PROJECT NAME

GLADIATOR

aion DIY GUITAR EFFECTS

BASED ON
King Tone Duellist

BUILD DIFFICULTY
Intermediate

EFFECT TYPE
Dual-channel overdrive

DOCUMENT VERSION
1.0.0 (2024-09-06)

PROJECT SUMMARY
A popular dual-channel overdrive that combines a stripped-down Tube Screamer with a hot-rodded Bluesbreaker.

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 29.

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

Film Capacitors

NAME	QTY
4n7	1
10n (0.01)	5
15n (0.015)	1
22n (0.022)	1
47n (0.047)	1
100n (0.1 or “µ1J100”)	2
220n (0.22)	3
1uF	2
4.7uF	1

Electrolytic Capacitors

NAME	QTY
100uF	4

MLCC Capacitors

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

ICs

NAME	QTY
RC4558P	2
8-pin socket	2

Other

NAME	QTY
DIP switch, 2-pos.	2

Resistors

NAME	QTY
47R	2
220R	1
1k	3
3k3	1
4k7	2
5k6	1
9k53	1
10k	2
20k	1
22k	2
28k7	1
30k1	1
47k	5
133k	1
220k	1
1M	2
2M2	2

Diodes

NAME	QTY
1N5817	1
1N914	2
BAS33	8

PACKING LIST (CONT.)

Potentiometers

NAME	QTY
20kW	1
25kB	1
100kA	2
150kC	1
1MA	1
Dust cover	6
Knob	6
Mounting nut, potentiometer, 0.44"	6
Lock washer, potentiometer, 0.5"	6
Outer washer, potentiometer, 0.475"	6

Other

NAME	QTY
LED bezel	2
LED, blue	1
LED, red	1
DC jack	1
Input/output jack	2
Mounting nut, I/O jack, 0.54"	4
Outer washer, I/O jack, 0.6"	2
Lock washer, I/O jack, 0.5" (thin)	2
Send/return jack, Neutrik NMJ6	2
Insulation washer, Neutrik NMJ6	2
Ferrule, chrome, Neutrik NMJ6	2
Enclosure	1
Enclosure screws	4
PCB, main circuit	1
PCB, footswitch	1
PCB, input/output/DC	1

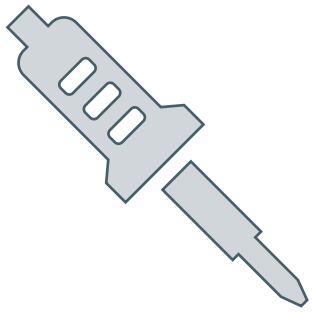
Switches

NAME	QTY
Toggle switch, SPDT on-off-on	2
Mounting nut, toggle switch, 0.36"	2
Lock washer, toggle switch, 0.4"	2
Dress nut, toggle switch, 0.375"	2
Stomp switch, 3PDT	2
Mounting nut, stomp switch, 0.6"	4
Lock washer, stomp switch, 0.6"	2
Dress nut, stomp switch, 0.77"	2

Wiring

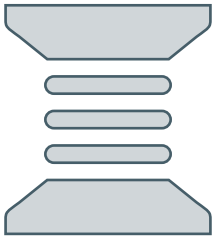
NAME	QTY
4-strand wire assembly, 60mm	2
6-strand wire assembly, 96mm	1
4-pin wire assembly header	2
6-pin wire assembly header	1

TOOLS NEEDED



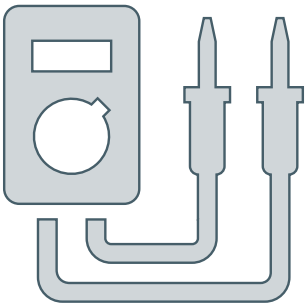
SOLDERING IRON

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.



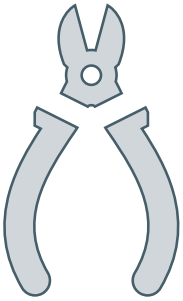
SOLDER

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.



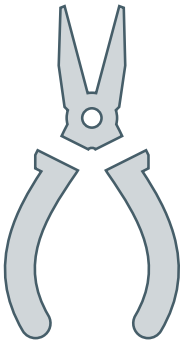
DIGITAL MULTIMETER (DMM)

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.



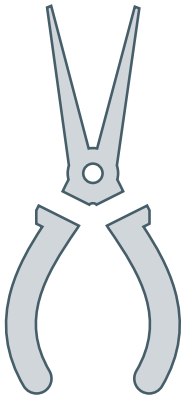
WIRE SNIPPERS

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



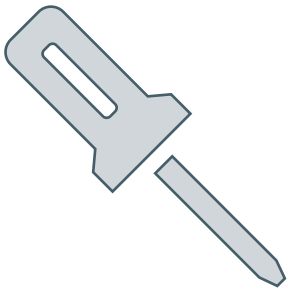
FLAT-NOSE PLIERS

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



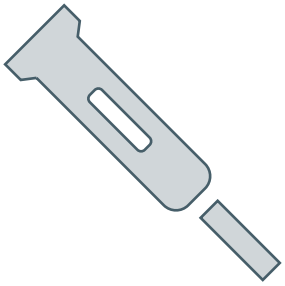
NEEDLE-NOSE PLIERS

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.



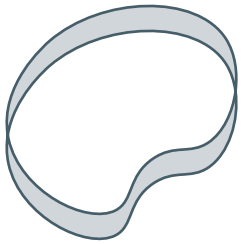
SCREWDRIVER (PHILLIPS)

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.



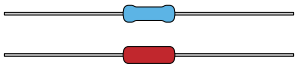
RUBBER BAND

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).

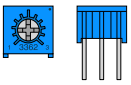
COMPONENT IDENTIFICATION

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.)

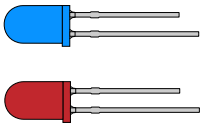
RESISTOR



TRIM POTENTIOMETER

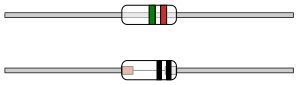


LED



SILICON DIODE

GERMANIUM DIODE



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

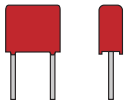
RECTIFIER DIODE

Some Schottky diodes also look like this.

SCHOTTKY DIODE

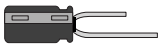
ZENER DIODE

FILM CAPACITOR



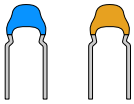
Not polarized. Color may vary by brand and type.

ELECTROLYTIC CAPACITOR



Polarized. The negative side is marked.

MLCC

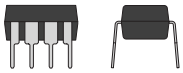


Not polarized. MLCC stands for "multi-layer ceramic capacitor."

TANTALUM CAPACITOR

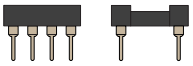
Polarized. The positive side is marked.

IC OR OP-AMP

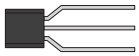


Charge pumps and delay chips also look like this. They may have more than 8 legs.

IC SOCKET



TRANSISTOR OR JFET



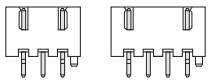
Some voltage regulators also look like this.

DIP SWITCH

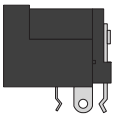
The number of switch positions may vary.

WIRE ASSEMBLY

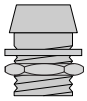
WIRE ASSEMBLY HEADER



DC JACK



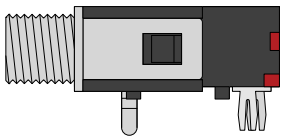
LED BEZEL



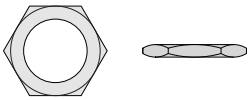
HARDWARE IDENTIFICATION

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

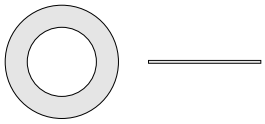


MOUNTING NUT



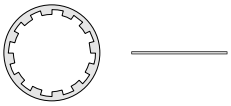
DIAMETER: 0.54" / 13.7mm

OUTER WASHER



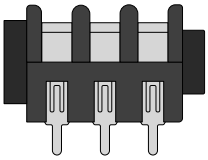
DIAMETER: 0.6" / 15.2mm

LOCK WASHER

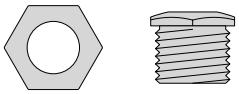


DIAMETER: 0.5" / 12.7mm

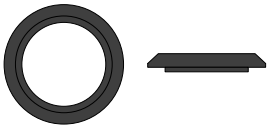
SEND/RETURN JACK



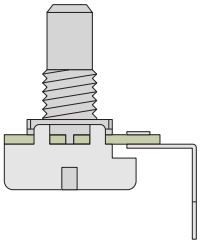
CHROME FERRULE



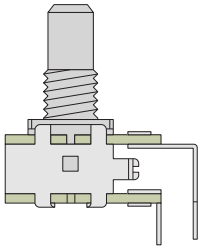
INSULATING WASHER



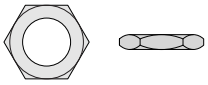
POTENTIOMETER (SINGLE)



POTENTIOMETER (DUAL)

