PROJECT NAME

EMERALD

aion
DIY GUITAR EFFECTS

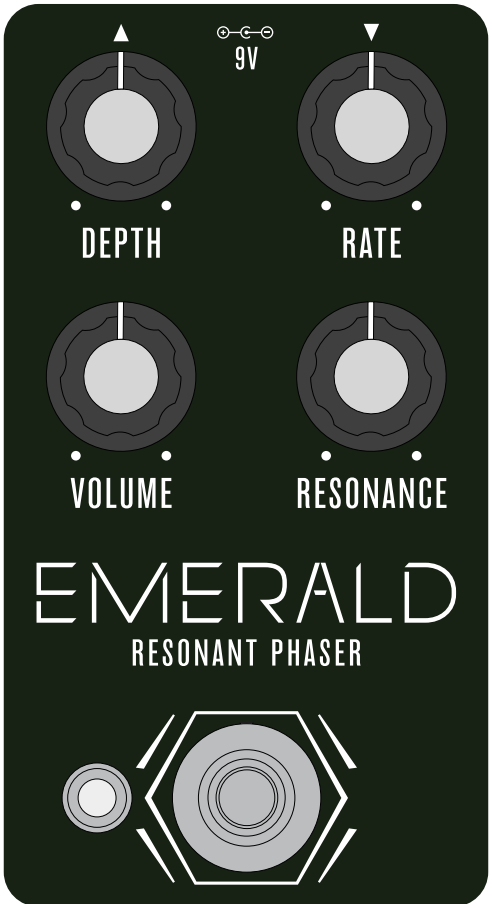
BASED ON
BOSS® PH-1r Phaser

BUILD DIFFICULTY
Intermediate

EFFECT TYPE
4-stage phaser

DOCUMENT VERSION
1.0.1 (2024-12-06)

PROJECT SUMMARY
A four-stage JFET-based phaser similar to the Phase 90, but with with a frequency-dependent feedback control that accentuates the phase effect.



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 pages 28-29.

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

Film Capacitors

NAME	QTY
10n (0.01)	9
33n (0.033)	1
47n (0.047)	1
1uF	4

Electrolytic Capacitors

NAME	QTY
47uF	1
100uF	1

Tantalum Capacitors

NAME	QTY
0.1uF (marked “104”)	1
15uF (marked “156” or 15µ)	1

MLCC Capacitors

NAME	QTY
47pF (marked “470”)	2
100pF (marked “101”)	1
100n (marked “104”)	1

Transistors

NAME	QTY
2N5088	1

Trimmers

NAME	QTY
100k trimmer	1

Resistors

NAME	QTY
10R	1
470R	1
1k5	2
1k8	1
4k7	1
10k	11
12k	1
22k	2
47k	6
68k	1
100k	4
150k	2
330k	8
470k	2
1M	1
2M2	2

Diodes

NAME	QTY
1N5817	1
1N914	1
1N5231B (marked “231B”)	1

JFETs

NAME
2SK208-GR, matched set of 4
16-pin header, right angle
16-pin socket

PACKING LIST (CONT.)

ICs

NAME	QTY
JRC4558D	3
TL022	1
8-pin socket	4

Other

NAME	QTY
LED bezel	1
LED, white	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

Potentiometers

NAME	QTY
10kB	1
100kA	1
100kB	1
1MC (C105)	1
Dust cover	4
Knob	4
Mounting nut, potentiometer, 0.44"	4
Lock washer, potentiometer, 0.5"	4
Outer washer, potentiometer, 0.475"	4

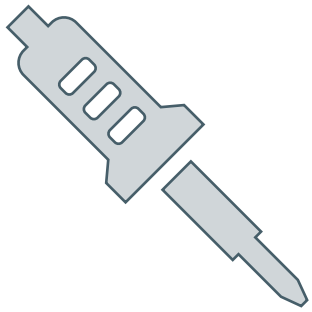
Switches

NAME	QTY
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

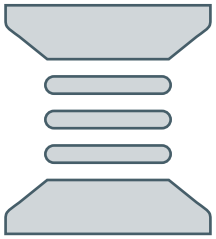
NAME	QTY
3-strand wire assembly, 70mm	2
4-strand wire assembly, 122mm	1
3-pin wire assembly header	2
4-pin wire assembly header	1

TOOLS NEEDED



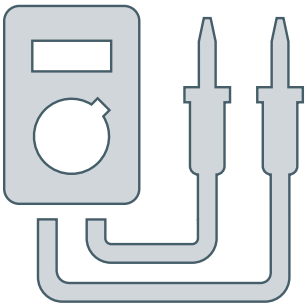
SOLDERING IRON

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



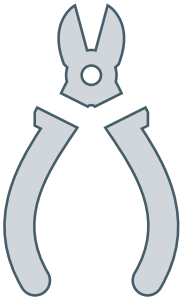
SOLDER

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



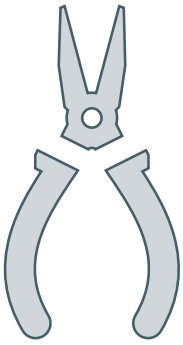
DIGITAL MULTIMETER (DMM)

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



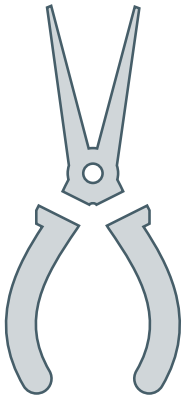
WIRE SNIPPERS

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



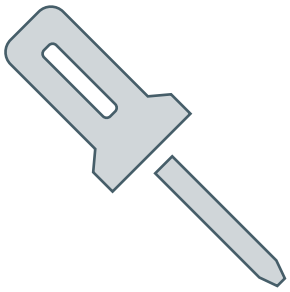
FLAT-NOSE PLIERS

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



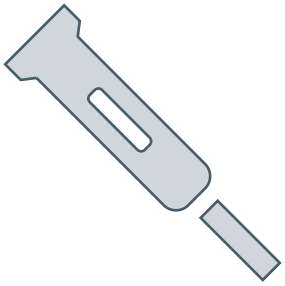
NEEDLE-NOSE PLIERS

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



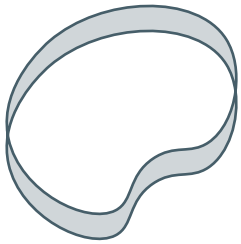
SCREWDRIVER (PHILLIPS)

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



FLAT SCREWDRIVER (SMALL)

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



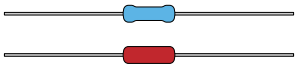
RUBBER BAND

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

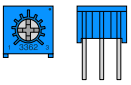
COMPONENT IDENTIFICATION

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

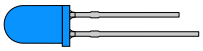
RESISTOR



TRIM POTENTIOMETER

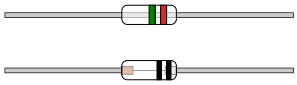


LED



SILICON DIODE

GERMANIUM DIODE



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

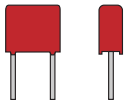
RECTIFIER DIODE

Some Schottky diodes also look like this.

SCHOTTKY DIODE

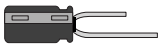
ZENER DIODE

FILM CAPACITOR



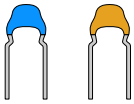
Not polarized. Color may vary by brand and type.

ELECTROLYTIC CAPACITOR



Polarized. The negative side is marked.

MLCC

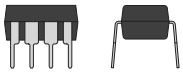


Not polarized. MLCC stands for "multi-layer ceramic capacitor."

TANTALUM CAPACITOR

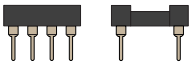
Polarized. The positive side is marked.

IC OR OP-AMP

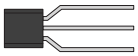


Charge pumps and delay chips also look like this. They may have more than 8 legs.

IC SOCKET

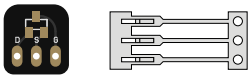


TRANSISTOR OR JFET



Some voltage regulators also look like this.

