PROJECT NAME

SOLARIS

aion
DIY GUITAR EFFECTS

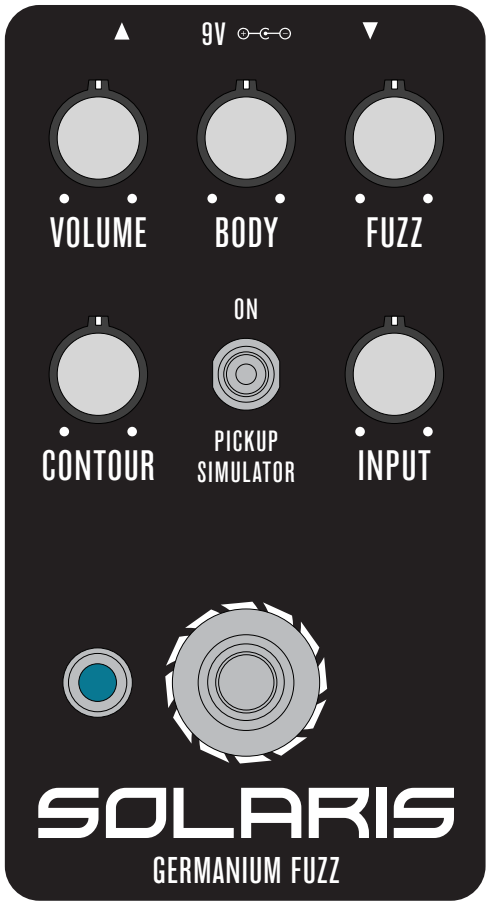
BASED ON
Dallas-Arbiter Fuzz Face

BUILD DIFFICULTY
Intermediate

EFFECT TYPE
Germanium fuzz

DOCUMENT VERSION
1.0.4 (2025-03-24)

PROJECT SUMMARY
A hot-rodded adaptation of the classic fuzz pedal made famous by Jimi Hendrix.



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 30.

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

Film Capacitors

NAME	QTY
1n	1
10n (0.01)	2
100n (0.1 or "µ1J100")	1
1uF	1

Electrolytic Capacitors

NAME	QTY
10uF	2
22uF	1
47uF	1
100uF	1

MLCC Capacitors

NAME	QTY
100n (marked "104")	2

Diodes

NAME	QTY
1N5817	1
1N4742A	1

Resistors

NAME	QTY
220R	1
1k	2
10k (see note)	3
100k	1
1M	2
2M2	1

Note: LEDR was changed from 4k7 to 10k in February 2025. Kits purchased before this time will use 4k7 for LEDR.

ICs

NAME	QTY
LT1054CP	1
8-pin socket	1

Transistors

NAME	QTY
J112 or PN4303	1
Transistors, germanium, matched for Fuzz Face	2

Trimmers

NAME	QTY
10k trimmer	1
100k trimmer	1

PACKING LIST (CONT.)

Potentiometers

NAME	QTY
1kB	1
1kC	1
100kB	1
250kB	1
500kA	1
Dust cover	5
Knob, 1/2”	5
Mounting nut, potentiometer, 0.44"	5
Lock washer, potentiometer, 0.5"	5
Outer washer, potentiometer, 0.475"	5

Other

NAME	QTY
LED bezel	1
LED, blue	1
9V battery snap	1
DC jack	1
Input/output jack	2
Mounting nut, jack, 0.54"	4
Outer washer, jack, 0.6"	2
Lock washer, jack, 0.5" (thin)	2
Enclosure	1
Enclosure screws	4
PCB, main circuit	1
PCB, footswitch	1
PCB, input/output/DC	1

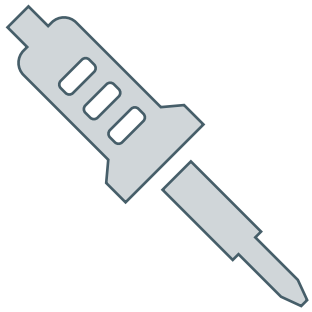
Switches

NAME	QTY
Toggle switch, DPDT on-on	1
Mounting nut, toggle switch, 0.36"	1
Lock washer, toggle switch, 0.4"	1
Dress nut, toggle switch, 0.375"	1
Stomp switch, 3PDT	1
Mounting nut, stomp switch, 0.6"	2
Lock washer, stomp switch, 0.6"	1
Dress nut, stomp switch, 0.77"	1

Wiring

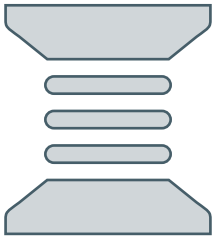
NAME	QTY
3-strand wire assembly, 70mm	2
4-strand wire assembly, 108mm	1
3-pin wire assembly header	2
4-pin wire assembly header	1

TOOLS NEEDED



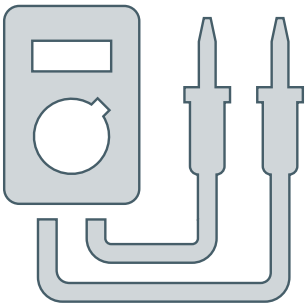
SOLDERING IRON

Temperature-adjustable is recommended. The optimum soldering temperature is 700-725° F (371-385° C) for leaded solder, or 750° F (400° C) for lead-free.



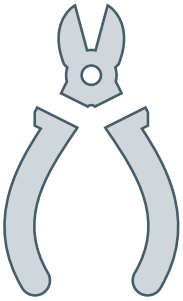
SOLDER

Preferably 63/37 or 60/40 leaded solder. Lead-free is more difficult to use, so if that's the only type you can get, it's best to watch tutorials that are specific to lead-free solder.



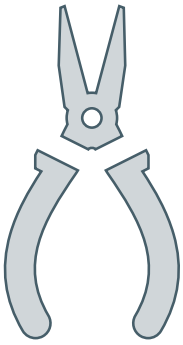
DIGITAL MULTIMETER (DMM)

Most cheap ones in the $10-30 range are fine for what we're doing. Make sure it has audible continuity testing (i.e. it beeps at the lowest resistance) and transistor hFE measurement.



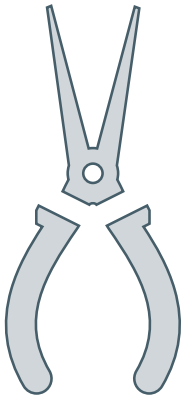
WIRE SNIPPERS

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



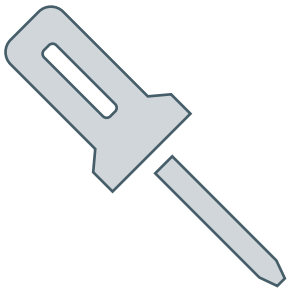
FLAT-NOSE PLIERS

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



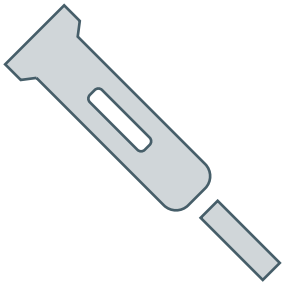
NEEDLE-NOSE PLIERS

These are used for bending leads on components and other general uses. Use the smaller type with a tip that's approximately 0.05" (1.25mm) wide.



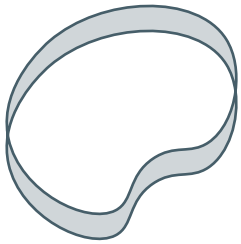
SCREWDRIVER (PHILLIPS)

Used for the enclosure screws. Get a powered driver if you'll be building a lot of pedals!



FLAT SCREWDRIVER (SMALL)

This is used for tightening the set screws on the knobs. The tip should be no more than 0.1" (2.5mm) wide.



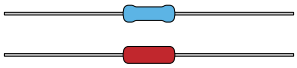
RUBBER BAND

Yes, a plain old rubber band. This is used to tighten the dress nut to avoid scratching or denting it (which can happen with metal tools).

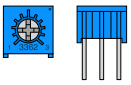
COMPONENT IDENTIFICATION

If you've never built a pedal before, you'll need to know what all the components are. These are shown actual size. (Not all of these types of components may be part of this kit.)

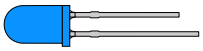
RESISTOR



TRIMMER POTENTIOMETER

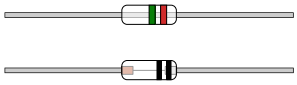


LED



SILICON DIODE

GERMANIUM DIODE



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

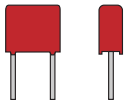
RECTIFIER DIODE

Some Schottky diodes also look like this.

SCHOTTKY DIODE

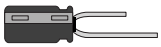
ZENER DIODE

FILM CAPACITOR



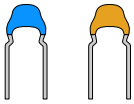
Not polarized. Color may vary by brand and type.

ELECTROLYTIC CAPACITOR



Polarized. The negative side is marked.

MLCC

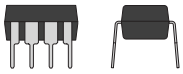


Not polarized. MLCC stands for "multi-layer ceramic capacitor."

TANTALUM CAPACITOR

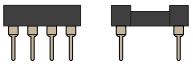
Polarized. The positive side is marked.

OP-AMP / IC

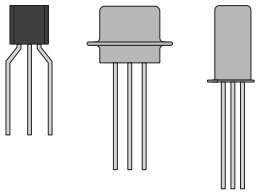


Charge pumps and delay chips also look like this. They may have more than 8 legs.