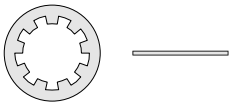


MOUNTING NUT



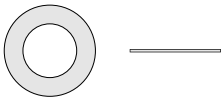
DIAMETER: 0.44" / 11.2mm

LOCK WASHER



DIAMETER: 0.5" / 12.7mm

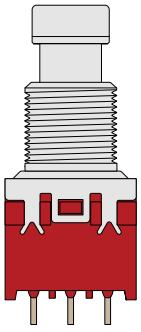
OUTER WASHER



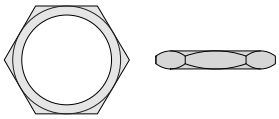
DIAMETER: 0.475" / 12mm

KNOB

FOOTSWITCH

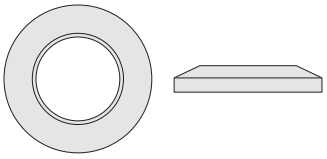


MOUNTING NUT



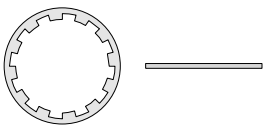
DIAMETER: 0.63" / 16mm

DRESS NUT



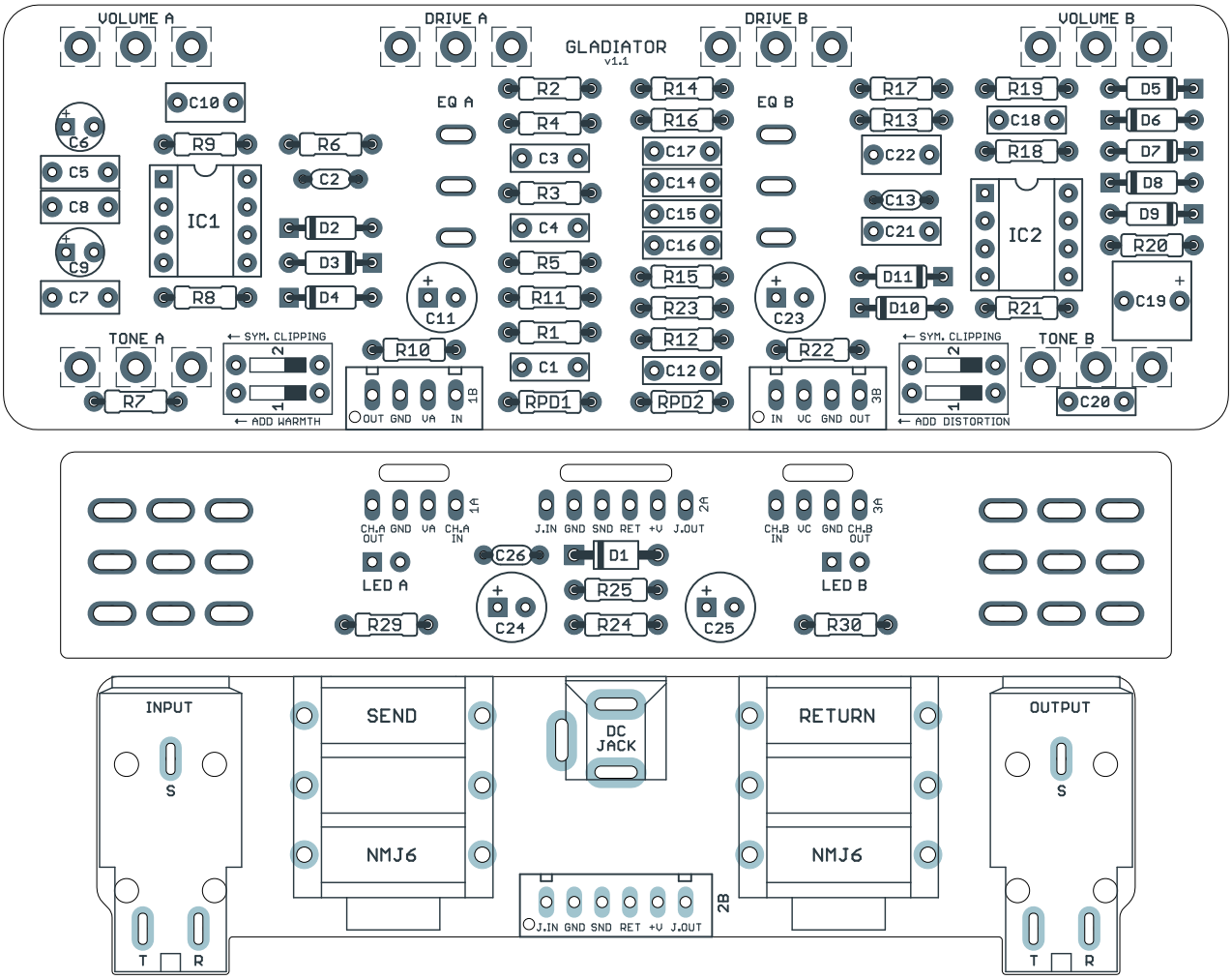
DIAMETER: 0.77" / 19.6mm

LOCK WASHER



DIAMETER: 0.63" / 16mm

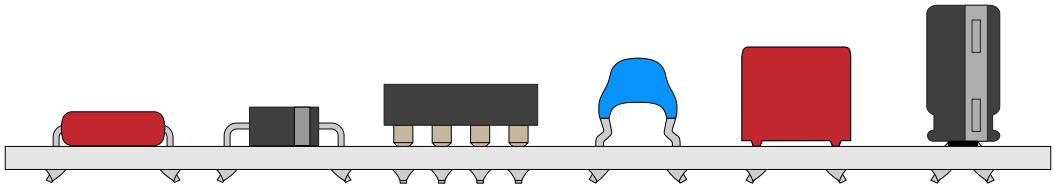
PCB ASSEMBLY OVERVIEW



Before you begin, snap apart the main PCB and footswitch board and break off the tabs from each using needle-nose or flat-head pliers. You'll be left with the three PCBs shown above.

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

1. Resistors & diodes
2. IC sockets & DIP switches
3. MLCC capacitors
4. Film capacitors
5. Electrolytic capacitors



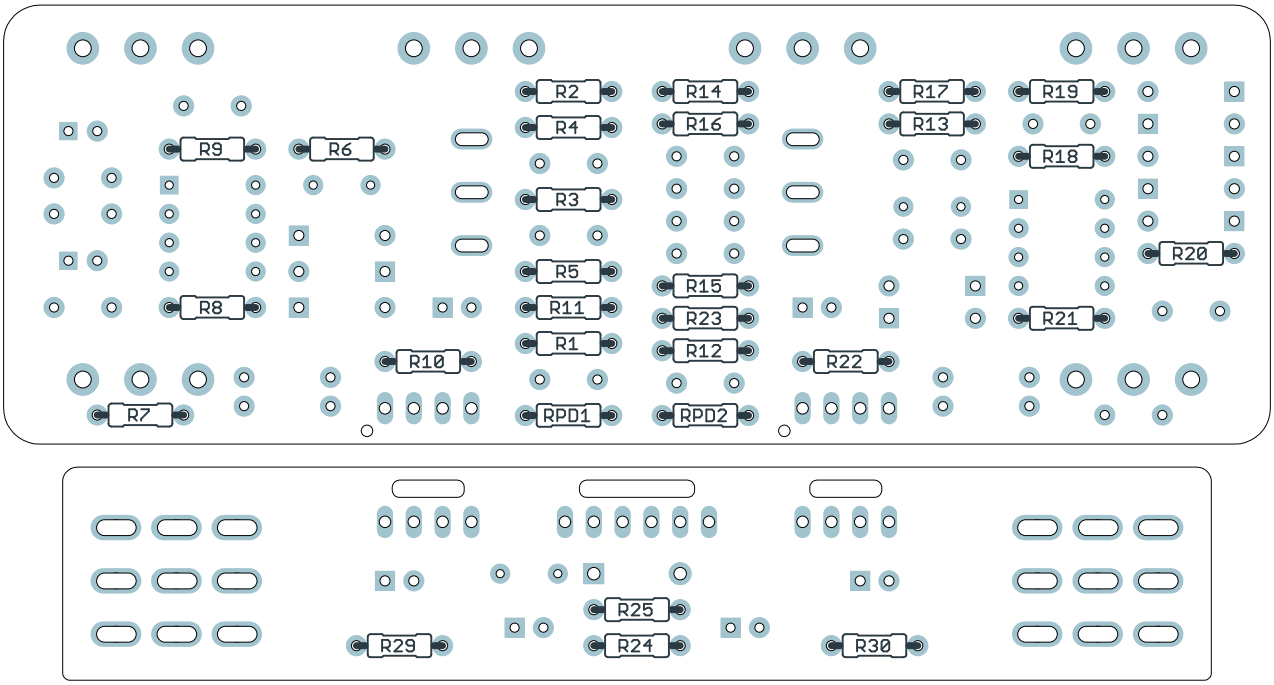
RESISTORS

PART	VALUE
R1	1M
R2	47k
R3	4k7
R4	3k3
R5	20k
R6	1k
R7	220R
R8	10k

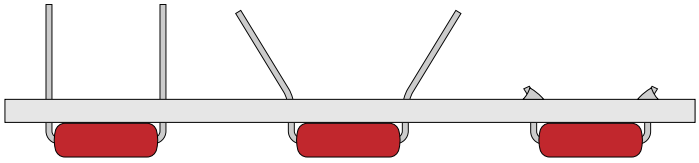
PART	VALUE
R9	1k
R10	47k
R11	47k
R12	1M
R13	133k
R14	30k1
R15	28k7
R16	4k7

PART	VALUE
R17	10k
R18	220k
R19	5k6
R20	1k
R21	9k53
R22	47k
R23	47k
R24	47R

PART	VALUE
R25	47R
R29	22k
R30	22k
RPD1	2M2
RPD2	2M2



Using the parts list above, populate the resistors by pushing them through the holes and bending the leads outward at an angle to hold them in place. Resistors are not polarized, so they will work in any direction. Turn the board upside-down to keep the components held in place while you solder.



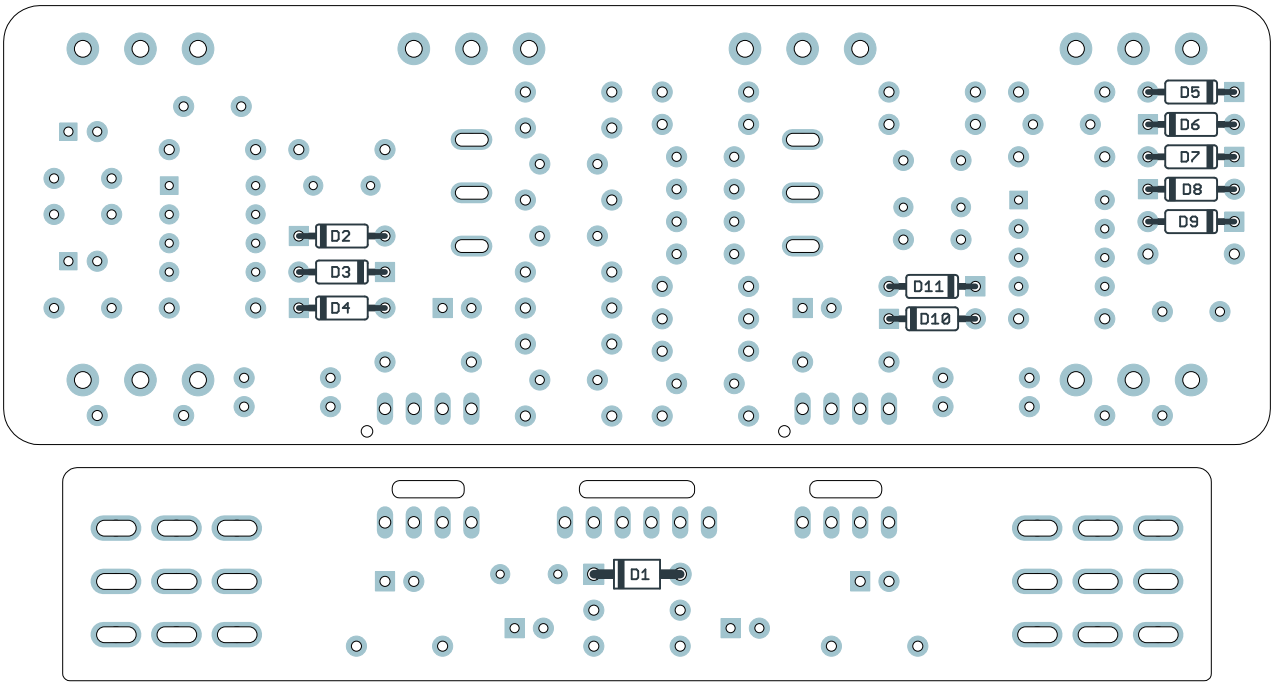
Don't try to do all of the resistors at once. You'll want to stop periodically flip the board and solder everything, then cut the leads using the wire snippers to make room for more.

DIODES

PART	VALUE
D1	1N5817
D2	BAS33
D3	BAS33
D4	BAS33

PART	VALUE
D5	BAS33
D6	BAS33
D7	BAS33
D8	BAS33

PART	VALUE
D9	BAS33
D10	1N914
D11	1N914

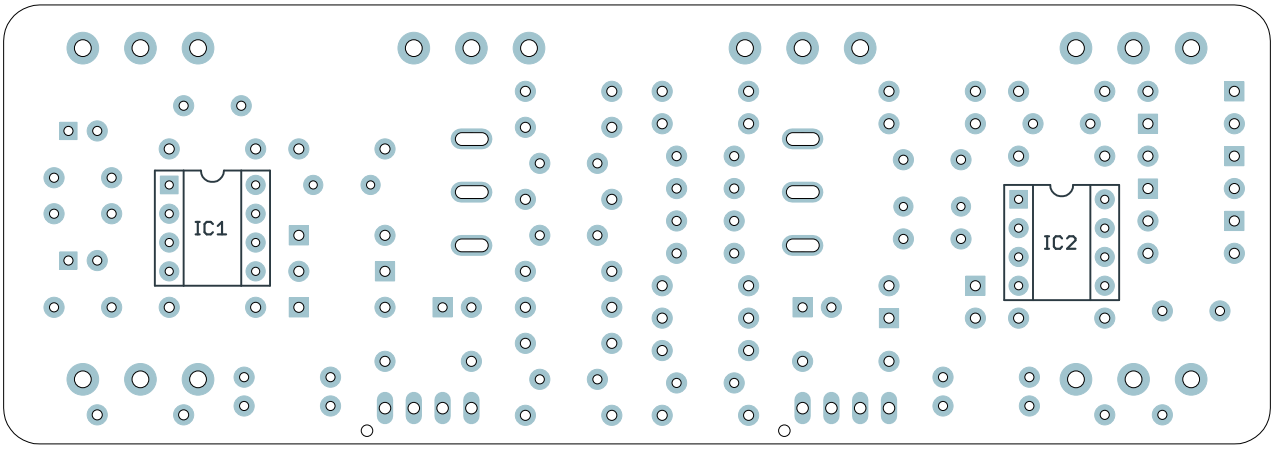


Next, you'll populate the diodes. Diodes are polarized, so make sure to identify the polarity band (which indicates the "cathode", or negative side) and match the band to the footprint on the PCB.