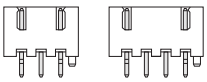
TRANSISTOR WITH ADAPTER



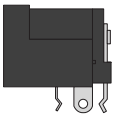
The pins will be soldered to the adapter during kit assembly.

WIRE ASSEMBLY

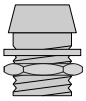
WIRE ASSEMBLY HEADER



DC JACK



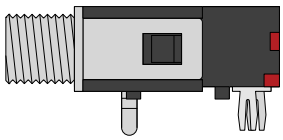
LED BEZEL



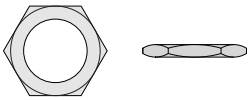
HARDWARE IDENTIFICATION

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

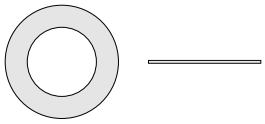


MOUNTING NUT



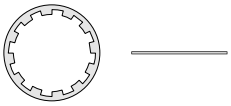
DIAMETER: 0.54" / 13.7mm

OUTER WASHER



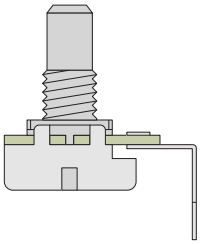
DIAMETER: 0.6" / 15.2mm

LOCK WASHER

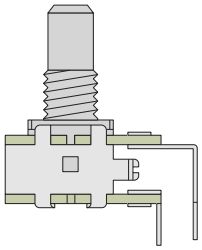


DIAMETER: 0.5" / 12.7mm

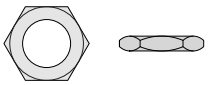
POTENTIOMETER (SINGLE)



POTENTIOMETER (DUAL)

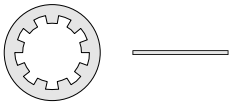


MOUNTING NUT



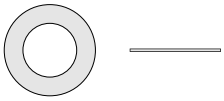
DIAMETER: 0.44" / 11.2mm

LOCK WASHER



DIAMETER: 0.5" / 12.7mm

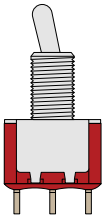
OUTER WASHER



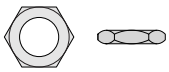
DIAMETER: 0.475" / 12mm

KNOB

TOGGLE SWITCH

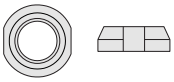


MOUNTING NUT



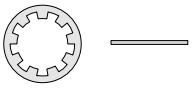
DIAMETER: 0.36" / 9.1mm

DRESS NUT



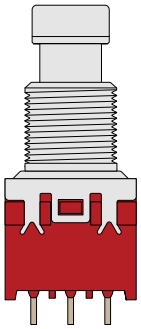
DIAMETER: 0.375" / 9.5mm

LOCK WASHER

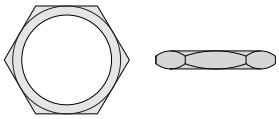


DIAMETER: 0.4" / 10.1mm

FOOTSWITCH

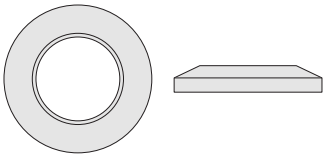


MOUNTING NUT



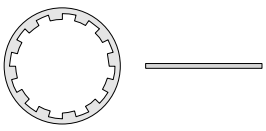
DIAMETER: 0.6" / 15.2mm

DRESS NUT



DIAMETER: 0.77" / 19.6mm

LOCK WASHER



DIAMETER: 0.6" / 15.2mm

PCB ASSEMBLY OVERVIEW

Now it's time to start building!

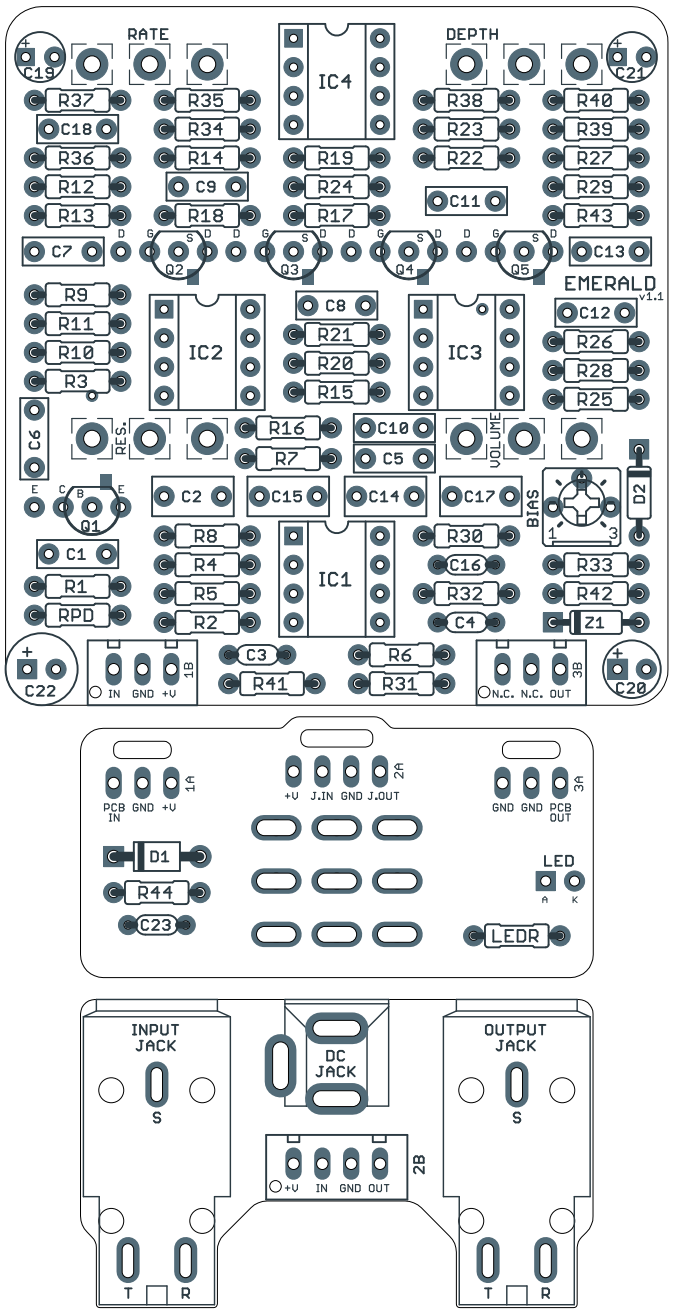
Before you begin, separate the PCBs into individual boards and break off the tabs from each using needle-nose or flat-head pliers. You should be left with the three PCBs shown to the right.

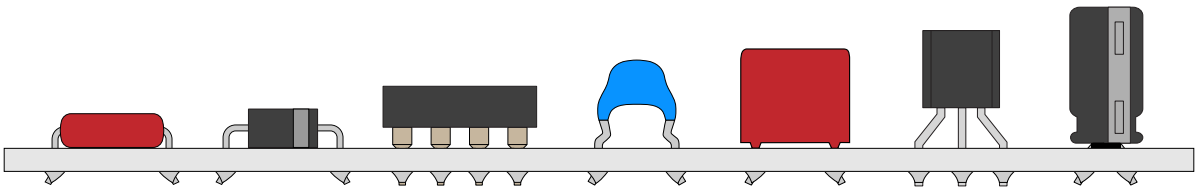
The general principle for PCB population is that you want to work in layers from shortest components (i.e. lowest-profile) to tallest so that when the PCB is upside-down, everything is making contact with the work surface and is held in place.

Generally speaking, you should populate the components in this order:

1. Resistors
2. Diodes
3. IC sockets
4. MLCC capacitors
5. Film capacitors
6. Transistors
7. Electrolytic capacitors

Not all of these component types are included in each kit, so skip them if they aren't applicable. Some types of film capacitors are taller than electrolytics, so those can be done last.





