PROJECT NAME

THESEUS

aion DIY GUITAR EFFECTS

BASED ON
Analogman King of Tone

BUILD DIFFICULTY
Intermediate

EFFECT TYPE
Dual-channel overdrive

DOCUMENT VERSION
1.0.0 (2023-02-02)

PROJECT SUMMARY
One of the earliest boutique "waitlist" pedals, this dual-channel drive is made up of two modified Marshall Bluesbreaker circuits in series.

IMPORTANT NOTE

This documentation is for the **kit** version of the project. If you purchased the PCB by itself, please use the PCB-only version of the documentation instead. The circuit is the same, but the instructions are completely different due to the specialized parts and assembly methods used in the kit.

TABLE OF CONTENTS

INTRODUCTION

If this is your first pedal, welcome to the hobby and thank you for choosing Aion FX. You've just joined a community of over 40,000 people around the world with a passion for building homemade noise machines using obsolete electronics technologies, and we're glad to have you!

If you've done this before, it's great to see you again and we're confident you'll find this build experience an enjoyable one.

Aion FX kits are designed to empower anyone to build a high-quality pedal, no matter the skill level. The pedalbuilding hobby has traditionally had a steep learning curve, but don't be overwhelmed—we've done all the hard work for you. All you need to do is follow these instructions and you'll be on your way to transforming your tone.

There are a few things to go over before you get started.

- **You're going to have to get your hands dirty**—there's no way around it. Nothing here comes preassembled, and you'll have to learn the skills to put it all together. This document will walk you through everything you need, but be prepared to learn a few things along the way.
- **This will take time**. Plan on about two hours start to finish. It may take even longer if it's your first time building. Don't rush it. If you find yourself getting frustrated or overwhelmed, take a break and come back in a couple of hours or the next day.
- **No direct technical support is offered.** There are several DIY forums and Facebook groups with thousands of members who enjoy troubleshooting and teaching. But please be sensitive to the fact that the staff at Aion FX is minimal, and every minute spent helping individuals in private is time that can't be spent on new project development.
- **There is no implied guarantee of a final product.** Aion FX provides the ingredients and the recipe, but you are responsible for putting everything together to make it work. We've tried to make the process as clear and accessible as possible, but it must be expressly stated that purchasing the kit is not a guarantee that you will end up with a working pedal.

It's recommended to read through all of the instructions before you start, particularly if you've never built a pedal before. If you familiarize yourself with the entire process ahead of time and you know what the goal looks like, each step will make more sense.

Now, on to the fun stuff!

PACKING LIST

This is a list of all the parts that are included with the kit, grouped by value. For a list of all the parts based on their PCB part numbers, please see page 29.

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

Film Capacitors

NAME	QTY
10n	8
22n	2
100n	2
1uF	2

Electrolytic Capacitors

NAME	QTY
1uF	2
100uF	4

MLCC Capacitors

NAME	QTY
100pF (marked “101”)	2
100n (marked “104”)	1

ICs

NAME	QTY
JRC4580D	2
8-pin socket	2

Other

NAME	QTY
Trimmer, 50k	2
DIP switch, 3-pos.	2

Resistors

NAME	QTY
47R	2
1k	4
6k8	4
10k	2
22k	2
27k	2
33k	2
47k	4
100k	2
220k	2
1M	6

Diodes

NAME	QTY
1N5817	1
1N914	4
BAS33	8

PACKING LIST (CONT.)

Potentiometers

NAME	QTY
25kB	2
100kA	2
100kB	2
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

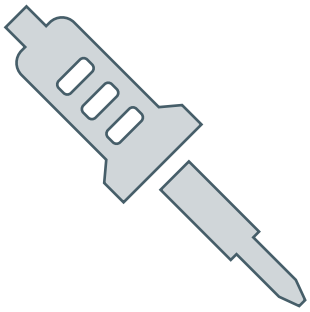
Switches

NAME	QTY
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

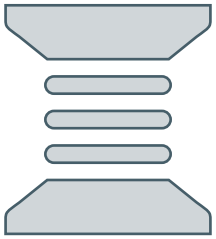
NAME	QTY
4-strand wire assembly, 60mm	2
6-strand wire assembly, 96mm	1
4-pin wire assembly header	2
6-pin wire assembly header	1

TOOLS NEEDED



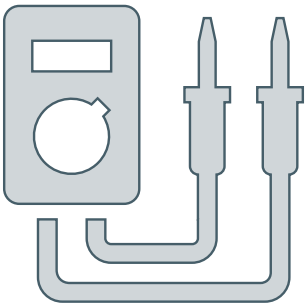
SOLDERING IRON

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.



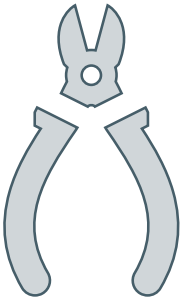
SOLDER

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.



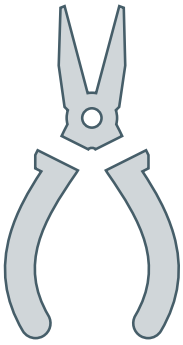
DIGITAL MULTIMETER (DMM)

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.



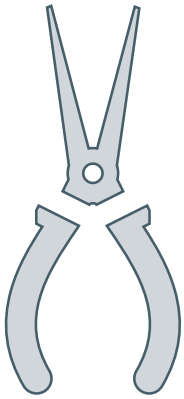
WIRE SNIPPERS

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



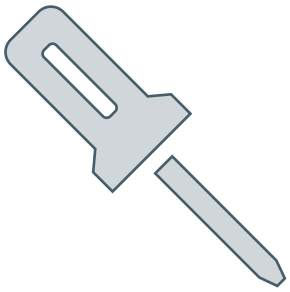
FLAT-NOSE PLIERS

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



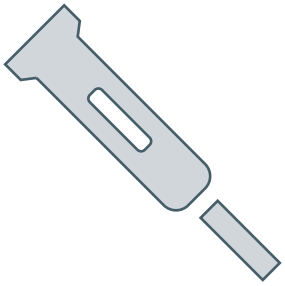
NEEDLE-NOSE PLIERS

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.



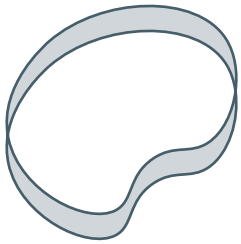
SCREWDRIVER (PHILLIPS)

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.



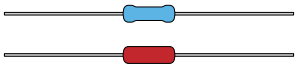
RUBBER BAND

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).

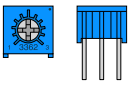
COMPONENT IDENTIFICATION

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.)

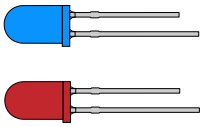
RESISTOR



TRIM POTENTIOMETER

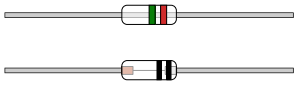


LED



SILICON DIODE

GERMANIUM DIODE



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

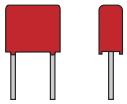
RECTIFIER DIODE

Some Schottky diodes also look like this.

SCHOTTKY DIODE

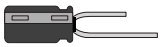
ZENER DIODE

FILM CAPACITOR



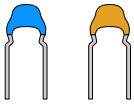
Not polarized. Color may vary by brand and type.

ELECTROLYTIC CAPACITOR



Polarized. The negative side is marked.

MLCC

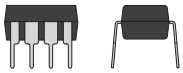


Not polarized. MLCC stands for "multi-layer ceramic capacitor."

TANTALUM CAPACITOR

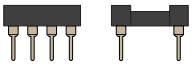
Polarized. The positive side is marked.

IC OR OP-AMP

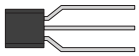


Charge pumps and delay chips also look like this. They may have more than 8 legs.

IC SOCKET



TRANSISTOR OR JFET



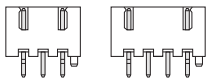
Some voltage regulators also look like this.

DIP SWITCH

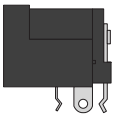
The number of switch positions may vary.

WIRE ASSEMBLY

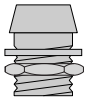
WIRE ASSEMBLY HEADER



DC JACK



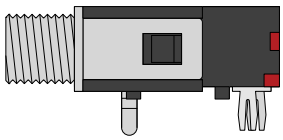
LED BEZEL



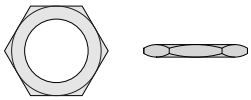
HARDWARE IDENTIFICATION

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

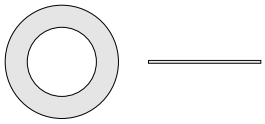


MOUNTING NUT



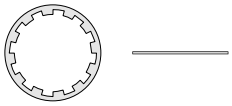
DIAMETER: 0.54" / 13.7mm

OUTER WASHER



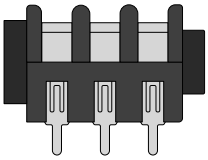
DIAMETER: 0.6" / 15.2mm

LOCK WASHER

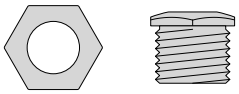


DIAMETER: 0.5" / 12.7mm

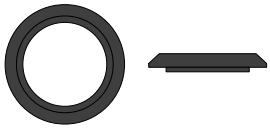
SEND/RETURN JACK



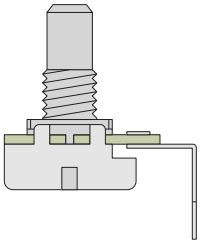
CHROME FERRULE



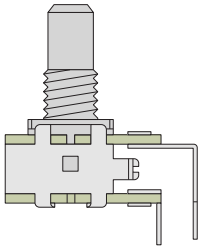
INSULATING WASHER



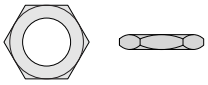
POTENTIOMETER (SINGLE)



POTENTIOMETER (DUAL)