The BAS33 and 1N914 diodes are very similar in appearance. The names are printed on the side, but if they're too hard to read, the easiest way of telling them apart is that the glass case of the BAS33 is a little longer.

SOCKETS & ICS

PART	VALUE
IC1	RC4558P
IC2	RC4558P

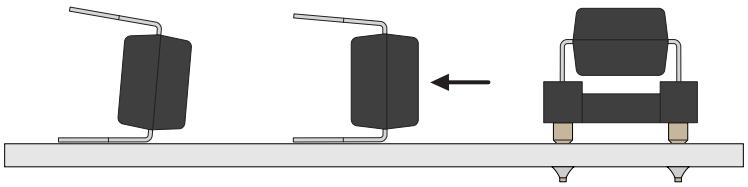


Next up are the sockets. You can't bend the leads of the socket as with the other components, so they won't stay in on their own until they're soldered. Flip the PCB over and use gravity to hold them in place.

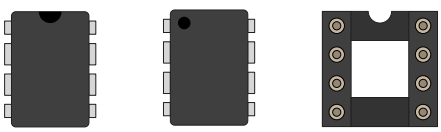
Installing the ICs

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

The legs of the ICs 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.

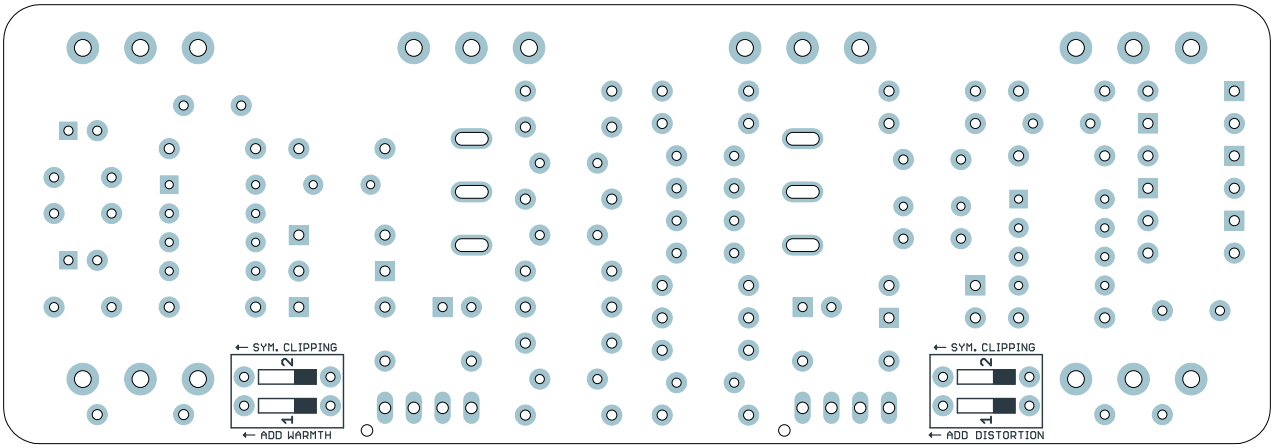


ICs may have two different orientation marks: either a dot in the upper-left or a half-circle notch in the middle of the top side. Some ICs have both marks. This shows which way the IC should be rotated when inserting it into a socket (the socket also has a half-circle notch).



DIP SWITCHES

PART	VALUE
SW1	2-position DIP switch
SW2	2-position DIP switch



Now we'll do the DIP switches. These are very slightly taller than the IC sockets, so they should be done after the sockets are soldered, but the process is the same. The legs aren't long enough to be bent, so just turn the PCB upside down and let it hold the DIP switch in place while you solder.

Make sure the "ON" text faces to the left. If they are installed backwards, they will still work, but the switch positions will be inverted compared to the labels on the PCB.

Using the DIP switches

The DIP switch modes modes are labeled on the PCB silkscreen. The switch on the left is for channel A, and the switch on the right is for channel B. To engage each mode, move the switch in the direction of the arrow. If the switch is moved away from the arrow, the mode is disengaged.

The default is OFF for all four settings. The description of each switch mode are as follows.

Channel A, Sym. Clipping: When engaged, it bypasses one of the soft-clipping diodes so there is only one in each direction.

Channel A, Add Warmth: When engaged, this shifts the range of the tone control to remove a lot of the brightness at the higher end of the range.

Channel B, Sym. Clipping: When engaged, it bypasses one of the soft-clipping diodes so there are two in each direction.

Channel B, Add Distortion: When engaged, the hard-clipping diodes are active.

Note that the hard-clipping diodes essentially override the soft-clipping diodes since they clip at a lower signal level. If Add Distortion is active, you'll notice little or no change in sound when engaging or disengaging Sym. Clipping.

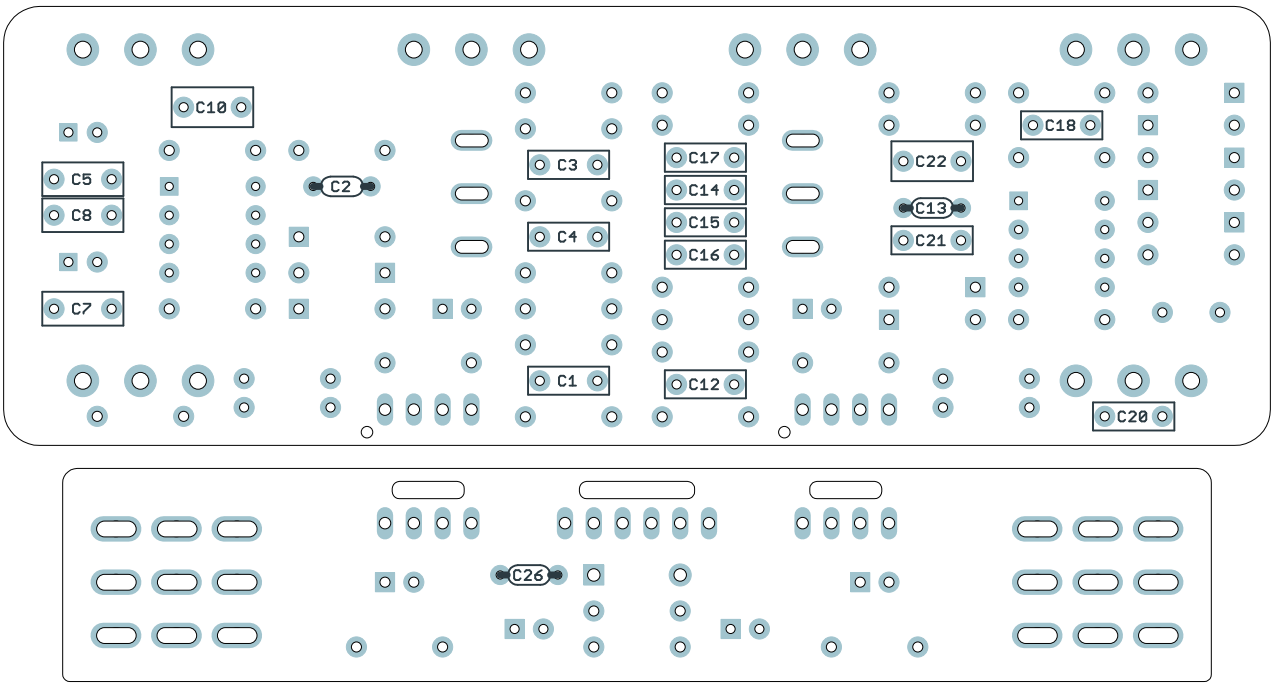
CAPACITORS (NON-POLARIZED)

PART	VALUE
C1	22n (0.022)
C2	47pF MLCC
C3	47n (0.047)
C4	100n (0.1)
C5	220n (0.22)

PART	VALUE
C7	220n (0.22)
C8	220n (0.22)
C10	1uF
C12	10n (0.01)
C13	100pF MLCC

PART	VALUE
C14	10n (0.01)
C15	10n (0.01)
C16	15n (0.015)
C17	4n7
C18	100n (0.1)

PART	VALUE
C20	10n (0.01)
C21	10n (0.01)
C22	1uF
C26	100n MLCC



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

C19 is a non-polarized film capacitor, but significantly larger than the others, so we will do this at the same time as the electrolytics on page 16.

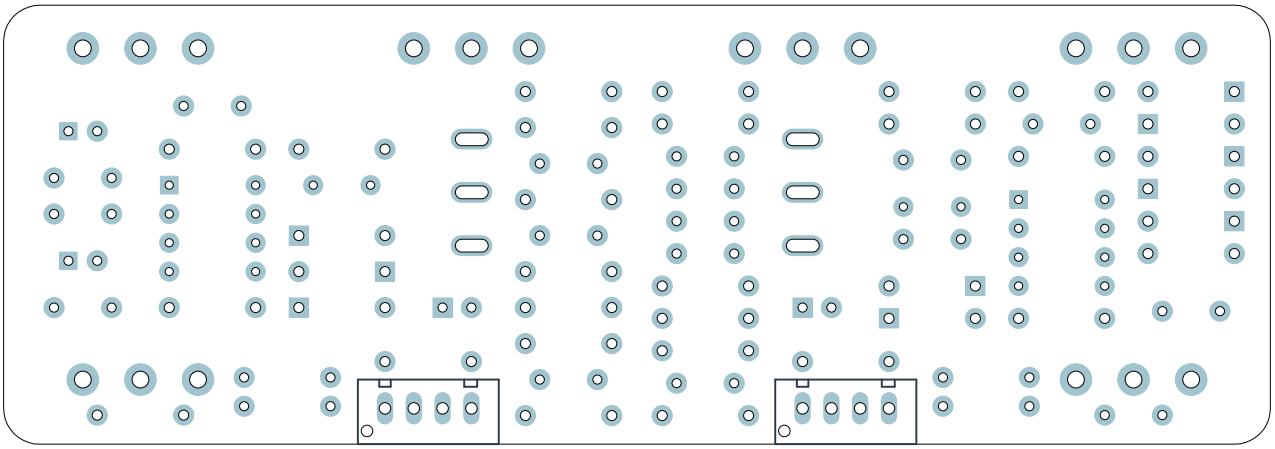
MLCCs and box capacitors are not polarized, so they will work in any direction, but to keep things neat, it's best to put them all facing the same way.

Note: The red box-film capacitors have the value printed on the side, while the blue box-film capacitors have the value printed on the top.

C4 and C18 are usually red, but may read "µ1J100" on the top rather than the side.

C2 (47pF) and C13 (100pF) MLCCs are always blue and typically come taped to cardboard. The value will be written on the cardboard. C26 (100n MLCC) is always yellow. The code is hard to read, but it can be identified by color.

