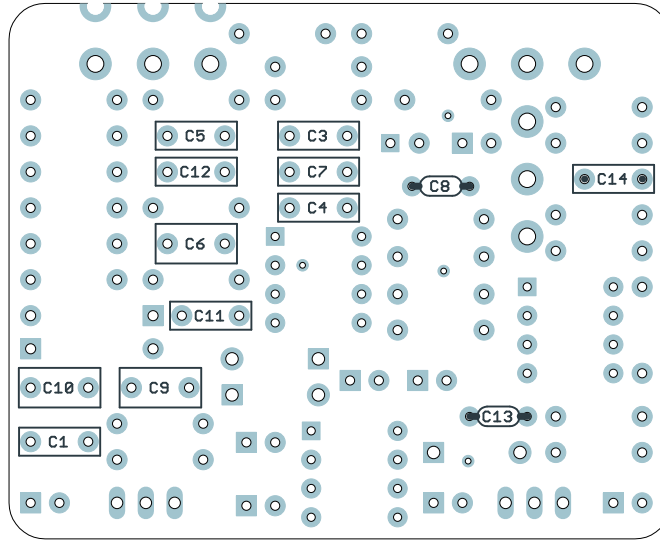


ICs may have two different orientation marks: either a dot in the upper-left or a half-circle notch in the middle of the top side. Some ICs have both marks. This shows which way the IC should be rotated when inserting it into a socket (the socket also has a half-circle notch).



## MAIN PCB: CAPACITORS (NON-POLARIZED)

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



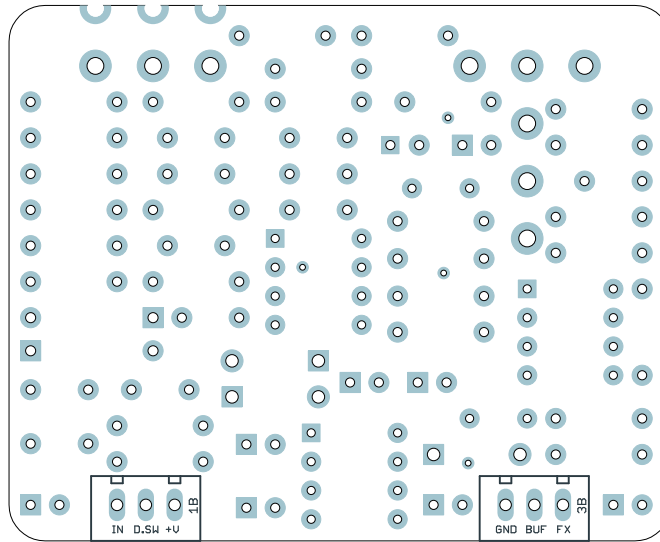
After the sockets come the box film and MLCC capacitors. These are all several different heights, but there aren't as many, so just do them all at once. Bend the leads at an angle to hold them in place.

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

**Note:** C11 and C14 are typically blue box-film capacitors. For these, the value is printed on the top rather than the side.

## MAIN PCB: WIRE HEADERS

---

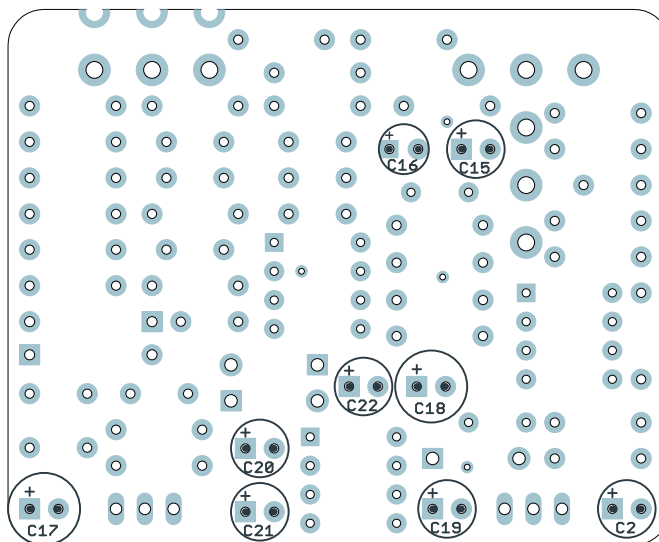


Install the two 3-pin headers (wire connectors) as shown above. These have a polarity pin, so as long as they are pressed all the way down, there's only one possible way to install them. They do fit pretty tightly in the holes, though, so press firmly.