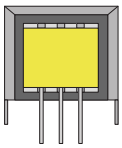
IC SOCKET



TRANSISTOR OR JFET

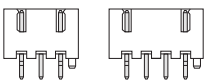


TRANSFORMER

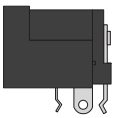


WIRE ASSEMBLY

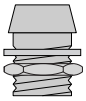
WIRE ASSEMBLY HEADER



DC JACK



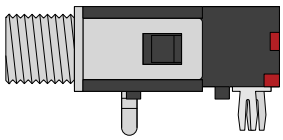
LED BEZEL



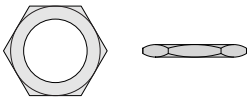
HARDWARE IDENTIFICATION

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

I/O JACK

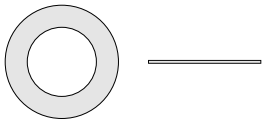


MOUNTING NUT



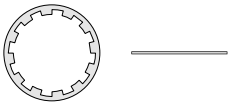
DIAMETER: 0.54" / 13.7mm

OUTER WASHER



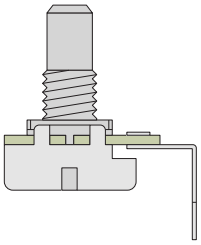
DIAMETER: 0.6" / 15.2mm

LOCK WASHER

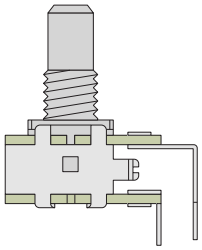


DIAMETER: 0.5" / 12.7mm

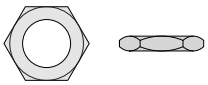
POTENTIOMETER (SINGLE)



POTENTIOMETER (DUAL)

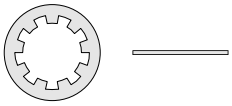


MOUNTING NUT



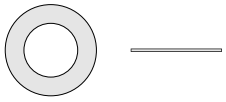
DIAMETER: 0.44" / 11.2mm

LOCK WASHER



DIAMETER: 0.5" / 12.7mm

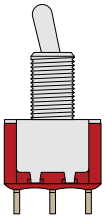
OUTER WASHER



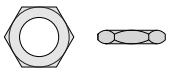
DIAMETER: 0.475" / 12mm

KNOB

TOGGLE SWITCH

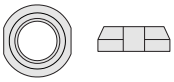


MOUNTING NUT



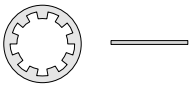
DIAMETER: 0.36" / 9.1mm

DRESS NUT



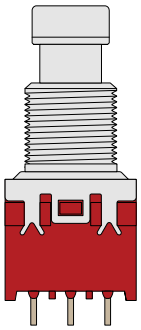
DIAMETER: 0.375" / 9.5mm

LOCK WASHER

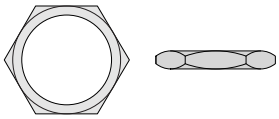


DIAMETER: 0.4" / 10.1mm

FOOTSWITCH

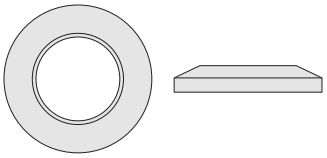


MOUNTING NUT



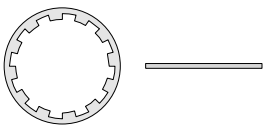
DIAMETER: 0.6" / 15.2mm

DRESS NUT



DIAMETER: 0.77" / 19.6mm

LOCK WASHER



DIAMETER: 0.6" / 15.2mm

PCB ASSEMBLY OVERVIEW

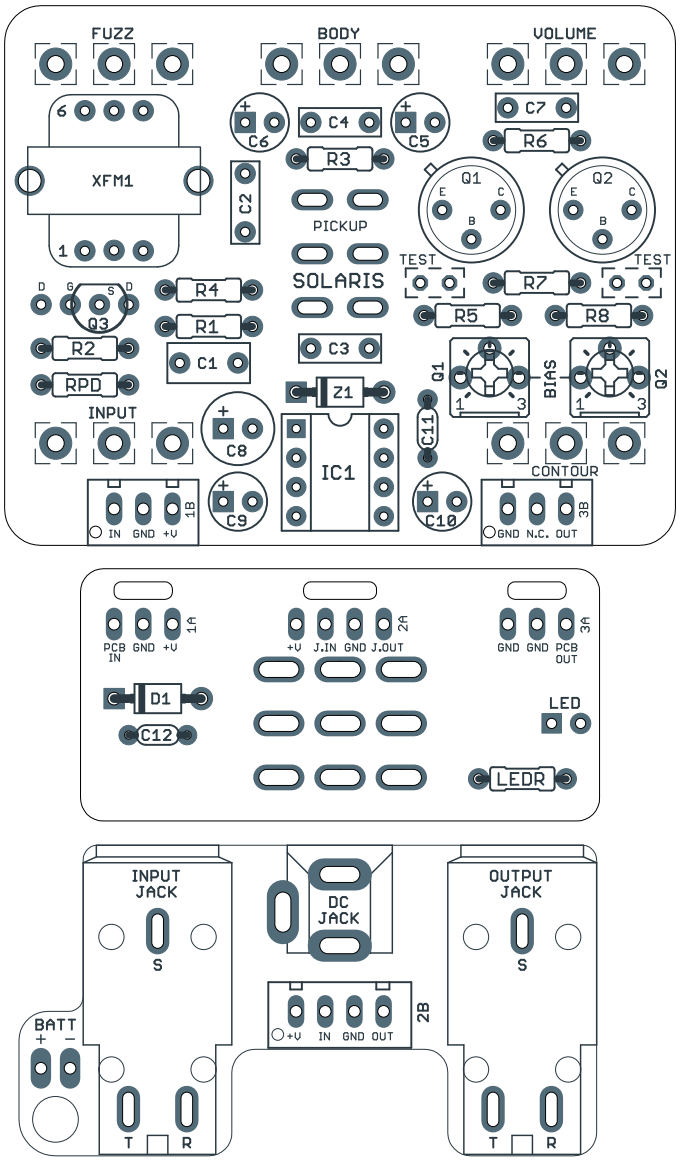
Now it's time to start building!

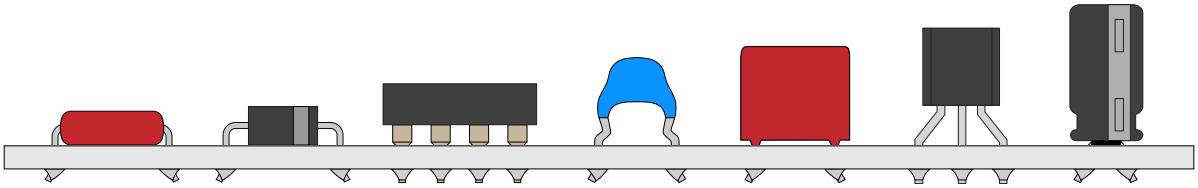
Before you begin, separate the PCBs into individual boards and break off the tabs from each using needle-nose or flat-head pliers. You should be left with the three PCBs shown to the right.

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, you should populate the components in this order:

1. Resistors
2. Diodes
3. IC sockets
4. Trimmers
5. MLCC capacitors
6. Film capacitors
7. Transistors (silicon)
8. Electrolytic capacitors
9. Transistors (germanium)
10. Transformer



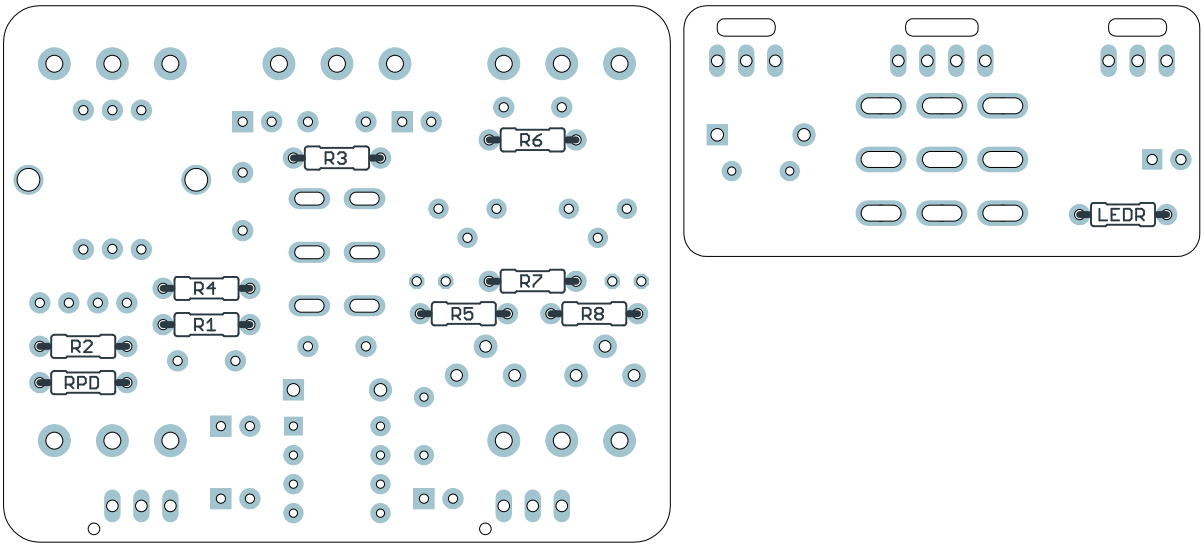


RESISTORS

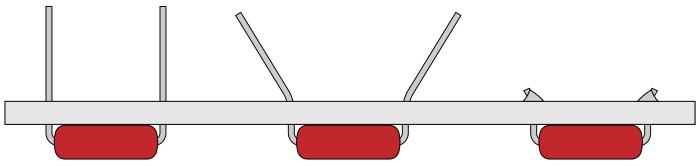
PART	VALUE
R1	1M
R2	1M
R3	10k
R4	1k
R5	10k

PART	VALUE
R6	100k
R7	1k
R8	220R
RPD	2M2
LEDR	10k

Note: LEDR was changed from 4k7 to 10k in February 2025. Kits purchased before this time will use 4k7 for LEDR.