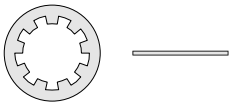


MOUNTING NUT



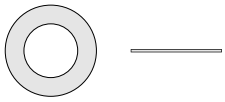
DIAMETER: 0.44" / 11.2mm

LOCK WASHER



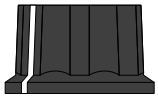
DIAMETER: 0.5" / 12.7mm

OUTER WASHER

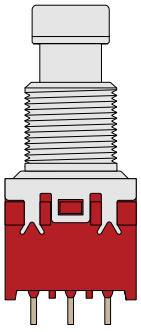


DIAMETER: 0.475" / 12mm

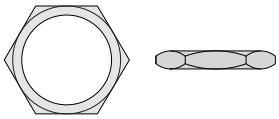
KNOB



FOOTSWITCH

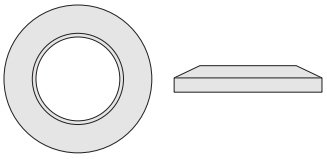


MOUNTING NUT



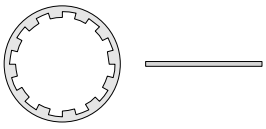
DIAMETER: 0.63" / 16mm

DRESS NUT



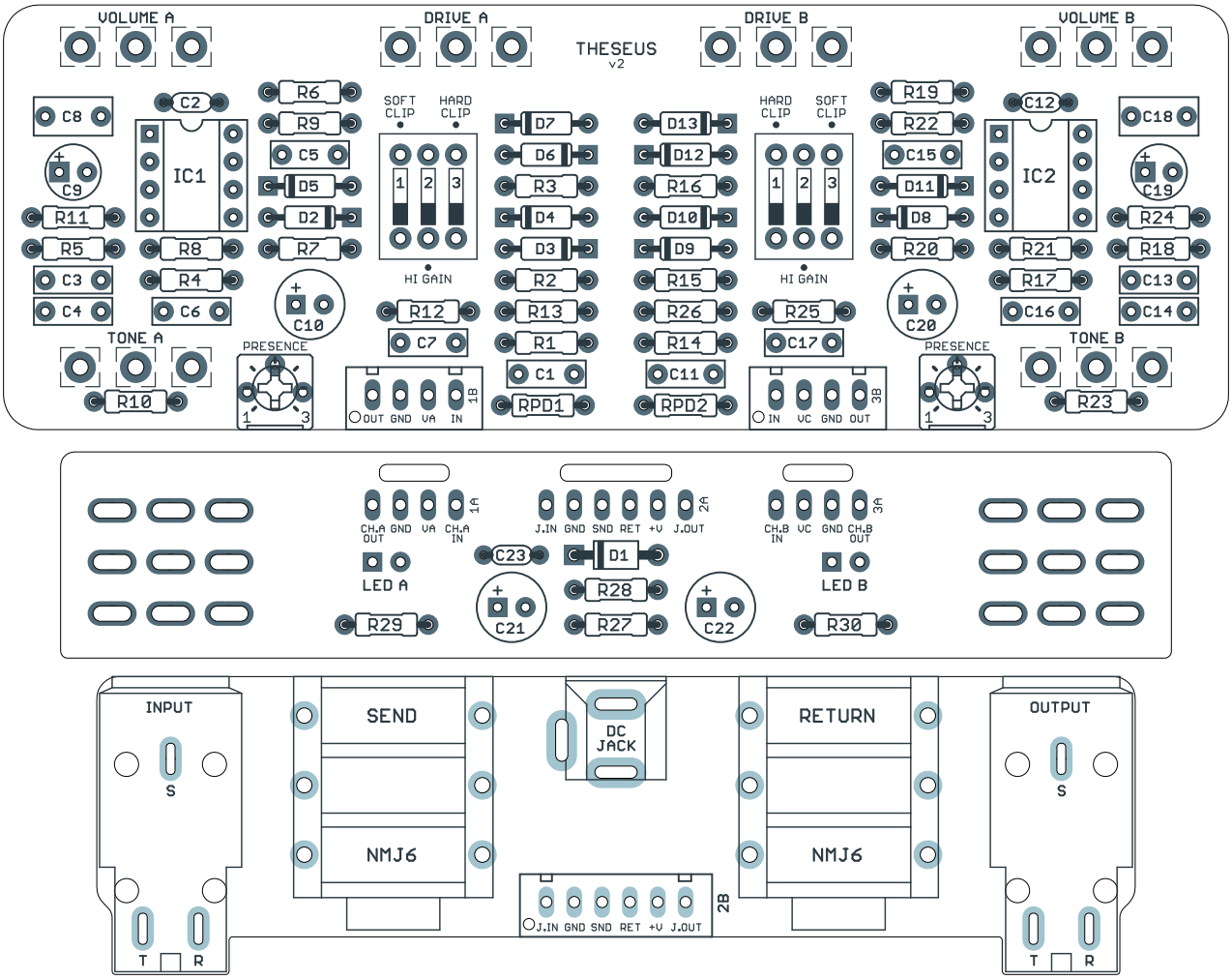
DIAMETER: 0.77" / 19.6mm

LOCK WASHER



DIAMETER: 0.63" / 16mm

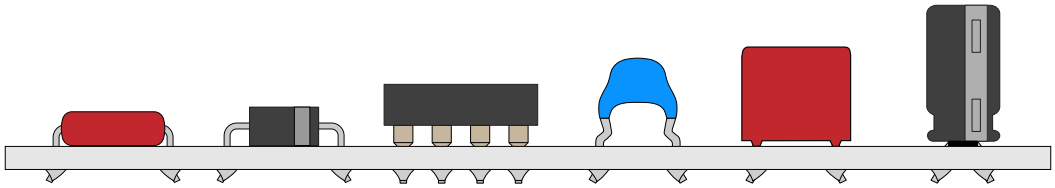
PCB ASSEMBLY OVERVIEW



To 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



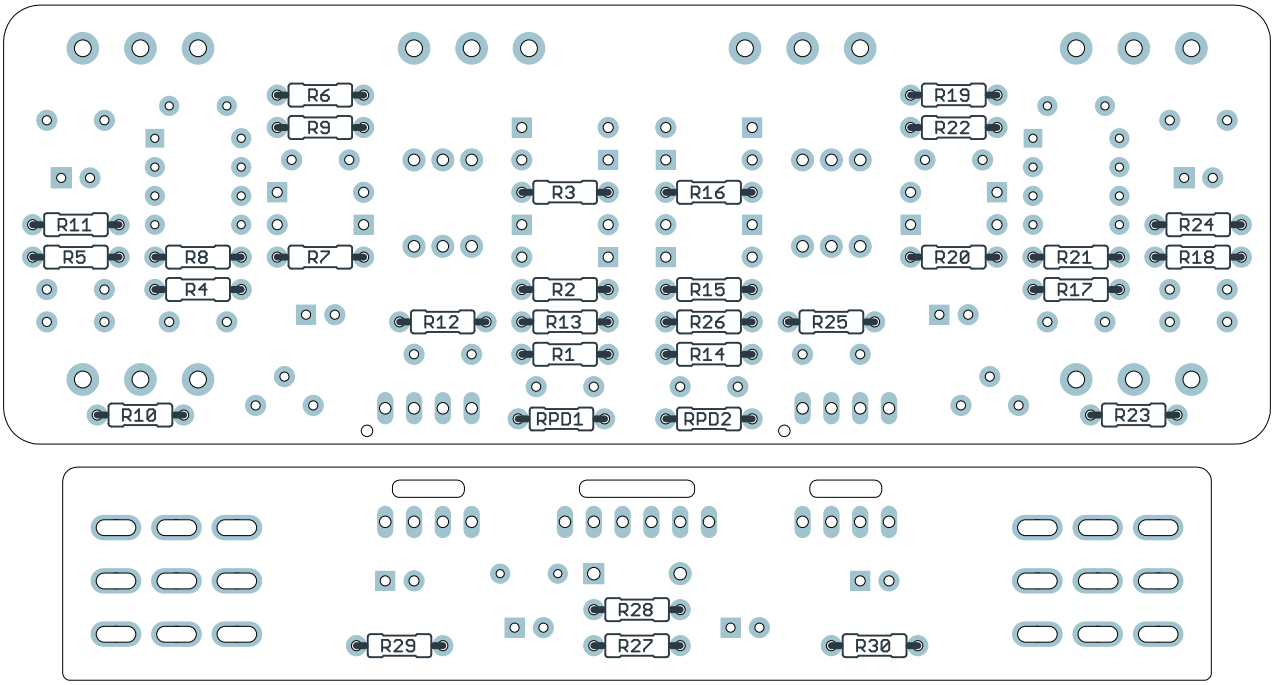
RESISTORS

PART	VALUE
R1	1M
R2	100k
R3	1k
R4	27k
R5	33k
R6	10k
R7	220k
R8	6k8

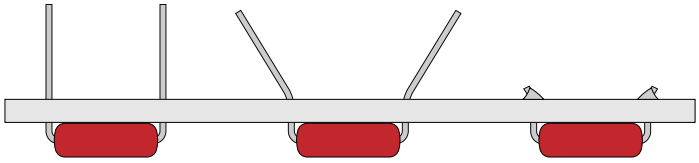
PART	VALUE
R9	1k
R10	6k8
R11	1M
R12	47k
R13	47k
R14	1M
R15	100k
R16	1k

PART	VALUE
R17	27k
R18	33k
R19	10k
R20	220k
R21	6k8
R22	1k
R23	6k8
R24	1M

PART	VALUE
R25	47k
R26	47k
R27	47R
R28	47R
R29	22k
R30	22k
RPD1	1M
RPD2	1M



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.

