PROJECT NAME

STRAYLIGHT

aion DIY GUITAR EFFECTS

BASED ON
Shin-ei/Univox Uni-Vibe®

BUILD DIFFICULTY
Advanced

EFFECT TYPE
Chorus/phaser/vibrato

DOCUMENT VERSION
1.0.0 (2026-08-21)

PROJECT SUMMARY
Originally designed as a simulation of a Leslie spinning speaker cabinet, it uses an incandescent bulb and LDRs to produce an asymmetrical phase-shift effect. It was most famously used 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 39.

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

Film Capacitors

NAME	QTY
4n7	1
15n (0.015)	1
220n (0.22 or µ22)	1
1uF	11

Electrolytic Capacitors

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

MLCC Capacitors

NAME	QTY
330pF	1
470pF	1
470n (marked “474”)	3

Diodes

NAME	QTY
1N4742A	1
1N5817	3
1N914	2
BAT48	1

Resistors

NAME	QTY
68R	1
1k2	1
2k2	3
3k3	2
4k7	14
6k8	1
10k	1
22k	3
47k	9
68k	1
75k	2
100k	8
220k	3
1M2	2
1M5	1
2M2	2

Transistors & Regulator

NAME	QTY
2N5087	1
2N5088	9
2N5089	1
MPSA13	2
78L15 regulator	1

ICs

NAME	QTY
LT1054CP	1
8-pin socket	1

PACKING LIST (CONT.)

Other

NAME	QTY
Trimmer, 500R (marked “501”)	1
Trimmer, 50k (marked “503”)	1
Trimmer, 250k (marked “254”)	1
LDR (PDV-P9203)	4
Lamp (2174)	1

Potentiometers

NAME	QTY
50kB	1
100kB	1
100kC dual	2
Dust cover	2
Knob	4
Mounting nut, potentiometer, 0.44"	4
Lock washer, potentiometer, 0.5"	4
Outer washer, potentiometer, 0.475"	4

Other

NAME	QTY
LED, white (5mm)	1
LED, red (3mm)	2
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
Enclosure	1
Enclosure screws	4

Switches & hardware

NAME	QTY
Slide switch, DPDT, EG1271	1
Toggle switch, SPDT on-on	2
Mounting nut, toggle switch, 0.36”	2
Lock washer, toggle switch, 0.4"	2
Dress nut, toggle switch, 0.375"	2
Stomp switch, 3PDT	2
Mounting nut, stomp switch, 0.6”	2
Knurled nut, stomp switch, 0.57"	2
Lock washer, stomp switch, 0.6"	2
Switch label plate, “ON/OFF”	1
Switch label plate, “FAST/SLOW”	1

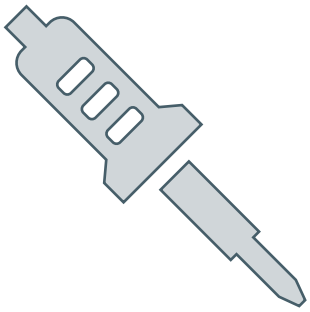
Wiring

NAME	QTY
4-strand wire assembly, 60mm	3
4-strand wire assembly, 122mm	1
4-pin wire assembly header	4

PCBs

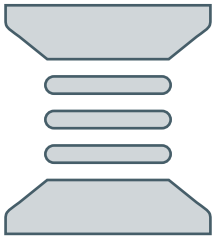
NAME	QTY
PCB, main circuit	1
PCB, bypass & footswitches	1
PCB, input/output/DC	1
PCB, light shield	1

TOOLS NEEDED



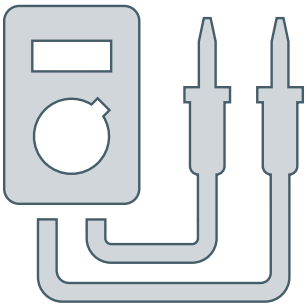
SOLDERING IRON

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.



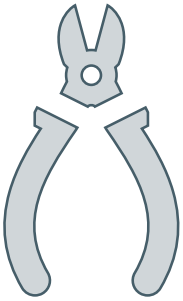
SOLDER

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.



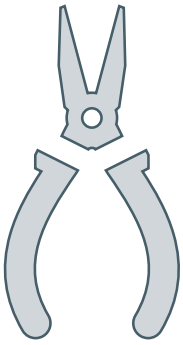
DIGITAL MULTIMETER (DMM)

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.



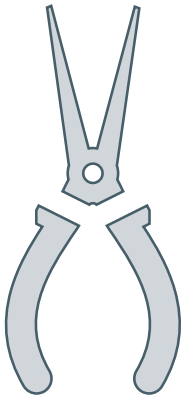
WIRE SNIPPERS

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



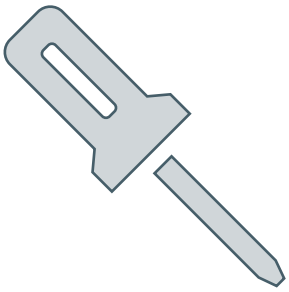
FLAT-NOSE PLIERS

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



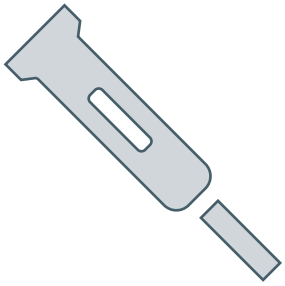
NEEDLE-NOSE PLIERS

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!



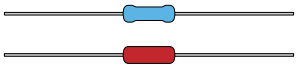
FLAT SCREWDRIVER (SMALL)

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

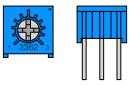
COMPONENT IDENTIFICATION

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.)

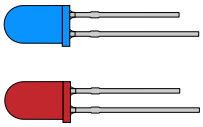
RESISTOR



TRIM POTENTIOMETER

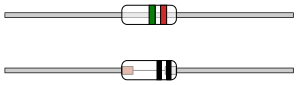


LED



SILICON DIODE

GERMANIUM DIODE



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

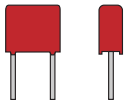
RECTIFIER DIODE

Some Schottky diodes also look like this.

SCHOTTKY DIODE

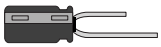
ZENER DIODE

FILM CAPACITOR



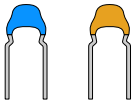
Not polarized. Color may vary by brand and type.

ELECTROLYTIC CAPACITOR



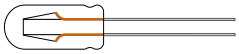
Polarized. The negative side is marked.

MLCC



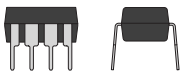
Not polarized. MLCC stands for "multi-layer ceramic capacitor."

LAMP



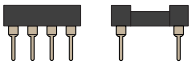
Not polarized. Very fragile.

IC OR OP-AMP

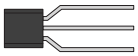


Charge pumps and delay chips also look like this. They may have more than 8 legs.

IC SOCKET

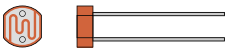


TRANSISTOR OR JFET



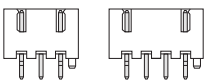
Some voltage regulators also look like this.

LDR (PHOTOCELL)

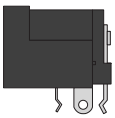


WIRE ASSEMBLY

WIRE ASSEMBLY HEADER



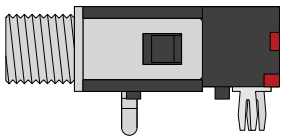
DC JACK



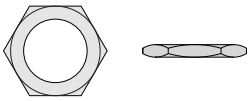
HARDWARE IDENTIFICATION

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

I/O JACK

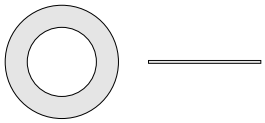


MOUNTING NUT



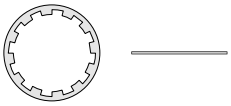
DIAMETER: 0.54" / 13.7mm

OUTER WASHER



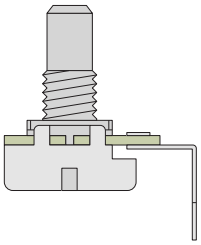
DIAMETER: 0.6" / 15.2mm

LOCK WASHER

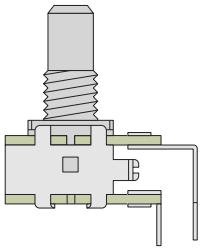


DIAMETER: 0.5" / 12.7mm

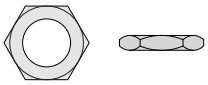
POTENTIOMETER (SINGLE)



POTENTIOMETER (DUAL)

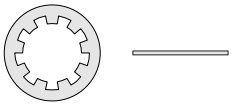


MOUNTING NUT



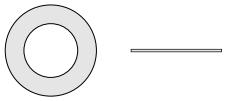
DIAMETER: 0.44" / 11.2mm

LOCK WASHER



DIAMETER: 0.5" / 12.7mm

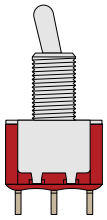
OUTER WASHER



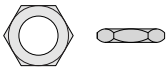
DIAMETER: 0.475" / 12mm

KNOB

TOGGLE SWITCH

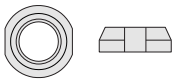


MOUNTING NUT



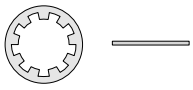
DIAMETER: 0.36" / 9.1mm

DRESS NUT



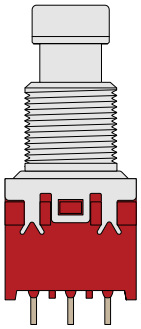
DIAMETER: 0.375" / 9.5mm

LOCK WASHER

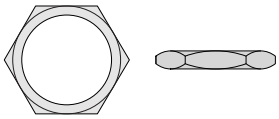


DIAMETER: 0.4" / 10.1mm

FOOTSWITCH

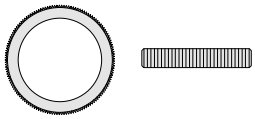


MOUNTING NUT



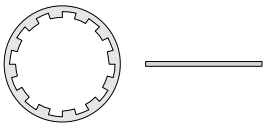
DIAMETER: 0.63" / 16mm

KNURLED NUT



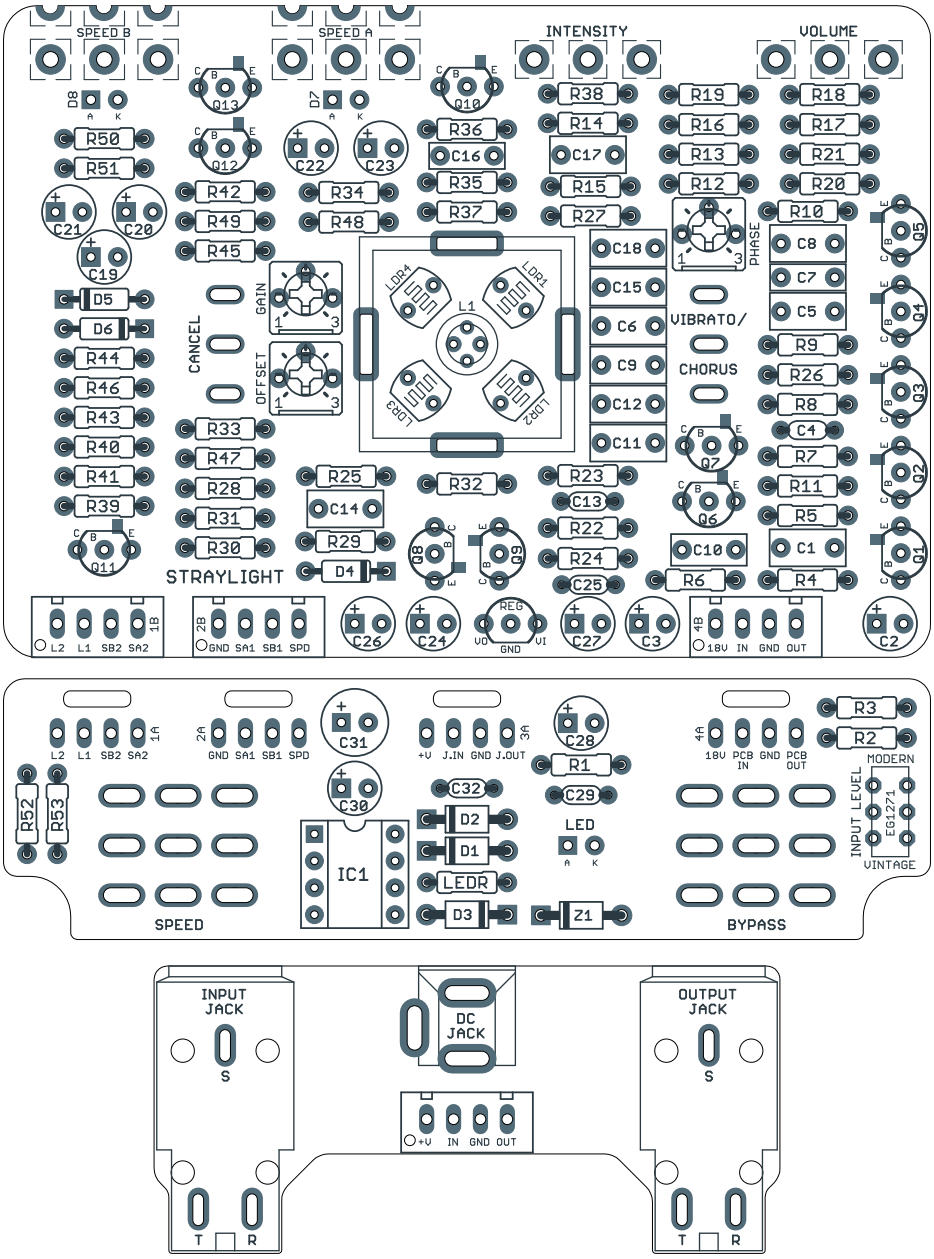
DIAMETER: 0.57" / 14.5mm

LOCK WASHER



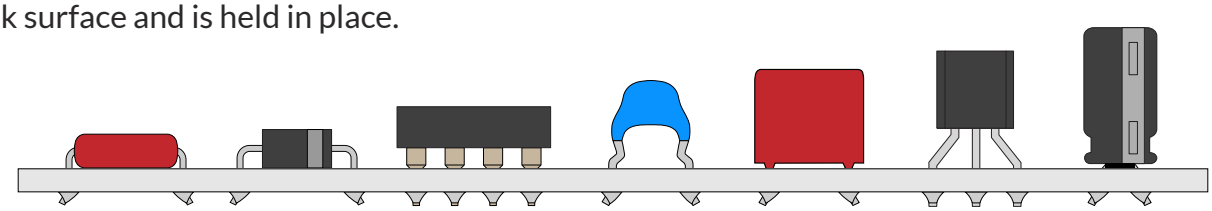
DIAMETER: 0.63" / 16mm

PCB ASSEMBLY OVERVIEW



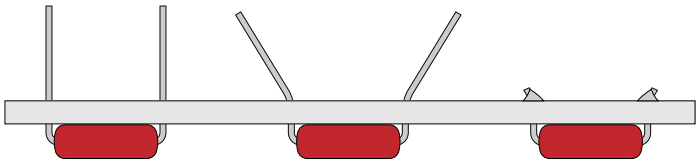
Before you begin, snap apart the main PCB and footswitch board and break off the tabs from each using needle-nose or flat-head pliers. You'll be left with the three PCBs shown above. (The light shield is also a PCB, but it's treated more like a component. See pages 21-23 for assembly.)