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



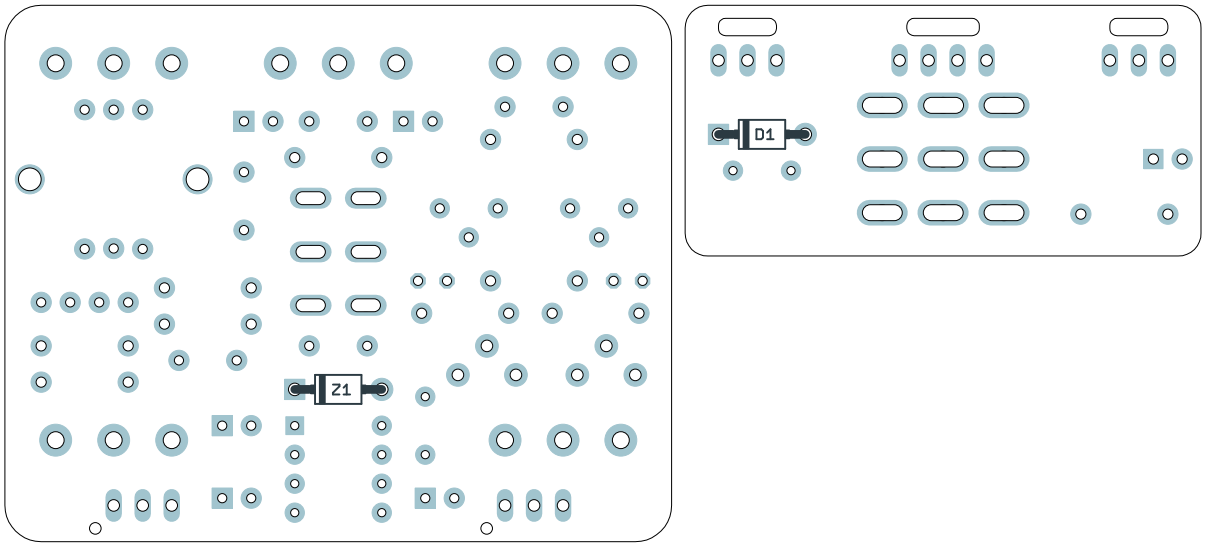
You'll use this same technique for most of the other components as well.

If it gets too crowded, just flip the board and solder everything you've done so far, then cut the leads using the wire snippers to make room for more.

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

DIODES

PART	VALUE
D1	1N5817
Z1	1N4742A

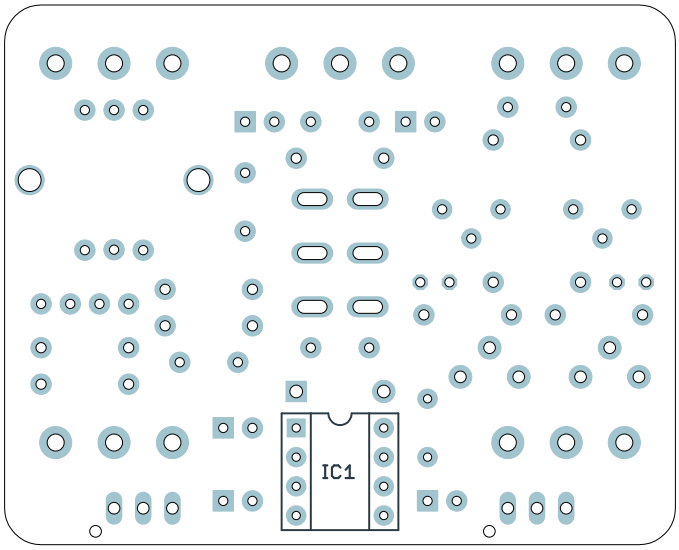


Next, you'll populate the diodes—only one on each board, so this step is pretty simple. The 1N5817 is black with a silver stripe, while the 1N4742A is orange with a black stripe.

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.

SOCKET & IC

PART	VALUE
IC1	LT1054CP



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

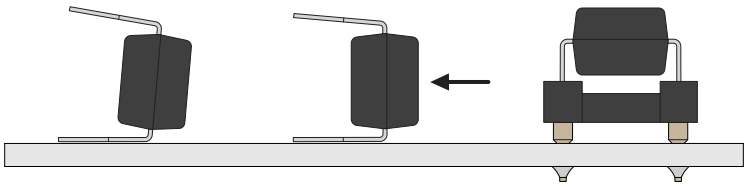
Again, it's much easier to do this with gravity holding them in place for you, so you'll want do them before you do any of the taller components.

Installing the IC

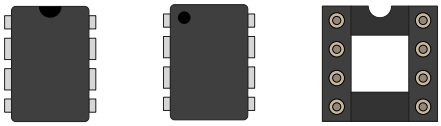
Don't insert the IC into the socket just yet. We will do this in a later step, after we've finished soldering the tallest components (the polarized capacitors). This information is just listed here for reference.

The legs of the IC are bent outward slightly during manufacturing, so they'll need to be bent back inward before they can be inserted into the sockets.

It's easiest to do this by laying the IC legs against the table and bending the body itself so all four legs on the side are straightened out at once. Then, flip it and do the other side.

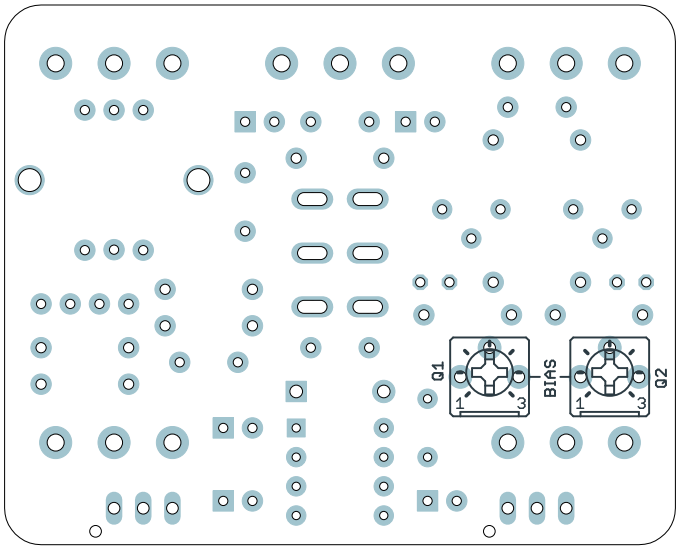


ICs may have two different orientation marks: either a dot in the upper-left or a half-circle notch in the middle of the top side. Some ICs have both marks. This shows which way the IC should be rotated when inserting it into a socket (the socket also has a half-circle notch).



TRIMMERS

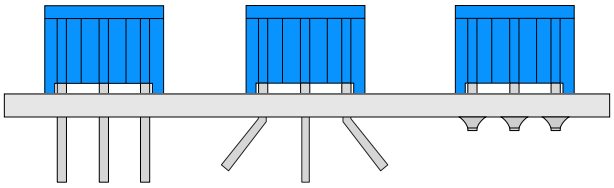
PART	VALUE
Q1 BIAS	100k (104)
Q2 BIAS	10k (103)



The bias trimmers come next. The two trimmers have different values, so you'll first need to identify them by the code on the side of the case. The text is laser-etched rather than printed, so if you have a hard time making it out, you can hold it at an angle to a light source and it should be much easier to see.

The 100k trimmer for Q1 BIAS is marked **P 104**, while the 10k trimmer for Q2 BIAS is **P 103**.

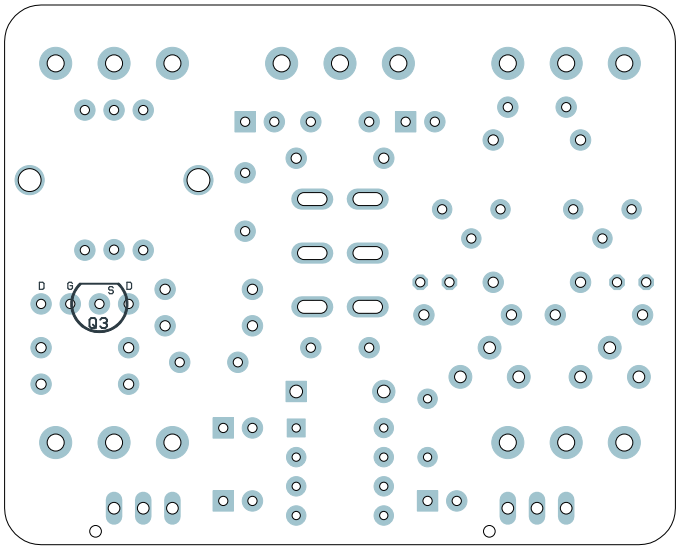
The two trimmers can be soldered like normal components, by bending the legs outward as shown:



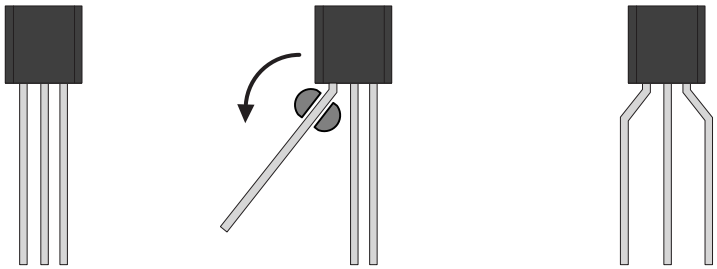
These trimmers are used to bias the germanium transistors. We will adjust them at the end once the pedal is fully assembled.

JFET TRANSISTOR

PART	VALUE
Q3	J112 or PN4303

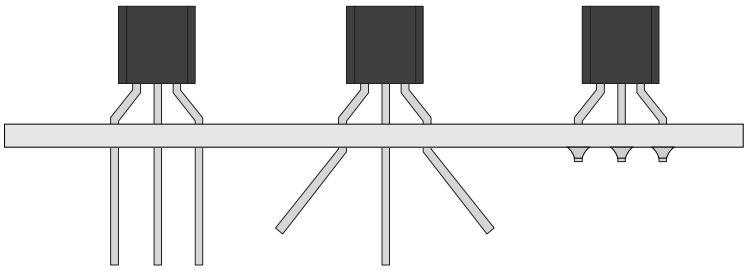


Now we'll do the JFET transistor. For the JFET, if the legs are not already bent into 0.1" spacing, use your needle-nose pliers to bend the outer two legs as shown.