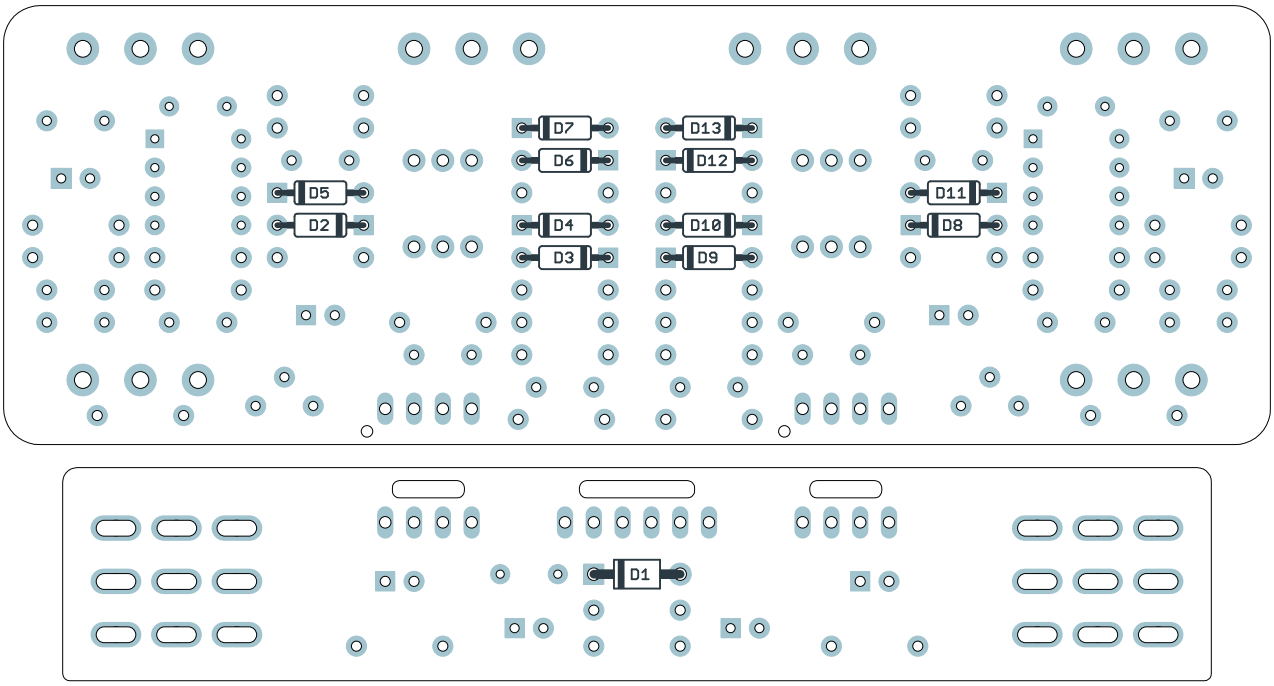
DIODES

PART	VALUE
D1	1N5817
D2	BAS33
D3	BAS33
D4	BAS33

PART	VALUE
D5	BAS33
D6	1N914
D7	1N914

PART	VALUE
D8	BAS33
D9	BAS33
D10	BAS33

PART	VALUE
D11	BAS33
D12	1N914
D13	1N914

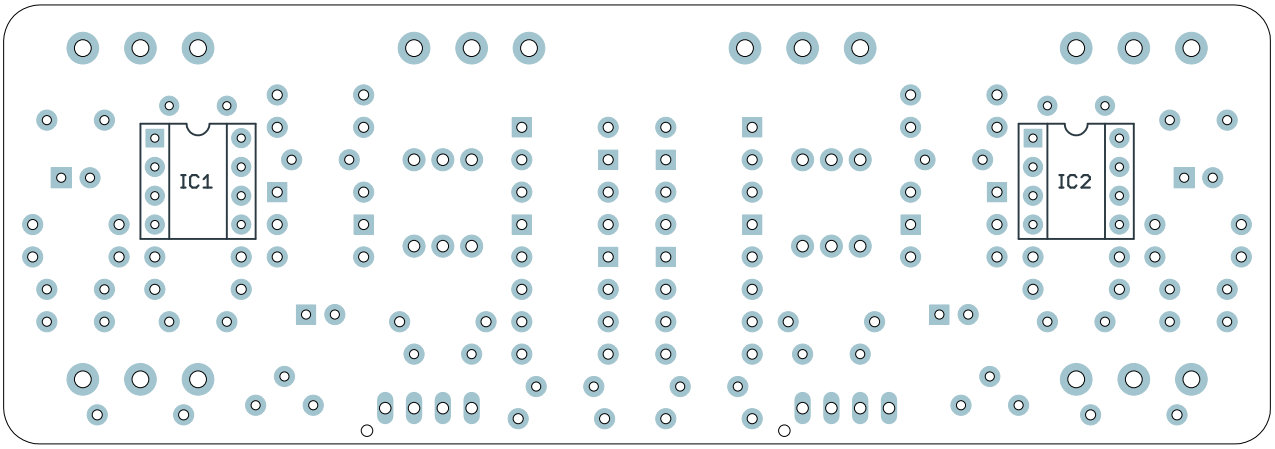


Next, you'll populate the diodes. Diodes are polarized, so make sure to identify the polarity band (which indicates the "cathode", or negative side) and match the band to the footprint on the PCB.

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	JRC4580D
IC2	JRC4580D

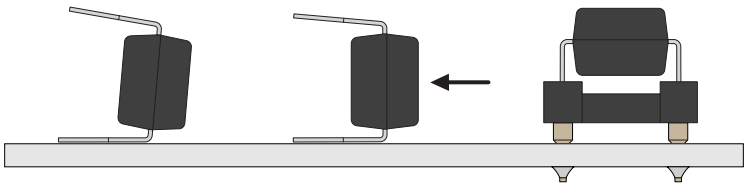


Next up are the sockets. You can't bend the leads of the socket as with the other components, so they won't stay in on their own until they're soldered. Flip the PCB over and use gravity to hold them in place.

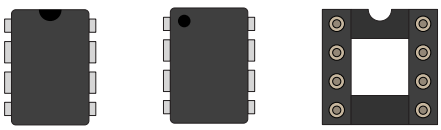
Installing the ICs

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.

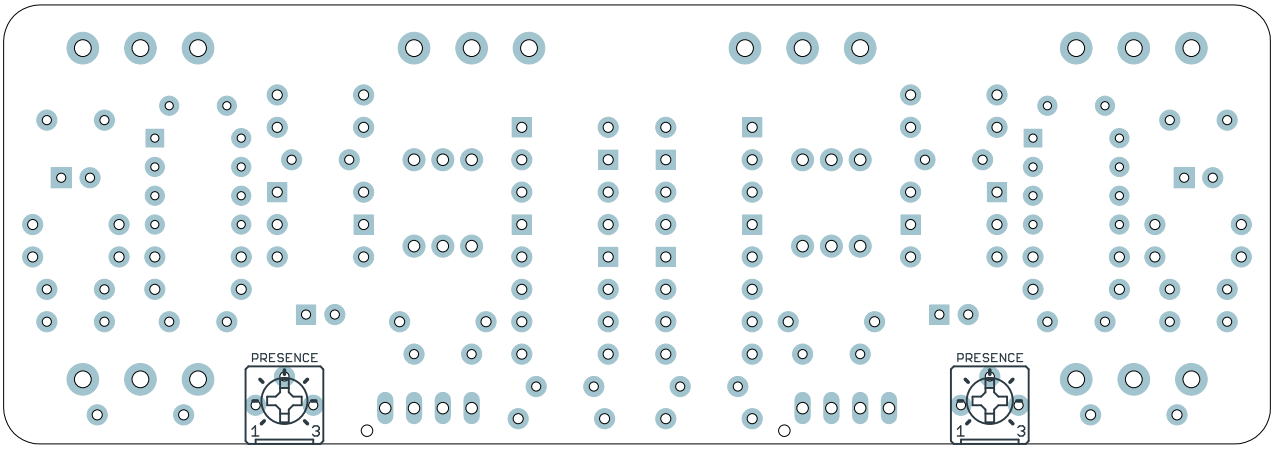


ICs may have two different orientation marks: either a dot in the upper-left or a half-circle notch in the middle of the top side. Some ICs have both marks. This shows which way the IC should be rotated when inserting it into a socket (the socket also has a half-circle notch).



TRIMMERS

PART	VALUE
PRESENCE A	50k trimmer
PRESENCE B	50k trimmer



Next up are the trimmers. The legs on these can be bent like normal components to hold them in place while soldering.

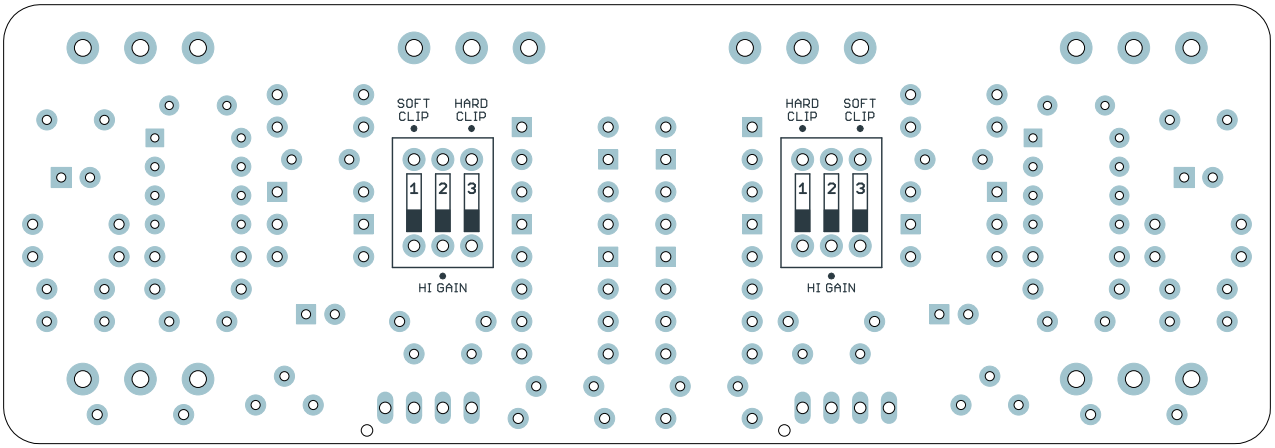
Using the trimmers

The original Bluesbreaker circuit has a fixed 2.3kHz treble cut immediately after the tone control. In the King of Tone, each channel's Presence trimmer essentially fades this hi-cut capacitor out of the circuit, reducing the impact of the treble cut as you turn it up. This can also increase the amount of noise or hiss.

The default position is full counter-clockwise for both, which is equivalent to the stock Bluesbreaker.

DIP SWITCHES

PART	VALUE
SW1	3-position DIP switch
SW2	3-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 up, toward the top of the PCB. If it's installed backwards, it 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 available modes are identical between the two channels. To engage each mode, move the switch toward the dot. If the switch is moved away from the dot, the mode is disengaged.

The description of each switch mode are as follows.

Hi Gain: This shifts the gain range of the drive knob. The default position is OFF. It's equivalent to the Hi Gain mode in the original King of Tone, but instead of being hardwired by customer request, it's selectable per channel.

Soft Clip: When engaged, the soft-clipping diodes are active. The default position is ON.

Hard Clip: When engaged, the hard-clipping diodes are active. The default position is OFF.

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

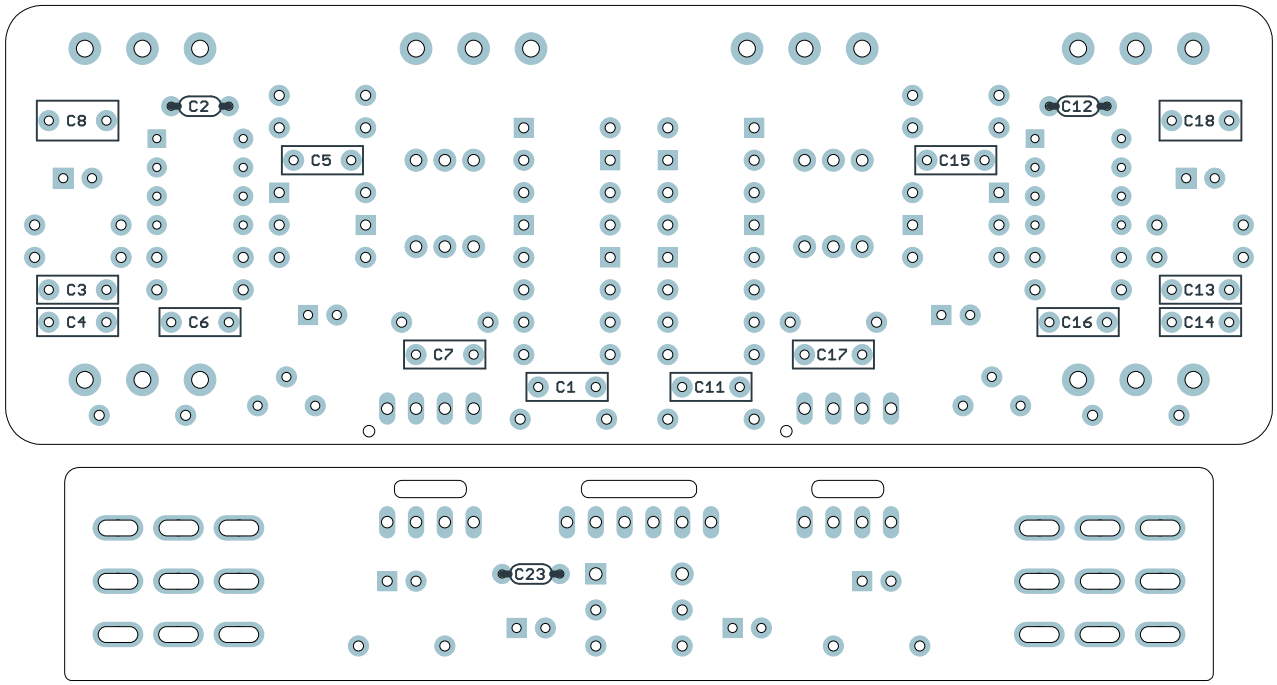
Disengage both "Soft Clip" and "Hard Clip" for boost mode. It's not a perfectly clean boost, and the op-amp itself will eventually clip at higher gain settings, but it's much cleaner than the stock mode.