WIRE HEADERS



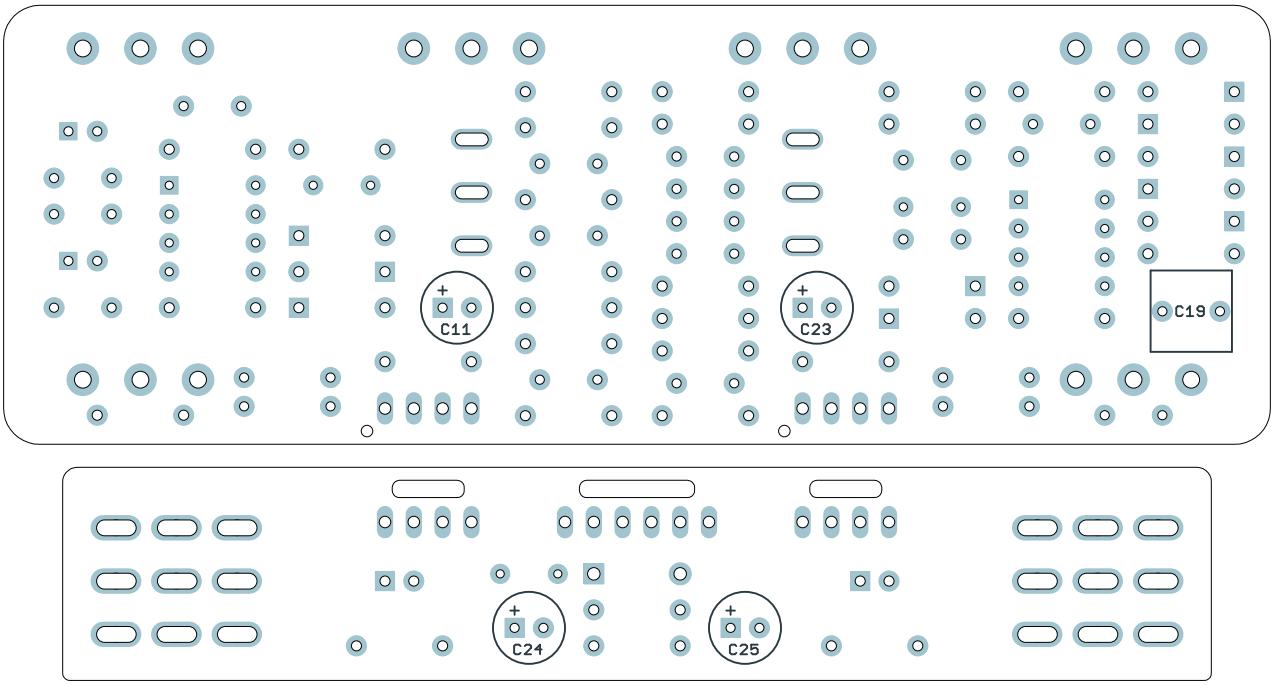
Install the two 4-pin headers (wire connectors) as shown above. These have a polarity pin, so as long as they are pressed all the way down, there's only one possible way to install them. They do fit pretty tightly in the holes, though, so press firmly.

There's also a 6-pin header on the I/O board that we will do in a later step.

CAPACITORS (POLARIZED)

PART	VALUE
C11	100uF electro
C19	4.7uF film
C23	100uF electro

PART	VALUE
C24	100uF electro
C25	100uF electro



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.

Along with the electrolytics, we will also install the C19 film capacitor since it is a similar height. This is a non-polarized capacitor and so the “+” symbol on the PCB can be ignored for the kit.

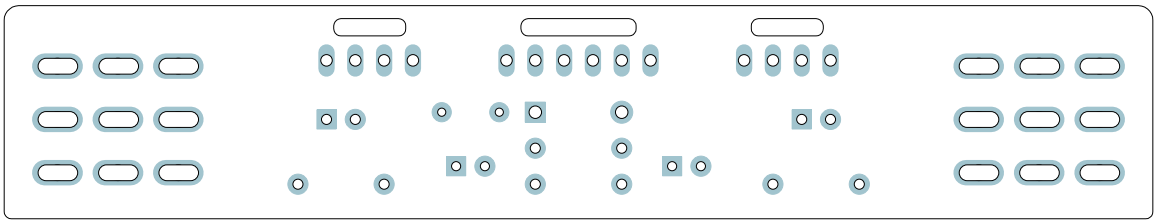
Note that C6 and C9 are not used in this kit and should be left empty on the PCB.

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

FOOTSWITCH PCB

PARTS

- 4-strand wire assembly (2)
- 6-strand wire assembly



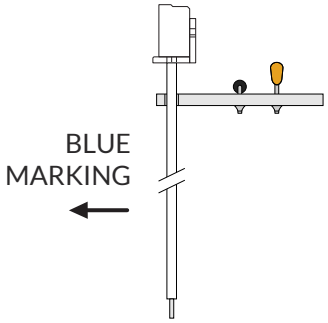
Next, it's time to finish up the footswitch board. You should have done most of the on-board components on this board in a previous step, but if not, go back and do those.

There will be one longer assembly with 6 wires and two shorter ones with 4 wires. The longer one goes in the middle and the shorter ones go on the left and right sides. The wire assemblies should then be soldered to the footswitch board as shown.

STEP 1

First, thread the wire through the strain-relief slots, with the blue side facing outward and the PCB's previously-installed components facing up.

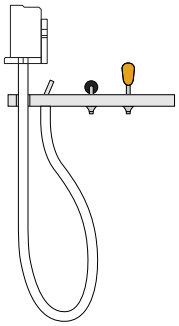
For now, pull it down through the slot as far as it can go.



STEP 2

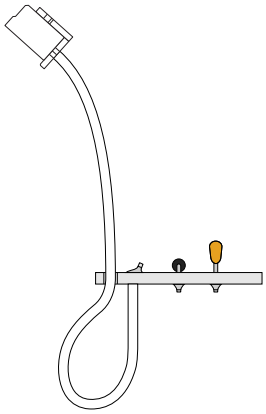
Next, bend the wires back upward and fit the ends of the wires into the solder pads.

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.

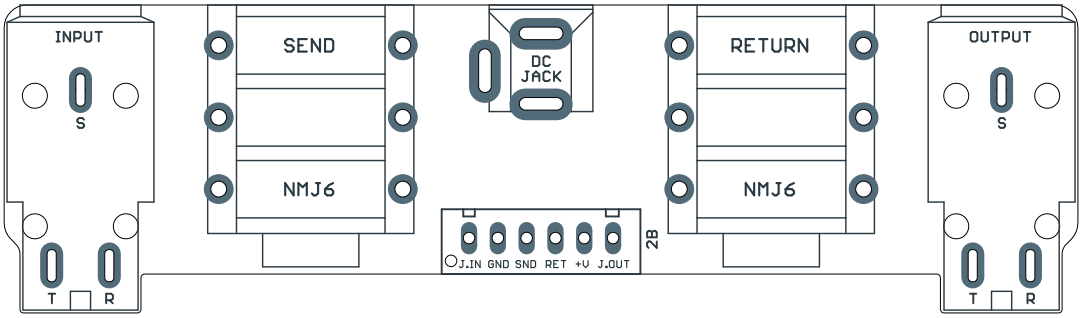


Once all three wire assemblies are soldered, set the footswitch PCB aside. We'll solder the actual footswitches and LEDs in a later step.

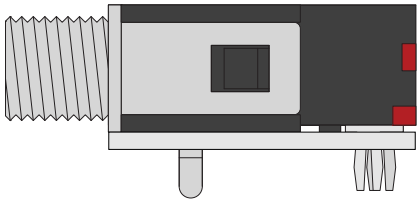
INPUT/OUTPUT PCB

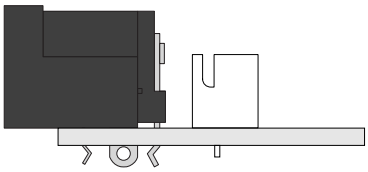
PARTS

- Input & output jacks
- Send & return jacks
- DC jack
- Wire header



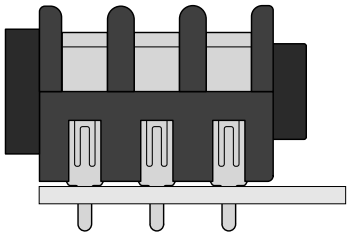
Next comes the input/output PCB. Find the two input/output jacks, the DC jack and the wire header and snap them in place. The PCB is designed for them to fit securely, so you can do them all at once before flipping and soldering.





Next, we'll do the send and return jacks, which are shorter and wider than the input/output jacks. These typically come in individual bags. Open the bags and set aside the included hardware for now.

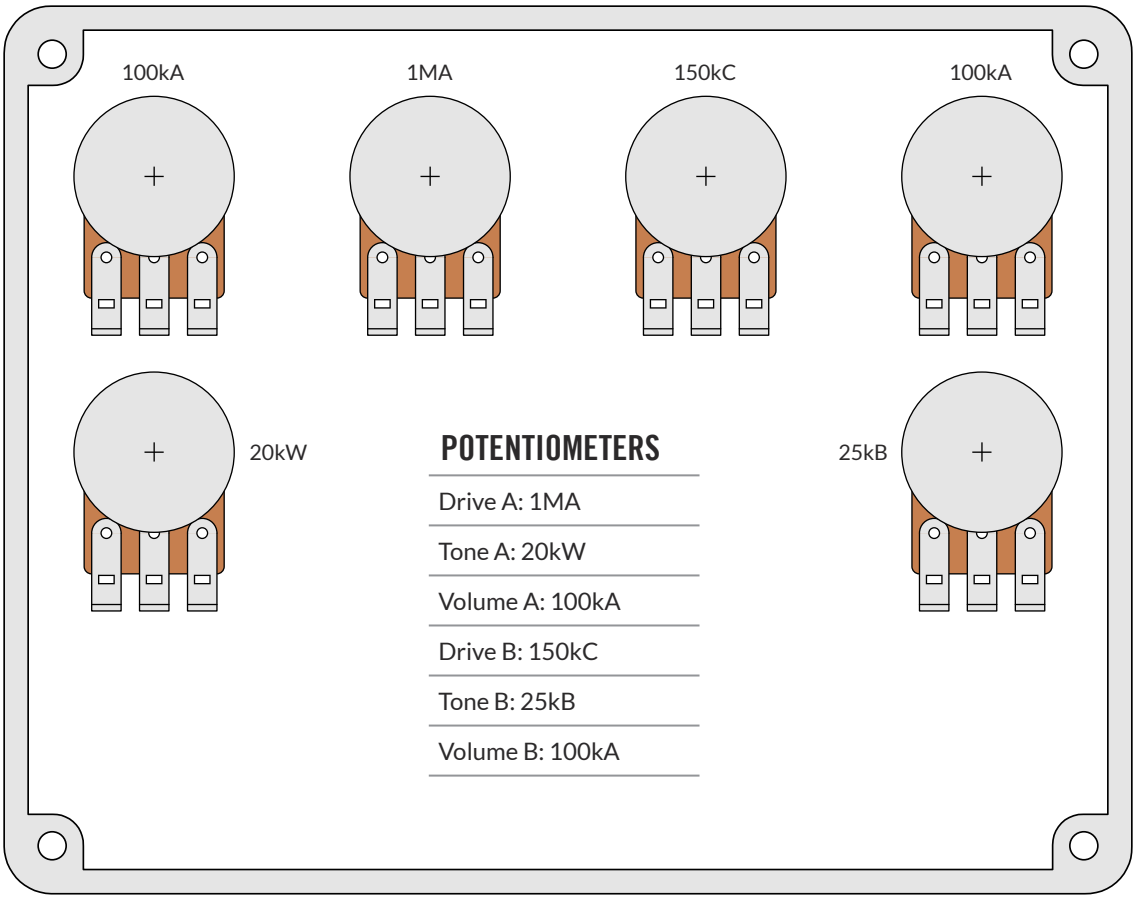
This type of jack does not snap in like the other components, so you'll have to set them in the holes and then flip the board upside down while holding them in place. Ensure the jack is facing forward, the same way as the input & output jacks, because it will fit in the holes either way.