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.

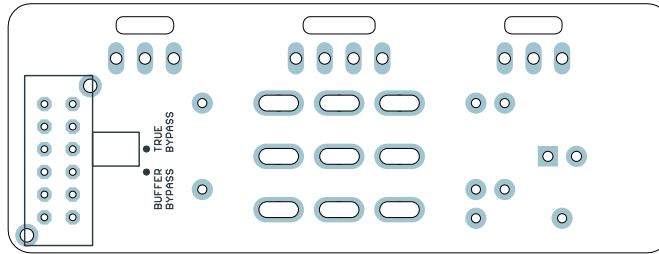
Next, populate the yellow tantalum capacitor (C16). Be very careful with this: unlike electrolytic capacitors, tantalum capacitors have the “+” (positive) side marked instead of the negative. Since tantalum capacitors aren’t commonplace in guitar pedals, many people instinctively reverse them, and as 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.

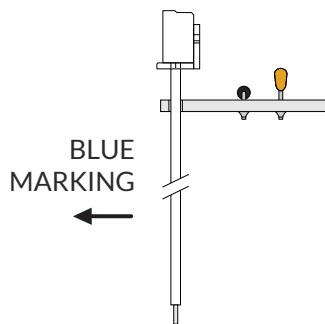
It's easiest to start with the slide switch. Fit it in as shown in the diagram above, with the slide lever facing to the right. Be careful—the pads are small. Make sure you don't accidentally “bridge” two pads together when soldering or you will have issues with the bypass.

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.

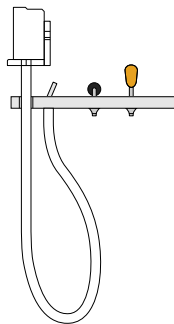
For now, pull it through as far as it can go.



## STEP 2

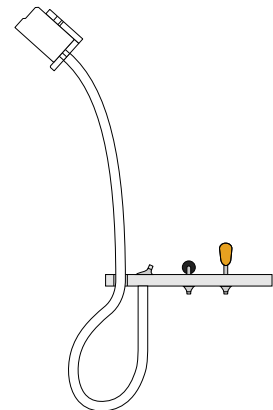
Next, bend the wires back upward and fit the ends of the wires into the solder pads.

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

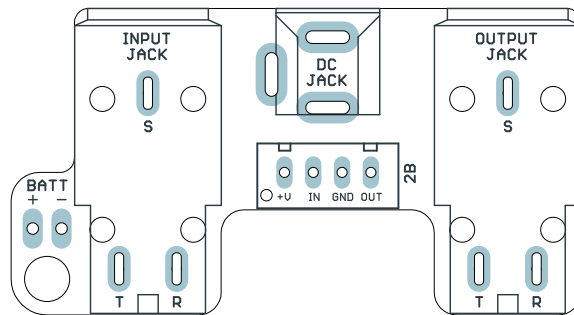
## PARTS

Input & output jacks

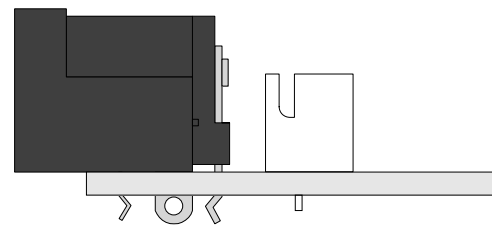
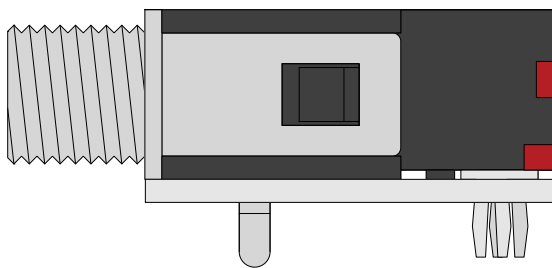
DC jack

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.