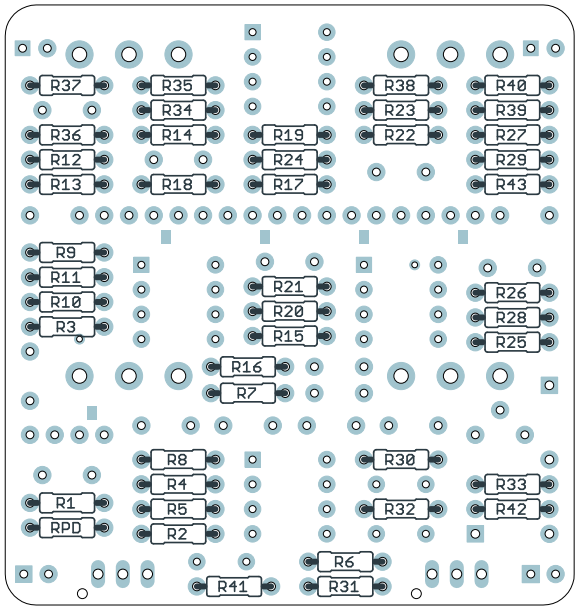
RESISTORS

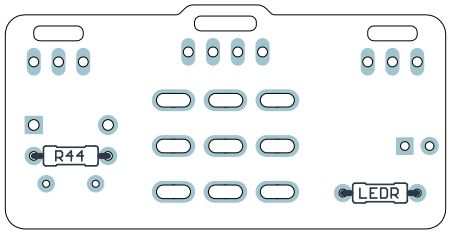
PART	VALUE
R1	10k
R2	470k
R3	10k
R4	47k
R5	47k
R6	47k
R7	12k
R8	1k5
R9	1k5
R10	10k
R11	10k
R12	100k

PART	VALUE
R13	330k
R14	330k
R15	10k
R16	10k
R17	100k
R18	330k
R19	330k
R20	10k
R21	10k
R22	100k
R23	330k
R24	330k

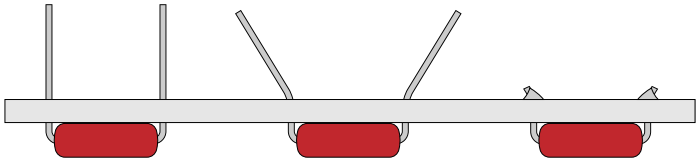
PART	VALUE
R25	10k
R26	10k
R27	100k
R28	330k
R29	330k
R30	22k
R31	47k
R32	47k
R33	470R
R34	150k
R35	470k
R36	4k7

PART	VALUE
R37	150k
R38	68k
R39	10k
R40	2M2
R41	1k8
R42	47k
R43	1M
R44	10R
RPD	2M2
LEDR	22k





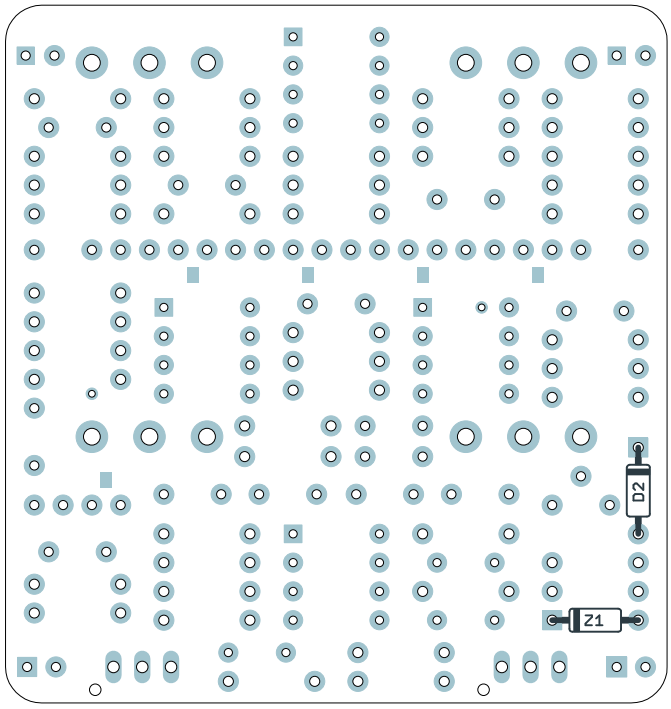
Using the parts list above, populate the resistors by pushing them through the holes and bending the leads outward at an angle to hold them in place. Resistors are not polarized, so they will work in any direction. Turn the board upside-down to keep the components held in place while you solder.

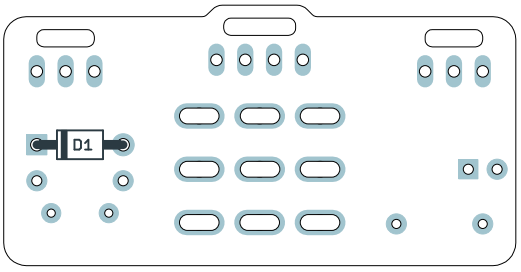


Don't try to do all of the resistors at once. You'll want to stop periodically flip the board and solder everything, then cut the leads using the wire snippers to make room for more.

DIODES

PART	VALUE
D1	1N5817
D2	1N914
Z1	1N5231B

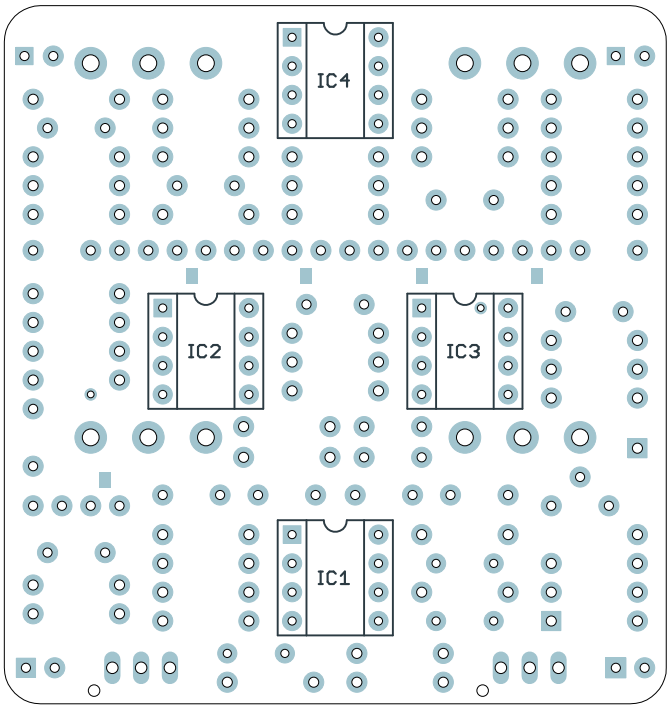




Next, you'll populate the diodes. Diodes are polarized, so make sure to identify the polarity band (which indicates the "cathode", or negative side) and match the band to the footprint on the PCB.

The value can be difficult to read, so they can alternately be identified by appearance. D1 (1N5817) is black and larger than the other two. D2 and Z1 are both orange and black and similar in size, but D2 (1N914) will have two paper tabs on the ends of the leads while Z1 (1N5231B) will be loose.

PART	VALUE
IC1	JRC4558D
IC2	JRC4558D
IC3	JRC4558D
IC4	TL022

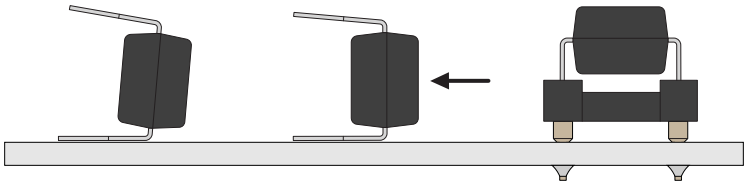


Next up are the IC sockets. You can't bend the leads of the sockets as with the other components, so they won't stay in on their own until they are soldered. Flip the PCB over and use gravity to hold them in place.