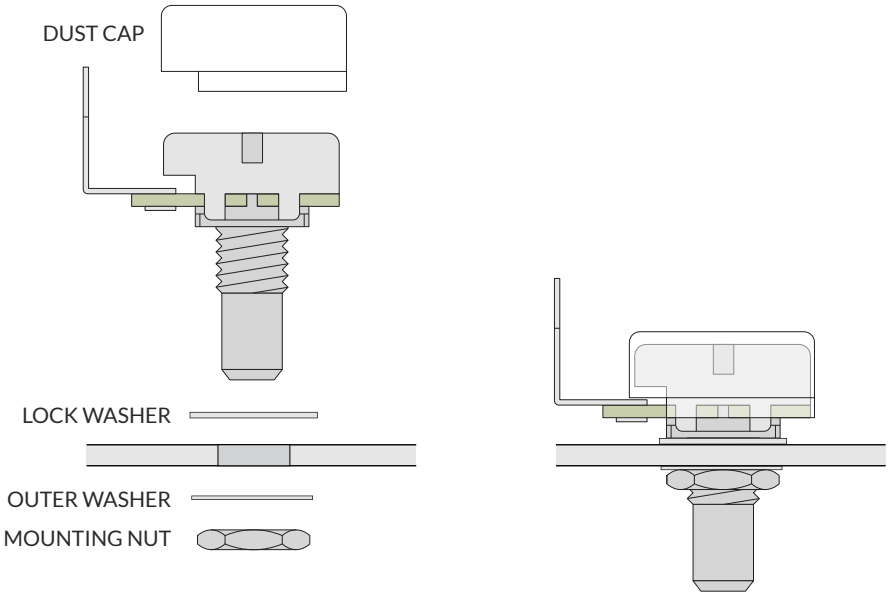
Solder each jack, taking care to make them as straight as possible relative to the white outline on the PCB. It's recommended to just solder one of the six pins at first, then check it. If it's out of alignment, the joint can be reheated to allow the jack to be adjusted. Once the next pin is soldered, it will stay aligned.

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

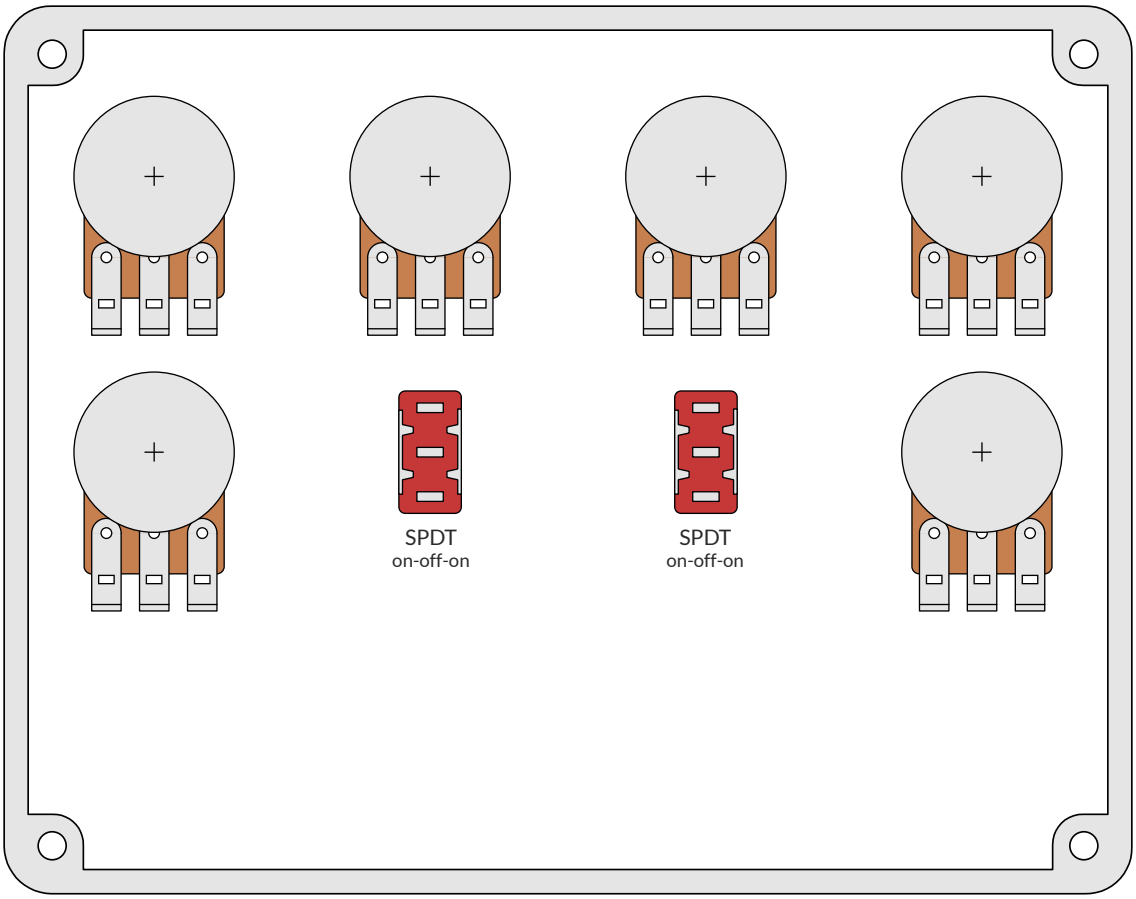
ENCLOSURE LAYOUT: POTENTIOMETERS



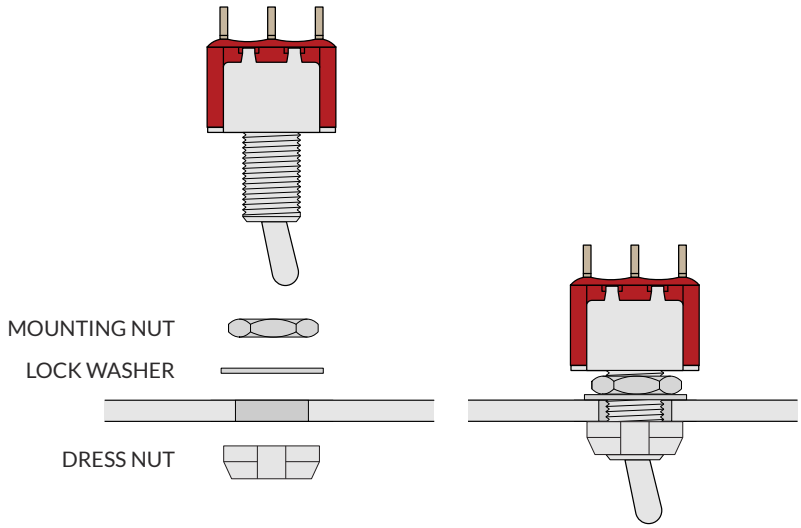
Attach the potentiometers to the enclosure as shown. Make sure they're aligned as straight as possible, then tighten the outer nut firmly. It can be helpful to use a second set of pliers to hold the potentiometer in place from the inside while tightening the nut from the outside.



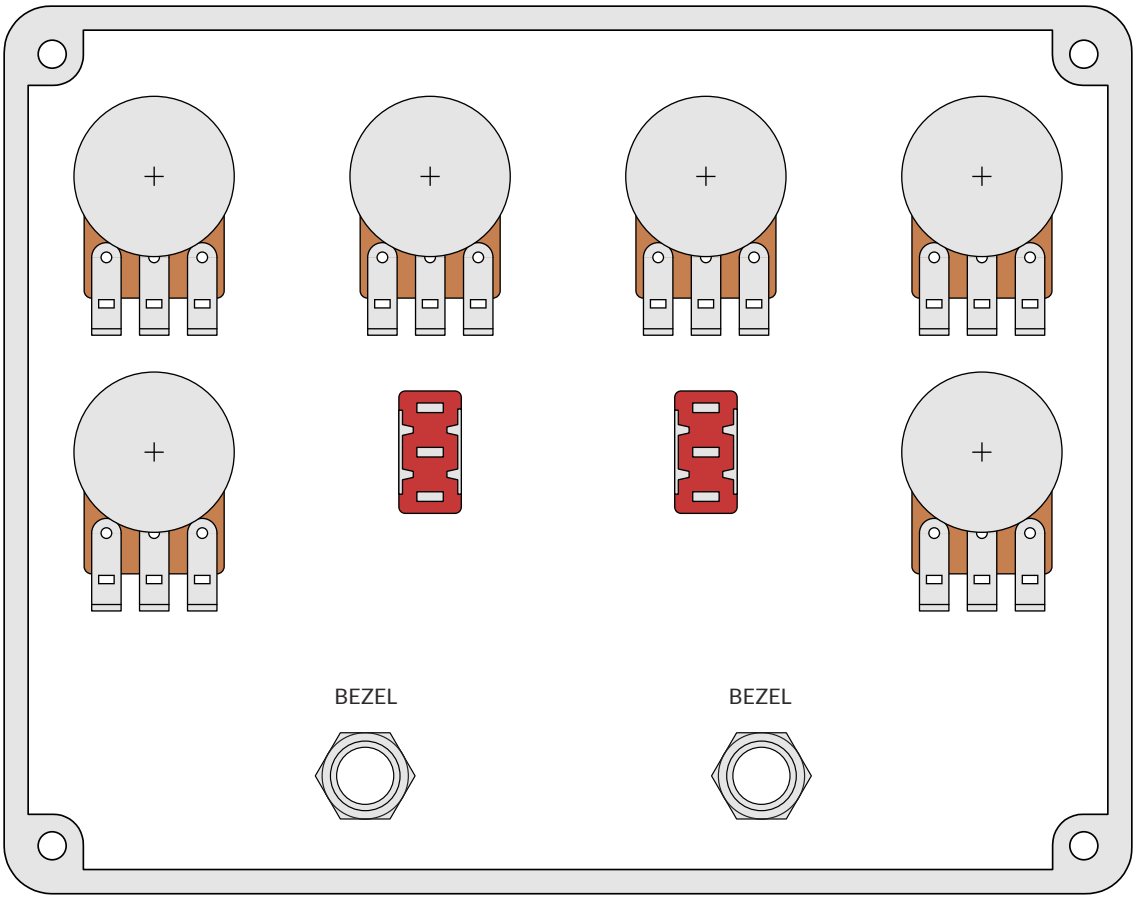
ENCLOSURE LAYOUT: TOGGLE SWITCHES



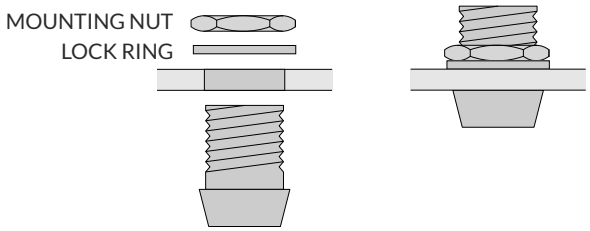
Attach the two toggle switches to the enclosure as shown, ensuring that they are aligned straight up and down. Use flat-nose pliers to grip the flat sides of the dress nut and tighten securely. It's easiest to hold it with your thumb and forefinger from the inside to prevent rotation while you tighten from the outside.