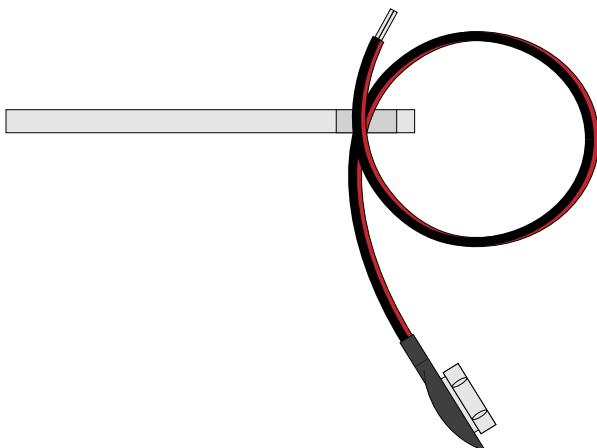


After you've soldered everything, make sure to **snip the leads on the I/O jacks as close as possible to the PCB**. There's not a lot of clearance between the bottom of this board and the top of the main PCB once everything is in place, and you don't want the pins to short against anything on accident.

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

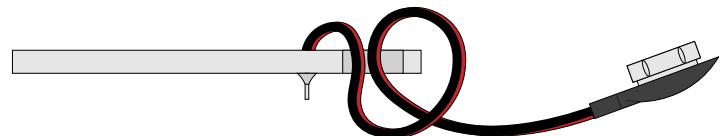
Thread the battery snap through the strain-relief hole twice so it forms a single loop.



## STEP 2

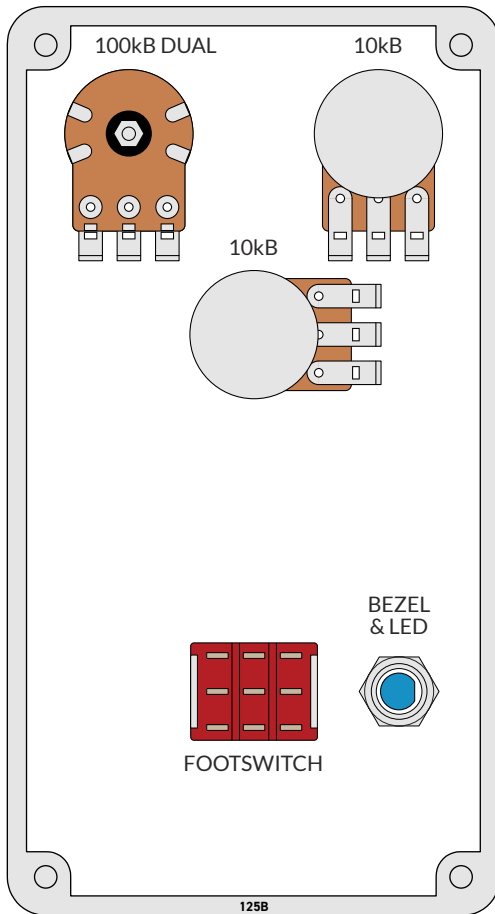
Bend the exposed wires back down and solder them into the pads. Red is positive (+), black is negative (-). After soldering, pull it tight.