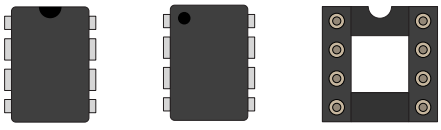
Installing the ICs

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

The legs of the ICs are bent outward slightly during manufacturing, so they'll need to be bent back inward before they can be inserted into the sockets. It's easiest to do this by laying the IC legs against the table and bending the body itself so all four legs on the side are straightened out at once. Then, flip it and do the other side.

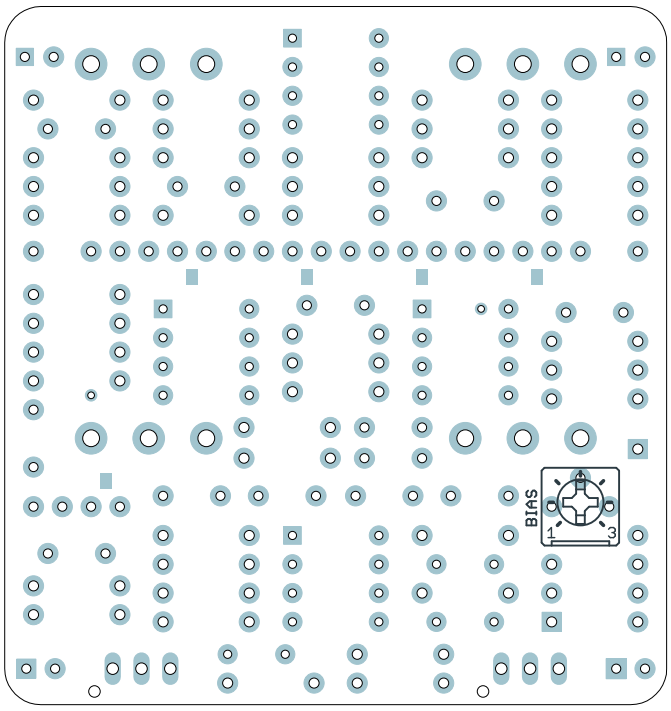


ICs may have two different orientation marks: either a dot in the upper-left or a half-circle notch in the middle of the top side. Some ICs have both marks. This shows which way the IC should be rotated when inserting it into a socket (the socket also has a half-circle notch).



TRIMMER

PART	VALUE
BIAS	100k trimmer



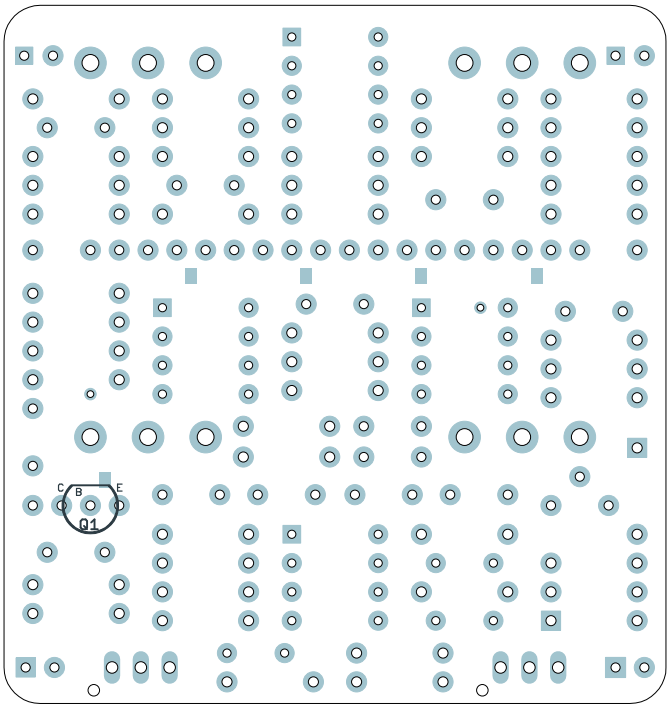
Next, we'll do the trimmer. Bend the legs outward to keep it in place while soldering.

Setting the trimmer

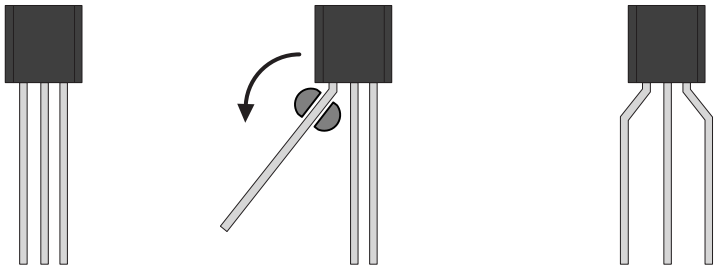
The trimmer will be used for the calibration process on page 26 and should not be adjusted after calibration is completed. For now, set it to the 50% position (12:00 noon).

TRANSISTOR

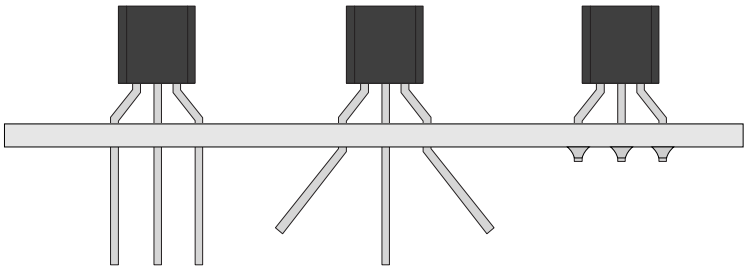
PART	VALUE
Q1	2N5088



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



Once installed, bend the two outer legs to hold it in place on the board. Then, solder them in place and clip the leads.



Note that there is an extra "E" pad next to the transistor outline. This is only used for NOS Japanese transistors with different pinouts and should be ignored for the kit build. Install the transistor as shown in the silkscreen on the PCB.

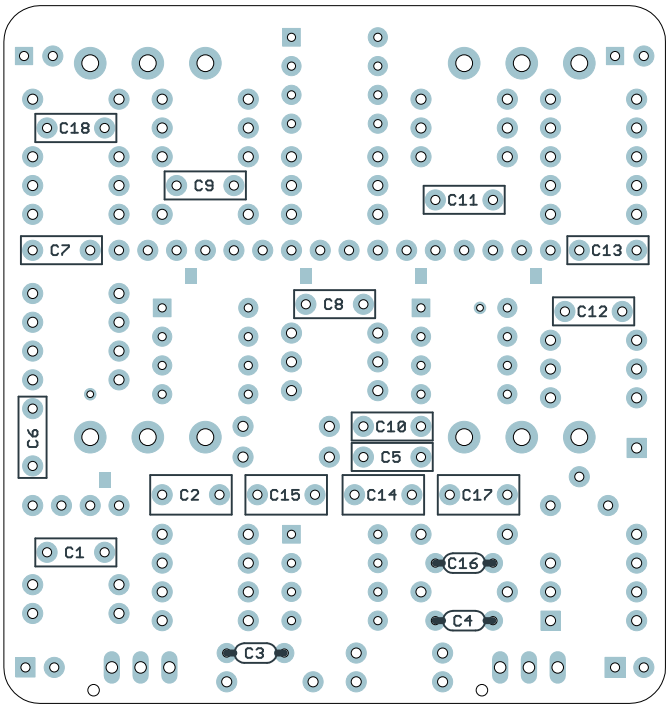
CAPACITORS (NON-POLARIZED)

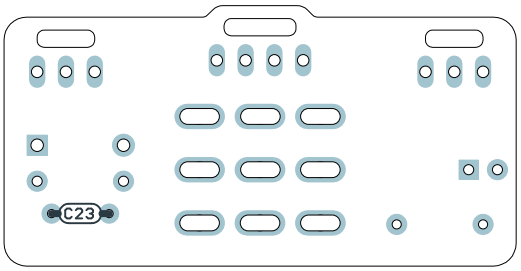
PART	VALUE
C1	47n (0.047)
C2	1uF
C3	47pF MLCC
C4	100pF MLCC
C5	33n (0.033)