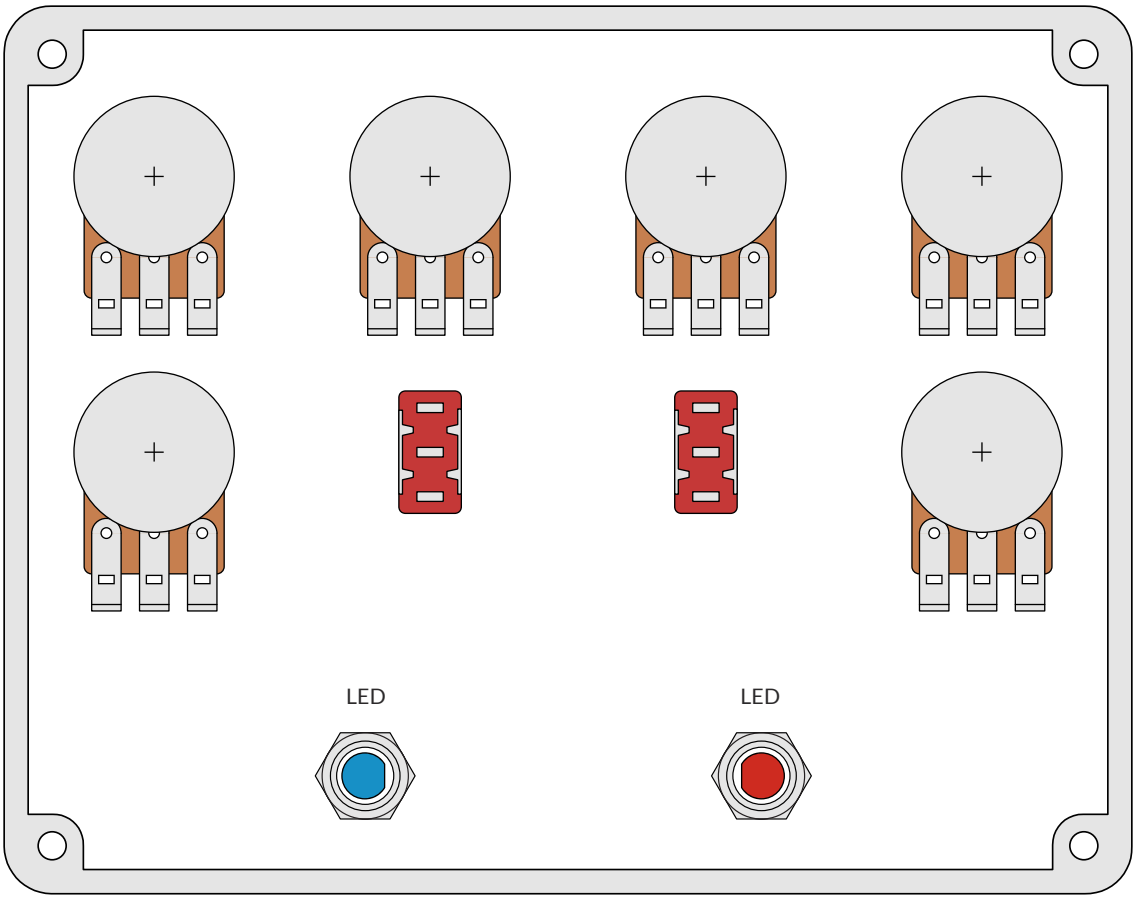
ENCLOSURE LAYOUT: LED BEZELS



Next, attach the LED bezels to the enclosure. You'll need to hold the bezel in place from the outside when tightening the nut.

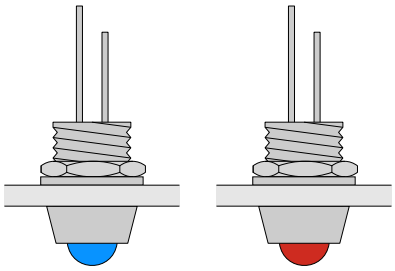


Be aware that the top of the bezel is fairly sharp. Try using a rubber band for grip instead of just pressing your finger against it.



Next, set the LEDs into the bezels, with the long leg facing toward the left. The blue one goes in the left bezel and the red in the right.

This will be reversed on the finished pedal when viewing it from the front, so the blue LED is for channel A and the red LED is for channel B.



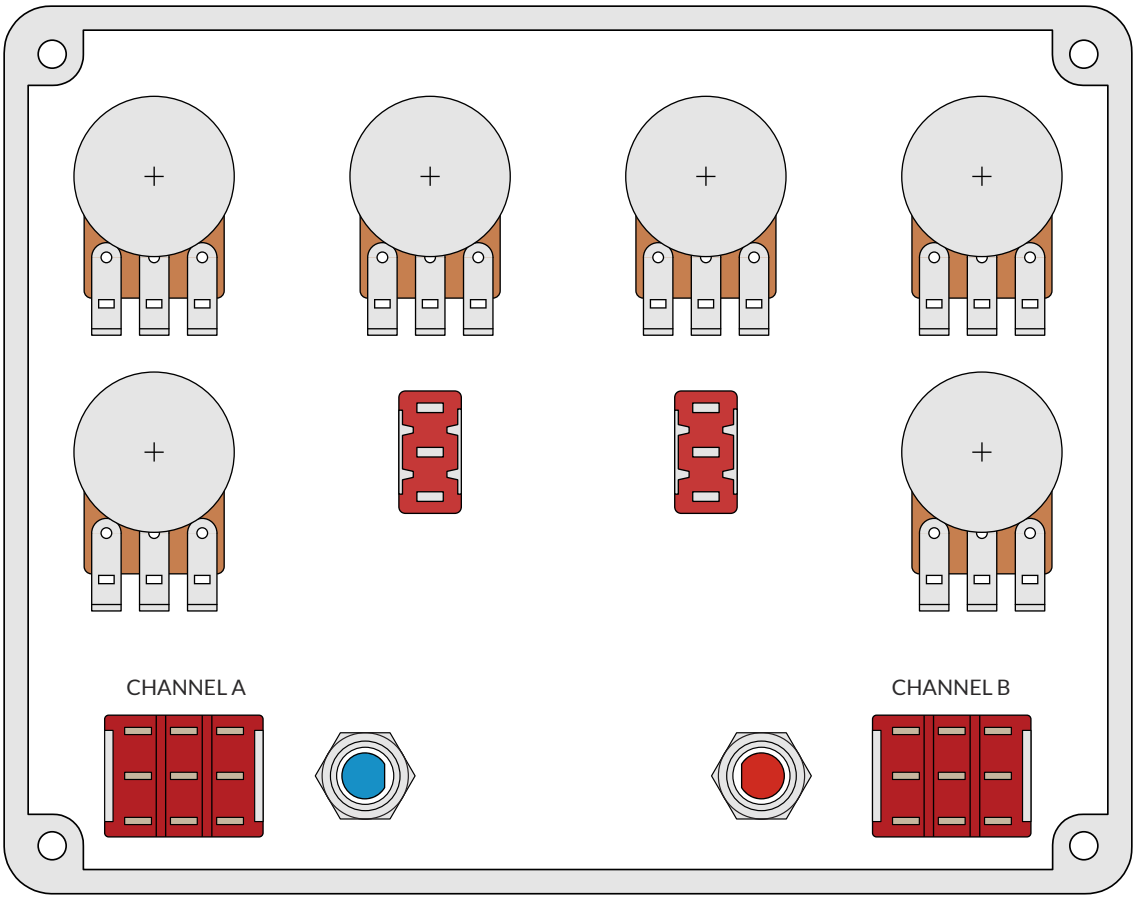
Red LED orientation

This simple step gets its own page because of one very important precaution regarding the red LED. The lens of an LED typically has a flattened side to mark the cathode (i.e. negative) pin, while the longer leg indicates the anode (i.e. positive) pin.

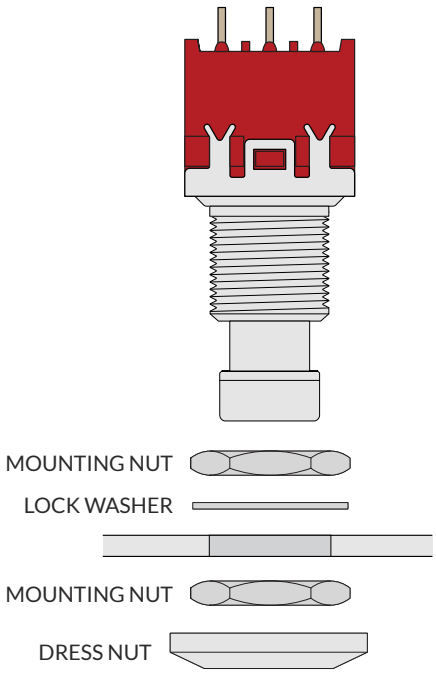
In sourcing LEDs for this kit and the Theseus, we spent a long time trying to find a model of red LED that was perfectly balanced in brightness with the blue LED that we use our other kits. After trying several different types, we found an exact match, but it had one strange characteristic: the manufacturer reverses the lens so the flat side marks the anode. The long leg still indicates the anode.

If both of the long legs face to the left, lining up with the square pads on the footswitch PCB, it will work as expected. However, the PCB itself shows the “correct” footprint (i.e. for a standard LED) on the underside, meaning this red LED’s flat side will not match the PCB.

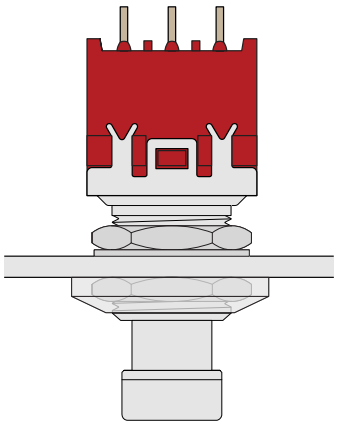
ENCLOSURE LAYOUT: FOOTSWITCHES



Next, attach the footswitches, ensuring the lugs are oriented horizontally as shown in the diagram.

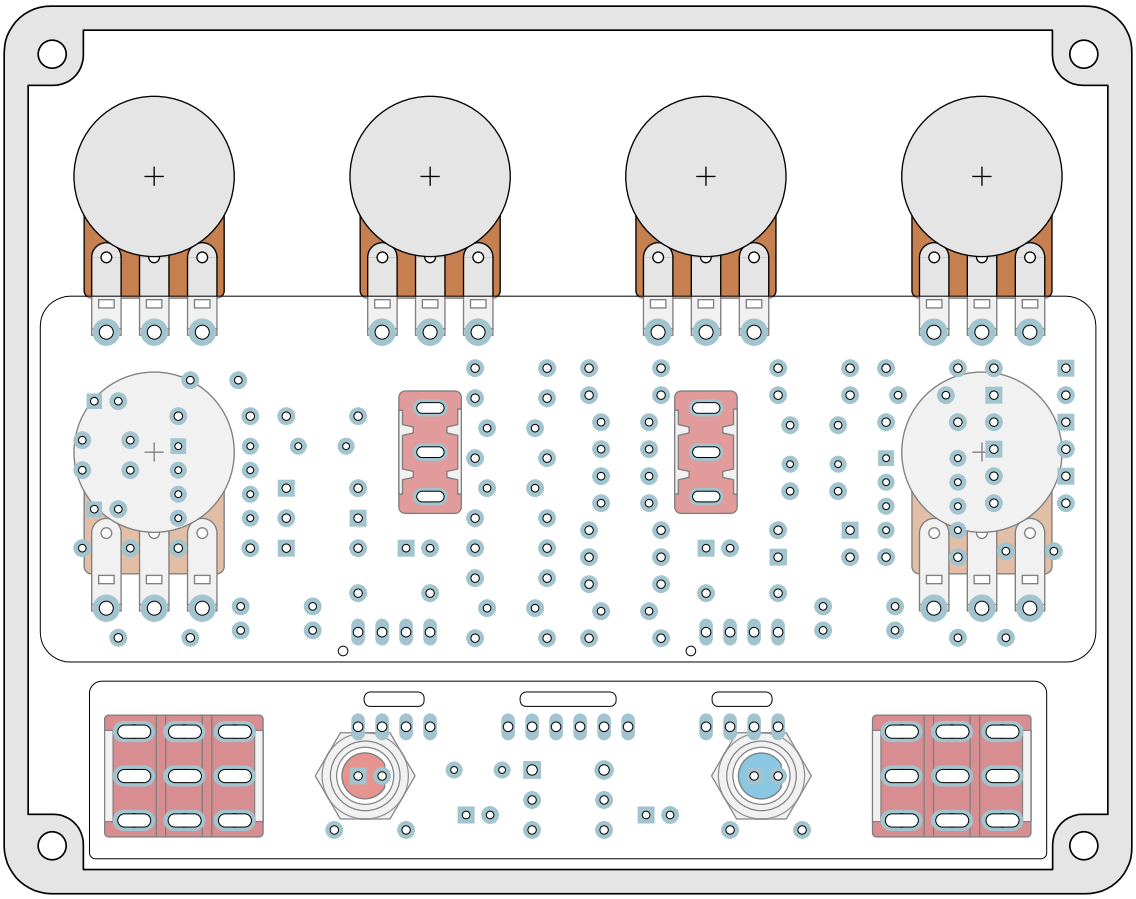


Note that the threading has a notch in one side. For the sake of appearance, it looks best if the notches are oriented the same on both switches, either up or down. However, the switches are functionally symmetrical so this doesn't make a difference to the operation.



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.

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 above. You may need to adjust the position of the potentiometers slightly if they don't line up with the holes.