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

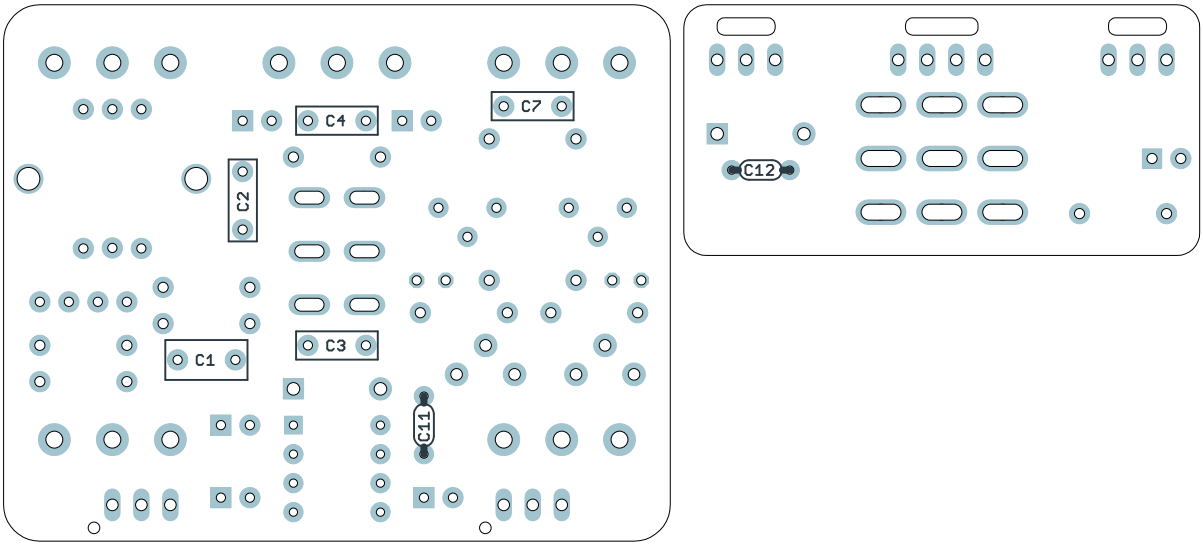
Bend the outer leads of the JFET to hold it in place on the board. Then, solder it and clip the leads.



CAPACITORS (NON-POLARIZED)

PART	VALUE
C1	1uF
C2	100n (0.1)
C3	1n
C4	10n (0.01)

PART	VALUE
C7	10n (0.01)
C11	100n MLCC
C12	100n MLCC



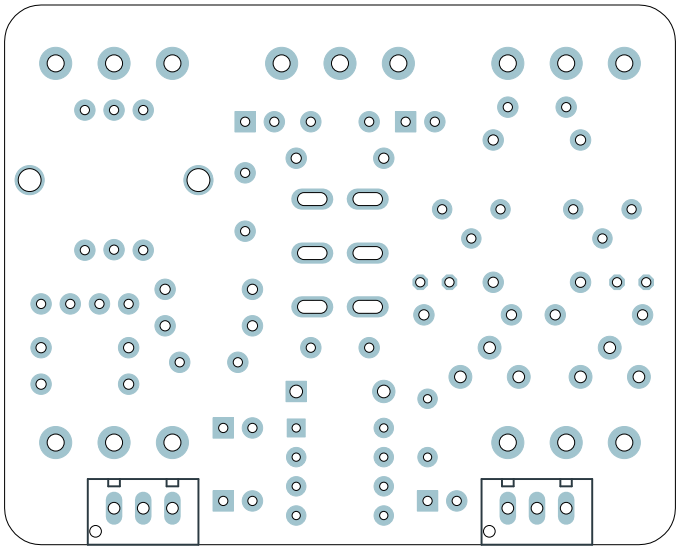
After the JFET come the box film and MLCC capacitors. These are all several different heights, but there aren't as many, so just do them all at once. Bend the leads at an angle to hold them in place.

MLCCs and box capacitors are not polarized and will work in any direction. To keep things neat, though, it's best to put them all facing the same way so the values can easily be read.

Note: The blue box film capacitors (C3 and usually C4 and C7) have the value printed on the top, while the red capacitors have the value on the side. The text on the side of the blue capacitors is not related to the value and can be ignored.

C2 is usually red, but may read "µ1J100" on the top rather than the side.

WIRE HEADERS

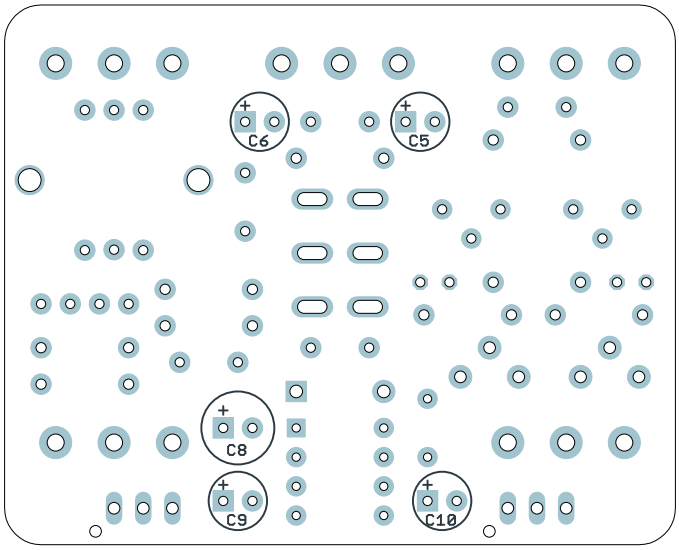


Install the two 3-pin headers (wire connectors) as shown above. These have a polarity pin, so as long as they are pressed all the way down, there's only one possible way to install them. They do fit pretty tightly in the holes, though, so press firmly.

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

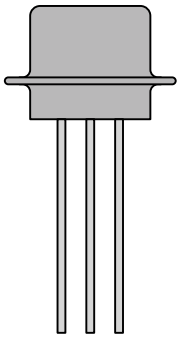
CAPACITORS (POLARIZED)

PART	VALUE
C5	10uF
C6	22uF
C8	100uF
C9	10uF
C10	47uF



Populate the electrolytic capacitors. 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.

GERMANIUM TRANSISTORS: INTRODUCTION



Next are the germanium transistors. These are very different than modern electronic components, so they need an introduction before we cover how to install them.

Silicon vs. germanium

Compared to silicon, germanium transistors are imprecise and inconsistent, even among the same part number. As a result, when classic fuzz circuits use germanium transistors, it's not the part number that's important, but the specifications of the individual device. Two transistors of different part numbers with identical gain and leakage will sound exactly the same. Conversely, not all transistors of the same part number will work in a particular circuit.

Because of this, be aware that this kit may include any of several different part numbers. In this document we will refer to them as Q1 and Q2, not by specific part numbers such as AC125 or МП16Б.

Matching

The transistors included in the Solaris kit have been measured and matched for best performance in the Fuzz Face circuit. The transistors come in two bags stapled together, and each bag has two numbers written on it that are used for the matching process.

It's not necessary for you to understand what these numbers mean, but if you're curious, the top number is the gain (h_{FE}) and the bottom number is the leakage current in microamps (μA). All you need to know for a successful build is that the transistor in the front is **Q1** and has a lower gain value. The second transistor is **Q2** and has a gain value approximately 40 to 60% higher than Q1.

It is extremely important that the transistors are not removed from the bags until it's time to install them, and only one at a time, to avoid inadvertently mixing them up. They are not visually distinguishable from each other, and outside of the bags there is no way to tell which is which. The pedal won't sound right if the transistors are used in the wrong positions.

Temperature sensitivity

Germanium is a delicate semiconductor material and very sensitive to overheating. If the transistor is overheated, it could be permanently damaged. It's recommended to mount the transistors so they are elevated above the PCB by about 3/8" (9.5mm) to allow some of the heat to dissipate before it reaches the body of the transistor. If it takes longer than two seconds for the solder to form a good joint, move to a different component and let the transistor cool a bit before trying again.

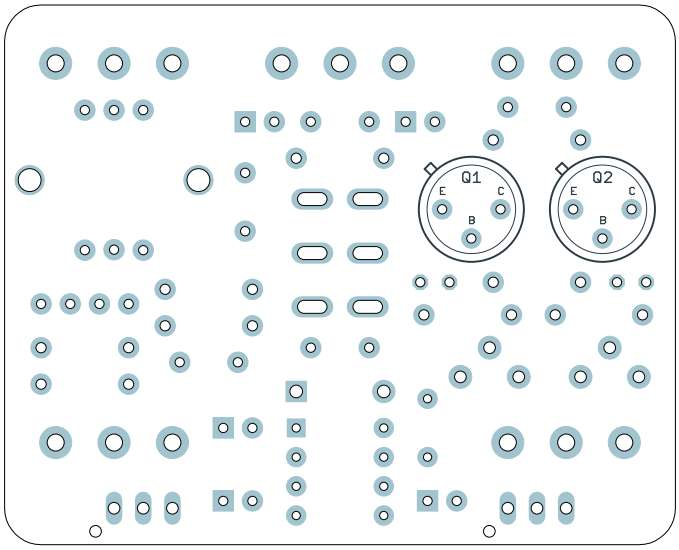
Old-stock precautions

Germanium transistors are not manufactured today except for some expensive industry-specific applications, so nearly all of the germanium transistors used in guitar pedals are old-stock. The transistors in this kit are at least 30 years old, and sometimes 50 or more.

While age has nothing to do with performance of the device itself, the leads may be corroded and solder may not adhere well unless they are cleaned first. If you notice any corrosion on the leads, use medium-grit sandpaper or a fine metal file to remove it, and ensure the leads are shiny before soldering.