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.



RESISTORS (FIRST SET)

Using the parts list below, 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.



It's recommended to only install 10-15 resistors at a time, then flip the board over, solder them, and trim the leads. This makes room for the next set.

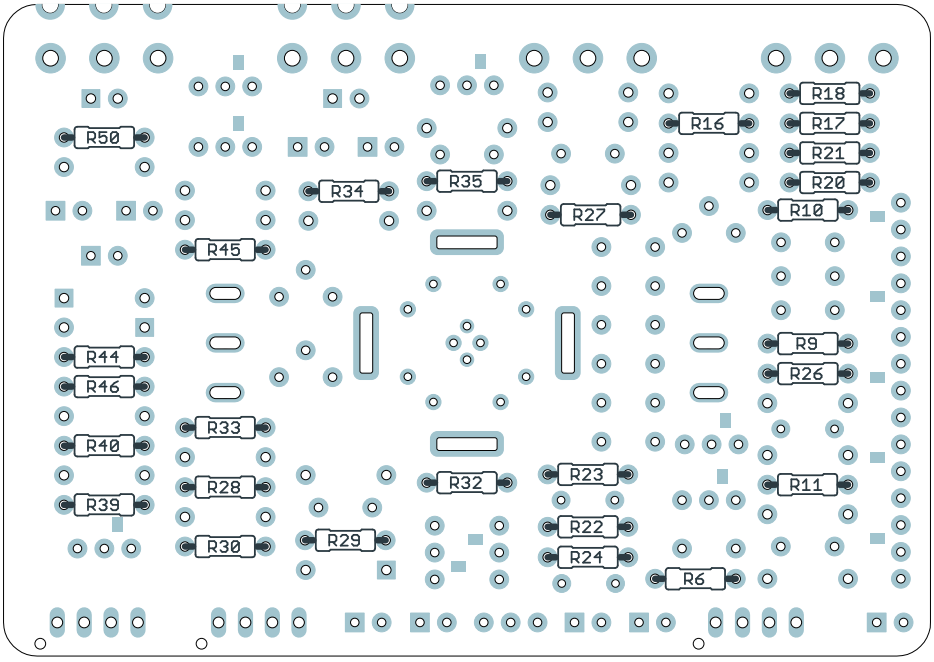
Due to the large number of resistors in this circuit, we've broken out the parts list into two separate groups of resistor values.

PART	VALUE
R9	4k7
R16	4k7
R20	4k7
R21	4k7
R22	4k7
R26	4k7
R27	4k7

PART	VALUE
R28	4k7
R32	4k7
R33	4k7
R34	4k7
R40	4k7
R44	4k7
R45	4k7

PART	VALUE
R6	100k
R17	100k
R18	100k
R23	100k
R24	100k
R29	100k
R30	100k

PART	VALUE
R35	100k
R10	3k3
R39	3k3
R50	10k
R11	1k2
R46	68R



RESISTORS (SECOND SET)

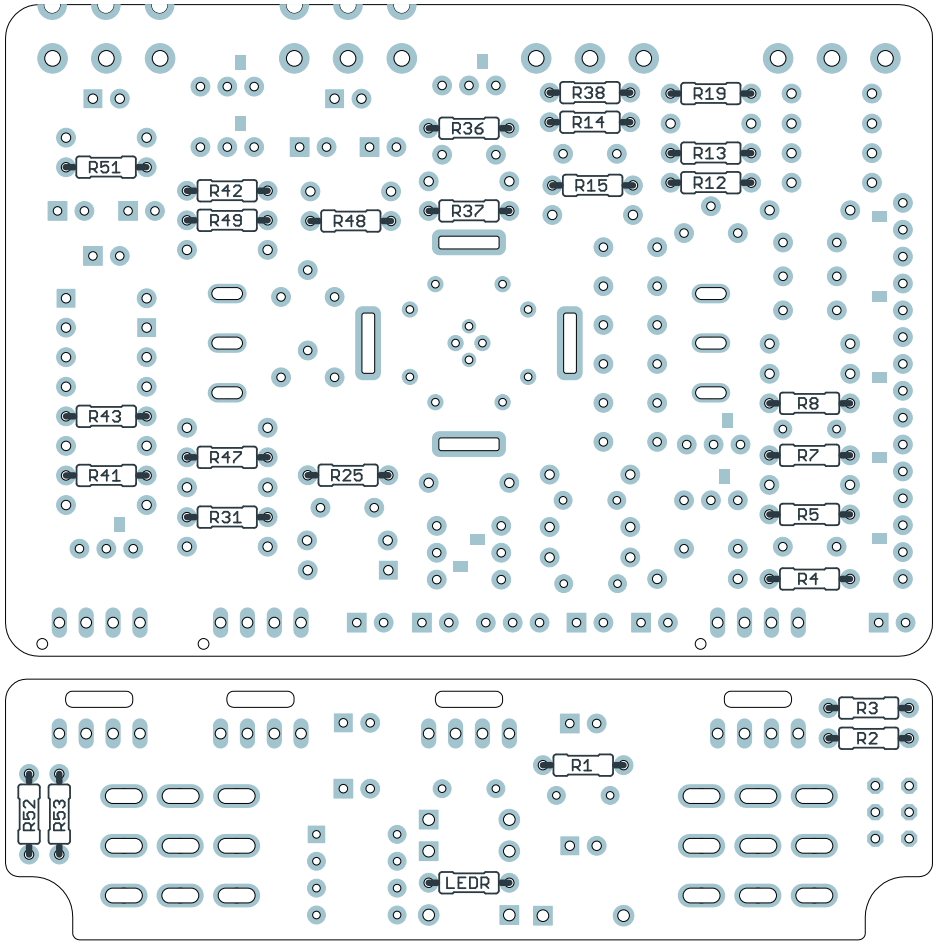
PART	VALUE
R1	22k*
R38	22k
LEDR	22k*
R2	47k*
R7	47k
R14	47k
R19	47k

PART	VALUE
R25	47k
R31	47k
R37	47k
R47	47k
R48	47k
R3	2M2*
R41	2M2

PART	VALUE
R4	1M2
R5	1M2
R8	6k8
R12	75k
R13	75k
R15	220k
R42	220k

PART	VALUE
R43	220k
R51	2k2
R52	2k2*
R53	2k2*
R36	68k
R49	1M5

An asterisk (*) denotes placement on the footswitch PCB.



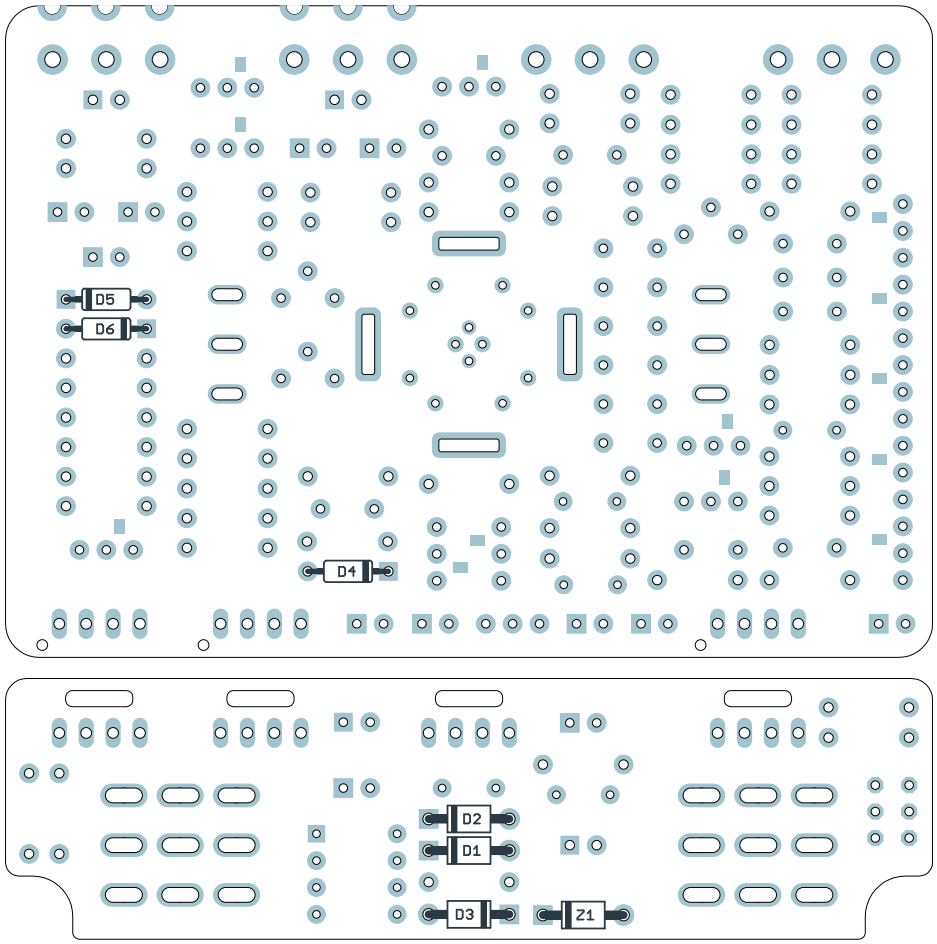
DIODES

PART	VALUE
D1	1N5817
D2	1N5817

PART	VALUE
D3	1N5817
D4	BAT48

PART	VALUE
D5	1N914
D6	1N914

PART	VALUE
Z1	1N4742A



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

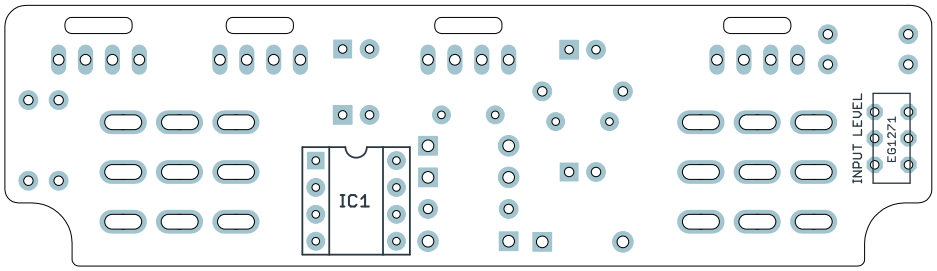
All four types of diodes are very different in appearance, but the names may be hard to read. They can be identified as follows:

- 1N5817 is large with a black body and silver printing.
- 1N4741A is large with an orange body and black stripe.
- 1N914 is small with an orange body and black stripe.
- BAT48 is small with a blue body and black stripe.

Note that the red indicator LEDs that show the rate are marked D7 and D8 on the PCB. These will be addressed in later steps on pages 25 and 30.

SOCKET, IC & SLIDE SWITCH

PART	VALUE
IC1	LT1054CP
INPUT	DPDT slide

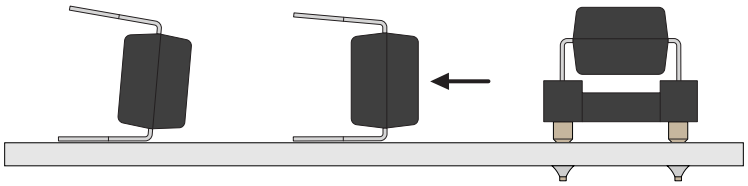


Next up are IC socket and slide switch, both located on the footswitch PCB. You can't bend the leads of these parts as with the other components, so they won't stay in on their own until they're soldered. Flip the PCB over and use gravity to hold them in place. Note that the slide switch is symmetrical, so there's no wrong way to install it.

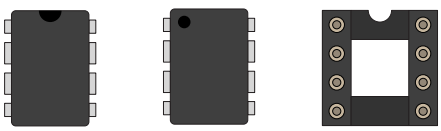
Installing the IC

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.