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.

Next, do the same thing with the footswitch board—the 3PDT footswitches and the LEDs. Before soldering, double-check to **make sure the long legs of each LED line up with the square pad on the PCB**, and refer back to page 22 for the precaution about the red LED's orientation.

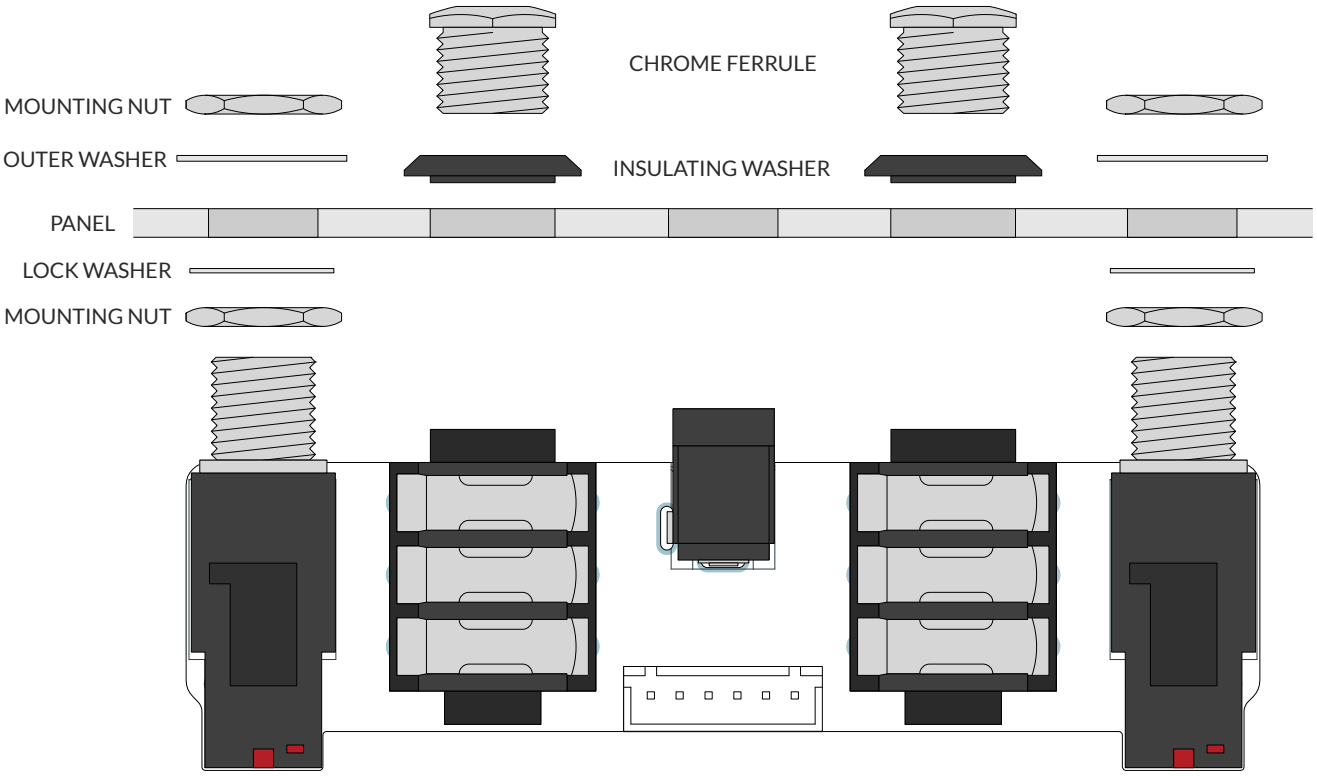
Why solder everything inside the enclosure before testing it?

Aion FX projects are designed to be extremely easy to remove from the enclosure for troubleshooting, with no desoldering required. As a result, it's much easier to just build the whole thing start to finish.

This method also helps ensure that all of the hardware is aligned to the holes on the enclosure and that the solder joints will not be stressed after installation.

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, but if there are any issues then it only takes a minute or two to disassemble, and it goes back together just as easily.

ENCLOSURE LAYOUT: INPUT/OUTPUT PCB

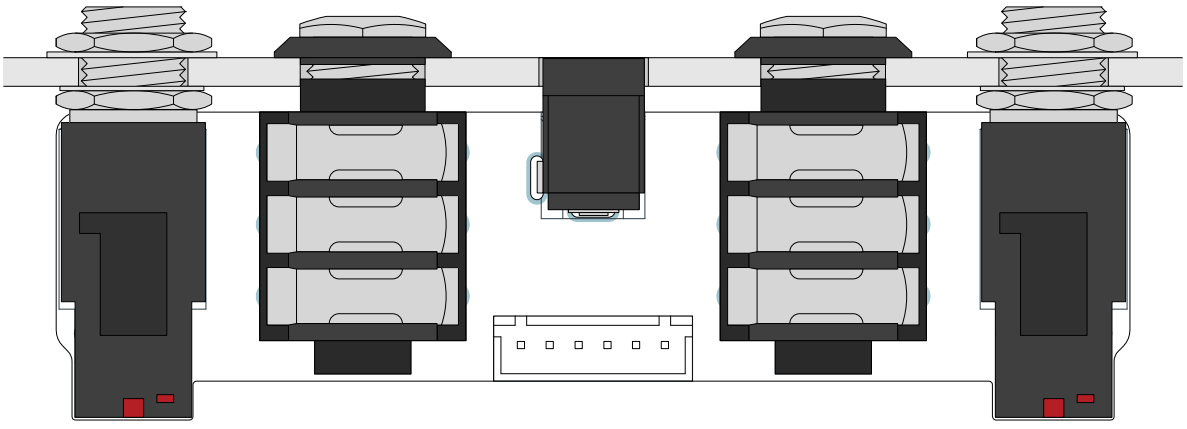


Affix the input/output PCB to the north-facing panel of the enclosure as shown.

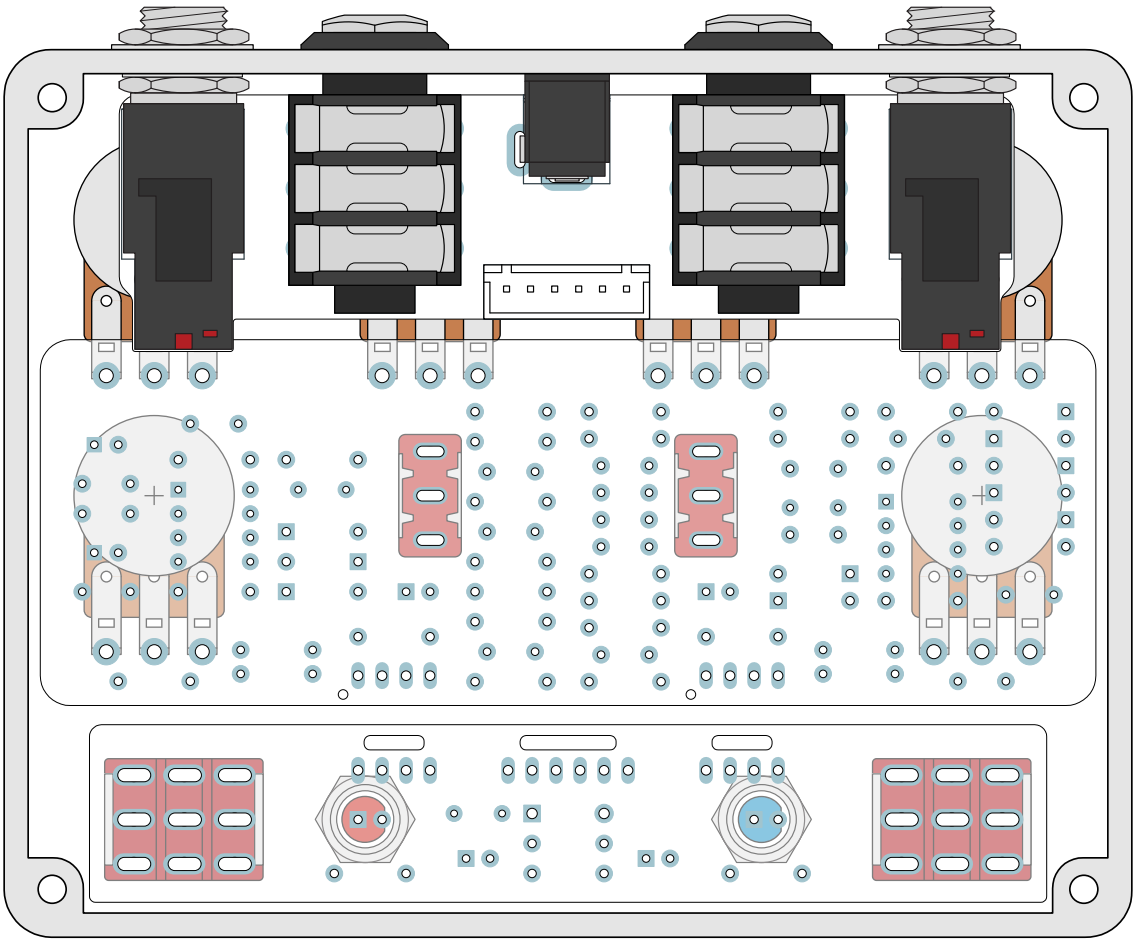
Note the use of two mounting nuts on each of the jacks, one inside and one outside. The inner nut acts as a spacer to set the DC jack flush with the outside of the enclosure. The inner nuts should be threaded as far down as they can go.

The send/return jacks will typically come with two or three additional black plastic washers that are different from the insulating washers shown in the diagram. These are not used in the kit.

The chrome ferrule for these jacks should be tightened by hand only. Only the outer jacks are actually supporting the PCB, so the send/return jacks just need their ferrules tight enough that they don't come loose. If the ferrule is too tight, it will strip the plastic threading on the inside of the jack.



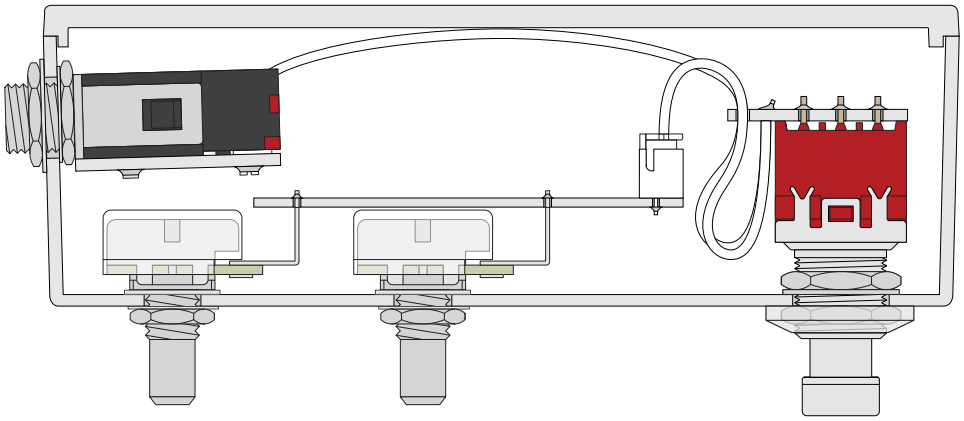
ENCLOSURE LAYOUT: FINISHED DIAGRAM



This is a complete diagram of the enclosure with all three PCBs in place.

FINAL TESTING & ASSEMBLY

Now, just plug the 3 wire assemblies into their respective headers and make sure they're secure. Here is a cross-section of the inside of the completed pedal.



At this point, you have completed the full circuit as far as the electrons are concerned. Plug in a 9-volt supply and test it out with a guitar and an amplifier.