PART	VALUE
C6	10n (0.01)
C7	10n (0.01)
C8	10n (0.01)
C9	10n (0.01)
C10	10n (0.01)

PART	VALUE
C11	10n (0.01)
C12	10n (0.01)
C13	10n (0.01)
C14	1uF
C15	1uF

PART	VALUE
C16	47pF MLCC
C17	1uF
C18	10n (0.01)
C23	100n MLCC





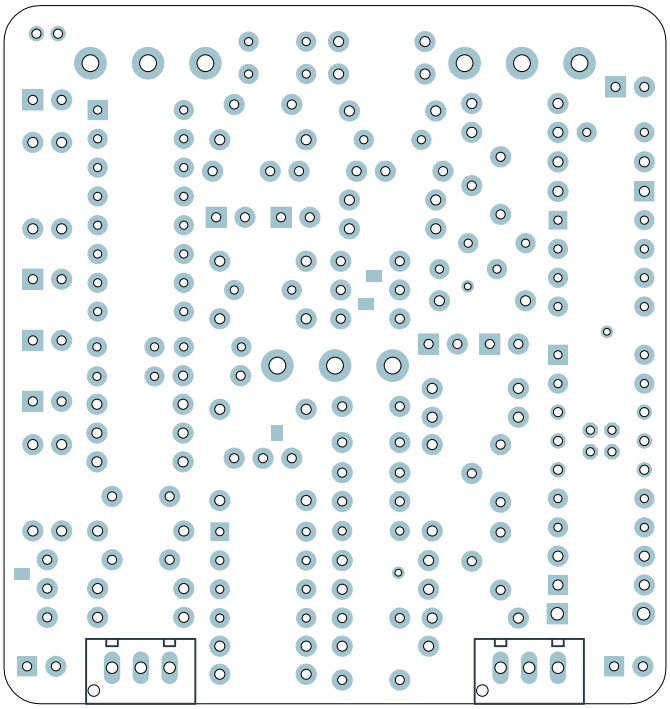
After the sockets come the box film and MLCC capacitors. These are all several different heights, so it's recommended to do them shortest to tallest. Bend the leads at an angle to hold them in place.

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: Depending on the type, the box film capacitors may have their value printed on either the top or the side. Usually the red ones have it printed on the side while the blue or gray ones have it on the top.

All of the MLCCs except C23 are blue MLCC capacitors taped to cardboard. For these, the value will be written on the cardboard. C23 (100n MLCC) is yellow and is most easily identified by color.

WIRE HEADER SOCKETS

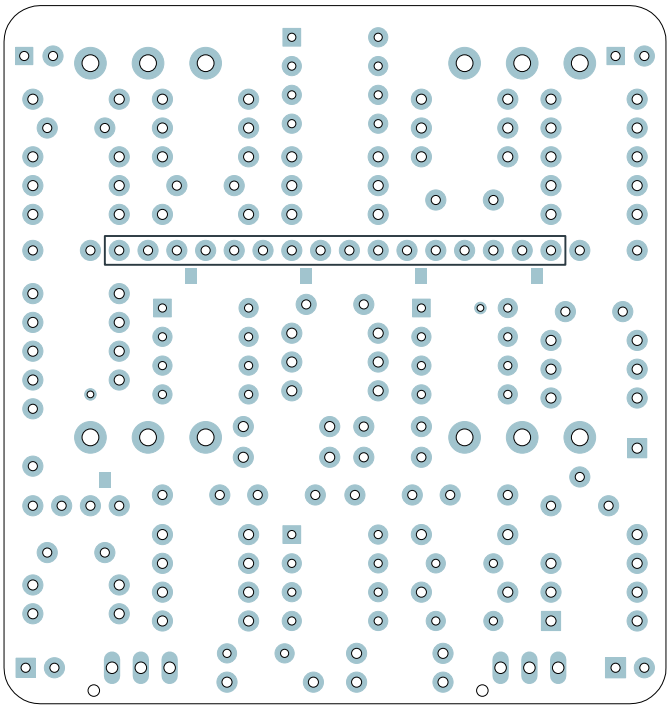


Install the two 3-pin header sockets (wire connectors) as shown above. These have a polarity pin, so as long as they are pressed all the way down, there's only one possible way to install them. They do fit pretty tightly in the holes, though, so press firmly.

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

Now is the time to go back to page 13 and insert the ICs into the sockets. These are the same height as the JFET socket in the next step, so it will help keep the PCB level while that is installed.

JFET SOCKET

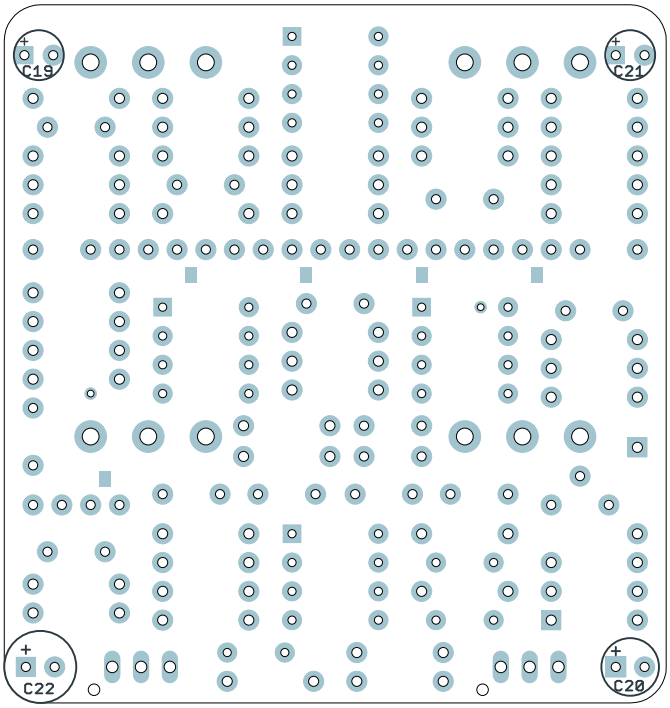


Remove the 16-pin header from the black socket and install the socket as shown above. With the header removed, it should be roughly the same height as the tallest non-polar capacitors and socketed ICs, so when you turn the PCB upside down to solder, it should balance pretty close to level.

There is no polarity for the socket. Just be careful that it is installed perpendicular to the PCB and not at an angle. It can be helpful to solder just a single pin at first so you can check the angle. This gives room to reflow the joint if it's not level with the board. Once you've soldered more than one joint, it will be very difficult to correct.

CAPACITORS (POLARIZED)

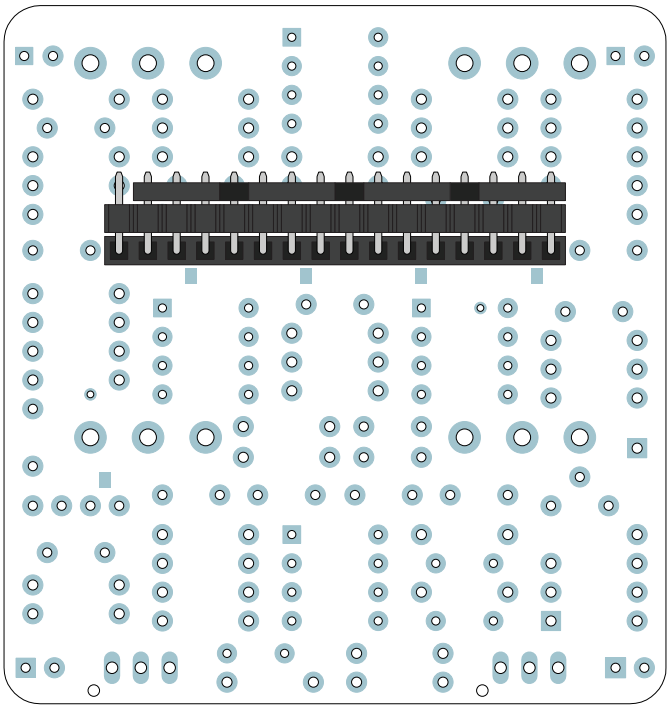
PART	VALUE
C19	15uF tantalum (156)
C20	47uF electrolytic
C21	0.1uF tantalum (104)
C22	100uF electrolytic



Populate the two electrolytic capacitors, C20 and C22. They are polarized (i.e. they will only work in one direction), so note the vertical mark that indicates the negative side. If one leg is longer, this is the positive leg and it fits in the square pad.