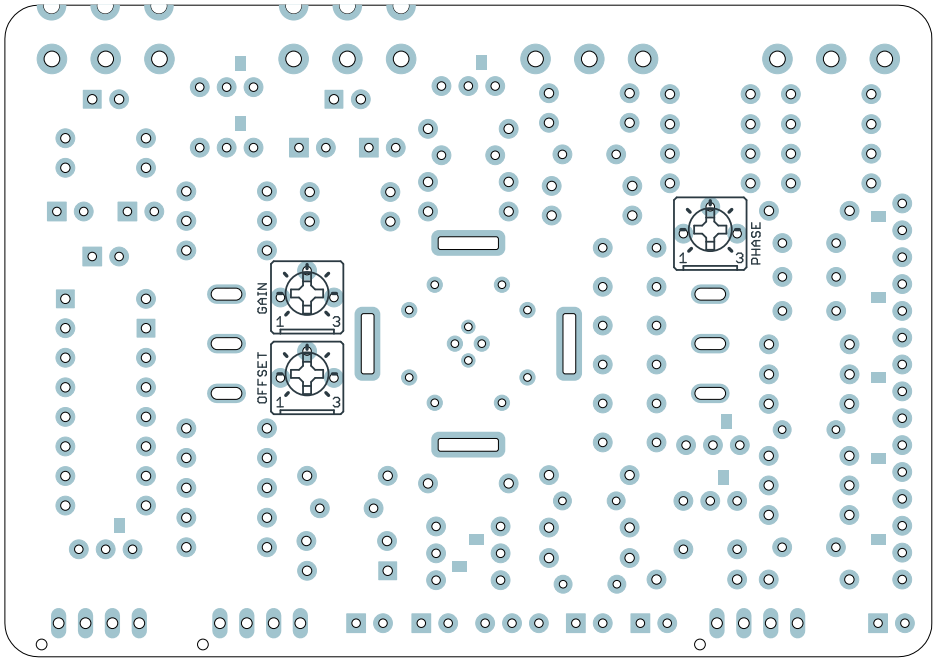


ICs may have two different orientation marks: either a dot in the upper-left or a half-circle notch in the middle of the top side. Some ICs have both marks. This shows which way the IC should be rotated when inserting it into a socket (the socket also has a half-circle notch).



TRIMMERS

PART	VALUE
GAIN	500R trimmer (marked “501”)
OFFSET	250k trimmer (marked “254”)
PHASE	50k trimmer (marked “503”)



Now install the three trimmers. The legs on these can be bent like normal components to hold them in place while soldering.

Setting the trimmers

The trimmers will be adjusted during the bias phase, which is covered on page 34.

For now, set “Phase” and “Offset” to the 12:00 (center) position and set “Gain” to around 10:00.

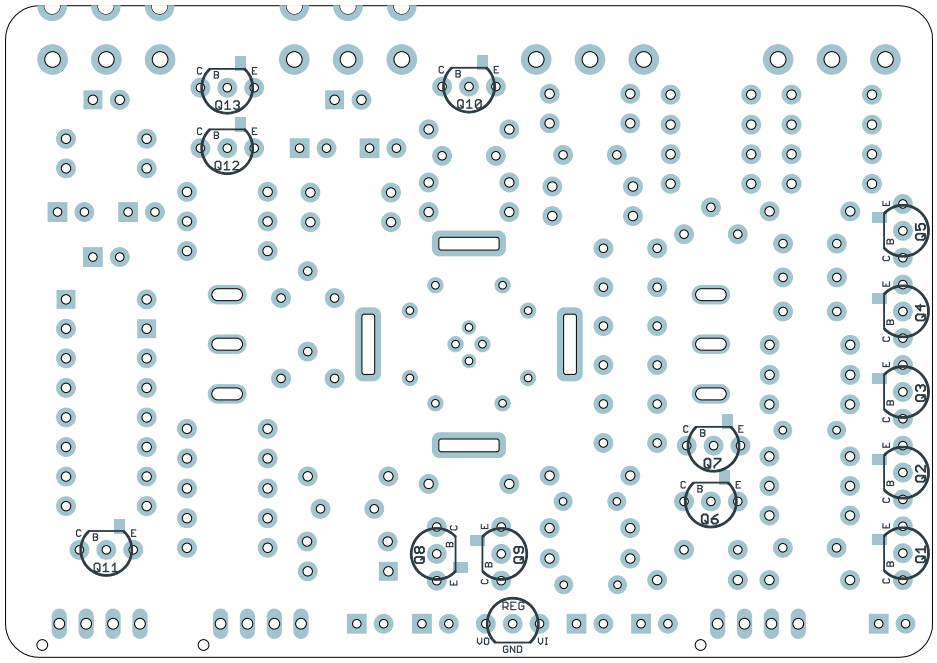
TRANSISTORS

PART	VALUE
Q1	2N5089
Q2	2N5088
Q3	2N5088
Q4	2N5088

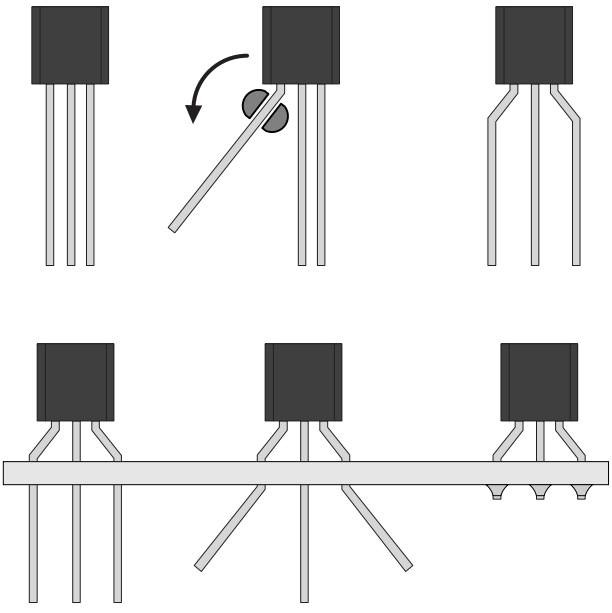
PART	VALUE
Q5	2N5088
Q6	2N5088
Q7	2N5088
Q8	2N5088

PART	VALUE
Q9	2N5088
Q10	2N5088
Q11	MPSA13

PART	VALUE
Q12	MPSA13
Q13	2N5087
REG	78L15



Now we'll do the transistors and regulator. If the legs are not already bent into 0.1" spacing, use your needle-nose pliers to bend the outer two legs as shown in the diagram to the right.



Note the flat side of the transistor and install them according to the orientation shown in the silkscreen.

These transistors do not need to be selected for any particular characteristics, so sockets are not necessary. You can just solder them directly to the board.

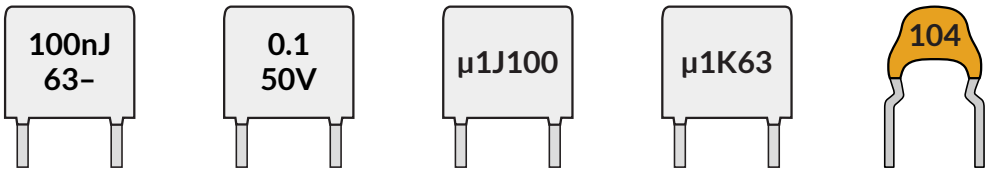
Bend the outer leads to hold the transistor in place on the board. Then, solder and clip the leads. They are all the same height, so as with other components, it's easiest to flip the board over and solder them all at once.

CAPACITOR IDENTIFICATION

Before the next step, we need to provide some information on how to identify the value of a capacitor. All of the supplied capacitors have the value printed on them, but there are multiple ways to format this value and no consistent standard between manufacturers. We use several different brands in our kits depending on availability, so it's not possible to give a reliable indicator of what the marking will be.

In audio circuits, most signal capacitors fall between 1 nanofarad (nF) and 1 microfarad (µF). Anything below that range is always given in picofarads (pF), and anything above is always the µF value. Within that range, the DIY audio community usually uses the nF value, often abbreviated to just "n".

In practice, though, a 100n capacitor may be marked in any of these ways:



All of the text has a purpose, but for this kit, you only need to know the capacitance value and the rest can be ignored. Here are a few basic rules that will help with this.

- **1µF = 1,000nF.** If it doesn't specify µF or nF, then always assume µF.

  0.1 = 0.1µF = 100n
  0.022 = 0.022µF = 22n

- **The "n" or "µ" designator may be used in place of a decimal point.** This is from the days of paper schematics and low-quality copies when a stray speck of ink could cause significant mistakes.

  3n3 = 3.3nF
  µ47 = 0.47 = 470n

- **"J" or "K" and everything afterward can be ignored.** This represents the tolerance (J = 5%; K = 10%) and the maximum voltage. We ensure that all of the capacitors in this kit meet the minimum requirements for the circuit, so you will not need to decode either of these ratings.

  µ1J100 or µ1K63 = µ1 = 0.1 = 100n
  1n0J = 1n0 = 1.0n = 1n

- **For film capacitors, the value may be printed on either the top or the front of the case.** Check the top first, and if it's blank, look at the front. If the value is printed on the top, then the front will have extraneous information such as the maximum voltage, series name, and production date code.

- **Ceramic capacitors (MLCCs) use a separate 3-digit code format.** Since these are often difficult to read without magnification, we don't rely on this code to identify MLCC capacitors.
  - Nearly all kits come with a single yellow 100n MLCC that is used for power filtering.
  - All other MLCCs are taped to cardboard strips.
  - If there is more than one other value in the kit, the values will be written on the cardboard to differentiate them. This value will be in the same format as the parts list, e.g. "100pF".

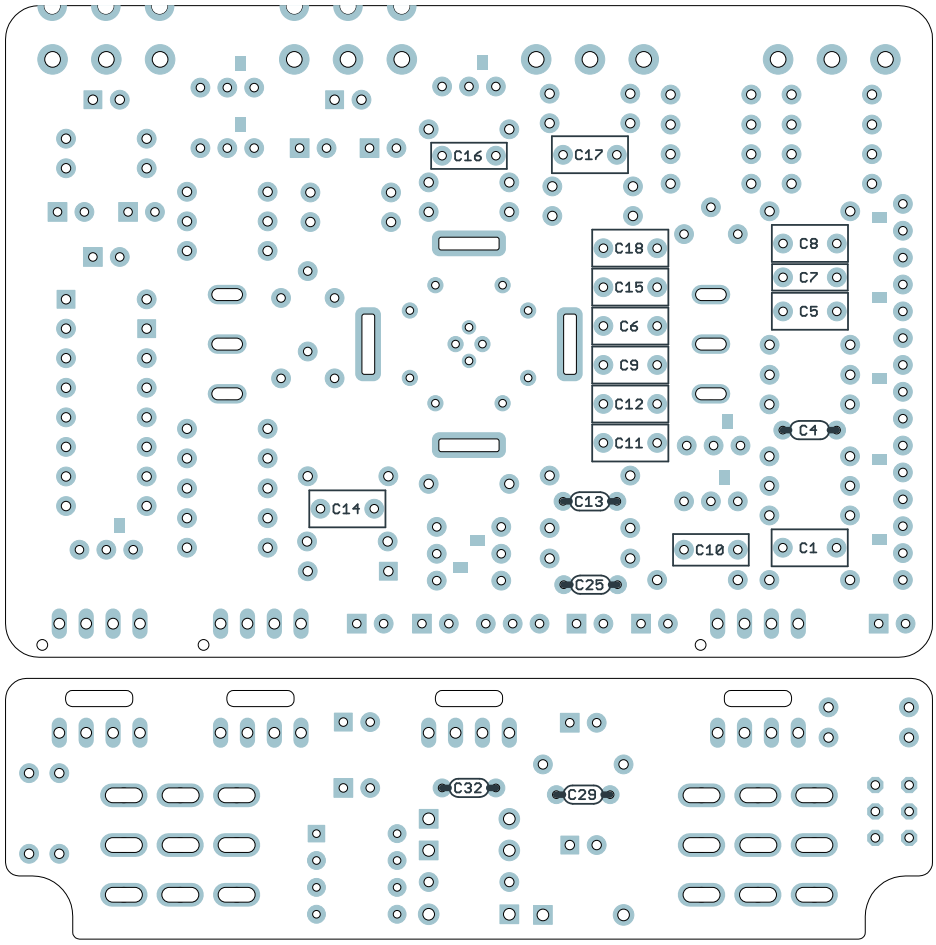
CAPACITORS (NON-POLARIZED)

PART	VALUE
C1	1uF
C4	330pF MLCC
C5	1uF
C6	1uF
C7	15n

PART	VALUE
C8	1uF
C9	1uF
C10	220n
C11	1uF
C12	1uF

PART	VALUE
C13	470pF MLCC
C14	1uF
C15	1uF
C16	4n7
C17	1uF

PART	VALUE
C18	1uF
C25	470n MLCC
C29	470n MLCC
C32	470n MLCC

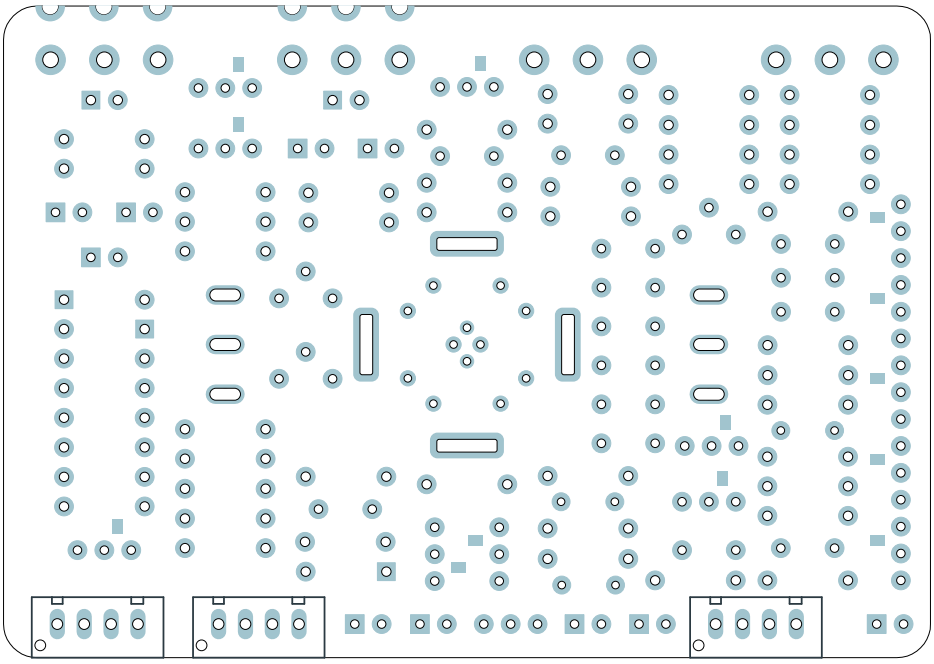


Now you can install the box film and MLCC capacitors. These are all several different heights, but there aren't as many, so just do them all at once. Bend the leads at an angle to hold them in place.

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: For the box film capacitors, the value may be printed on either the top or the side. See previous page for capacitor identification. The MLCCs will be taped to cardboard with the value written on it.