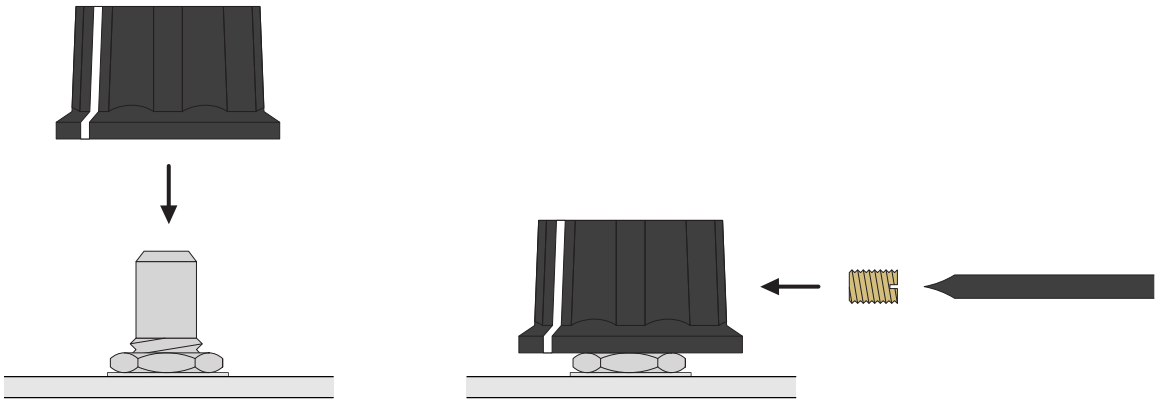
Test the bypass switch a few times, then start turning the knobs and see if everything sounds OK. If it works, great! If not, don't be discouraged. See page 30 for troubleshooting info.

Finishing touches

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

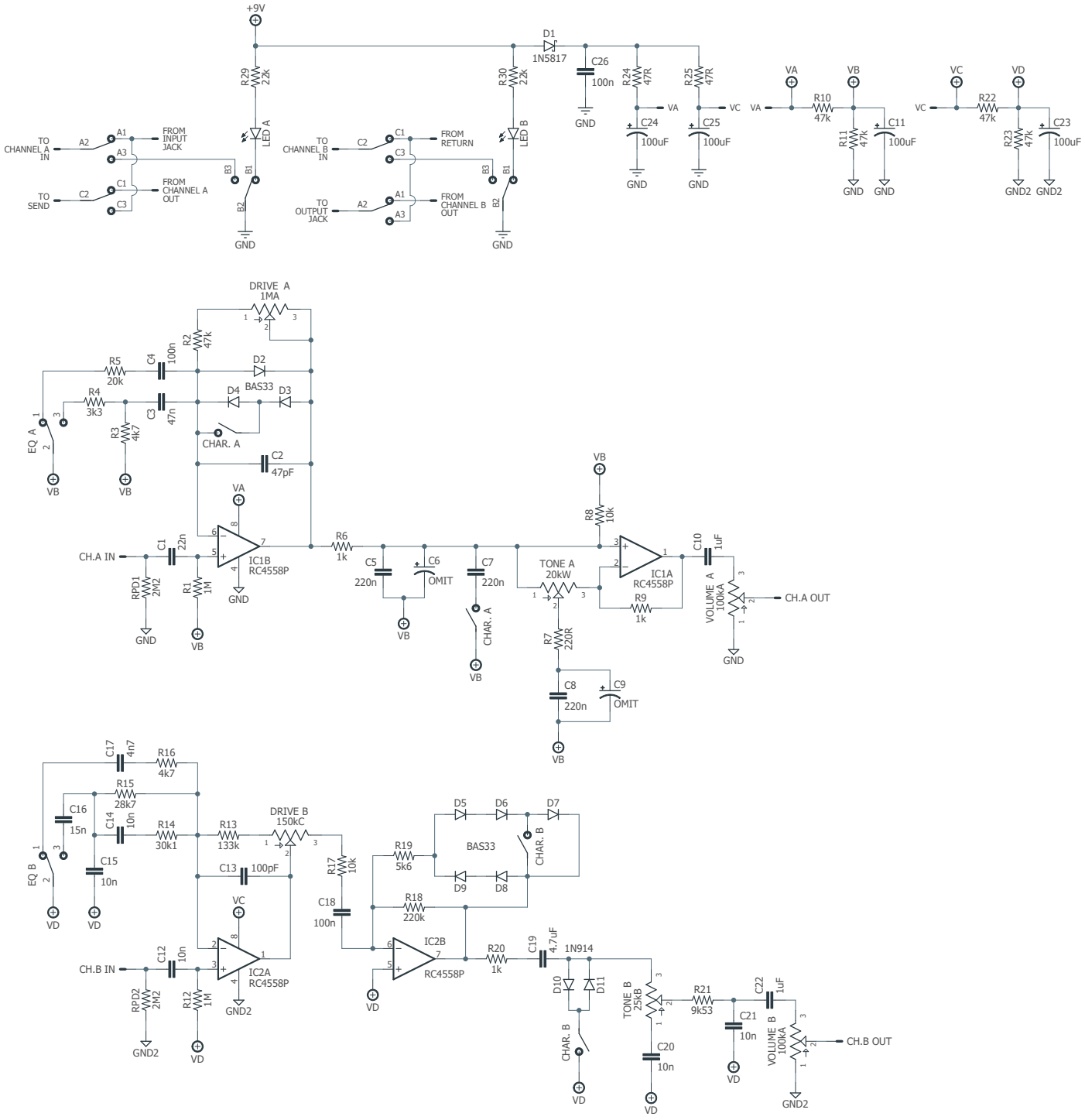
Using a small flat-head screwdriver (no more than 0.1" / 2.5mm in diameter), firmly tighten the set screw until it presses against the shaft of the potentiometer and holds the knob in place.

Be careful not to over-tighten or you may damage the set screw. But if it's not tight enough, the knob will be more likely to fall off or lose its alignment with the markings on the enclosure.



Last, just close the panel on the back using the four screws. That's it!

SCHEMATIC


+9V
D1
1N5817
C26
100n
R29
22k
R30
22k
R24
47R
R25
47R
VA
VB
VC
VD
R10
47k
R11
47k
C11
100uF
R22
47k
R23
47k
C23
100uF
C24
100uF
C25
100uF
GND
GND2
TO CHANNEL A IN
FROM INPUT JACK
TO SEND
FROM CHANNEL A OUT
LED A
TO CHANNEL B IN
FROM RETURN
TO OUTPUT JACK
FROM CHANNEL B OUT
LED B
DRIVE A
1MA
R2
47k
R5
20k
C4
100n
D2
D4
BAS33
D3
R4
3k3
C3
47n
R3
4k7
EQ A
CHAR. A
C2
47pF
CH.A IN
C1
22n
IC1B
RC4558P
RPD1
2M2
R1
1M
R6
1k
C5
220n
C6
OMIT
C7
220n
TONE A
20kW
R8
10k
IC1A
RC4558P
R9
1k
C10
1uF
VOLUME A
100kA
CH.A OUT
R7
220R
C8
220n
C9
OMIT
DRIVE B
150kC
R17
10k
C17
4n7
R16
4k7
R15
28k7
C16
15n
C14
10n
R14
30k1
R13
133k
C13
100pF
EQ B
C15
10n
C18
100n
D5
D6
D7
D9
D8
BAS33
CHAR. B
R19
5k6
R18
220k
CH.B IN
C12
10n
IC2A
RC4558P
RPD2
2M2
R12
1M
IC2B
RC4558P
R20
1k
C19
4.7uF
1N914
D10
D11
TONE B
25kB
R21
9k53
C22
1uF
C21
10n
C20
10n
VOLUME B
100kA
CH.B OUT

FULL PARTS LIST

Resistors

PART	VALUE
R1	1M
R2	47k
R3	4k7
R4	3k3
R5	20k
R6	1k
R7	220R
R8	10k

PART	VALUE
R9	1k
R10	47k
R11	47k
R12	1M
R13	133k
R14	30k1
R15	28k7
R16	4k7

PART	VALUE
R17	10k
R18	220k
R19	5k6
R20	1k
R21	9k53
R22	47k
R23	47k
R24	47R

PART	VALUE
R25	47R
R29	22k
R30	22k
RPD1	2M2
RPD2	2M2

Capacitors

PART	VALUE
C1	22n film
C2	47pF MLCC
C3	47n film
C4	100n film
C5	220n film
C7	220n film
C8	220n film
C10	1uF film

PART	VALUE
C11	100uF electro
C12	10n film
C13	100pF MLCC
C14	10n film
C15	10n film
C16	15n film
C17	4n7 film
C18	100n film

PART	VALUE
C19	4.7uF film
C20	10n film
C21	10n film
C22	1uF film
C23	100uF electro
C24	100uF electro
C25	100uF electro
C26	100n MLCC

ICs

PART	VALUE
IC1	RC4558P
IC2	RC4558P

Diodes

PART	VALUE
D1	1N5817
D2	BAS33
D3	BAS33
D4	BAS33
D5	BAS33
D6	BAS33

PART	VALUE
D7	BAS33
D8	BAS33
D9	BAS33
D10	1N914
D11	1N914

Potentiometers

PART	VALUE
Drive A	1MA
Tone A	20kW
Volume A	100kA
Drive B	150kC
Tone B	25kB
Volume B	100kA

Switches

PART	VALUE
SPDT on-off-on (2)	
3PDT stomp (2)	
2-pos. DIP switch (2)	

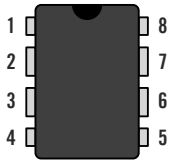
TROUBLESHOOTING INFORMATION

If you finish building the kit and find that it doesn't work right, we've written a separate in-depth Troubleshooting Guide that applies to all of our kits. The main troubleshooting process is covered there. Here you will find information specific to this kit that will help with that process.

Voltages

The following voltages are taken from our prototype unit using a **9.60V** supply. Your measured voltages won't be exactly the same due to variance in power supplies and component tolerances. However, if you see anything more than +/-0.5V from the listed voltages, it's a good indicator of an issue, and the exact voltages can help narrow it down.

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



IC1

PIN	VOLTAGE
1	5.03
2	5.03
3	5.03
4	0
5	4.51
6	5.09
7	5.04
8	9.23

IC2

PIN	VOLTAGE
1	4.73
2	4.78
3	4.35
4	0
5	4.62
6	4.62
7	4.58
8	9.22

SUPPORT

Aion FX does not offer direct support for these projects beyond the provided documentation. Replacements and refunds cannot be offered unless it can be shown that the circuit or documentation are in error or that the included components are non-functional.

Where to get help

The three best places to ask for help are the DIY Stompboxes forum, the DIY Stompboxes Facebook group, and the r/diypedals subreddit. These communities have more than 150,000 members between them and they are very accommodating to new builders.

When posting a troubleshooting request, always include the following:

1. A thorough description of the problem you are experiencing
2. A photo of the inside of the pedal
3. A list of all the measured voltages of each of the pins, described on the previous page

While we cannot offer direct, private support, you may send a link to your public troubleshooting thread to Aion FX using the contact form on the website. There is no guarantee that we will be able to join the discussion and help solve your problem, but this improves the chances.

It benefits the whole community if the troubleshooting process is public because then people who have the same issue in the future may come across it when searching. And if you do get help, remember to pay it forward! The best way to learn new skills is to help others. Even if you've only built one pedal, you have more experience than someone who is brand new, so you have something to offer.

RESALE TERMS

These kits may be used for commercial endeavors in any quantity unless otherwise noted. It's okay to sell individual builds locally or online, or even to offer a service to build pedals based on these kits.

No direct attribution is necessary, though a link back is always greatly appreciated. The only usage restriction is that you cannot "goop" the PCB or otherwise obscure the source. In other words: you don't have to go out of your way to advertise the fact that you use Aion FX kits, but please don't go out of your way to hide it. The guitar effects industry needs more transparency, not less!

LEGAL INFORMATION

All trademarks are property of their respective owners.

Any use of trademarks is for comparative advertising purposes only under fair use. It is not an endorsement of this product by the trademark holders.

These kits are intended to be built by the customer. Aion FX is not responsible for language that may be used by the customer in the marketing or resale of the finished product.

DOCUMENT REVISIONS

1.0.0 (2024-09-06)
Initial release.