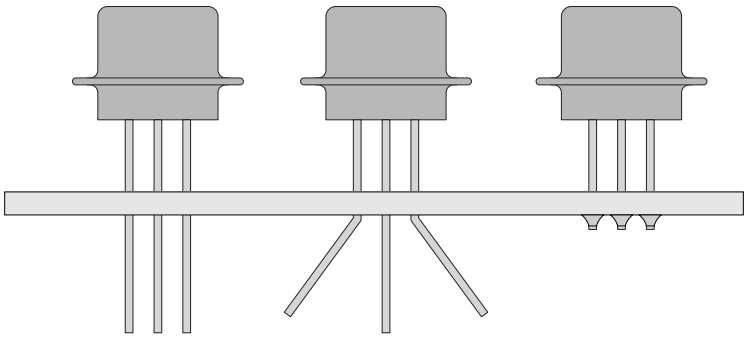
GERMANIUM TRANSISTORS

PART	VALUE
Q1	Germanium
Q2	Germanium

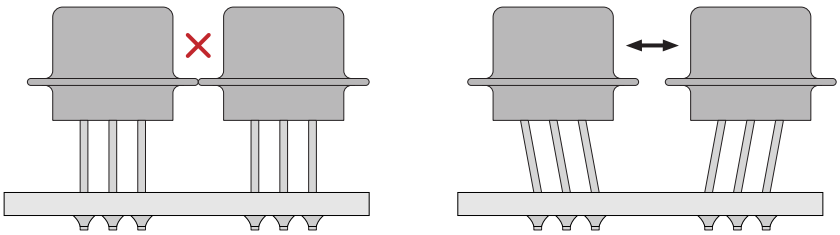


Now that we've covered the basics of germanium transistors, it's time to install them. Remove Q1 (the front transistor with a lower gain value) from the bag. As mentioned on the previous page, it's important to leave Q2 in its bag until Q1 has been soldered so they don't get mixed up.

Insert it into the pads on the PCB, being mindful of the "V" pattern of the pins. Keep it raised about 3/8" (9.5mm) above the PCB, then bend the legs on the bottom side to keep it attached. Turn the PCB over and let it hang down while you solder it in place. Be quick and make sure the transistor casing doesn't get too hot or the transistor could be damaged.

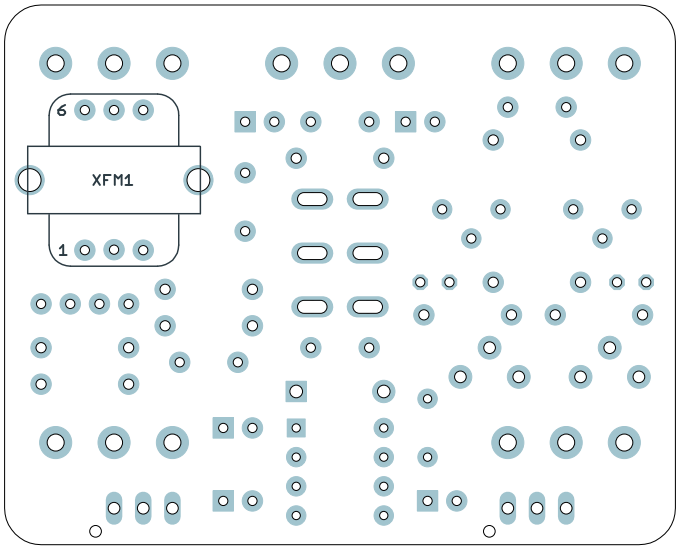


Now, repeat the same step with Q2, the one labeled with the higher gain value. Once both have been soldered, you'll need to make sure the transistors aren't making physical contact with each other. The base pins are connected to the metal casing, so anything they touch will be shorted against the base. Bend them slightly away from each other and they should remain that way.



TRANSFORMER

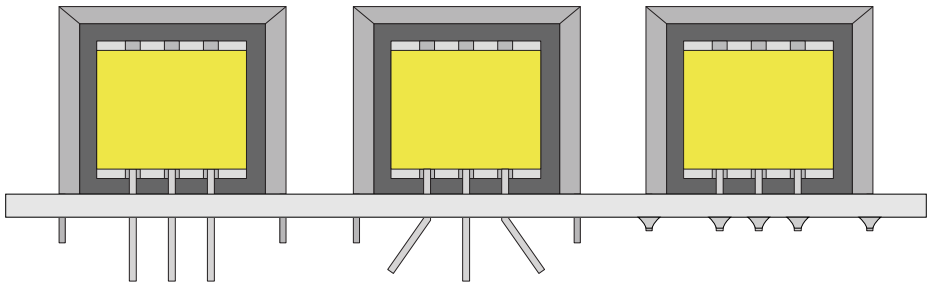
PART	VALUE
XFM1	42TL019



The transformer is the last of the on-board components. Before installing it to the PCB, scrape off any wax on the leads. You don't need to be terribly thorough since it will melt away with the heat of the soldering iron, but we don't want the wax getting in the way of a good solder joint.

The primary and secondary coils of the transformer have different properties, so the transformer must be facing the right direction or it won't work correctly. In this case, the primary is marked with a "P" and the top of the nylon bobbin is colored black on that side. The primary side should line up with the pad marked "1" (in other words, facing toward the bottom of the PCB).

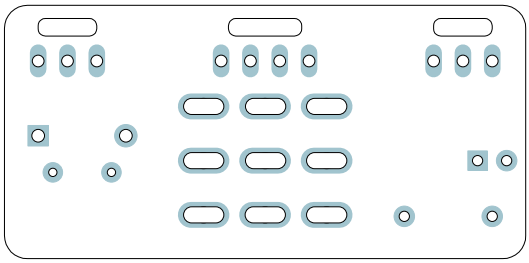
As with earlier components, first bend the six legs to hold it in place as shown in the diagram below. Solder the legs and the two outer tabs and then clip them short.



Now that we've finished with the main PCB, go back to page 12 and insert the IC into the socket.

FOOTSWITCH PCB

PARTS
3-strand wire assembly (2)
4-strand wire assembly



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

There will be one longer assembly with 4 wires and two shorter ones with 3 wires. The longer one goes in the middle and the shorter ones go on the left and right sides. The wire assemblies should then be soldered to the footswitch board as shown.

STEP 1

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

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

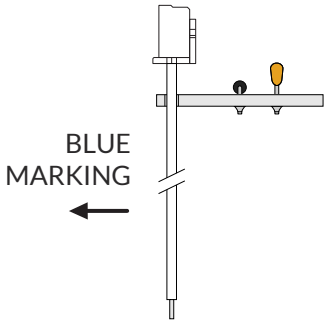
STEP 2

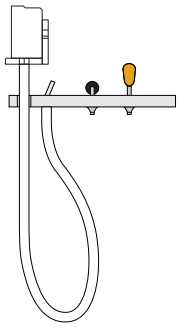
Next, bend the wires back upward and fit the ends of the wires into the solder pads.

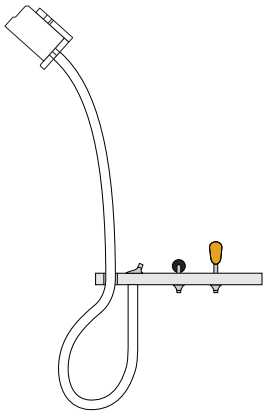
On the top side of the PCB, bend the exposed wires backward so it holds the wire in place. Pull the header back up through the slot partway.

STEP 3

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



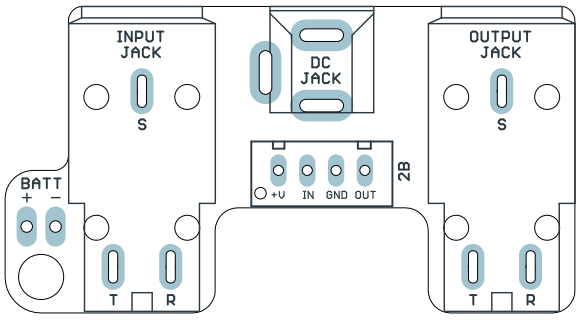




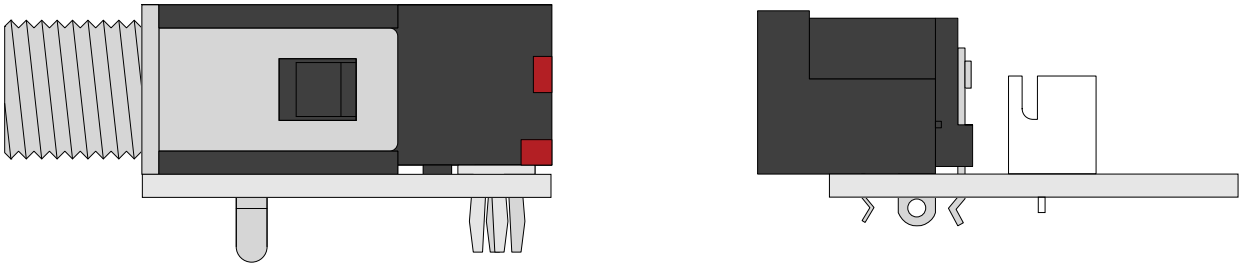
Once all three wire assemblies are soldered, set the footswitch PCB aside. We'll solder the actual footswitch and LED in a later step.

INPUT/OUTPUT PCB

PARTS
Input & output jacks
DC jack
Wire header
9V battery snap



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

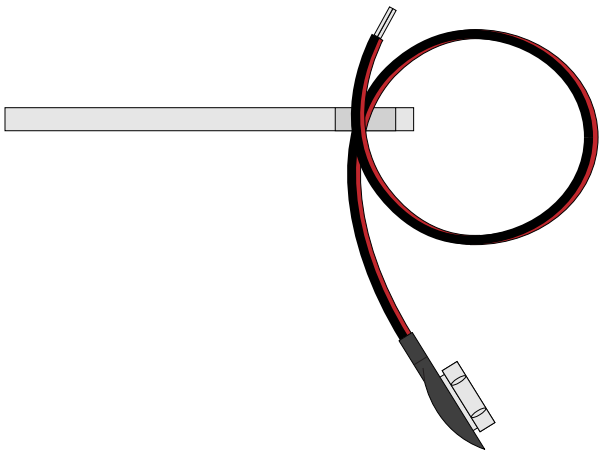


After you've soldered everything, make sure to **snip the leads on the I/O jacks as close as possible to the PCB**. There's not a lot of clearance between the bottom of this board and the top of the main PCB once everything is in place, and you don't want the pins to short against anything on accident.

Next, we'll hook up the 9V battery connector. **This is optional.** Not everyone uses batteries. But, if you do, this pedal should last a long time on a single 9V so you won't need to change it very often.

STEP 1

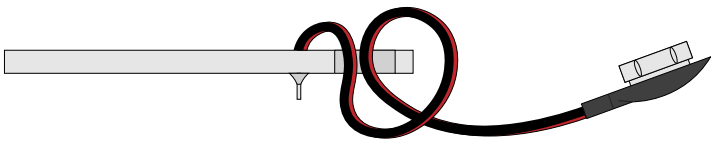
Thread the battery snap leads through the strain-relief hole twice so it forms a single loop.