WIRE HEADERS



Install the three 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 another 4-pin header on the I/O board that we will do in a later step.

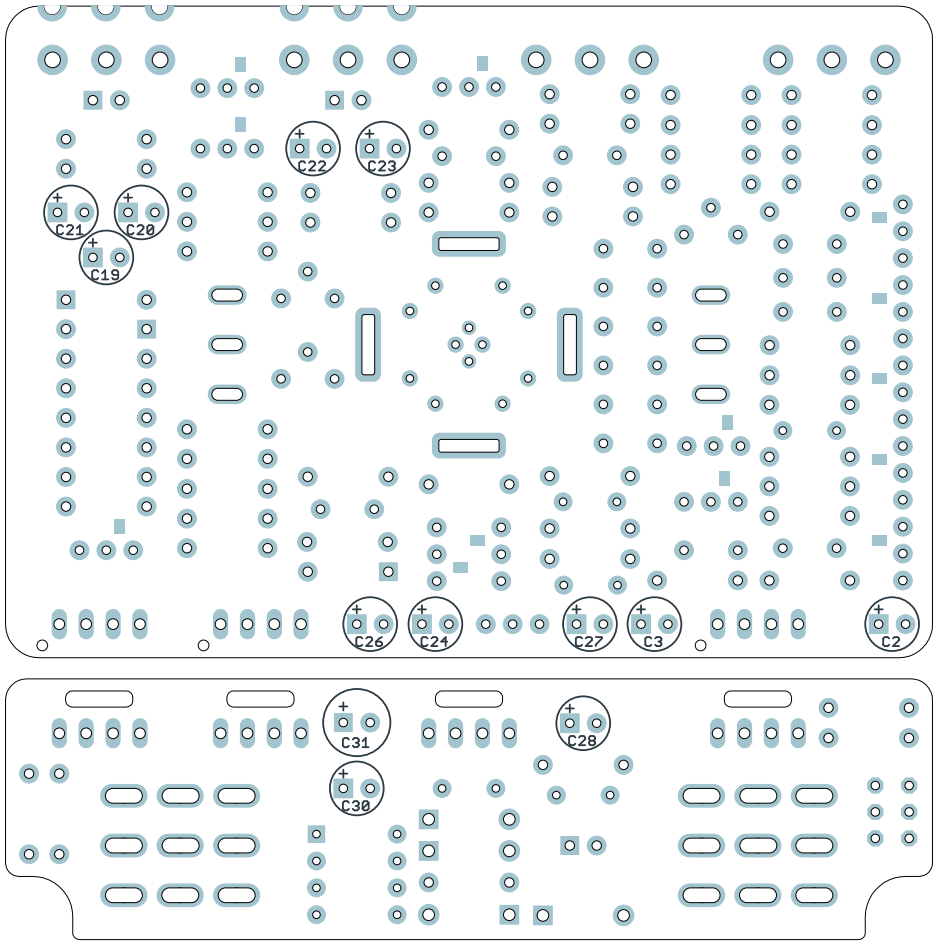
CAPACITORS (POLARIZED)

PART	VALUE
C2	1uF
C3	10uF
C19	1uF
C20	1uF

PART	VALUE
C21	1uF
C22	10uF
C23	10uF

PART	VALUE
C24	47uF
C26	47uF
C27	47uF

PART	VALUE
C28	47uF
C30	10uF
C31	100uF



Populate the electrolytic capacitors. These are the tallest of the standard components, so we save them for last except for the optical components (LDRs, lamp and light shield).

They are polarized, meaning 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.

For the 1uF electrolytic capacitors, the value may be hard to read, so they can be identified by size. They are significantly smaller than the other three types.

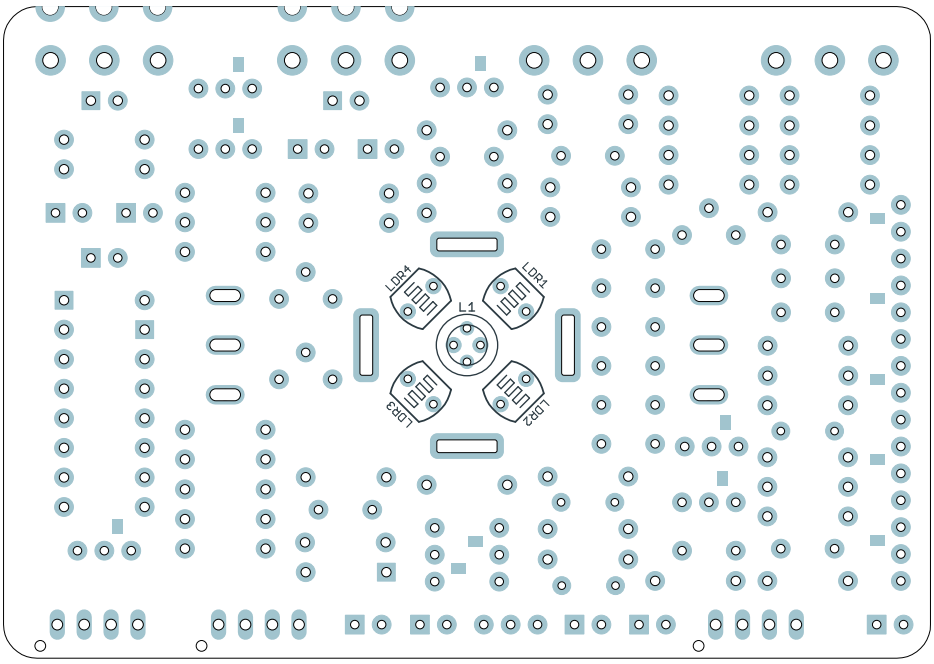
Before moving onto the optical components, it's recommended to go back to page 13 and insert the IC (LT1054) into the socket on the footswitch PCB.

LAMP AND LDRS

PART	VALUE
LDR1	PDV-P9203
LDR2	PDV-P9203

PART	VALUE
LDR3	PDV-P9203
LDR4	PDV-P9203

PART	VALUE
LAMP	2174



The LDRs and lamp are the last of the components on the main PCB. All five pieces are unmarked, but see page 7 for illustrations if you're not sure which is which.

Install the four LDRs (photocells) in the positions marked LDR1-LDR4. They are all identical, so it doesn't matter which one goes in which spot. The positioning doesn't really matter due to the reflective light shield, but we recommend installing them about 1/4" (6mm) above the PCB, facing straight up. Some Uni-Vibe clones point them inward directly toward the lamp, and this is fine too.

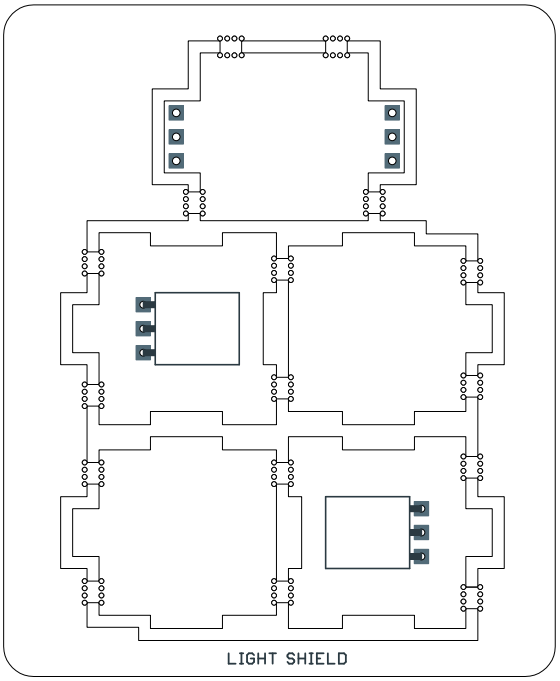
Next, install the lamp into the space in the center marked "L1". Use the left and right pads and ignore the pads on the top and bottom. The leads are very thin and the body is glass, so use caution when handling.

Incandescent lamps are not polarized, so the orientation does not matter, but install it as close to the PCB as possible so it can't move around.

LIGHT SHIELD

The original Uni-Vibe enclosed the lamp and LDRs in a reflective sheet-metal box that dispersed the light internally so that all four LDRs were evenly exposed. It also blocked most outside light so the effect would still work when the enclosure lid was removed for service.

For our implementation of the Uni-Vibe, we designed a solution that uses interlocking PCBs to build a box that can be soldered in place. The inside is reflective, and the top is removable. It comes in a single panel with five sub-PCBs that can be easily separated.



The following parts are included for the light shield:

QTY	PART
1	Light shield PCB (panelized)
2	Pin header, right angle, female
2	Header strip, 3 pin

Assembly instructions

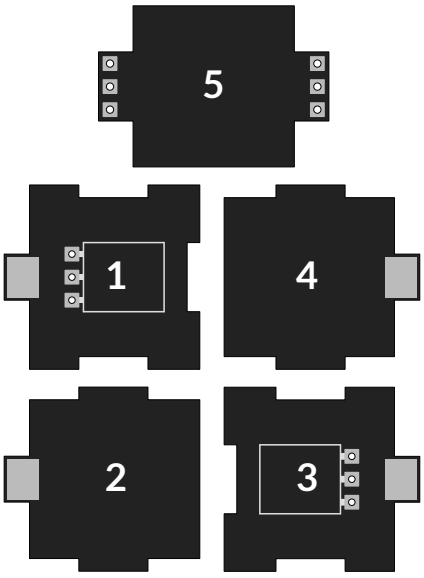
The light shield is straightforward to put together, but there are a few ways to mess it up, so make sure to read all of the instructions before you start soldering.

Before you begin

Separate all five of the PCBs from the frame. It's easiest to push from the center so that they snap and swing outward. The outer frame is flexible, so they should come out without much force.

Then, using needlenose or flat-nose pliers, snap off the mouse-bite tabs from each PCB.

Try to avoid touching the plated reverse side of the panel since fingerprints will diminish the reflectivity.



Panel identification

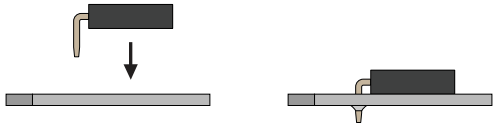
We will refer to the individual panels PCBs using the numbers shown to the right. Note that panels 1 & 3 are identical, as are panels 2 & 4, so don't worry if you lose track of which is which.

LIGHT SHIELD, CONT.

Step 1: Solder the female headers

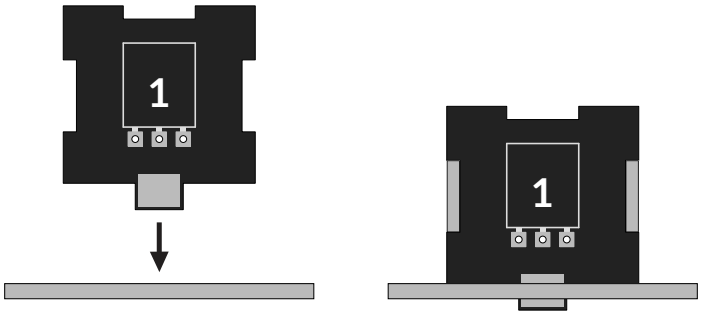
Panels 1 and 3 each have silkscreen outlines showing right-angle female headers. These should be installed to the black side. You can solder them from either the top or the bottom side of the PCB.

It's recommended to solder the middle pin first, then check to make sure it's aligned with the silkscreen. If not, heat up the joint and shift it accordingly. Only solder the other two pins if it's aligned or else it will be hard to correct later.



Step 2: Install panels 1-4 to the PCB

Fit the four panels together, then fit them loosely into the PCB to test the fit. The tab that is plated on both sides fits into the slot on the PCB. It's recommended to use a rubber band around the perimeter to hold the four panels together during this step.



Important: Make sure panels 1 & 3 are the "north" and "south" panels, with 2 & 4 facing "east" and "west", otherwise the header sockets won't have enough clearance due to the trimmers on the left and the capacitors on the right. The Straylight PCB layout intentionally uses resistors to the north and south of the box outline to leave space for the headers.

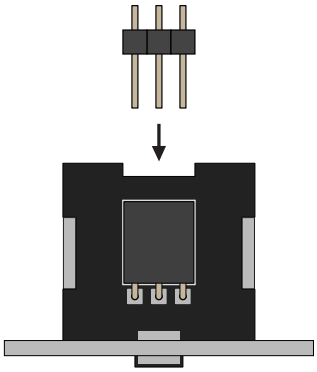
Step 3: Solder the tabs

Once the panels are interlocked and straight, you can flip the whole PCB upside down and solder the tabs from the bottom. They are plated on both sides, so solder as much as you can and make sure it flows into the slot.

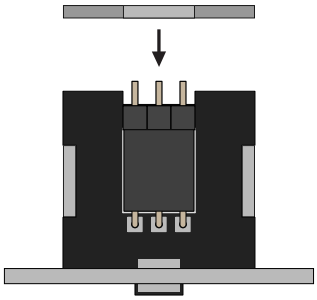
The solder won't go all the way around due to the unplated PCB material on the edges, but the connection will still be very rigid. You can also add some solder from the component side of the PCB if you want to make it even stronger.

Step 4: Solder the header pins to the lid (panel 5)

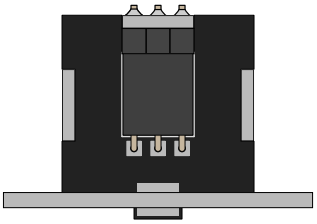
Follow the sequence below to install the header pins to the lid. By doing it this way, you can ensure that the pins are perfectly aligned with the header sockets.



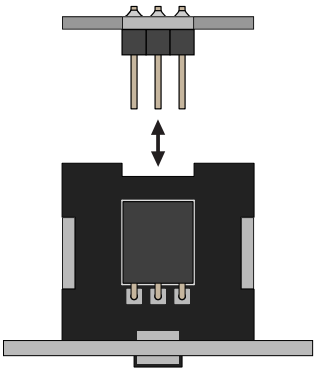
4a: Press the header pins all the way down into the sockets.