CAPACITORS (NON-POLARIZED)

PART	VALUE
C1	22n film
C2	100pF MLCC
C3	10n film
C4	10n film
C5	100n film
C6	10n film

PART	VALUE
C7	10n film
C8	1uF film
C11	22n film
C12	100pF MLCC
C13	10n film
C14	10n film

PART	VALUE
C15	100n film
C16	10n film
C17	10n film
C18	1uF film
C23	100n MLCC



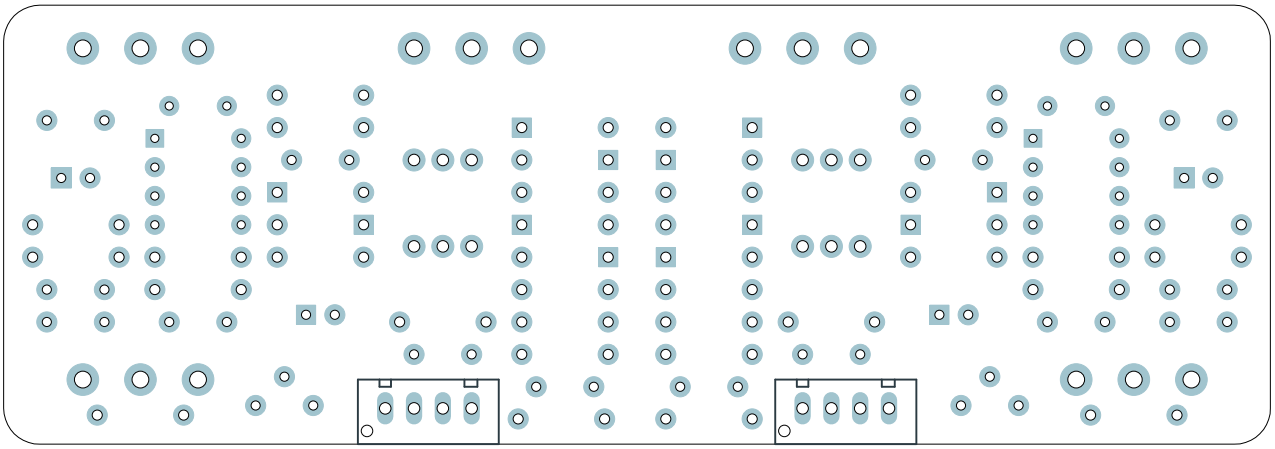
After the sockets come the box film and MLCC capacitors. These are all several different heights, but there aren't as many, so just do them all at once. Bend the leads at an angle to hold them in place.

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

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.

C2 and C12 (100pF MLCCs) are always blue and typically come taped to cardboard. C23 (100n MLCC) is always yellow. They can be identified by color since these are the only two values of MLCC.

WIRE HEADERS



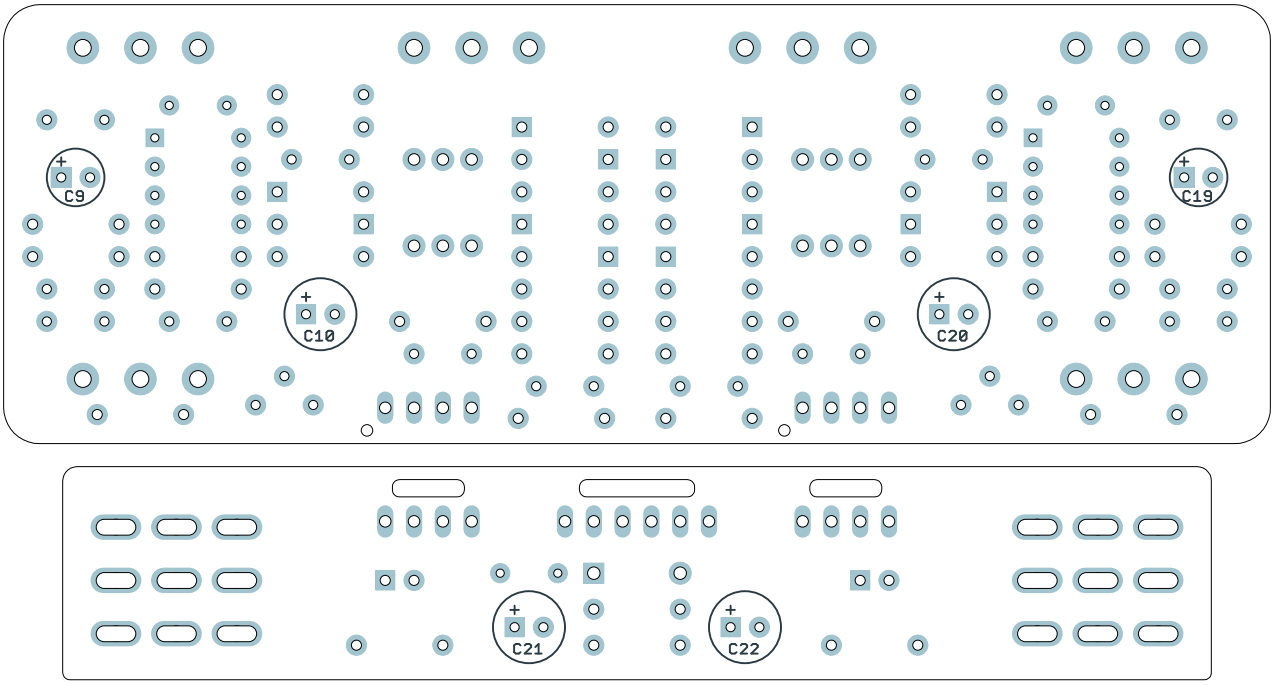
Install the two 4-pin headers (wire connectors) as shown above. These have a polarity pin, so as long as they are pressed all the way down, there's only one possible way to install them. They do fit pretty tightly in the holes, though, so press firmly.

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

CAPACITORS (POLARIZED)

PART	VALUE
C9	1uF electro
C10	100uF electro
C19	1uF electro

PART	VALUE
C20	100uF electro
C21	100uF electro
C22	100uF electro



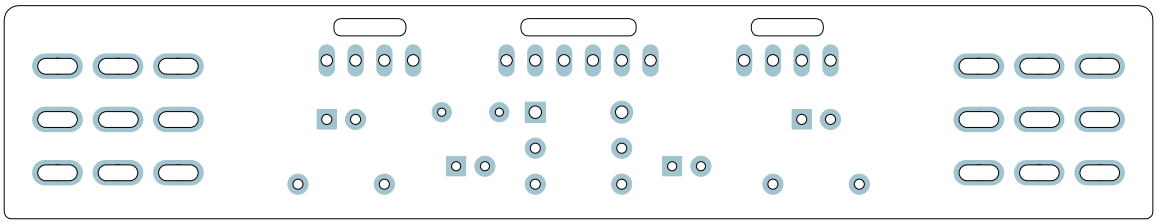
Populate the electrolytic capacitors. These are the tallest components, so we save them for last. They are polarized (i.e. they will only work in one direction), so note the vertical mark that indicates the negative side. The longer leg is positive and fits in the square pad.

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



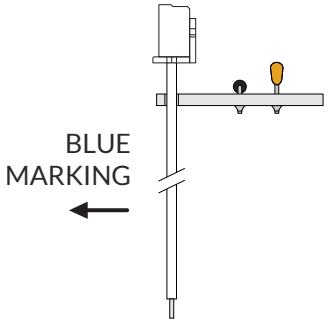
Next, it's time to finish up the footswitch board. You should have done most of the on-board components on this board in a previous step, but if not, go back and do those.

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.

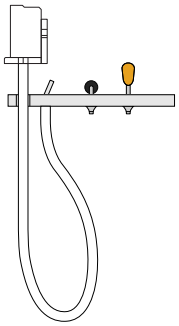
For now, pull it down through the slot as far as it can go.



STEP 2

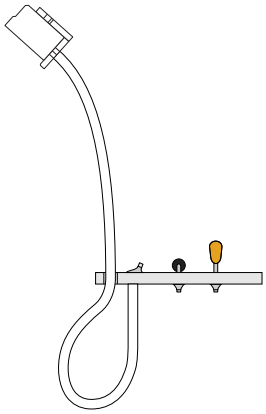
Next, bend the wires back upward and fit the ends of the wires into the solder pads.

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.

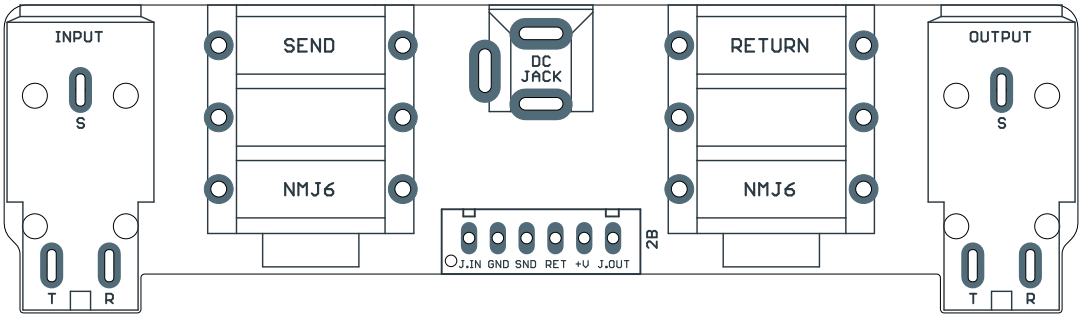


Once all three wire assemblies are soldered, set the footswitch PCB aside. We'll solder the actual footswitches and LEDs in a later step.