STEP 2

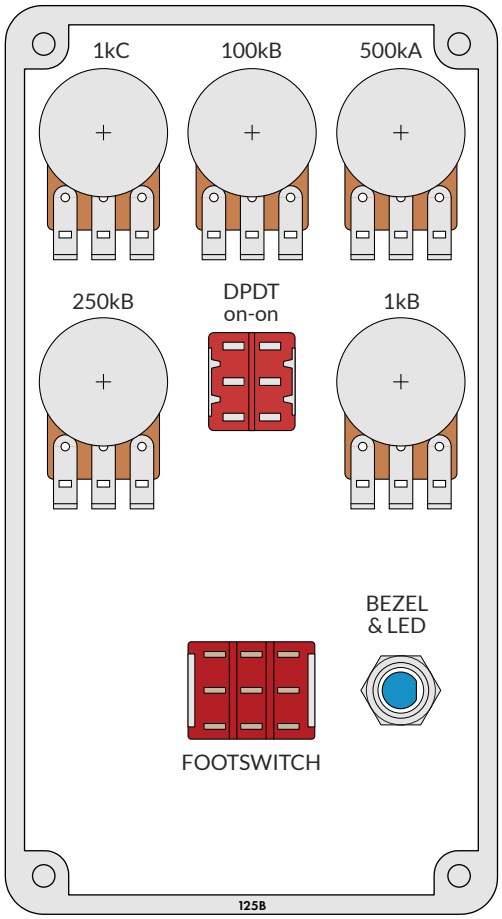
Bend the exposed wires back down and solder them into the pads. Red is positive (+), black is negative (-). After soldering, pull it tight.

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

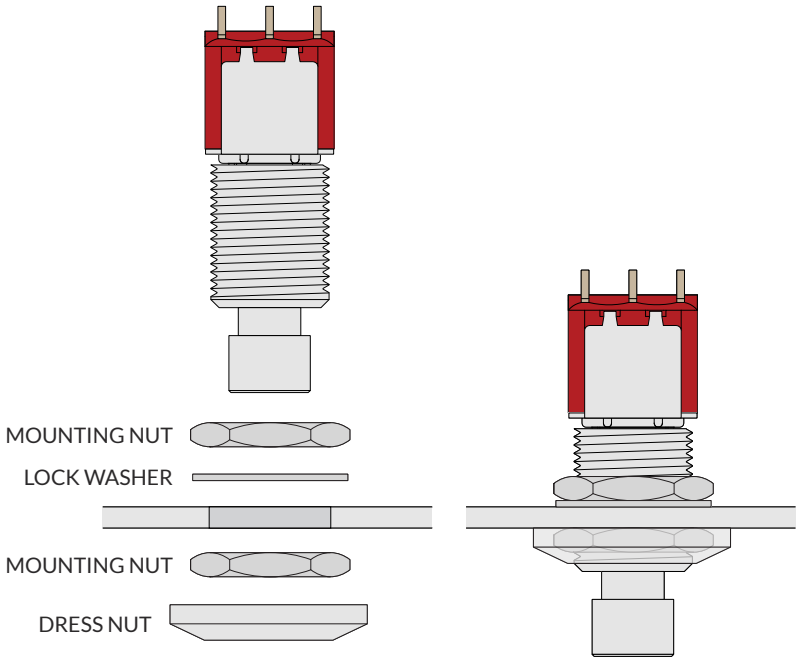


ENCLOSURE LAYOUT: PANEL MOUNTS

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

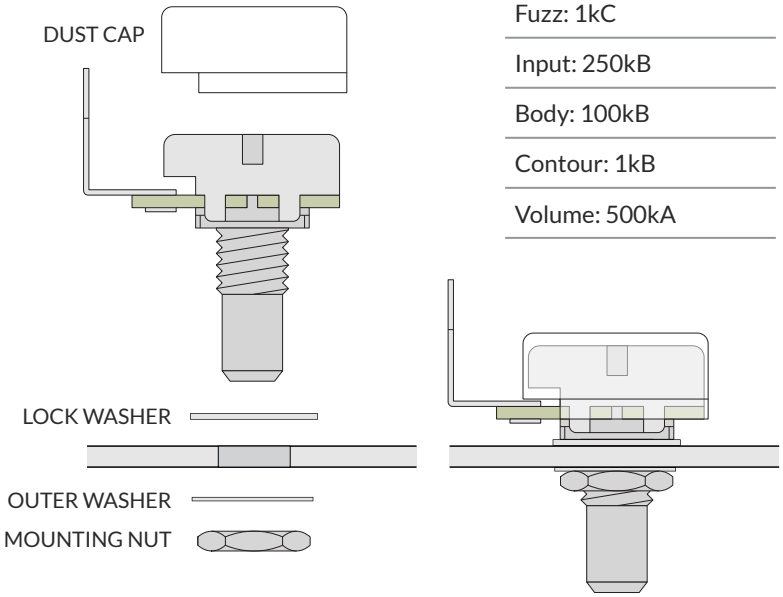


FOOTSWITCH



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

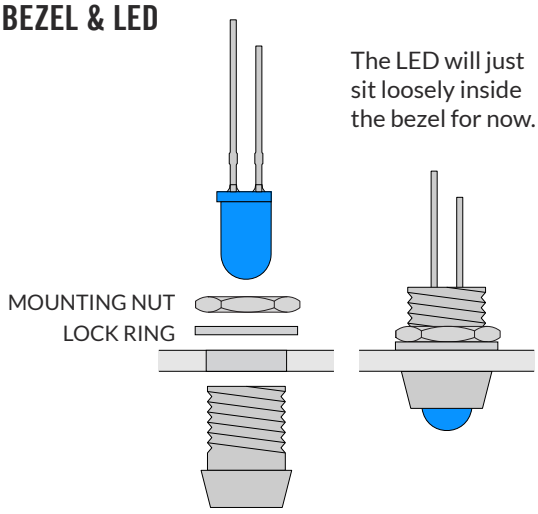
POTENTIOMETERS



Fuzz: 1kC
Input: 250kB
Body: 100kB
Contour: 1kB
Volume: 500kA

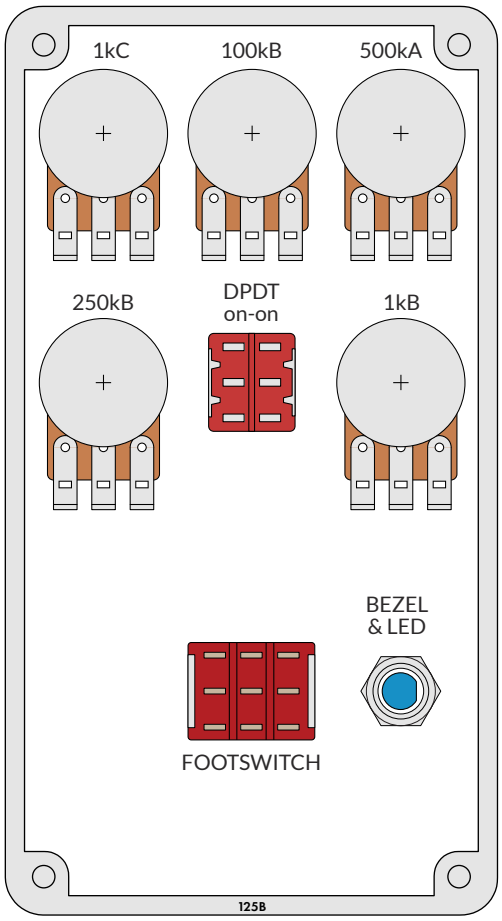
BEZEL & LED

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

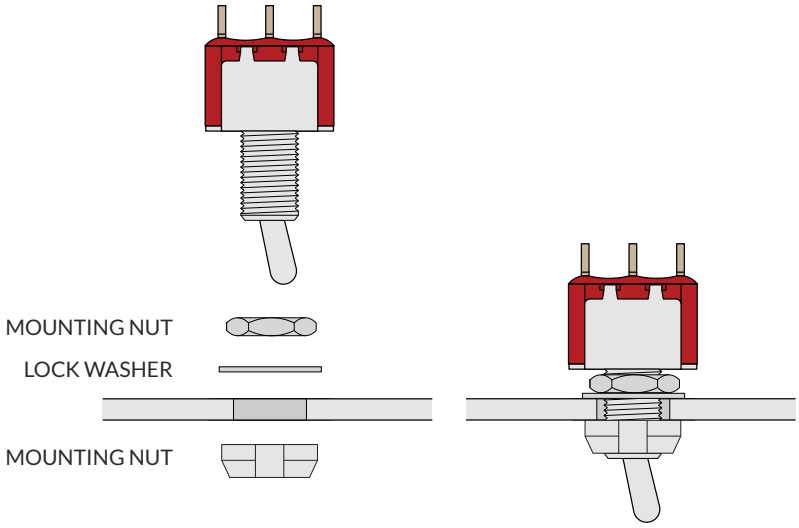


You'll need to hold the bezel in place when tightening the nut. The top of the bezel is fairly sharp, so try using a rubber band for grip instead of pressing your finger against the bottom.

ENCLOSURE LAYOUT: PANEL MOUNTS (CONT.)

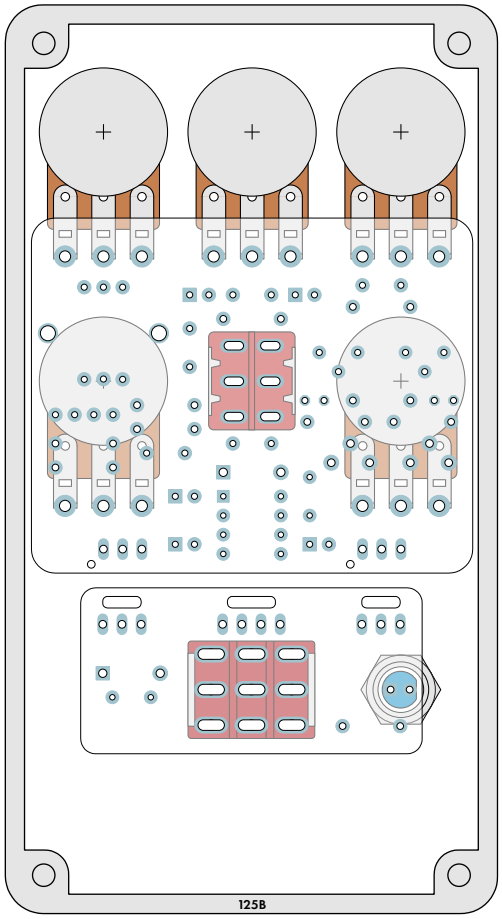


TOGGLE SWITCHES



The dress nut acts as a mounting nut, unlike the footswitch dress nut. Use flat-nose pliers on the flat sides of the nut to tighten securely.