4b: Fit the lid (panel 5) onto the pins, ensuring the fit is tight and it is level with the top of the box.



4c: Solder the pins from the top.



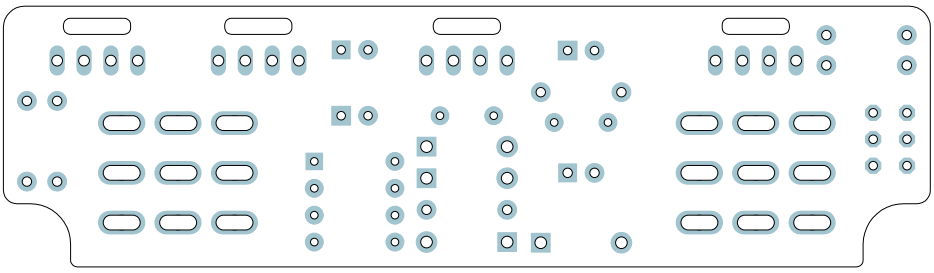
4d: Remove the lid and reinstall to test the fit.

That's it! Now you have a fully reflective light shield around the lamp and LDRs, and the lid is easily removed if you want to experiment with different components or if you need to troubleshoot.

FOOTSWITCH PCB

PARTS

- 4-strand wire assembly, 60mm (3)
- 4-strand wire assembly, 122mm (1)



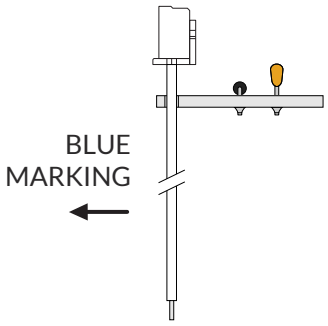
Next, it's time to finish up the footswitch board. You should have done most of the on-board components on this board in a previous step, but if not, go back and do those.

There will be one long assembly and three shorter ones, all 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.

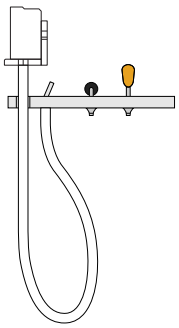
For now, pull it down through the slot as far as it can go.



STEP 2

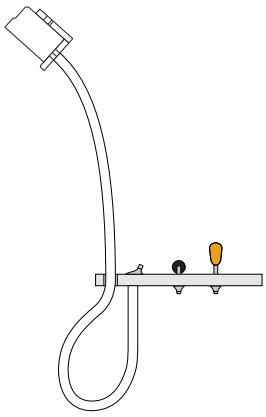
Next, bend the wires back upward and fit the ends of the wires into the solder pads.

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

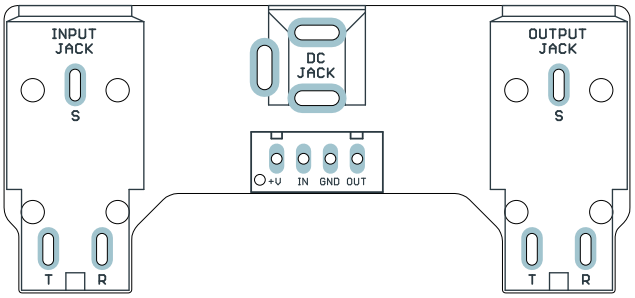
Then, solder the wires from the top. This is the trickiest part of the whole build. You want to solder the pads without touching the iron to the wires themselves and risking burning through the insulation. It helps to use a sharp or narrow tip on the soldering iron.



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

INPUT/OUTPUT PCB

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

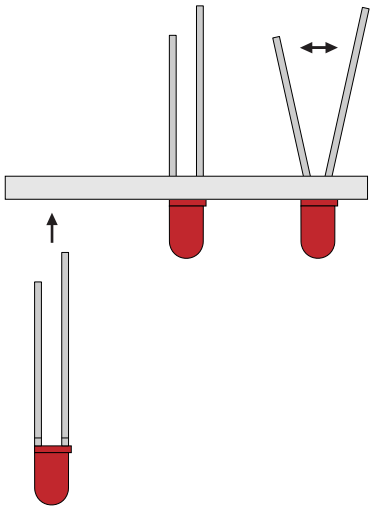


Next comes the input/output PCB. Find the two input/output jacks, the DC jack and the wire header and snap them in place. The PCB is designed for them to fit securely, so you can do them all at once before flipping and soldering.

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.

POSITIONING THE LEDS

Before soldering the hardware on the following pages, insert the three LEDs as shown in the diagram to the right. Bend the legs outward slightly to hold them in place for the next few steps.

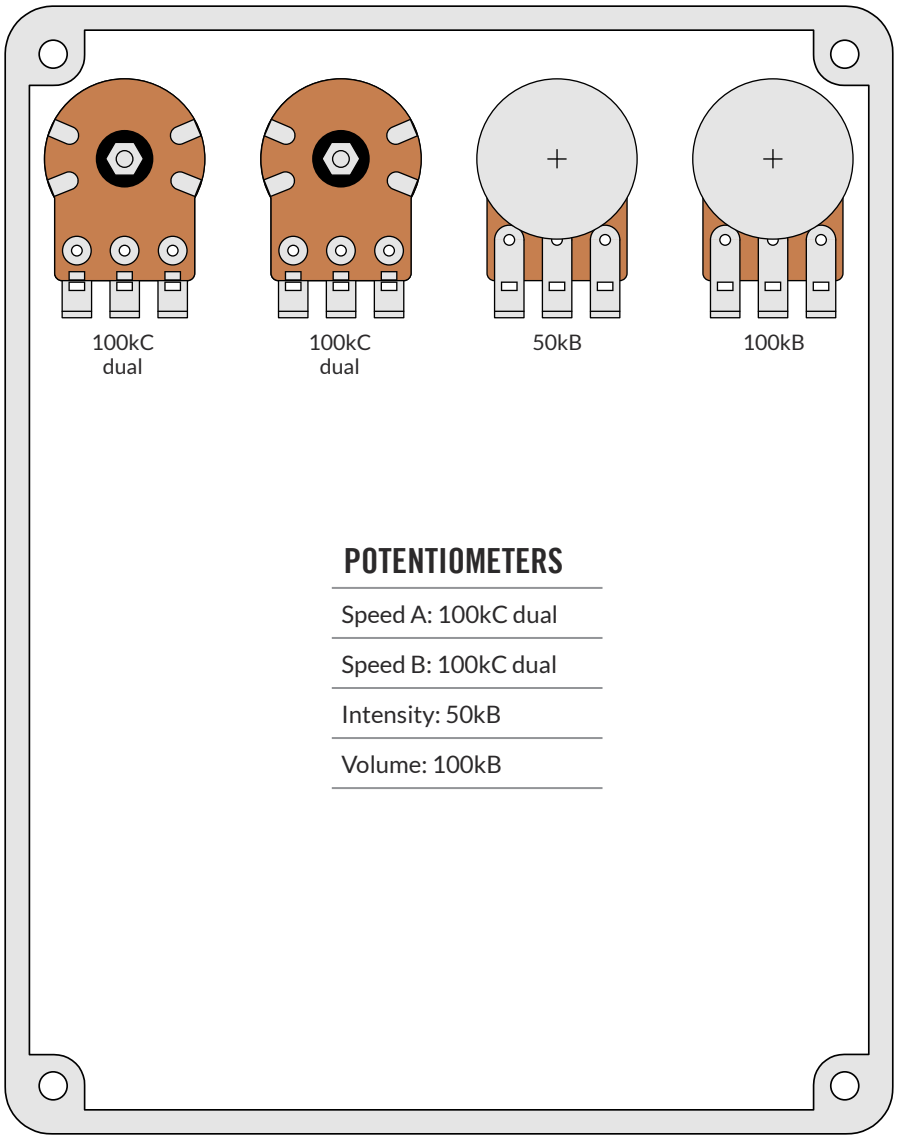


Important: Do not solder these until page 30! For now, we just want to hold them in place so they can be easily positioned after the PCB is soldered inside the enclosure.

D7 and D8 are small red LEDs on the main board, and the larger white indicator LED is on the footswitch PCB next to the bypass switch.

The long leg is the anode (A) and the short leg is the cathode (K). For all three LEDs, the long leg goes in the square pad and the short leg goes in the round pad.

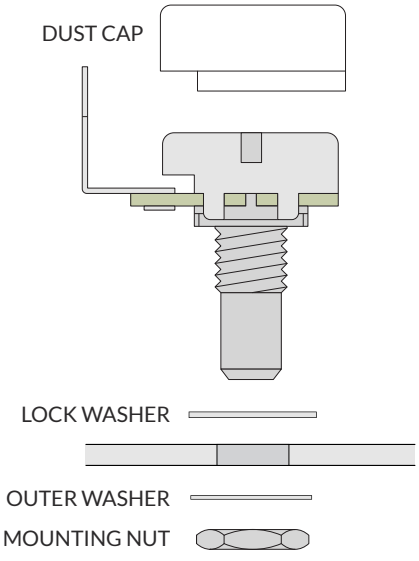
ENCLOSURE LAYOUT: POTENTIOMETERS

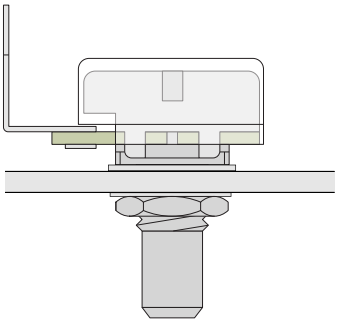


POTENTIOMETERS
Speed A: 100kC dual
Speed B: 100kC dual
Intensity: 50kB
Volume: 100kB

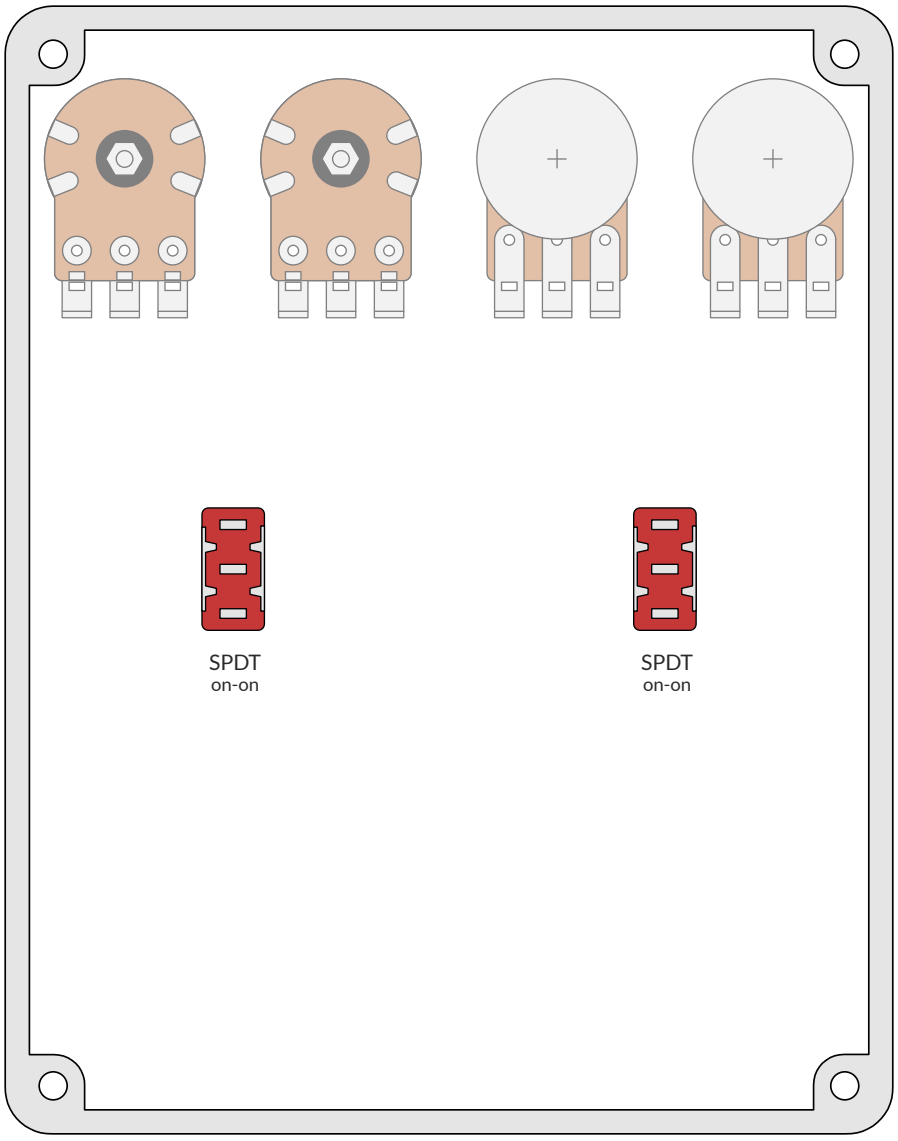
Attach the potentiometers to the enclosure as shown, using the diagram to the right as a guide. Make sure they're aligned as straight as possible, then tighten the outer nut firmly. It can be helpful to use a second set of pliers to hold the potentiometer in place from the inside while tightening the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

Note that the dust caps are only used for the single potentiometers, not the two duals.