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

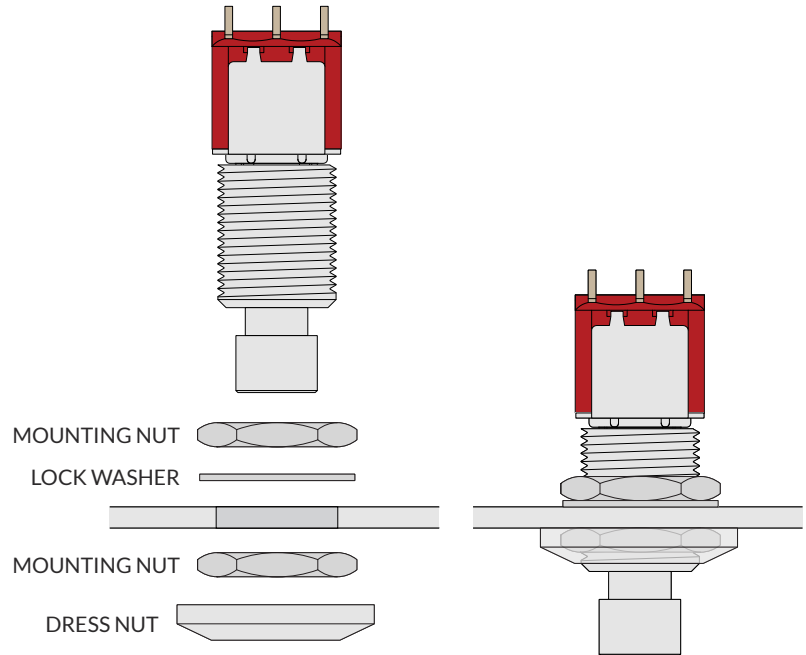


# ENCLOSURE LAYOUT: PANEL MOUNTS

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



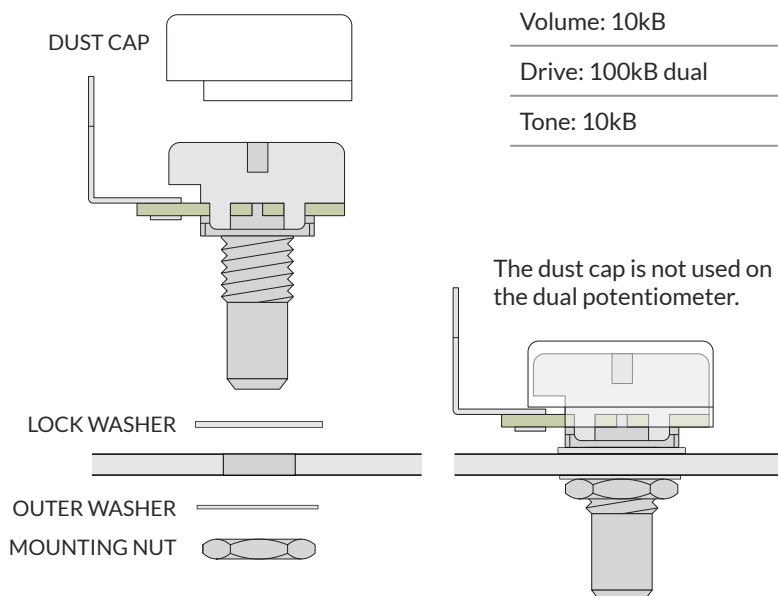
## FOOTSWITCH



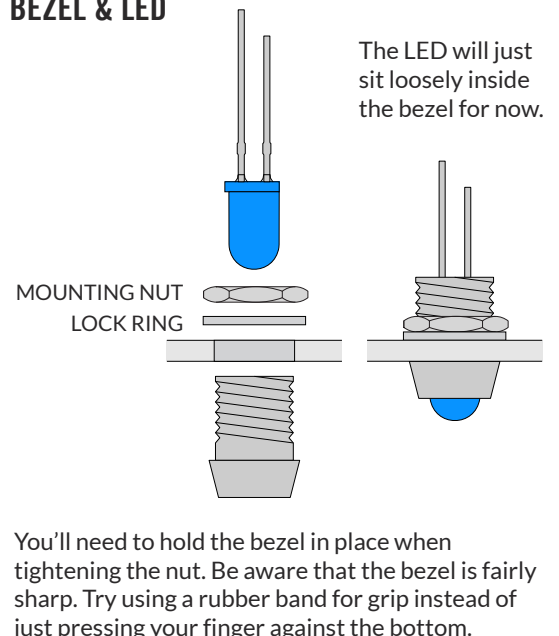
The dress nut fits over the top of the mounting nut and is for aesthetic purposes only. Wrap a rubber band around it to use as a grip when tightening. Do not use metal tools on it or you run the risk of scratching or denting it.