ENCLOSURE LAYOUT: MAIN & FOOTSWITCH PCBS



After all the components are affixed to the enclosure as shown on the previous page, place the main PCB on top of the potentiometers and toggle switches as in the diagram to the left.

You may need to adjust the position of the potentiometers and toggle slightly if they are not aligned straight.

Once all of the pins are through their holes and the PCB is laying flat, solder each of the pins from the top. Be careful not to touch any of the surrounding components with the soldering iron.

After you've finished soldering the pots, **clip the leads as close as you can to the main PCB**. This is more important with the two uppermost pots because the input/output PCB overlaps them and you need to avoid any of the components shorting. (The toggle switch lugs do not need to be clipped.)

Next, move to the footswitch board and solder the 3PDT switch.

The LED is last. Before soldering the LED, double-check to **make sure the flat side of the LED is facing to the right**, as shown in the diagram, and that the short leg is coming through the pad on the right. It won't work if it's turned the other way. Then, clip the leads of the LED.

Why solder everything inside the enclosure before testing it?

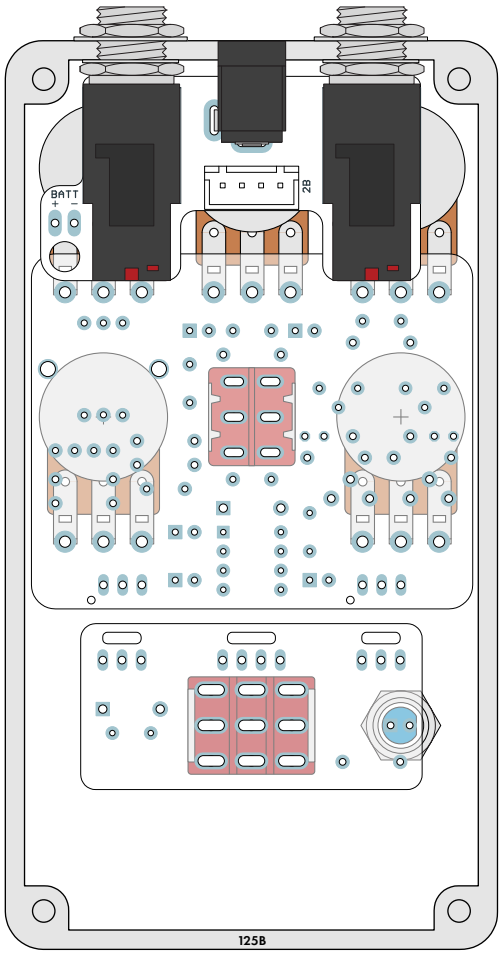
"Rock it before you box it" is conventional wisdom in pedalbuilding, and you'll often hear it recommended that builders should test the circuit before putting everything inside the enclosure. However, Aion FX projects are designed to be extremely easy to remove from the enclosure for troubleshooting,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

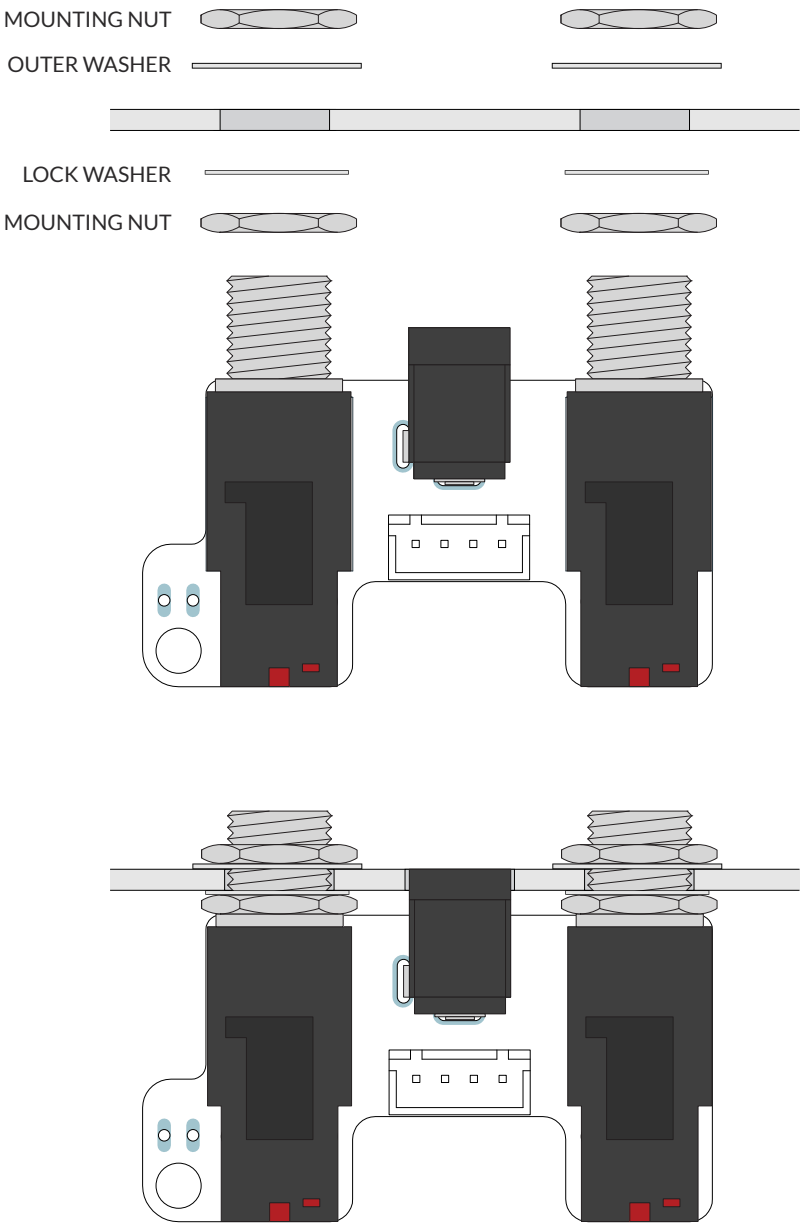
If you've read the documentation carefully and followed all the instructions, there's a good chance you will get it right the first time!

ENCLOSURE LAYOUT: INPUT/OUTPUT PCB

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

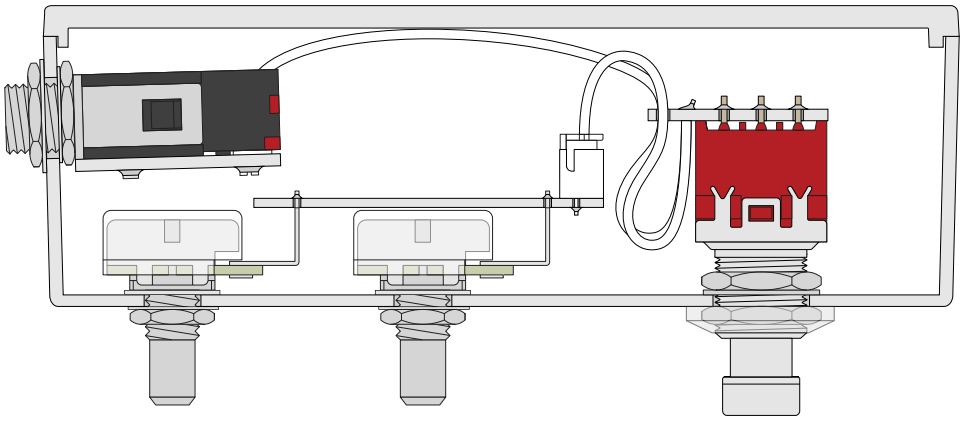
Note the use of two mounting nuts on each of the jacks, one inside and one outside. The inner nut acts as a spacer to set the DC jack flush with the outside of the enclosure. The inner nuts should be threaded as far down as they can go.





FINAL ASSEMBLY & BIASING

After everything is in place, 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, but we still need to bias the transistors before it will sound right.

Biasing

The trimmers allow for easy biasing of the two transistors without having to swap out resistors.

As a starting point, using a small screwdriver, turn the Q1 bias trimmer to 9:00 and the Q2 trimmer to around 2:00. Set the Contour knob just above 9:00. Then, with a multimeter, touch the black and red leads to the two pads marked "TEST" below Q1. Turn the Q1 trimmer until the multimeter reads **-0.7V** (either positive or negative depending on which lead is touching which pad).

Next, moving to the test pads under Q2, turn the Q2 bias trimmer until the multimeter shows **-4.5V** (again, either positive or negative). Then, measure each leg on all three of the transistors. You're looking for something near these voltages.

- **Q1:** Emitter 0V (left leg); Base -0.2V (center leg); Collector -0.7V (right leg)
- **Q2:** Emitter -0.5V (left leg); Base -0.7V (center leg); Collector -4.5V (right leg)

The voltages don't need to be anywhere near exact, this is just a benchmark. Let your ears be the judge. Some people prefer the Q2 voltage to be higher, around 5.5V.

Note that the Contour knob directly adjusts the bias of Q2, so the voltages will not match the above unless the knob is set to 25% rotation.

Testing

Plug in a 9-volt supply and test it out with a guitar and an amplifier.