INPUT/OUTPUT PCB

PARTS

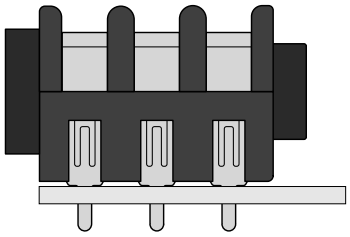
- Input & output jacks
- Send & return jacks
- DC jack
- Wire header



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.

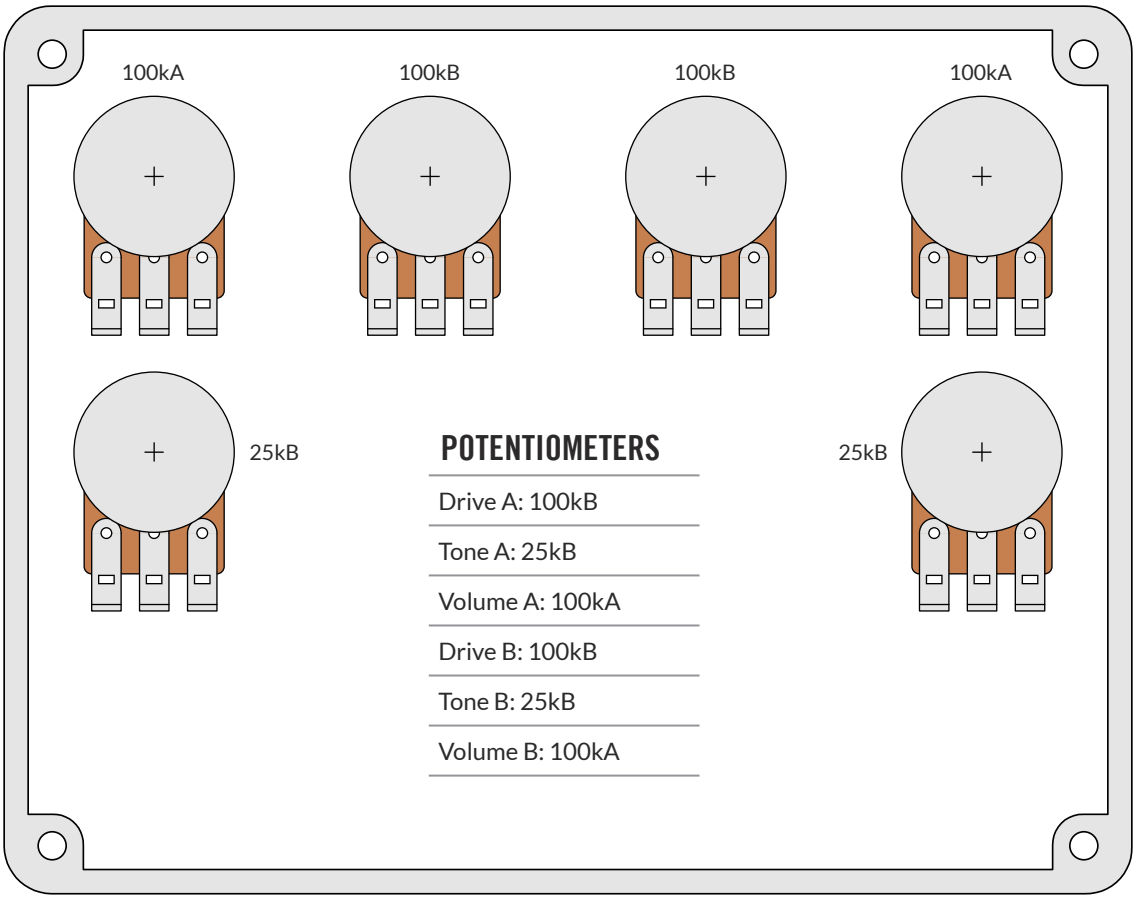
This type of jack does not snap in like the other components, so you'll have to set them in the holes and then flip the board upside down while holding them in place. Ensure the jack is facing forward, the same way as the input & output jacks, because it will fit in the holes either way.



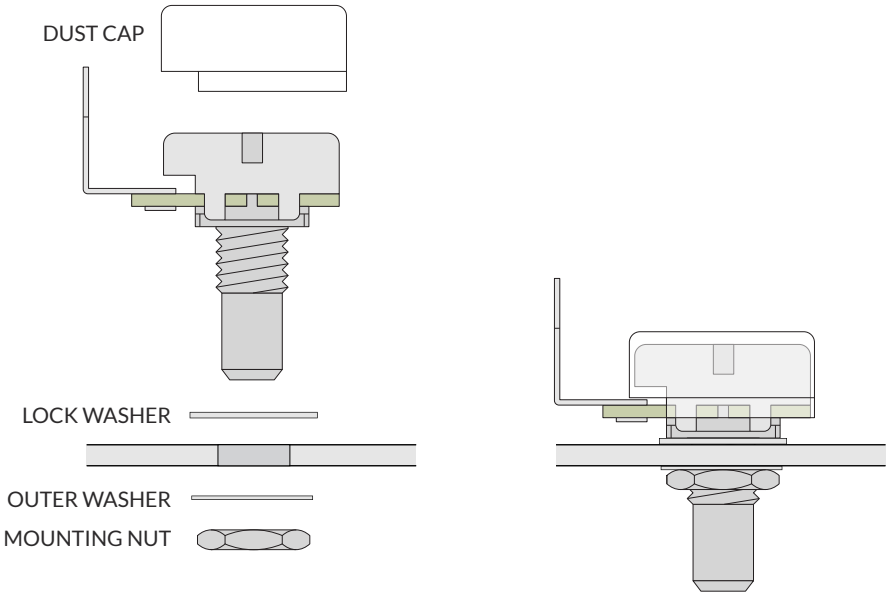
Solder each jack, taking care to make them as straight as possible relative to the white outline on the PCB. It's recommended to just solder one of the six pins at first, then check it. If it's out of alignment, the joint can be reheated to allow the jack to be adjusted. Once the next pin is soldered,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

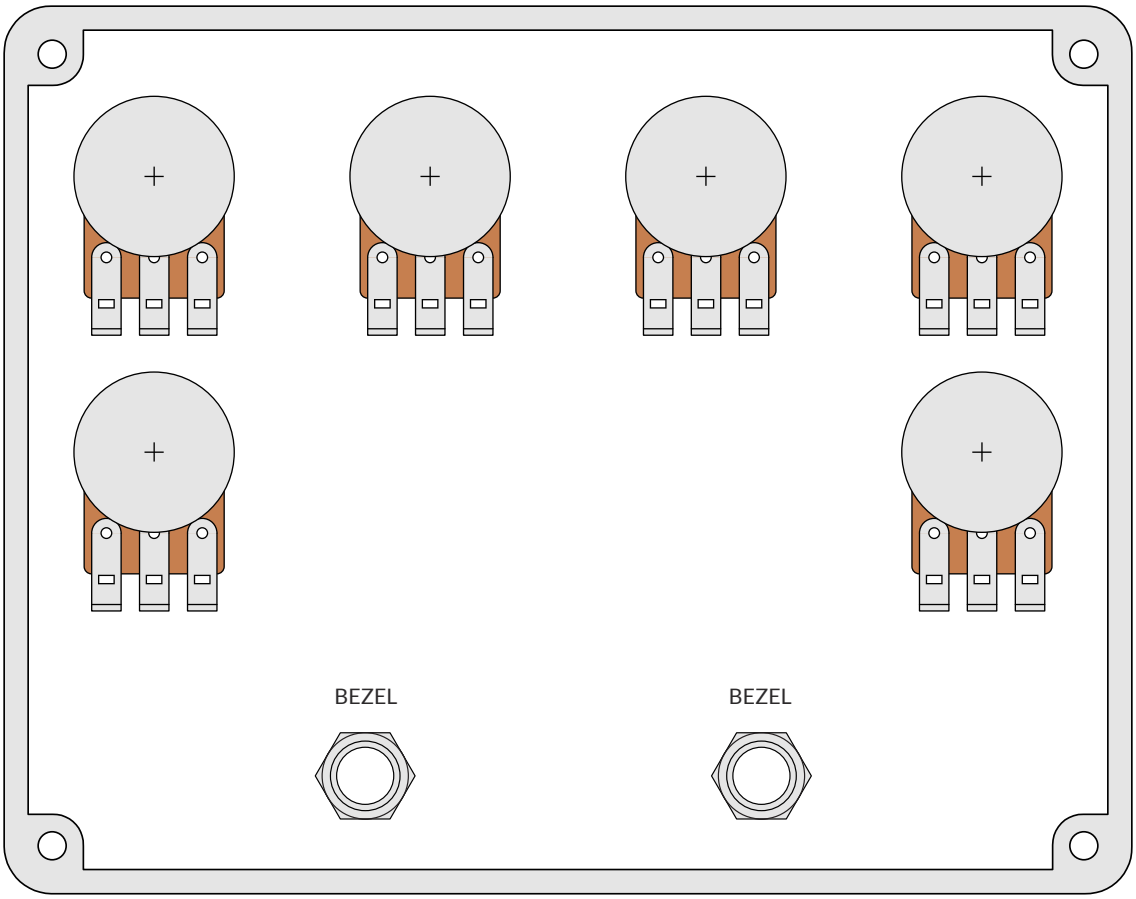
ENCLOSURE LAYOUT: POTENTIOMETERS



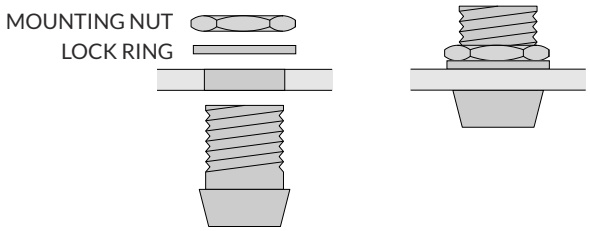
Attach the potentiometers to the enclosure as shown. Make sure they're aligned as straight as possible, then tighten the outer nut firmly. It can be helpful to use a second set of pliers to hold the potentiometer in place from the inside while tightening the nut from the outside.