## POTENTIOMETERS

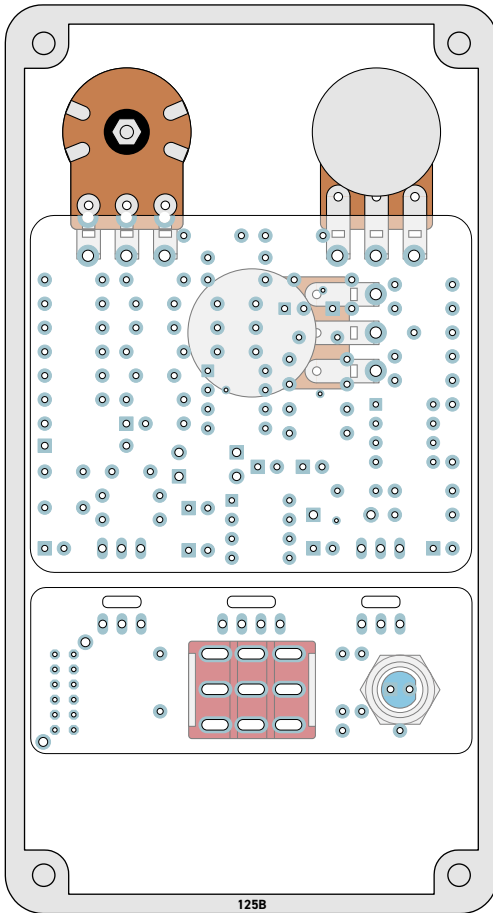
Volume: 10kB  
Drive: 100kB dual  
Tone: 10kB



## BEZEL & LED



## 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.

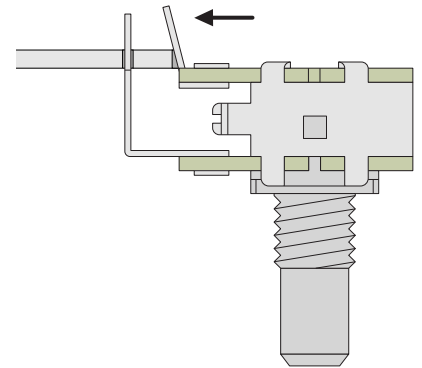
Next, do the same thing with the footswitch board—the 3PDT footswitch and the LED.

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?