Next, populate the yellow tantalum capacitors, C19 and C21. Be very careful with this: unlike electrolytic capacitors, tantalum capacitors have the "+" (positive) side marked instead of the negative. Since tantalum capacitors aren't commonplace in guitar pedals, many people instinctively reverse them, and as a result, their build has issues. Like electrolytics, the longer leg still goes in the square pad.

JFET MODULE

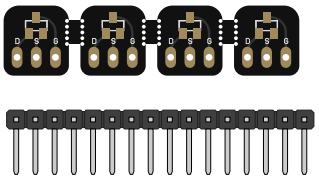


Next, the matched JFET module needs to be soldered to the 16-pin header. Make sure to look at all of the diagrams carefully before soldering! If the short and long pins are turned around, the module won't fit in the socket properly.

STEP 1

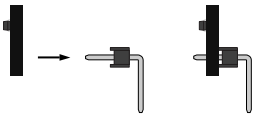
Snap the extra tabs off of the JFET module if applicable, but leave the four sub-PCBs connected together.

Identify the short and long pins of the right-angle pin header.



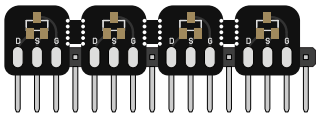
STEP 2

With the component side facing out, insert the short pins into the JFET module as shown. The spare pin should be on the right side as you look from the front (refer to step 3 diagram).



STEP 3

Solder the pins to the module. Four of the pins will not be soldered: three in between the JFETs, and one to the right of the module when facing it straight on.

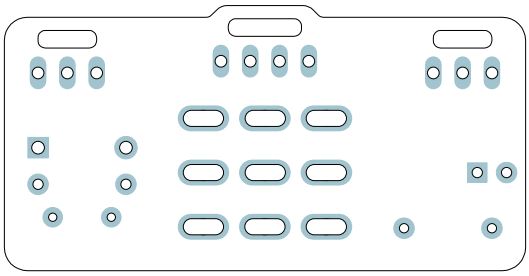


STEP 4

Insert the JFET module into the header socket as shown in the PCB diagram above. Press it down as far as it will go. The component side faces the top of the main PCB and the spare pin is on the far left side.

FOOTSWITCH PCB

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



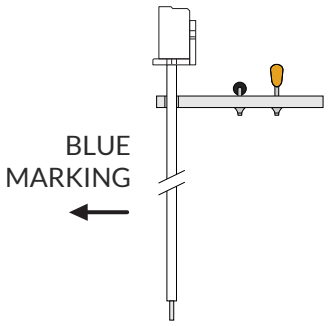
Next, it's time to finish up the footswitch board. You should have done most of the on-board components on this board in a previous step, but if not, go back and do those.

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.

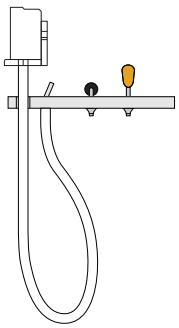
For now, pull it through as far as it can go.



STEP 2

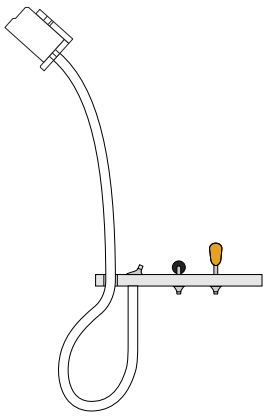
Next, bend the wires back upward and fit the ends of the wires into the solder pads.

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

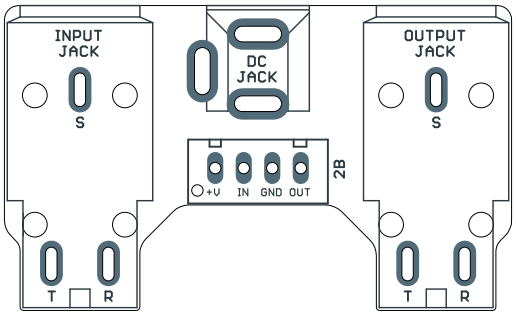
Then, solder the wires from the top. This is the trickiest part of the whole build. You want to solder the pads without touching the iron to the wires themselves and risking burning through the insulation. It helps to use a sharp or narrow tip on the soldering iron.



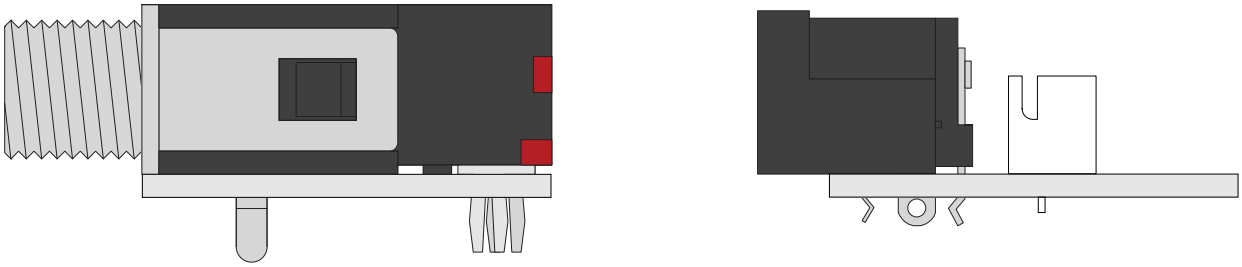
Once all three wire assemblies are soldered, set the footswitch PCB aside. We'll solder the actual footswitch and LED in a later step.

INPUT/OUTPUT PCB

PARTS
Input & output jacks
DC jack
Wire header



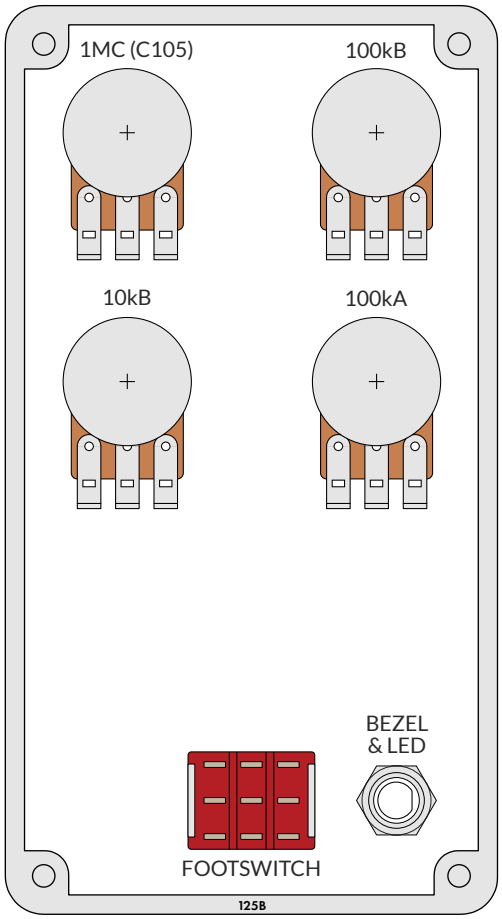
Almost done! Get the two input/output jacks, the DC jack and the wire header and snap them in place. The PCB is designed for them to fit securely, so you can do them all at once before flipping and soldering.