ENCLOSURE LAYOUT: LED BEZELS

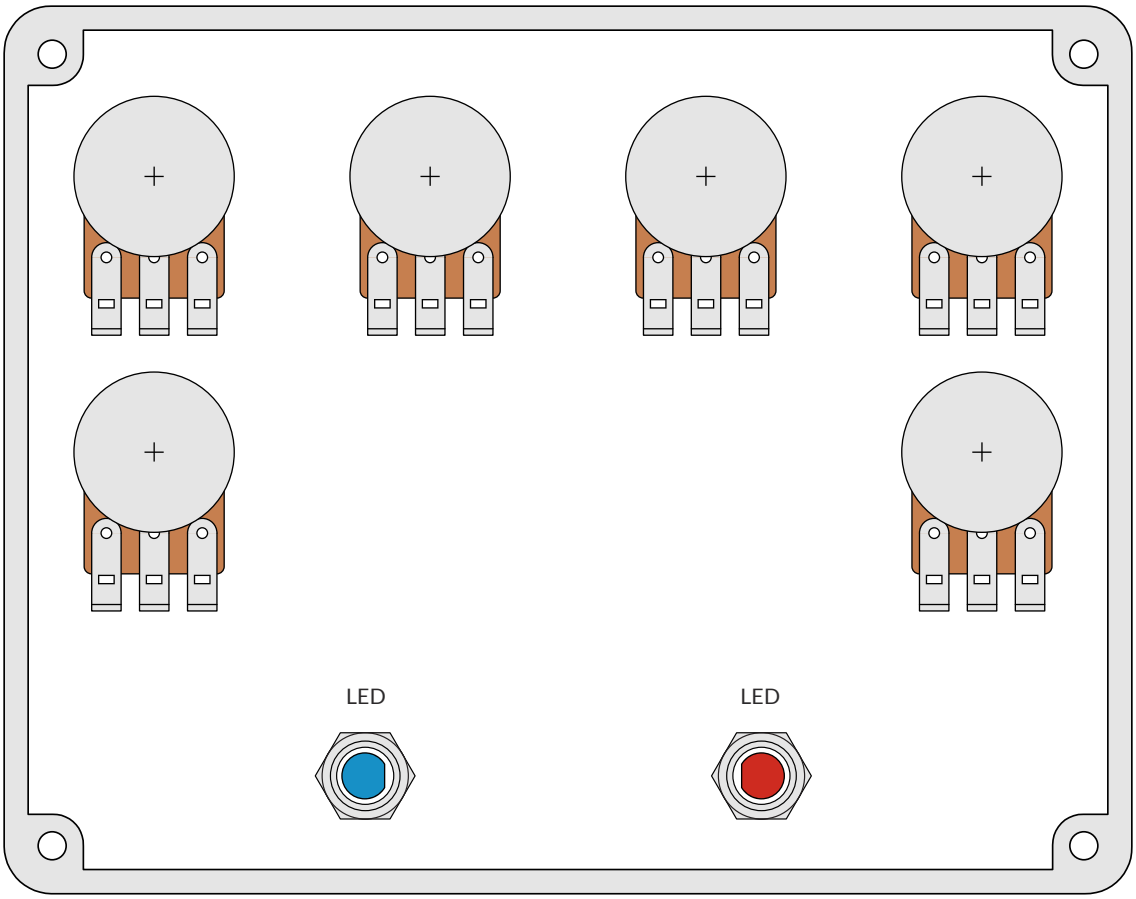


Next, attach the LED bezels to the enclosure. You'll need to hold the bezel in place from the outside when tightening the nut.



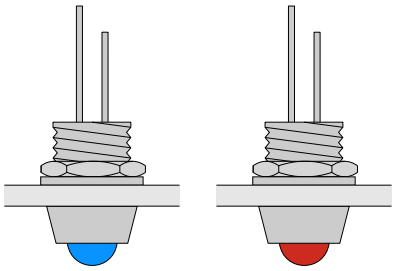
Be aware that the top of the bezel is fairly sharp. Try using a rubber band for grip instead of just pressing your finger against it.

ENCLOSURE LAYOUT: LEDS



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.



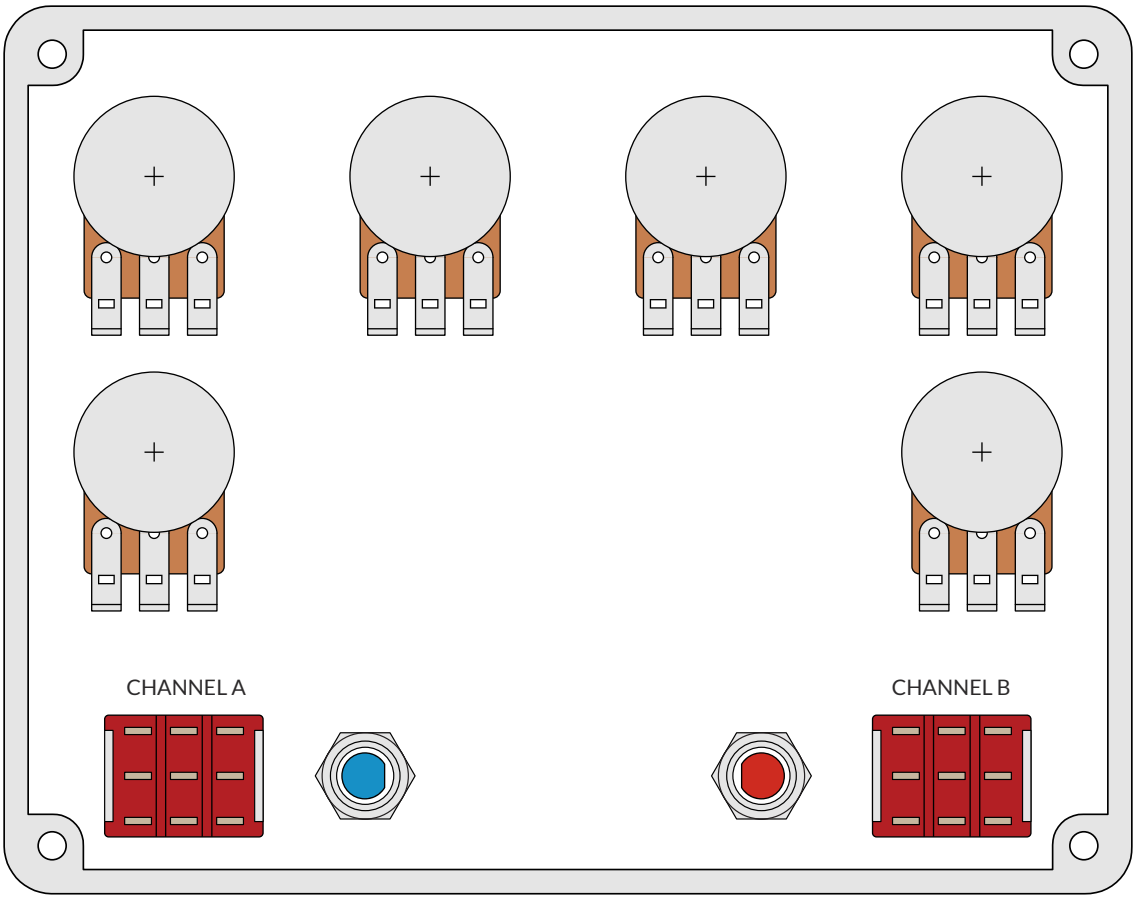
Red LED orientation

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.

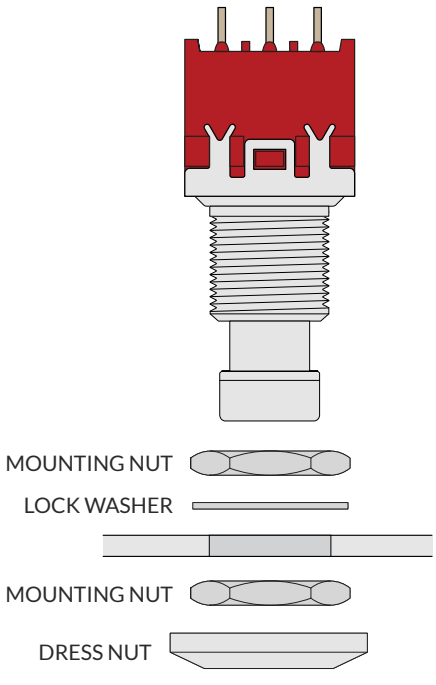
In sourcing LEDs for the Theseus, we spent a long time trying to find a model of red LED that was perfectly balanced in brightness with the blue LED that we use in all our kits. After trying several different types, we finally found an exact match, but it had one strange characteristic: the manufacturer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

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.

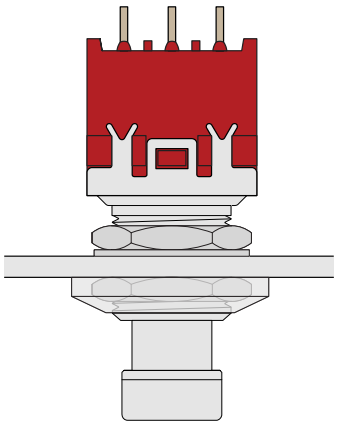
ENCLOSURE LAYOUT: FOOTSWITCHES



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

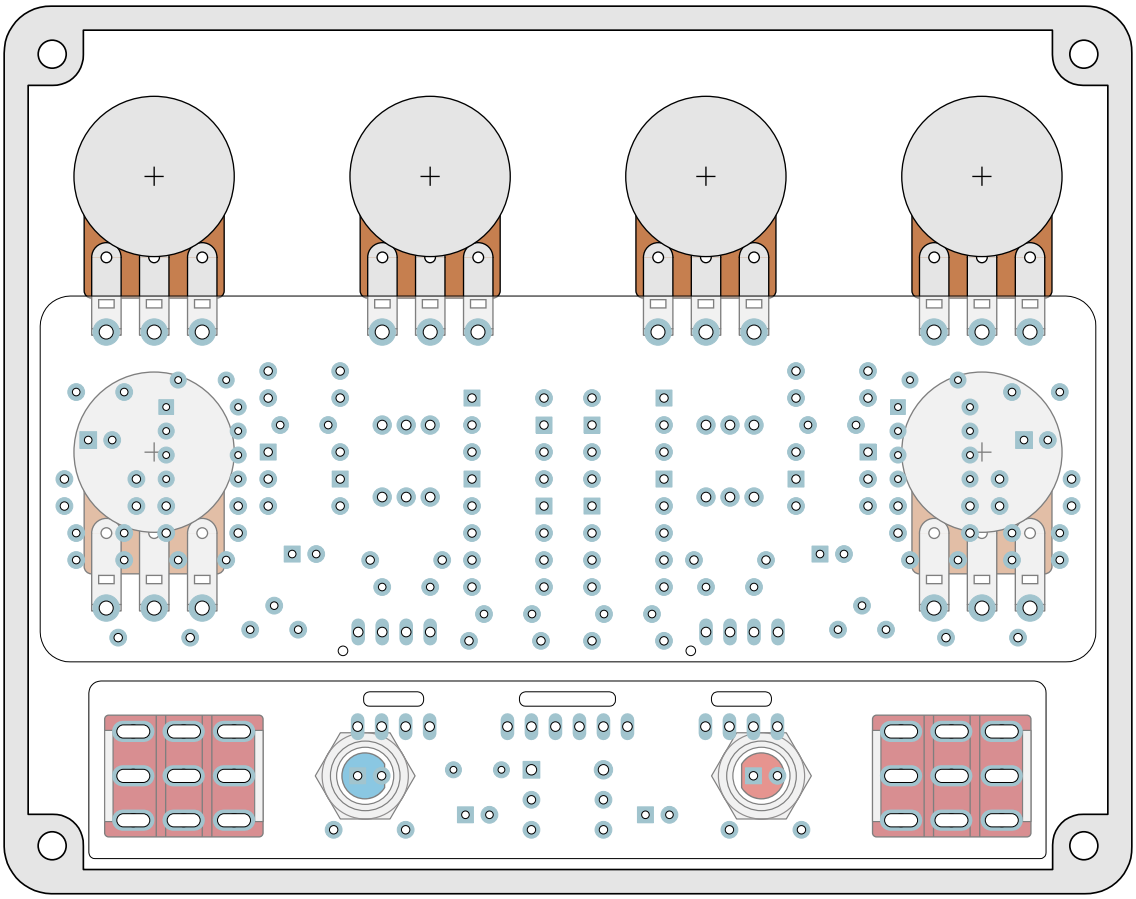


Note that the threading has a notch in one side. For the sake of appearance, it looks best if the notches are oriented the same on both switches, either up or down. However, the switches are functionally symmetrical so this doesn't make a difference to the operation.