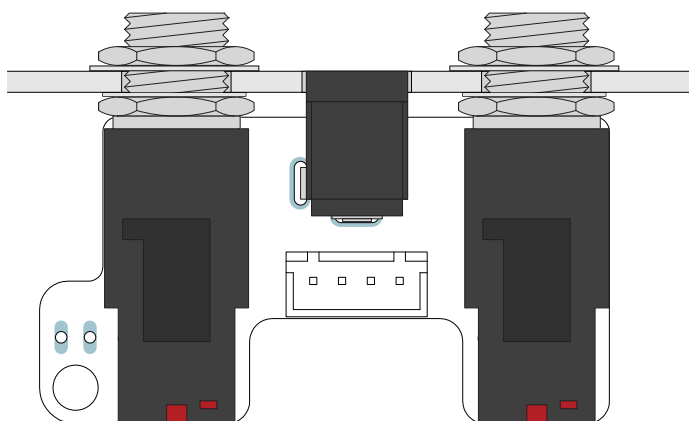
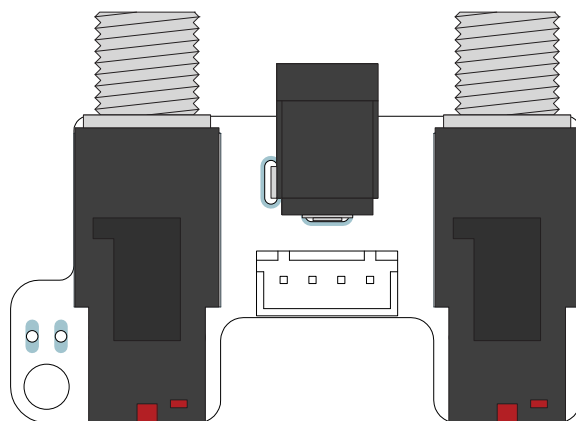
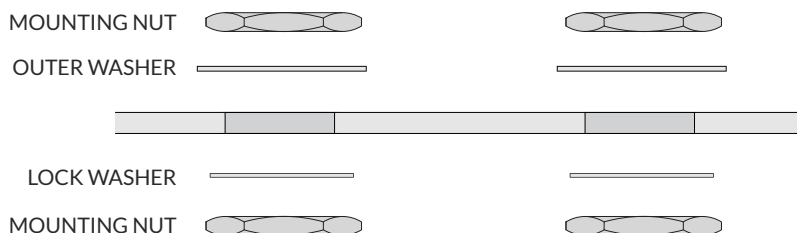
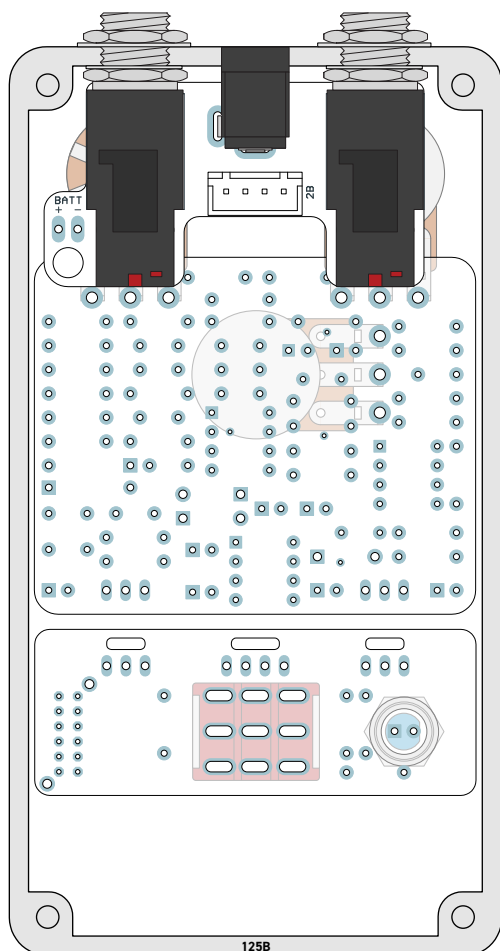
“Rock it before you box it” is conventional wisdom in pedalbuilding, and you'll often hear it recommended that builders should test the circuit before putting everything inside the enclosure. However, Aion FX projects are designed to be extremely easy to remove from the enclosure for troubleshooting,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