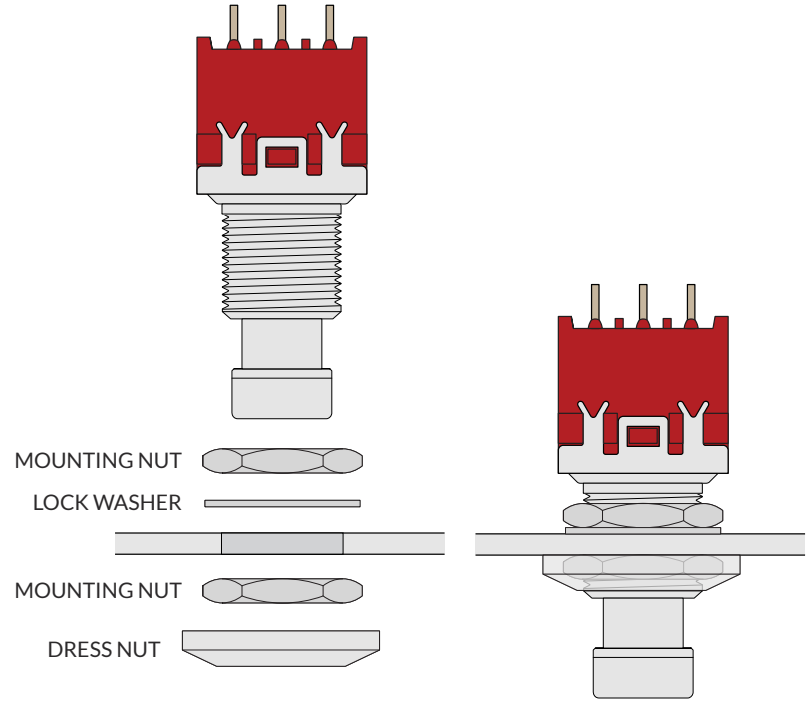
After you've soldered everything, make sure to **snip the leads on the I/O jacks as close as possible to the PCB**. There's not a lot of clearance between the bottom of this board and the top of the main PCB once everything is in place, and you don't want the pins to short against anything on accident.

ENCLOSURE LAYOUT: PANEL MOUNTS

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

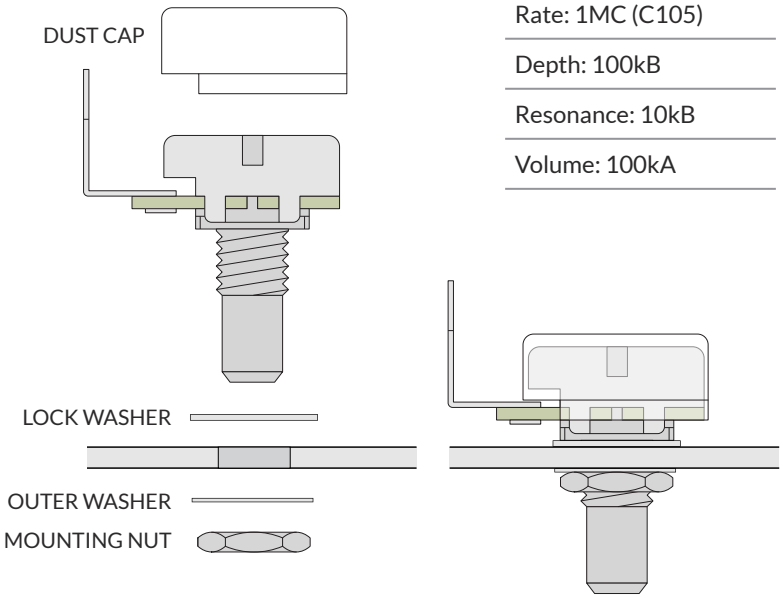


FOOTSWITCH



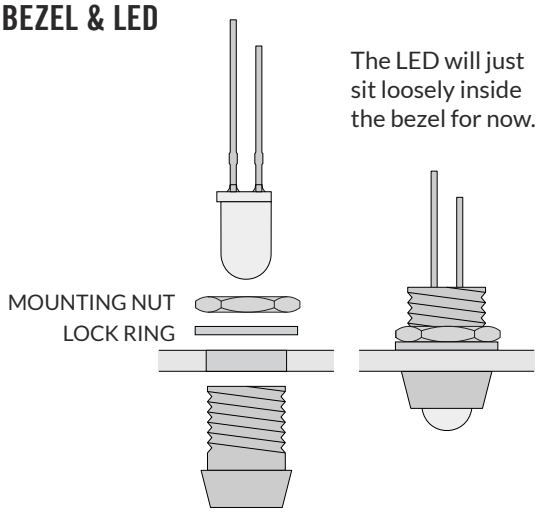
The dress nut fits over the top of the mounting nut and is for aesthetic purposes only. Wrap a rubber band around it to use as a grip when tightening. Avoid using metal tools on it or you run the risk of scratching or denting it.

POTENTIOMETERS



Rate: 1MC (C105)
Depth: 100kB
Resonance: 10kB
Volume: 100kA

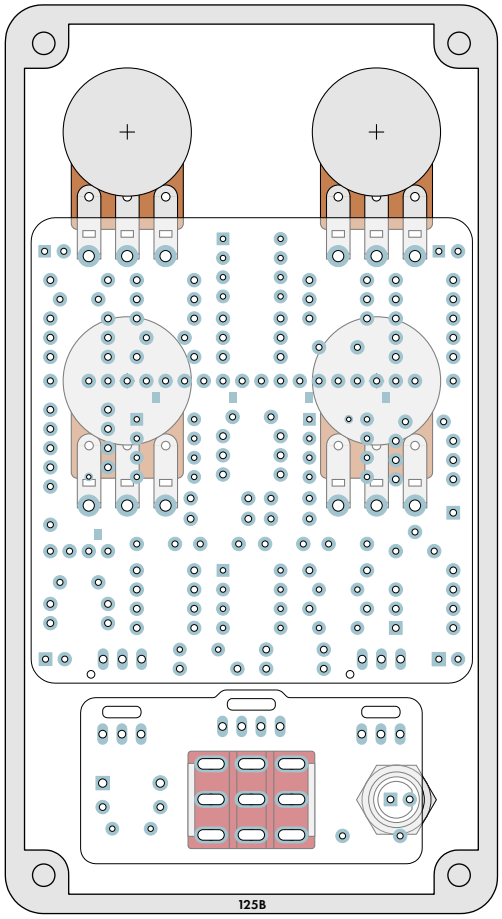
BEZEL & LED



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

You'll need to hold the bezel in place when tightening the nut. Be aware that the bezel is fairly sharp. Try using a rubber band for grip instead of just pressing your finger against the bottom.

ENCLOSURE LAYOUT: MAIN & FOOTSWITCH PCBS



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

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

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

After you've finished soldering the pots, **clip the leads as close as you can to the main PCB**. This is important because the input/output PCB overlaps them and you need to avoid any of the components shorting.

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

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

Why solder everything inside the enclosure before testing it?