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.

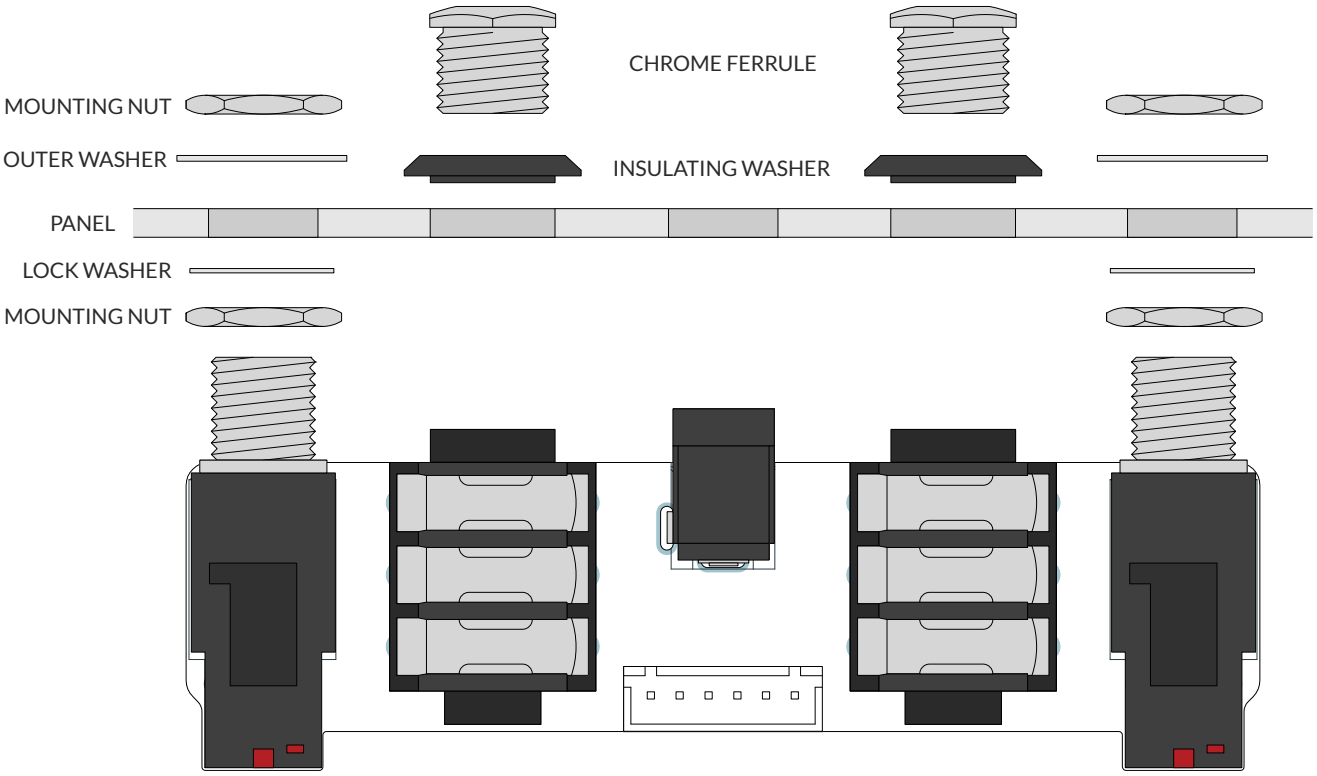
Why solder everything inside the enclosure before testing it?

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

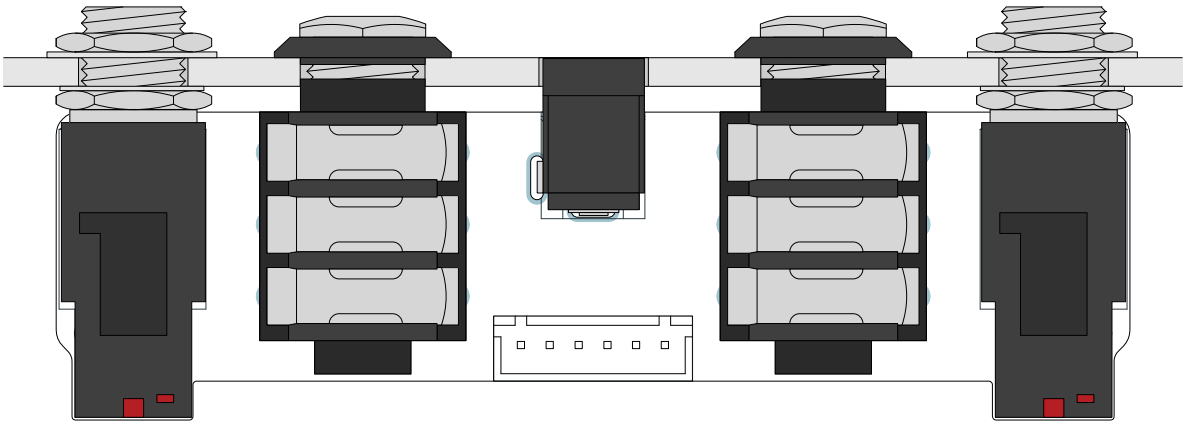


Affix the input/output PCB to the north-facing panel of the enclosure as shown.

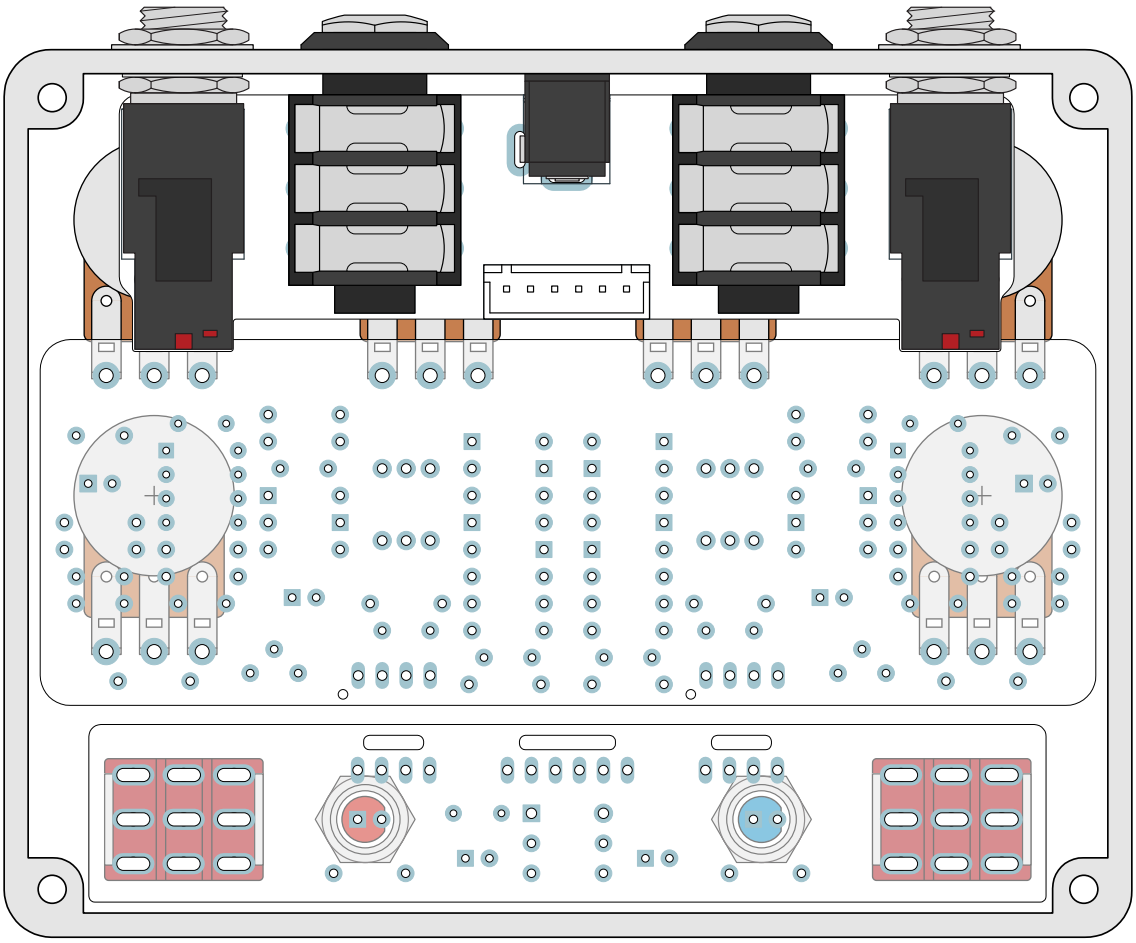
Note the use of two mounting nuts on each of the jacks, one inside and one outside. The inner nut acts as a spacer to set the DC jack flush with the outside of the enclosure. The inner nuts should be threaded as far down as they can go.

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.



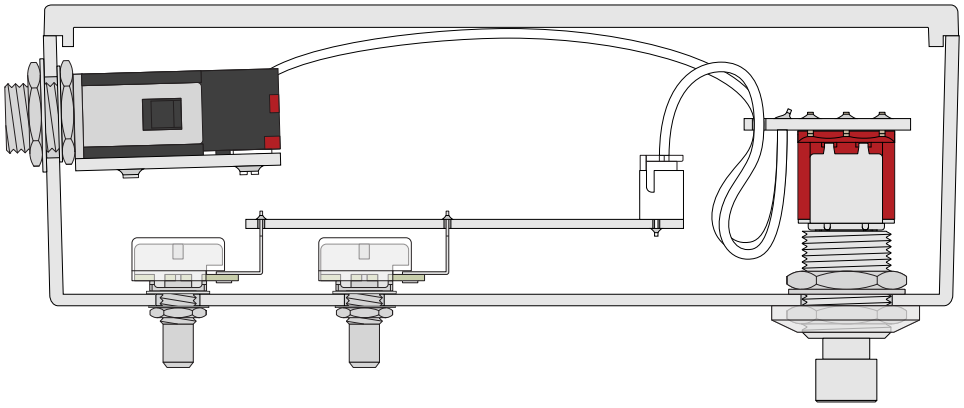
ENCLOSURE LAYOUT: FINISHED DIAGRAM



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. 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.