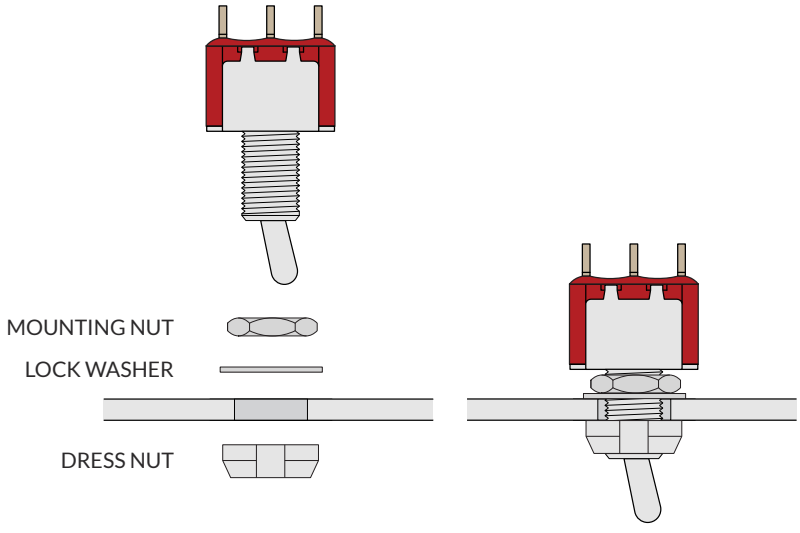
ENCLOSURE LAYOUT: TOGGLE SWITCHES



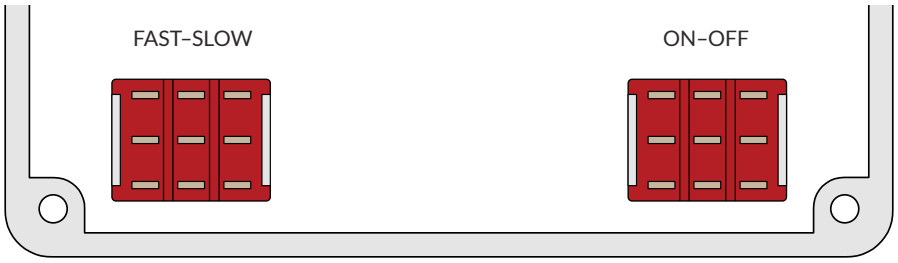
Attach the two toggle switches as shown, using the diagram to the right as a guide.

Both toggles are the same type. 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.

Use flat-nose pliers on the flat sides of the dress nut to tighten securely.



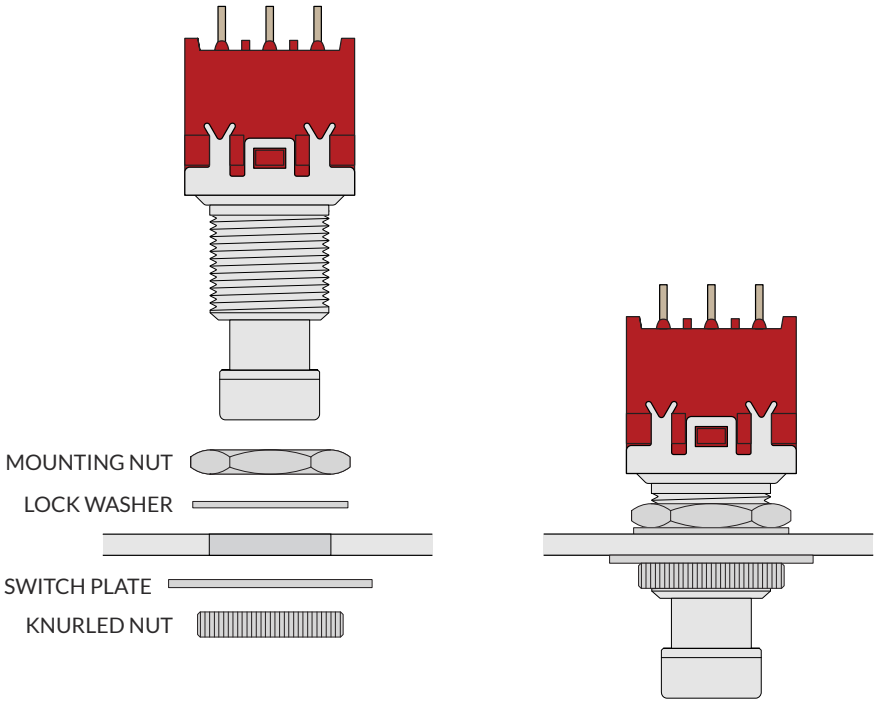
ENCLOSURE LAYOUT: FOOTSWITCHES



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

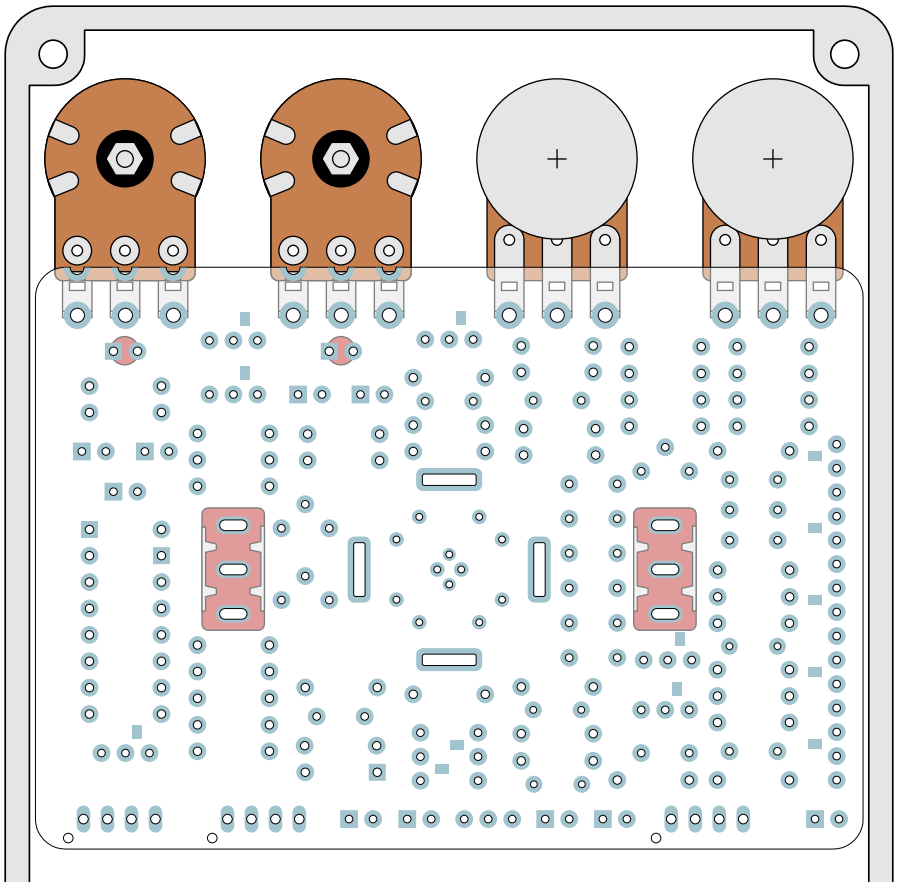
Each footswitch has a corresponding switch plate which indicates the footswitch's function and serves as a washer. The plates have a tab that matches the notch in the footswitch's threaded shaft. These notches should be oriented upward (i.e. toward the potentiometers) when the footswitch is installed so that the label is right side up.

The "Fast–Slow" plate should be affixed to the footswitch on the left (relative to the diagram above, an inside view of the enclosure), and the "On–Off" plate goes with the switch on the right. This will be inverted on the face of the enclosure so that "On–Off" is on the left and "Fast–Slow" is on the right.



The knurled nut should be tightened by hand as much as possible, then finished with needle-nose pliers with serrated teeth. This won't provide as much of a grip as the flat edges of a standard nut, so be careful not to squeeze too hard or over-tighten.

ENCLOSURE LAYOUT: MAIN PCB



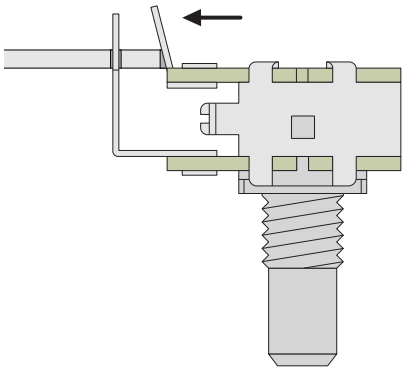
After all the hardware is affixed to the enclosure as shown on the previous pages, place the main PCB on top of the potentiometers and switches as in the diagram above, with the component side facing upward. You may need to adjust the position of the hardware slightly if they don't line up with the holes.

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.

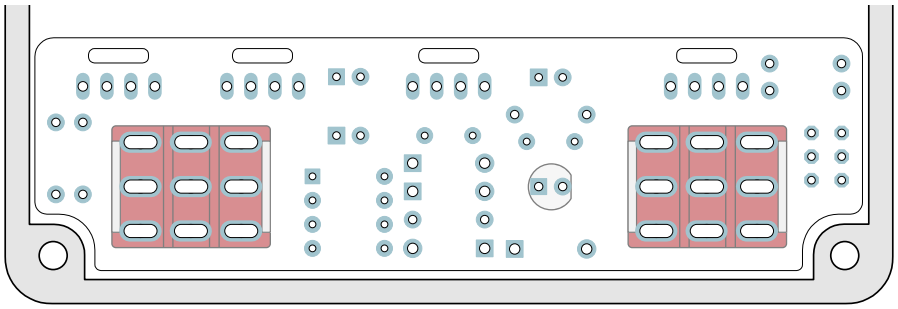
Soldering the dual-gang potentiometers

The upper pads for the dual-gang gain potentiometers appear to be cut in half on the PCB. **This is intentional.** They're called *plated half-holes* or *castellated holes*, and they are used so that the PCB can lay flat across the pots instead of angling upward for the dual pots.

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



ENCLOSURE LAYOUT: FOOTSWITCH PCB



Now, do the same with the footswitch PCB. Place the PCB on top of the two switches and the legs of the LED. The components and wire assemblies should face up.

Once the PCB is laying flat, solder the lugs of each of the footswitches. Take extra care not to touch the wires with the iron since the plastic insulation will melt quickly.

Soldering the LEDs

After the pots and switches have been soldered, press down on the legs of the two red LEDs in the upper-left of the PCB until they poke through the small holes on the face of the enclosure. The outward tension of the legs will hold them in place. Press them all the way down until they stop, and then pull back up slightly, around 1/16th of an inch. (This is purely for cosmetics, but it looks better this way.)

Now, do the same with the white bypass indicator LED, again pulling it up slightly before soldering.

Why solder everything inside the enclosure before testing it?

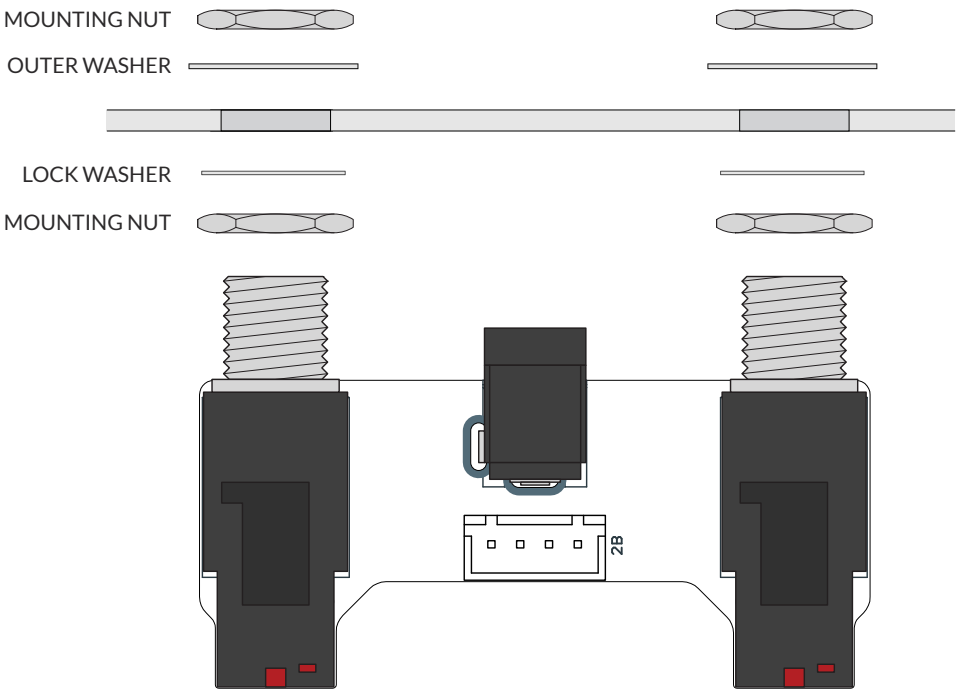
“Rock it before you box it” is conventional wisdom in pedalbuilding, and you’ll often hear it recommended that builders should test the circuit before putting everything inside the enclosure.

Aion FX projects are designed to be easy to remove from the enclosure for troubleshooting, with no desoldering required. As a result, it’s much simpler 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.

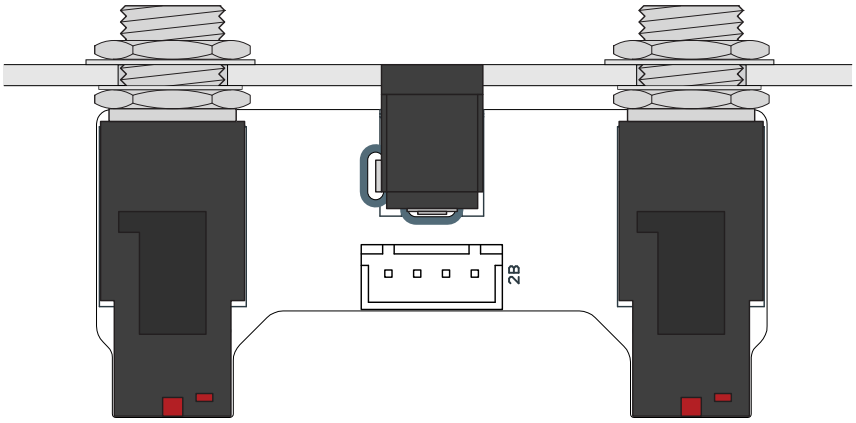
If you’ve read the documentation carefully and followed all the instructions, there’s a good chance you will get it right the first time, but if there are any issues then it only takes a minute or two to disassemble, and it goes back together just as easily.