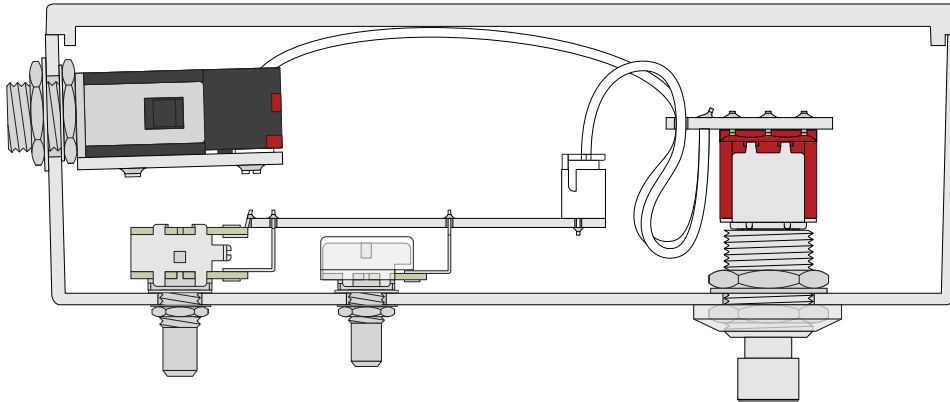
Affix the input/output PCB to the north-facing panel of the enclosure as shown.

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, 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.

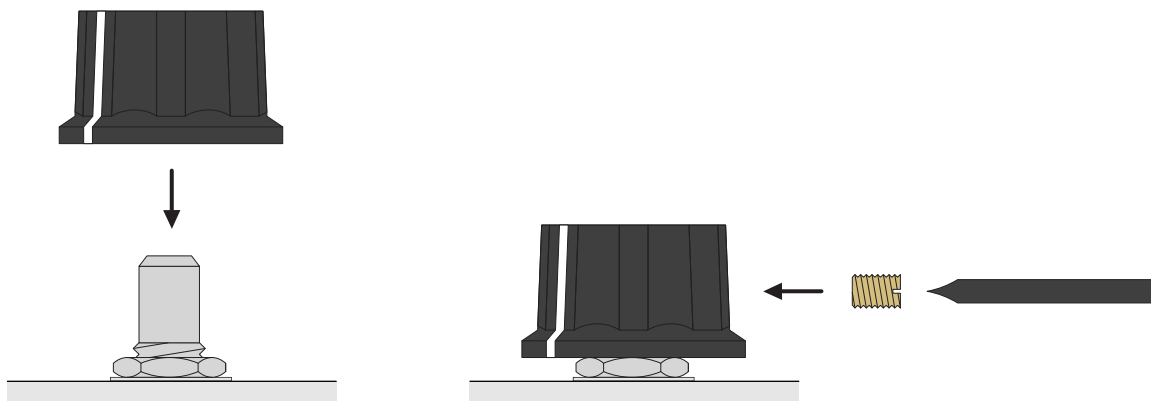
Test the bypass switch a few times, then start turning the knobs and see if everything sounds OK. If it works, great! If not, don't be discouraged. See page 24 for troubleshooting info.