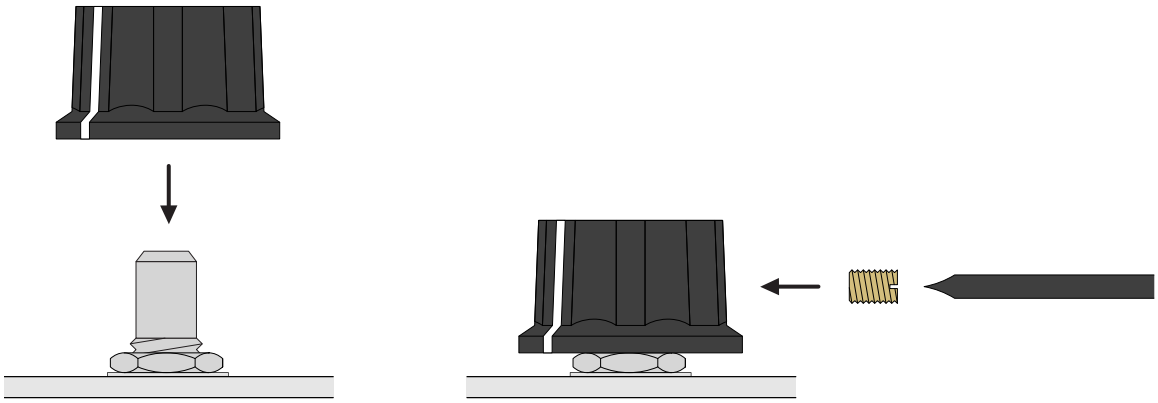
Test the bypass switch a few times, then start turning the controls and see if everything sounds OK. If it works, great! If not, don't be discouraged. See page 31 for troubleshooting info.

Finishing touches

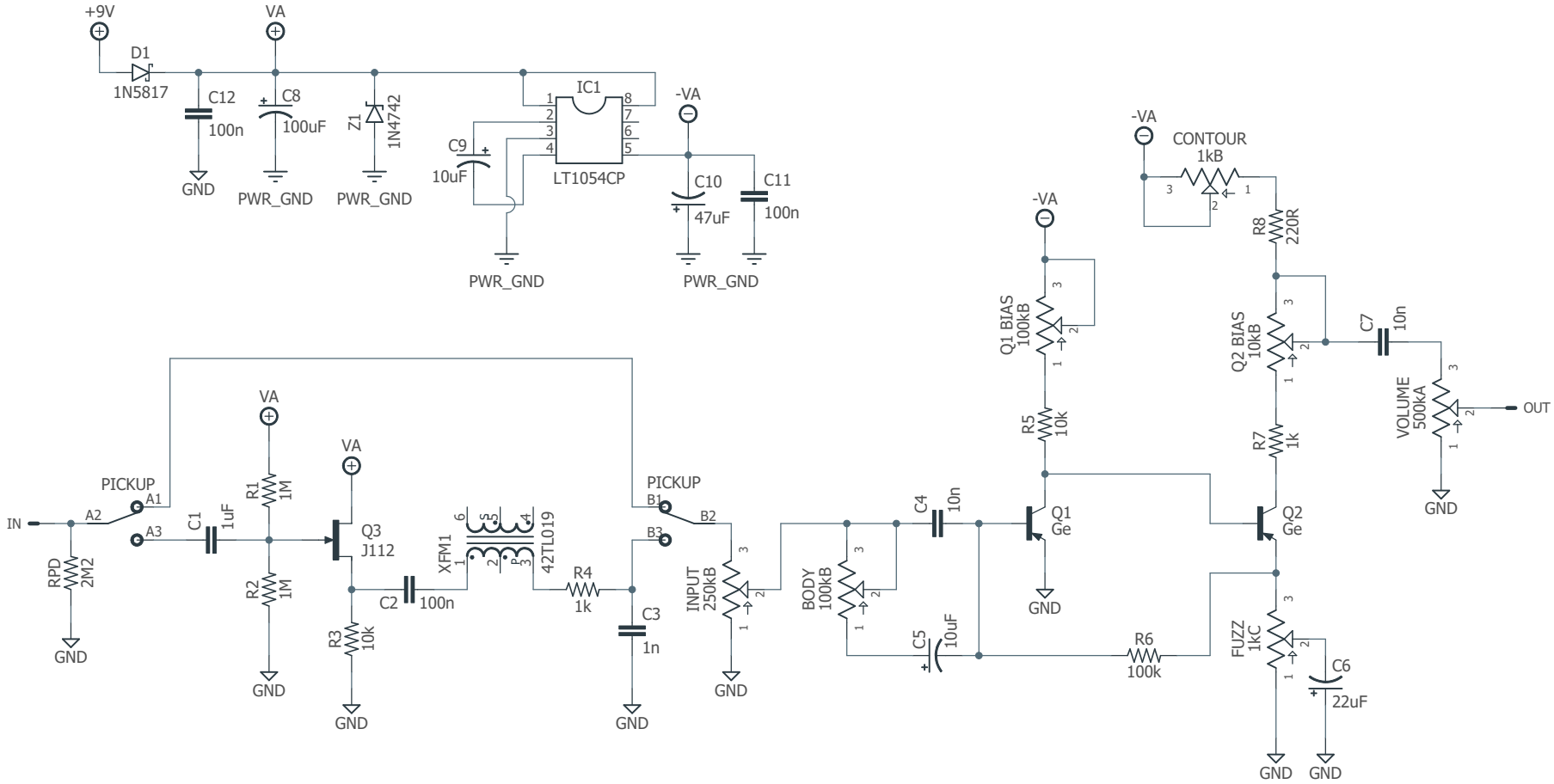
Now, just a couple of things for the final assembly. Turn the shafts all fully counter-clockwise, then put on the knob and rotate until the indicator line is aligned with the dot on the enclosure that shows the zero point. Affix the knobs to each of the potentiometer shafts as shown in the diagram below.

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!


+9V
D1
1N5817
C12
100n
GND
VA
C8
100uF
PWR_GND
Z1
1N4742
PWR_GND
C9
10uF
IC1
LT1054CP
-VA
C10
47uF
C11
100n
PWR_GND
PWR_GND
-VA
CONTOUR
1kB
R8
220R
-VA
Q1 BIAS
100kB
R5
10k
Q1
Ge
GND
Q2 BIAS
10kB
C7
10n
R7
1k
Q2
Ge
VOLUME
500kA
OUT
GND
IN
PICKUP
A1
A2
A3
RPD
2M2
GND
C1
1uF
R1
1M
R2
1M
GND
VA
Q3
J112
R3
10k
GND
C2
100n
XFM1
42TL019
R4
1k
C3
1n
GND
PICKUP
B1
B2
B3
INPUT
250kB
GND
BODY
100kB
C4
10n
C5
10uF
R6
100k
FUZZ
1kC
C6
22uF
GND
GND

FULL PARTS LIST

Resistors

PART	VALUE
R1	1M
R2	1M
R3	10k
R4	1k
R5	10k

PART	VALUE
R6	100k
R7	1k
R8	220R
RPD	2M2
LEDR	10k

Capacitors

PART	VALUE
C1	1uF
C2	100n (0.1)
C3	1n (0.001)
C4	10n (0.01)
C5	10uF electro
C6	22uF electro

PART	VALUE
C7	10n (0.01)
C8	100uF electro
C9	10uF electro
C10	47uF electro
C11	100n MLCC
C12	100n MLCC

Transistors

PART	VALUE
Q1	Germanium
Q2	Germanium
Q3	J112 or PN4303

Diodes

PART	VALUE
Z1	1N4742A
D1	1N5817

IC

PART	VALUE
IC1	TC1044SCPA or LT1054CP

Potentiometers

PART	VALUE
Fuzz	1kC
Input	250kB
Body	100kB
Contour	1kB
Volume	500kA

Switches

PART
DPDT on-on
3PDT stomp

Transformer

PART
42TL019

TROUBLESHOOTING INFORMATION

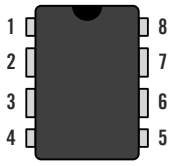
If you finish building the kit and find that it doesn't work right, we've written a separate in-depth Troubleshooting Guide that applies to all of our kits. The main troubleshooting process is covered there. Here you will find information specific to this kit that will help with that process.

The transistors won't bias correctly.

If you can't get the right voltages on the transistors as described on page 27, there are a few steps to diagnose the issue.

Is the charge pump putting out the correct voltage?

The charge pump (IC1) is an inverter that converts +9V to -9V so the circuit can be used with a standard center-negative power supply. If it's not inverting properly, nothing else will work.



First, set your multimeter to DC mode with a range of 20V or higher. Touch the black lead to a ground point for the circuit. The easiest spot is inside a tapped screw hole in one of the corners of the enclosure. This way the probe stays in place without needing to use alligator clips. The circuit must of course be fully installed in the enclosure for this to work.

Then, touch the red probe to the lower-right pin of IC1 (pin 5) and read the voltage. Note that IC pins are labeled counter-clockwise from the upper-left, as shown in the diagram to the right. It should measure the inverse of the supply voltage, or slightly less.

If the voltage is not -9V, then there's an issue with the charge pump itself or the surrounding parts.

Are the transistors touching each other, the side of the enclosure, or any other nearby component?

Some transistors have the metal case connected to one of the pins, so anything making contact with the transistor will interfere with its operation. Make sure the transistors are angled away from each other and away from the side of the enclosure, and that they don't make contact with the underside of the lid when it's closed.

The voltages are close, but not exact.

Is the collector voltage correct? The other voltages are given as rough guidelines, but they may be slightly different depending on the properties of the transistor. The collector voltage is the one we're concerned with. If it's right, then the others should be OK.

Also, make sure the Contour knob is set to approximately 25% rotation (just above 9:00) when biasing Q2. This knob directly adjusts the bias of Q2 which happens to change the midrange tone in a useful way. At 25% rotation, it has the stock Fuzz Face resistance of 470 ohms, which is what the bias voltages are based on.

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

1.0.4 (2025-03-24)
Added J112 as an alternate part for Q3.

1.0.3 (2025-02-12)
Updated LEDR (LED current-limiting resistor) to 10k to reduce brightness.

1.0.2 (2024-08-08)
Added link to troubleshooting guide on page 31.

1.0.1 (2022-07-01)
Add LT1054 as an alternate for IC1.

1.0.0 (2022-01-21)
Initial release.