ENCLOSURE LAYOUT: INPUT/OUTPUT PCB

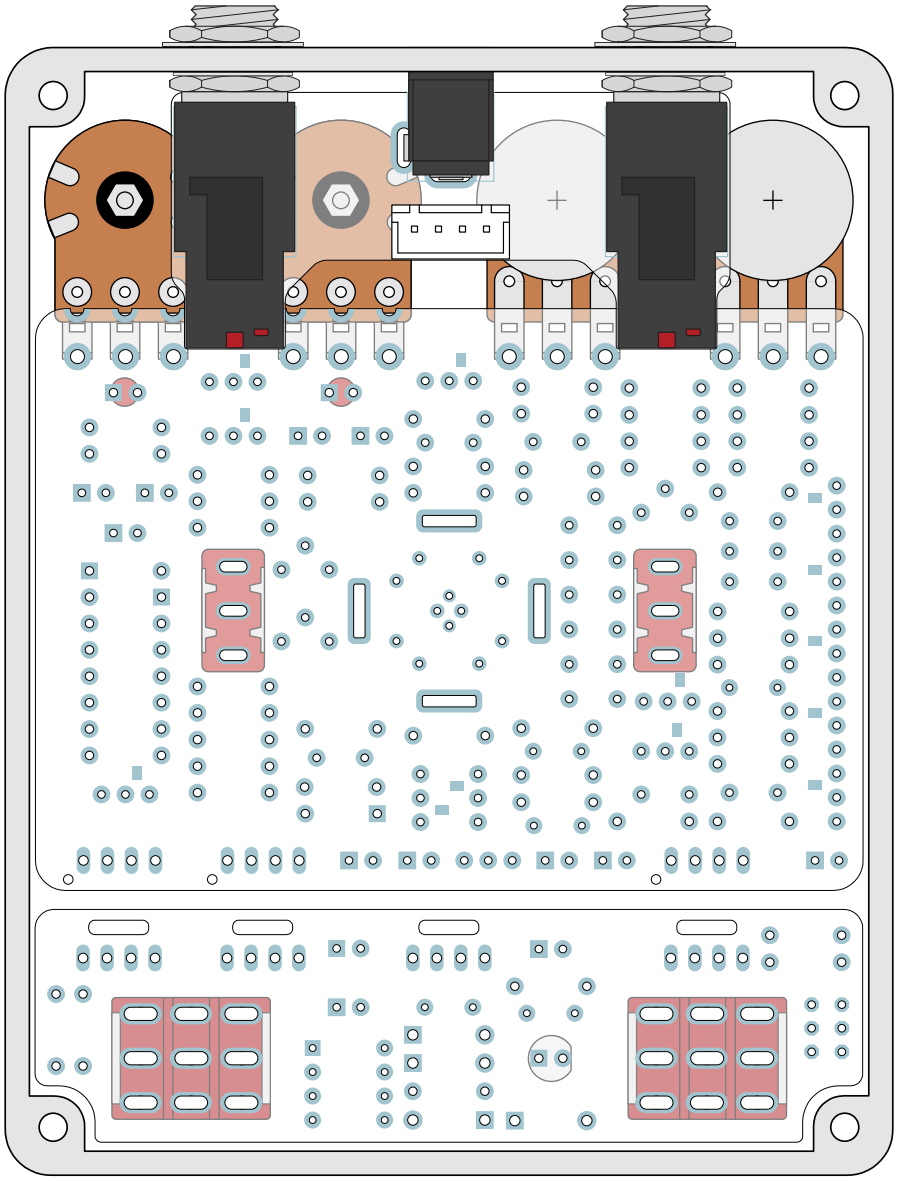


Affix the input/output PCB to the north-facing panel of the enclosure as shown.

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.

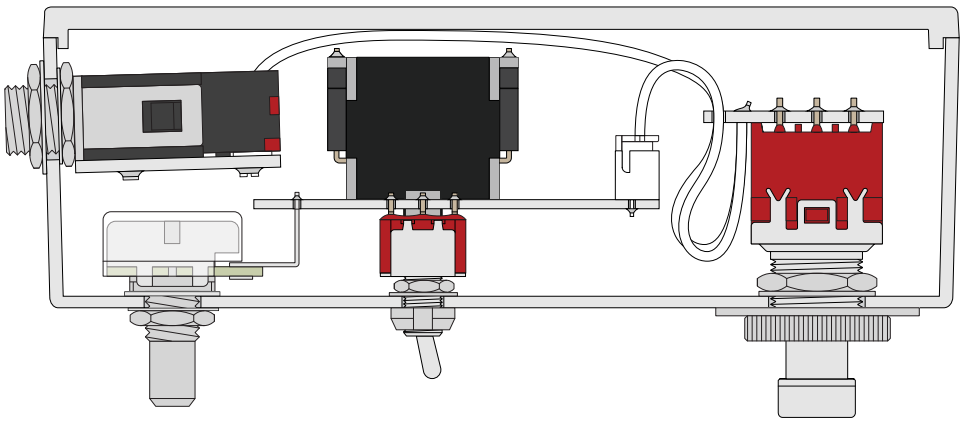


ENCLOSURE LAYOUT: FINISHED DIAGRAM



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

Now, just plug the 4 wire assemblies into their respective headers and make sure they're secure. Here is a cross-section of the inside of the completed pedal.



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

Test the bypass switch a few times, then leave it in effect mode. With the enclosure lid open, volume up, and LFO switch off, try waving your hand in front of the light shield. You should hear a change in sound. (You can remove the lid of the light shield if you want, but enough light comes through the edges of the shield that the effect should still be noticeable with the lid on.)

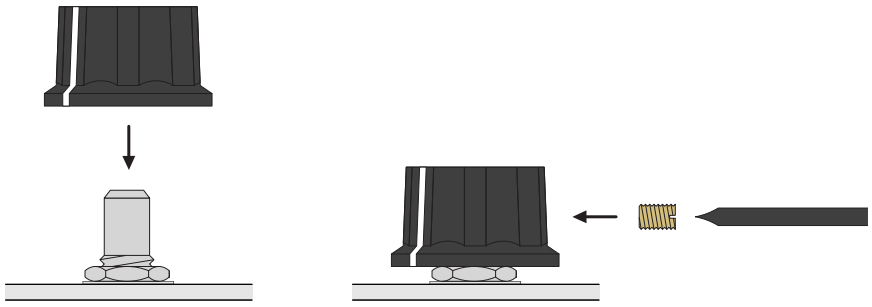
This confirms that the audio path is functioning properly. If it doesn't sound right, jump to page 41 and troubleshoot the audio path before moving onto the biasing on the following pages. We'll evaluate the remaining functionality as part of the biasing process.

Installing the knobs

Even though we're not finished, it's helpful to have the knobs installed for the calibration process. Turn the potentiometer 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.

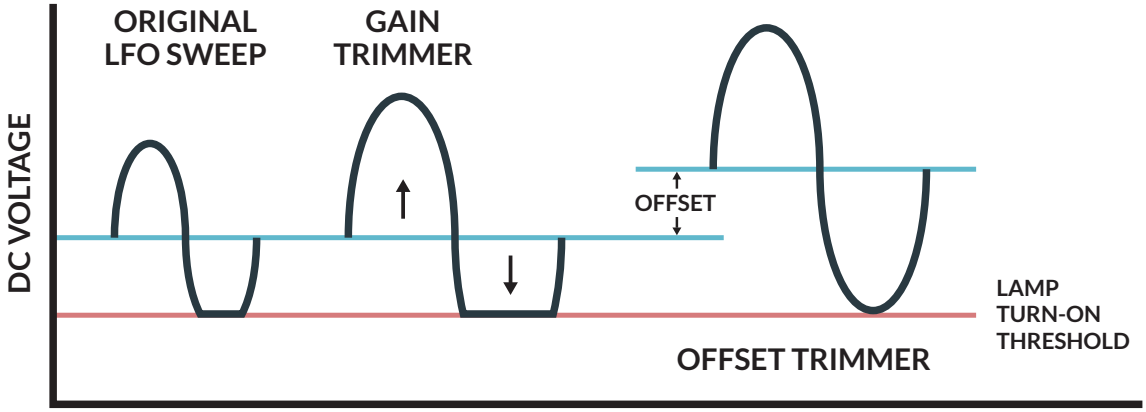
Be careful not to over-tighten or you may damage the set screw. But if it's not tight enough, the knob will be more likely to fall off or lose its alignment with the markings on the enclosure.



SETTING THE BIAS

There are two trimmers on the PCB used to calibrate the circuit. “Gain” and “Offset” directly control the lamp brightness and behavior and can be used to compensate for variations in lamps and LDRs.

The behavior of these two trimmers is illustrated in the following diagram:

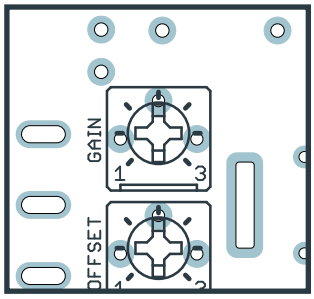


The **Gain** trimmer adjusts the overall amplitude range of the lamp’s sweep, making it brighter at the top end of the sweep and darker at the bottom.

The lamp has a fixed voltage threshold where it turns on, so the **Offset** trimmer is used to shift the range upward so that it only turns off at the very bottom of the sweep. If Gain is adjusted again, Offset will likely need to be as well.

The next sections discuss the function of each trimmer in more detail.

Gain trimmer



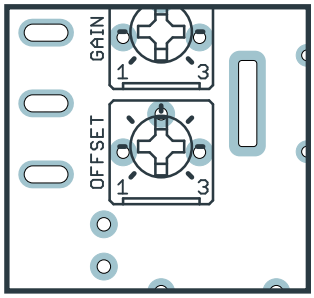
This sets the gain of the lamp driver. As you turn it up, the lamp will increase in average brightness.

- Start with this trimmer at 12:00 rotation (50%). Ensure the “Offset” and “Phase” trimmers are at 12:00.
- Turn the “Intensity” knob all the way down, then adjust the trimmer until the lamp is dim but glowing. In the units we’ve built, the ideal range is typically around 3-4:00.
- Then, turn “Intensity” all the way back up and listen.

Note that even with the light shield installed, a small amount of light will still find its way in if you are testing in direct light with the enclosure lid removed, and this may affect the sound. It’s recommended to replace the enclosure lid loosely when listening.

There’s no target voltage range or calibration spec, so just use your ears and leave it where it sounds best to you.

Offset trimmer



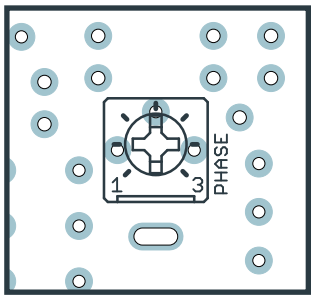
The Offset trimmer adjusts the DC bias point of the LFO going into the lamp driver. Lamps have a fixed turn-on threshold voltage, and this threshold may differ from lamp to lamp. This trimmer allows the LFO voltage range to be adjusted until the lamp cuts off at the very bottom.

With the "Speed" control set low (whichever of the two knobs is currently active), start with the trimmer at around noon and then adjust until the lamp goes dark at the bottom of the sweep.

The goal is to have the biggest dynamic range between the top and bottom of the sweep. Be aware that due to the thermal properties of the lamp, it may not be able to get fully dark on faster speeds, which is why it's best to calibrate at the slowest settings.

The Gain and Offset trimmers are interactive with each other, so you may need to go back and forth a few times between the two adjustments before you get it right.

Phase trimmer



This Phase trimmer does not impact the lamp's behavior, but sets the balance of phased signal to dry signal when in Chorus mode. For best phasing, these two signals should be as close in amplitude as possible, but due to component tolerances and other factors, they don't always line up optimally.

This should be the last trimmer you set, after the other two are complete.

- Set it to 12:00 (50%) to start with.
- Set the "Mode" switch to "Chorus" and set the "Intensity" control to full.
- Then adjust this trimmer to the left and right while listening for an improvement in the phasing. Leave it where the phased sound is deepest. If you don't notice any difference, just leave it at 12:00.

This trimmer modification was originally developed by the DIY scene in the late 1990s when 5% resistors were standard. Today, pedalbuilders can easily find 1% resistors for the same price or cheaper, and in fact our kits use 0.5% resistors. This means that there is much less potential for variation between the dry and modulated signals and usually the 12:00 position is best.

USAGE

The Straylight has four knobs:

- **Speed 1** and **Speed 2** set the speed of the modulation cycle, with a footswitch to select between the two settings. Both knobs have the exact same range. The transition is smooth, and if there's enough difference in the two speed settings, there's a slight ramping effect.
- **Intensity** sets the depth of the modulation effect.
- **Volume** is the output volume of the effect.

There are also two toggle switches:

- **Chorus/Vibe** selects between vibrato mode (effect only) and chorus mode (effect mixed with the dry signal). Vibe mode works best on faster speeds, since the slow phase shift by itself is not as noticeable without the dry signal as a reference point.
- **Cancel** disables the lamp modulation, but still allows the signal to pass through all of the static phase stages. This is how the original Uni-Vibe was bypassed, and it does provide a subtle coloration to the tone compared to standard true bypass.

Lastly, the **Input Level** slide switch on the footswitch PCB selects between Vintage and Modern mode. The original Uni-Vibe had a very significant volume drop at the input, cutting the signal by 1/3, which was responsible for the weak output.

The "Modern" mode eliminates this volume drop, but if you want to replicate what is usually considered a design flaw, then "Vintage" mode will behave exactly like the original.

Power supply

The Straylight should only be powered with a standard 9V adapter. It has an internal voltage doubler so that it runs on 18V internally. Do not try to run it on a higher voltage!