### Finishing touches

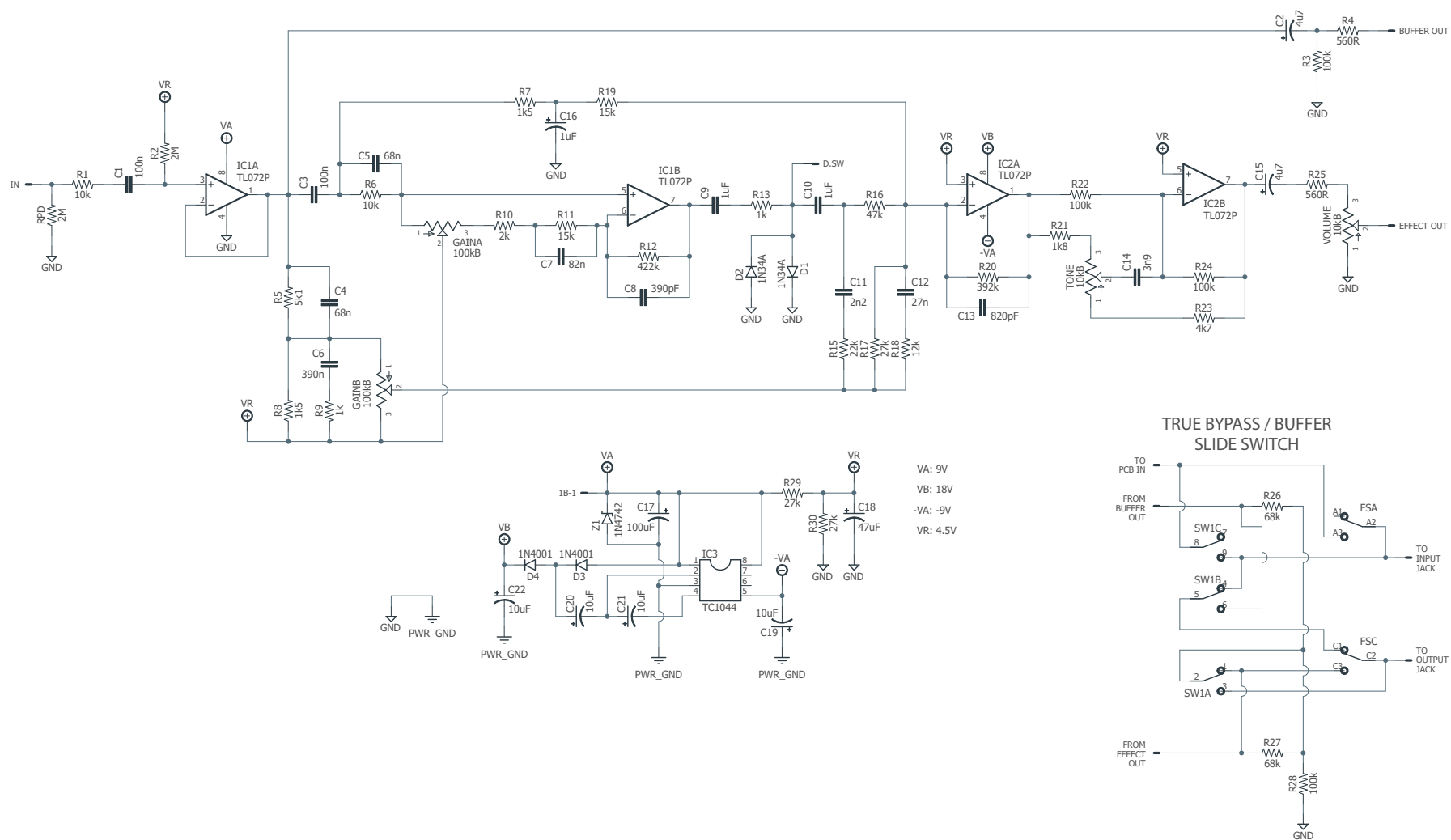
Now, just a couple of things for the final assembly. Turn the shafts all fully counter-clockwise, then put on the knob and rotate until the indicator line is aligned with the dot on the enclosure that shows the zero point. Affix the knobs to each of the potentiometer shafts as shown in the diagram below.

Using a small flat-head screwdriver (no more than 0.1" / 2.5mm in diameter), 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!



# 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	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 or 4k7

## 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	D9E
D2	D9E
D3	1N4001
D4	1N4001
Z1	1N4742A

## ICs

PART	VALUE
IC1	TL072
IC2	TL072
IC3	TC1044S or LT1054

## Potentiometers

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

## Switches

PART
4PDT slide
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.

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

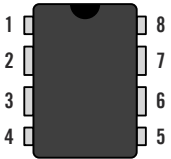
Check that C16, the tantalum capacitor, is oriented correctly. Many people install this incorrectly due to the fact that tantalums have the positive leg marked instead of the negative leg like electrolytics, so they instinctively assume that the marking indicates negative.

## 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 9.86V supply. Your measured voltages won't be exactly the same due to variance in power supplies and component tolerances. However, if you see anything 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.

### 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 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

---

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.

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

## DOCUMENT REVISIONS

---

### **1.0.5 (2023-11-23)**

Updated LEDR (LED current-limiting resistor) to 10k to reduce brightness.

### **1.0.4 (2022-07-01)**

Add LT1054 as an alternate for IC3.

### **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.