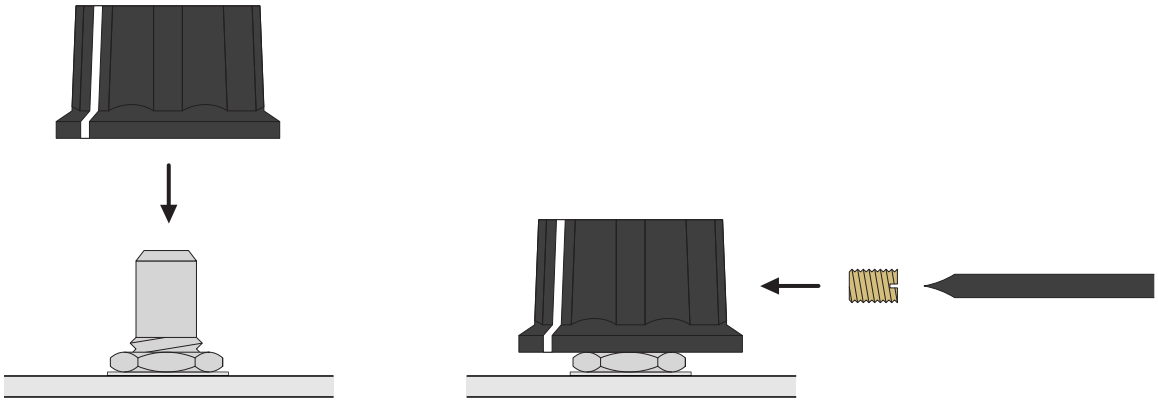
Test the bypass switch a few times, then start turning the knobs and see if everything sounds OK. If it works, great! If not, don't be discouraged. See page 30 for troubleshooting info.

Finishing touches

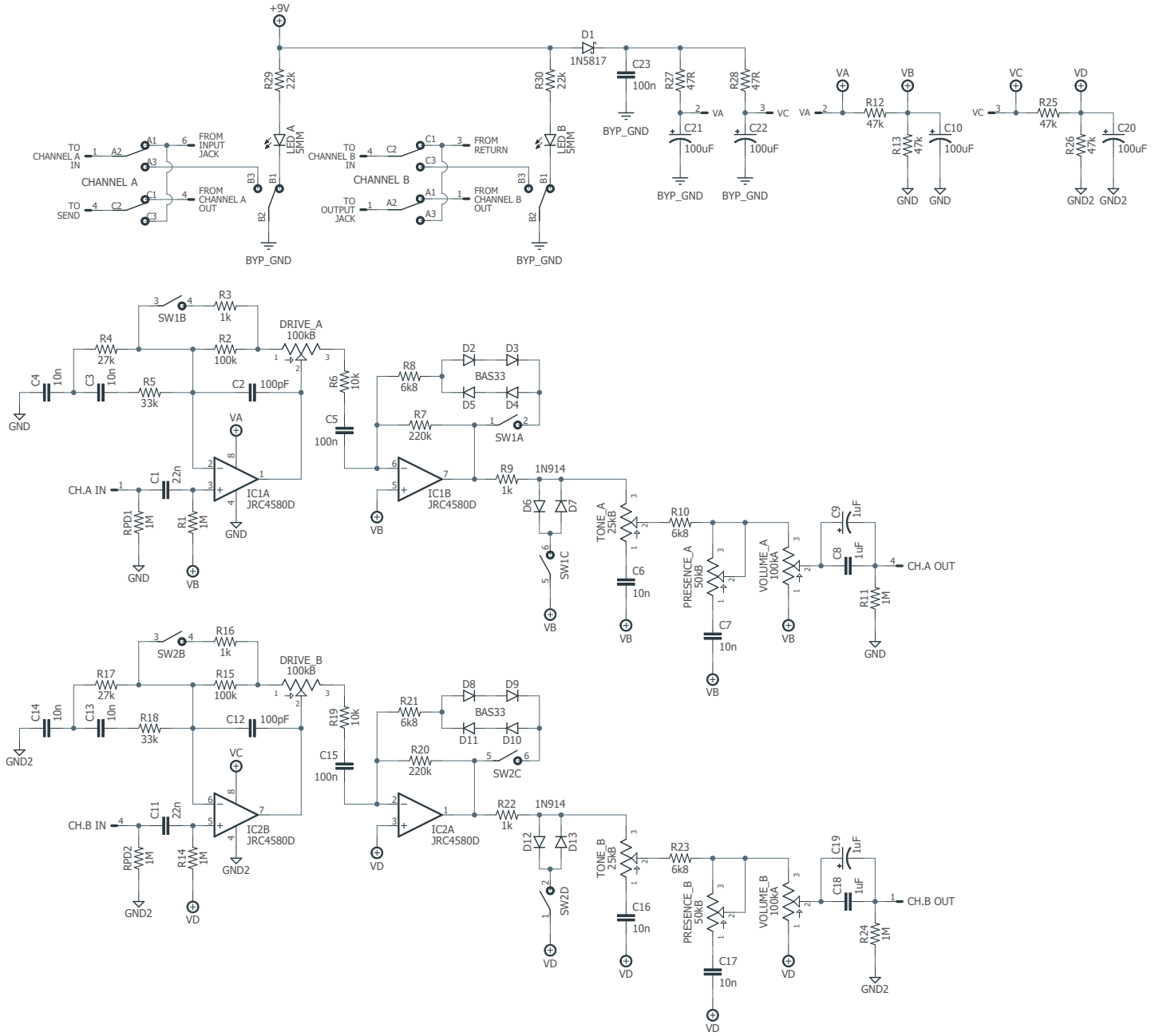
Now, just a couple of things for the final assembly. Turn the shafts all fully counter-clockwise, then put on the knob and rotate until the indicator line is aligned with the dot on the enclosure that shows the zero point. Affix the knobs to each of the potentiometer shafts as shown in the diagram below.

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

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



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


+9V
D1
1N5817
C23
100n
BYP_GND
R29
22k
R30
22k
R27
47R
R28
47R
VA
VB
VC
VD
R12
47k
R13
47k
C10
100uF
R25
47k
R26
47k
C20
100uF
C21
100uF
C22
100uF
GND
GND2
LED A
5MM
LED B
5MM
TO
CHANNEL A
IN
FROM
INPUT
JACK
CHANNEL A
FROM
CHANNEL A
OUT
TO
SEND
TO
CHANNEL B
IN
FROM
RETURN
CHANNEL B
TO
OUTPUT
JACK
FROM
CHANNEL B
OUT
SW1B
SW1A
SW1C
SW2B
SW2C
SW2D
R3
1k
DRIVE_A
100kB
R4
27k
R2
100k
R5
33k
C2
100pF
C3
10n
C4
10n
R6
10k
C5
100n
R8
6k8
R7
220k
D2
D3
D4
D5
BAS33
IC1A
JRC4580D
IC1B
JRC4580D
CH.A IN
C1
22n
RPD1
1M
R1
1M
R9
1k
1N914
D6
D7
TONE_A
25kB
R10
6k8
C6
10n
PRESENCE_A
50kB
C7
10n
VOLUME_A
100kA
C8
1uF
C9
1uF
R11
1M
CH.A OUT
R16
1k
DRIVE_B
100kB
R17
27k
R15
100k
R18
33k
C12
100pF
C13
10n
C14
10n
R19
10k
C15
100n
R21
6k8
R20
220k
D8
D9
D10
D11
IC2A
JRC4580D
IC2B
JRC4580D
CH.B IN
C11
22n
RPD2
1M
R14
1M
R22
1k
1N914
D12
D13
TONE_B
25kB
R23
6k8
C16
10n
PRESENCE_B
50kB
C17
10n
VOLUME_B
100kA
C18
1uF
C19
1uF
R24
1M
CH.B OUT

FULL PARTS LIST

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

Resistors

PART	VALUE
R1	1M
R2	100k
R3	1k
R4	27k
R5	33k
R6	10k
R7	220k
R8	6k8

PART	VALUE
R9	1k
R10	6k8
R11	1M
R12	47k
R13	47k
R14	1M
R15	100k
R16	1k

PART	VALUE
R17	27k
R18	33k
R19	10k
R20	220k
R21	6k8
R22	1k
R23	6k8
R24	1M

PART	VALUE
R25	47k
R26	47k
R27	47R
R28	47R
R29	22k
R30	22k
RPD1	1M
RPD2	1M

Capacitors

PART	VALUE
C1	22n film
C2	100pF MLCC
C3	10n film
C4	10n film
C5	100n film
C6	10n film
C7	10n film
C8	1uF film

PART	VALUE
C9	1uF electro
C10	100uF electro
C11	22n film
C12	100pF MLCC
C13	10n film
C14	10n film
C15	100n film
C16	10n film

PART	VALUE
C17	10n film
C18	1uF film
C19	1uF electro
C20	100uF electro
C21	100uF electro
C22	100uF electro
C23	100n MLCC

ICs

PART	VALUE
IC1	JRC4580D
IC2	JRC4580D

Diodes

PART	VALUE
D1	1N5817
D2	BAS33
D3	BAS33
D4	BAS33
D5	BAS33
D6	1N914
D7	1N914

PART	VALUE
D8	BAS33
D9	BAS33
D10	BAS33
D11	BAS33
D12	1N914
D13	1N914

Potentiometers

PART	VALUE
Drive A	100kB
Tone A	25kB
Volume A	100kA
Drive B	100kB
Tone B	25kB
Volume B	100kA

Switches

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

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.

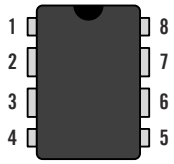
First, does the pedal sound right? If you aren't getting any sound, you probably have a power issue with the whole circuit that is not specific to the LED, so you'll want to look elsewhere for the problem.

If it does pass a signal, and especially if one LED works and the other doesn't, it's likely just the LED itself. Go back to page 22 and make sure to read the precaution about the red LED. If either LED is installed backwards, it won't illuminate and you'll have to re-solder it the right way. (Reversing it will not typically cause permanent damage.)

All other issues

For any other problems, the next step is to measure the voltages on each pin of the ICs using a digital multimeter. 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 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 **9.6V** 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 voltages can help narrow it down.

IC1

PIN	VOLTAGE
1	4.66
2	4.67
3	4.25
4	0
5	4.57
6	4.57
7	4.56
8	9.15

IC2

PIN	VOLTAGE
1	4.53
2	4.54
3	4.54
4	0
5	4.22
6	4.64
7	4.64
8	9.15

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

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 (2023-02-02)
Initial release.