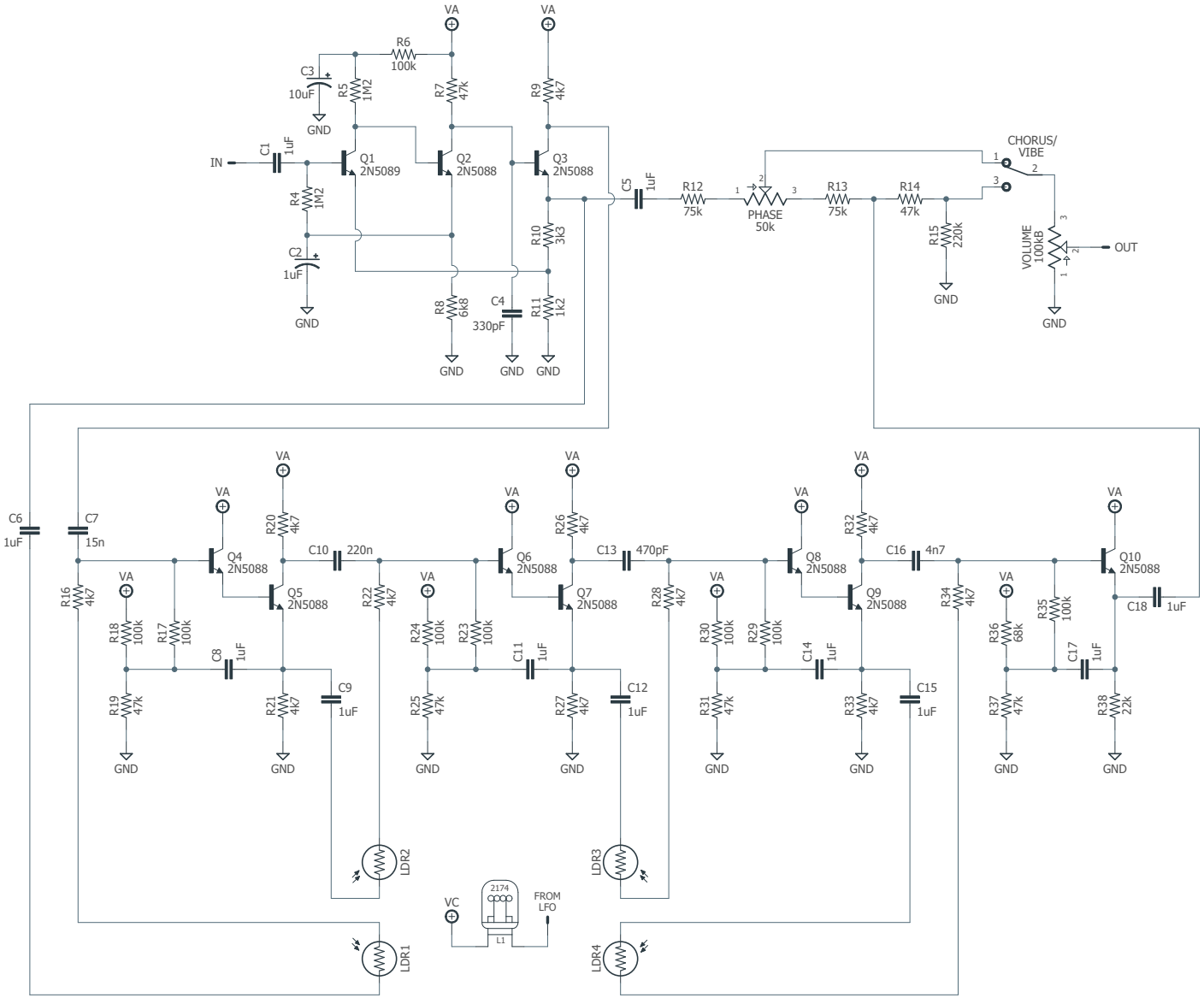
If you accidentally use 12V, e.g. from a Voodoo PP2+, it will most likely not cause any immediate damage, but it will stress the components and could lead to early failures of the power supply circuitry. But 15V or 18V will burn out the protection diode and could damage many other components in the circuit.

Additionally, since the lamp runs in current mode, the overall current draw of the pedal will fluctuate with the lamp brightness. This can cause issues with unregulated power supplies since the voltage is dependent on the current draw. It's strongly recommended to use a regulated power supply so that the voltage is stable. (Most supplies these days are regulated, but many older or low-quality ones are not.)

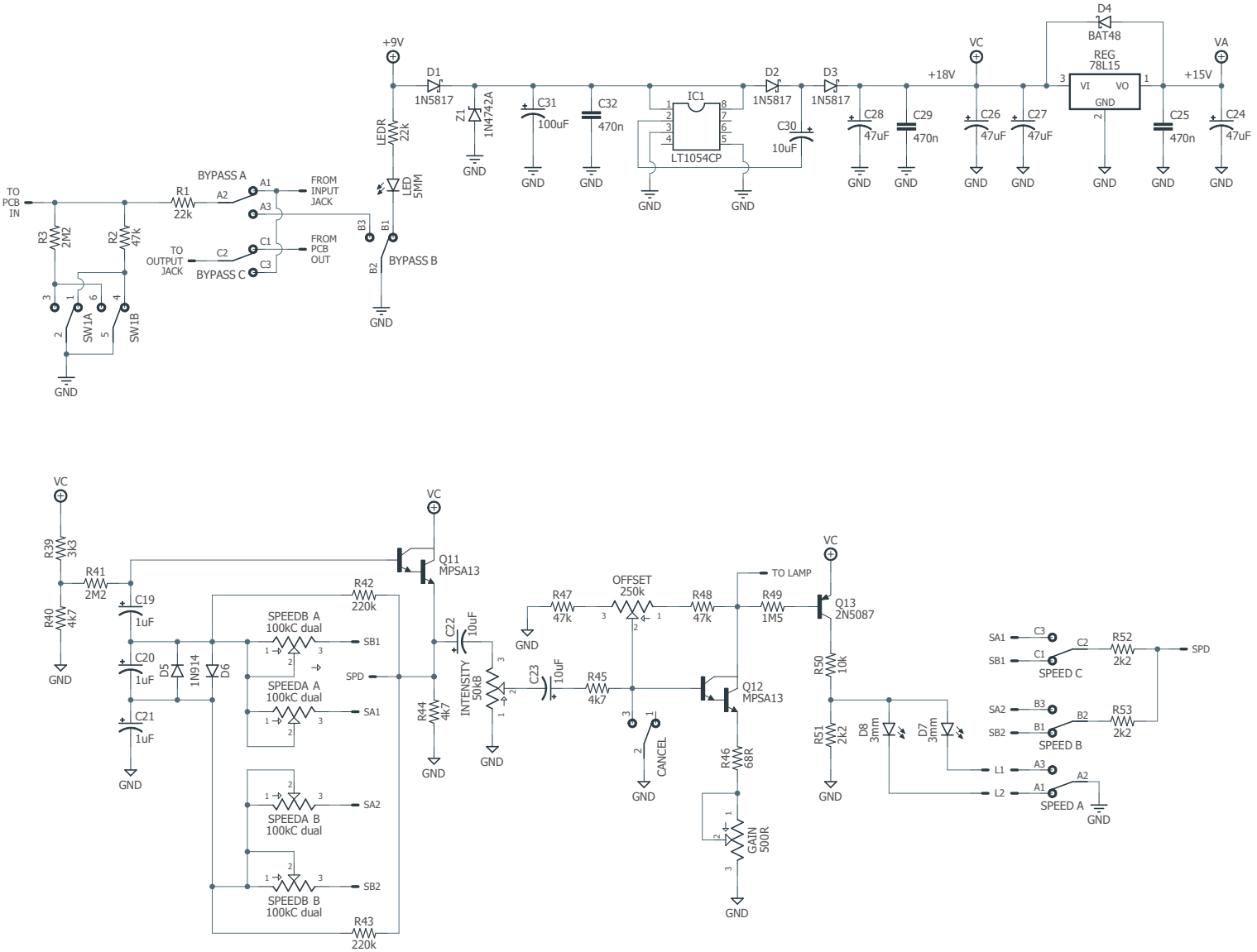
Cancel switch

If you routinely leave your pedals plugged in for days or weeks, it's recommended to engage the Cancel switch when not playing. This will dramatically extend the life of the lamp.

SCHEMATIC (AUDIO PATH)



SCHEMATIC (POWER, LFO & BYPASS)



FULL PARTS LIST

Resistors

PART	VALUE
R1	22k
R2	47k
R3	2M2
R4	1M2
R5	1M2
R6	100k
R7	47k
R8	6k8
R9	4k7
R10	3k3
R11	1k2
R12	75k
R13	75k
R14	47k

PART	VALUE
R15	220k
R16	4k7
R17	100k
R18	100k
R19	47k
R20	4k7
R21	4k7
R22	4k7
R23	100k
R24	100k
R25	47k
R26	4k7
R27	4k7
R28	4k7

PART	VALUE
R29	100k
R30	100k
R31	47k
R32	4k7
R33	4k7
R34	4k7
R35	100k
R36	68k
R37	47k
R38	22k
R39	3k3
R40	4k7
R41	2M2
R42	220k

PART	VALUE
R43	220k
R44	4k7
R45	4k7
R46	68R
R47	47k
R48	47k
R49	1M5
R50	10k
R51	2k2
R52	2k2
R53	2k2
LEDR	22k

Capacitors

PART	VALUE
C1	1uF film
C2	1uF electro
C3	10uF electro
C4	330pF MLCC
C5	1uF film
C6	1uF film
C7	15n film
C8	1uF film

PART	VALUE
C9	1uF film
C10	220n film
C11	1uF film
C12	1uF film
C13	470pF MLCC
C14	1uF film
C15	1uF film
C16	4n7 film

PART	VALUE
C17	1uF film
C18	1uF film
C19	1uF electro
C20	1uF electro
C21	1uF electro
C22	10uF electro
C23	10uF electro
C24	47uF electro

PART	VALUE
C25	470n MLCC
C26	47uF electro
C27	47uF electro
C28	47uF electro
C29	470n MLCC
C30	10uF electro
C31	100uF electro
C32	470n MLCC

FULL PARTS LIST, CONT.

Diodes

PART	VALUE
Z1	1N4742A
D1	1N5817
D2	1N5817
D3	1N5817
D4	BAT48

PART	VALUE
D5	1N914
D6	1N914
D7	3mm LED, red
D8	3mm LED, red
LED	5mm LED, white

IC & Socket

PART	VALUE
IC1	LT1054CP
IC1-S	DIP-8 socket

Trimmers

PART	VALUE
GAIN	500R
OFFSET	250k
PHASE	50k

Lamp & LDRs

PART	VALUE
L1	2174 lamp
LDR1	PDV-P9203
LDR2	PDV-P9203
LDR3	PDV-P9203
LDR4	PDV-P9203

Potentiometers

PART	VALUE
Speed A	100kC dual
Speed B	100kC dual
Intensity	50kB
Volume	100kB

Switches

PART	VALUE
3PDT stomp (2)	
DPDT slide switch	

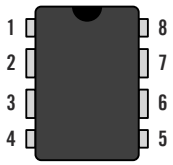
TROUBLESHOOTING INFORMATION

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

Voltages

The following voltages are taken from our prototype unit using a **9.60V** supply. Your measured voltages won't be exactly the same due to variance in power supplies, bias and component tolerances.

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



IC1

PIN	VOLTAGE
1	9.30
2	4.99
3	0
4	0
5	0
6	2.54
7	1.41
8	9.30

Q1

PIN	VOLTAGE
C	2.08
B	1.41
E	1.01

Q2

PIN	VOLTAGE
C	4.33
B	2.08
E	1.51

Q3

PIN	VOLTAGE
C	11.02
B	4.33
E	3.74

Q4

PIN	VOLTAGE
C	14.92
B	4.72
E	4.34

Q5

PIN	VOLTAGE
C	11.17
B	4.34
E	3.75

Q6

PIN	VOLTAGE
C	14.92
B	4.72
E	4.34

Q7

PIN	VOLTAGE
C	11.16
B	4.34
E	3.76

Q8

PIN	VOLTAGE
C	14.92
B	4.72
E	4.34

Q9

PIN	VOLTAGE
C	11.16
B	4.34
E	3.76

Q10

PIN	VOLTAGE
C	14.92
B	5.96
E	5.45

Q11

PIN	VOLTAGE
C	~17-17.5V
B	~6-10V
E	~7-11V

Q12

PIN	VOLTAGE
C	~8-10V
B	~4-5V
E	~3-4V

Q13

PIN	VOLTAGE
C	~11-13V
B	~16-17V
E	~17-17.5V

REG

PIN	VOLTAGE
VI	~17-17.5V
GND	0
VO	14.92

Voltages marked with a ~ are moving voltages that will change as the LFO oscillates. The exact ranges are not particularly important. The main thing to check is that they are moving along with the Speed setting.

TROUBLESHOOTING INFORMATION, CONT.

Along with the general troubleshooting guide linked on the previous page, there are a few other troubleshooting steps that are specific to this circuit.

Voltage variances

Many of the voltages on the previous page, particularly those marked with ~ in the bottom row, are highly dependent on the bias settings and knob positions. If they are outside the listed range, it is not necessarily an indicator of an issue. As long as they're moving, they're telling us what we need to know.

The two most important voltages are the VI and VO pins of the regulator in the bottom-center of the main PCB. If the VI pin measures the correct voltage, that will tell you that the voltage doubler is working. If the VO pin is correct, that will tell you that the regulator is working. The LFO runs off the unregulated voltage from the doubler, while the audio path runs off the regulated voltage.

Note that the unregulated voltage will fluctuate a small amount (around 0.5V) when the LFO is operational. This is due to the variable current draw of the lamp. The audio path uses regulated voltage so that these changes in the supply will not impact the audio.

Lamp lifespan

Current-production lamps have a rated lifespan of between 10,000 and 20,000 hours. The constant pulse of the LFO reduces this lifespan. It's strongly recommended to keep the pedal disconnected from power when not in use, which will lengthen the working lifespan by years if not decades. Alternately, you can turn off the LFO with the toggle switch when not in use, which will have the same effect.

Either way, though, it's important to be aware that the lamp is the shortest-lifespan component in the circuit. It's not fragile by any means, but just keep it in the back of your mind that if the phasing stops working at any point, the first thing to check is whether the lamp is working underneath the light shield.

Additionally, be aware that the rate LEDs get their control voltage directly from the lamp. If the lamp is damaged, it won't pass voltage and the LEDs will not function. This is another indicator that the lamp needs to be replaced.

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 page 41

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

Uni-Vibe® is a registered trademark of Dunlop Manufacturing, Inc.

All other trademarks are property of their respective owners.

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

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

DOCUMENT REVISIONS

1.0.0 (2026-08-21)
Initial release.