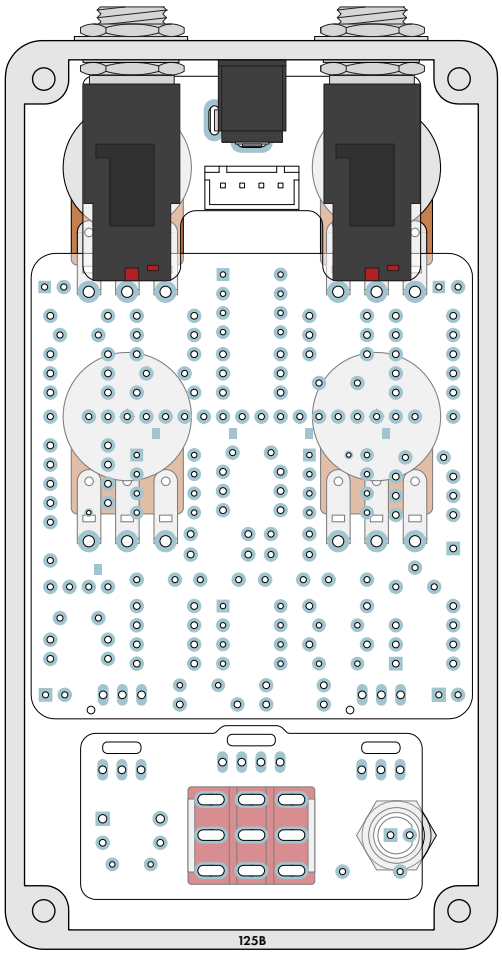
"Rock it before you box it" is conventional wisdom in pedalbuilding, and you'll often hear it recommended that builders should test the circuit before putting everything inside the enclosure. However, Aion FX projects are designed to be extremely easy to remove from the enclosure for troubleshooting,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

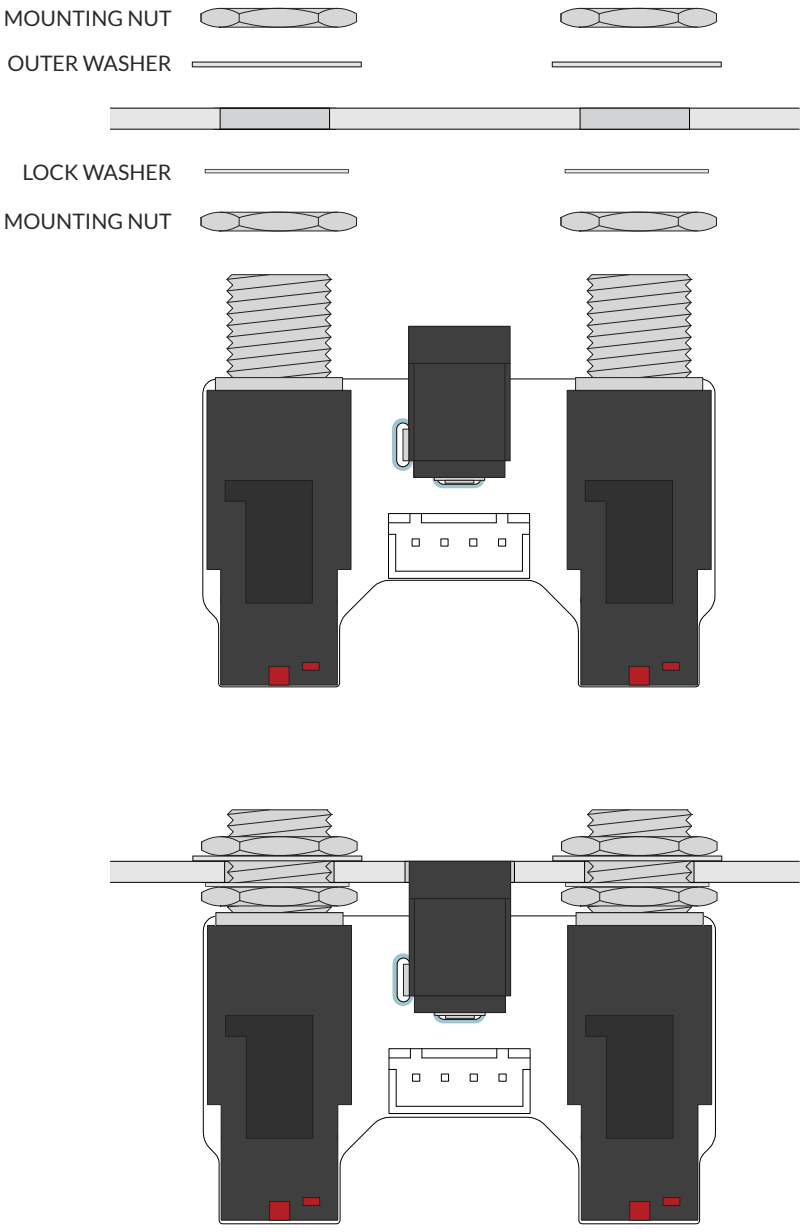
If you've read the documentation carefully and followed all the instructions, there's a good chance you will get it right the first time!

ENCLOSURE LAYOUT: INPUT/OUTPUT PCB

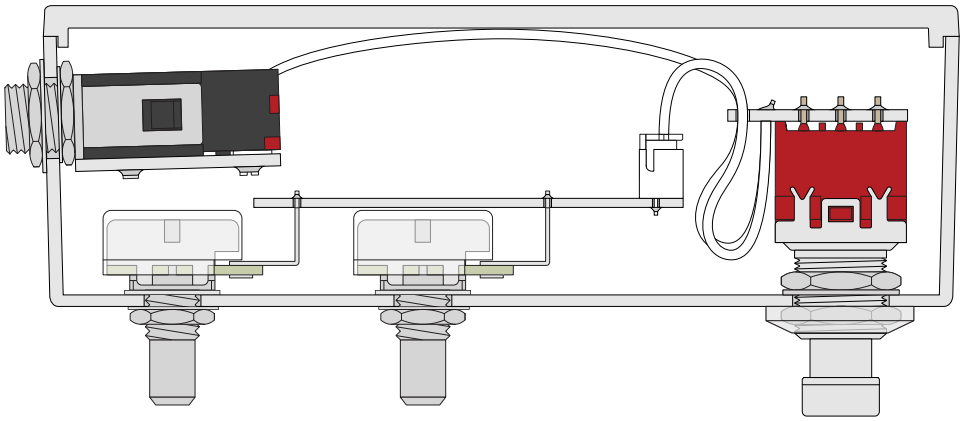
Affix the input/output PCB to the north-facing panel of the enclosure as shown.

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.





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

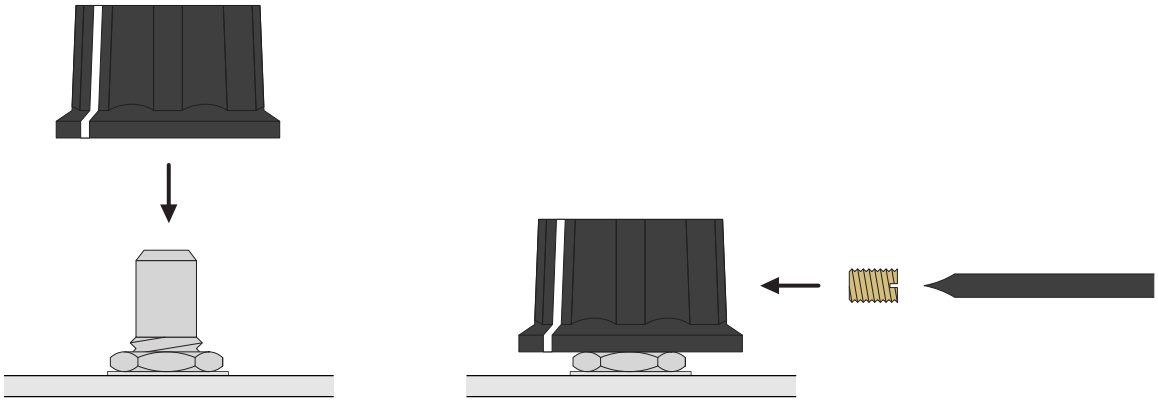
Test the bypass switch a few times, then leave it in effect mode and listen while you turn the knobs. Note that it has not been biased yet, so we're only testing to make sure it passes signal so we can proceed to the next step. We'll evaluate the sound as part of the biasing process.

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

The bias trimmer adjusts the LFO voltage range so it is best suited to the JFETs being used.

The bias can easily be set by ear: with the Depth control set to maximum and Rate to medium speed, just adjust the trimmer until you've got the best phasing sound with a wide frequency range and no audible distortion. This generally won't be too far off from the 12:00 position.

If you've got an oscilloscope and signal generator, the PH-1r service manual gives the full factory setup procedure. But for this circuit it's definitely not necessary. You probably won't get any better results with an oscilloscope than by using your ears.

FINAL ASSEMBLY

If everything calibrated OK, you're all done! Affix the back panel with the four screws provided and make some music.

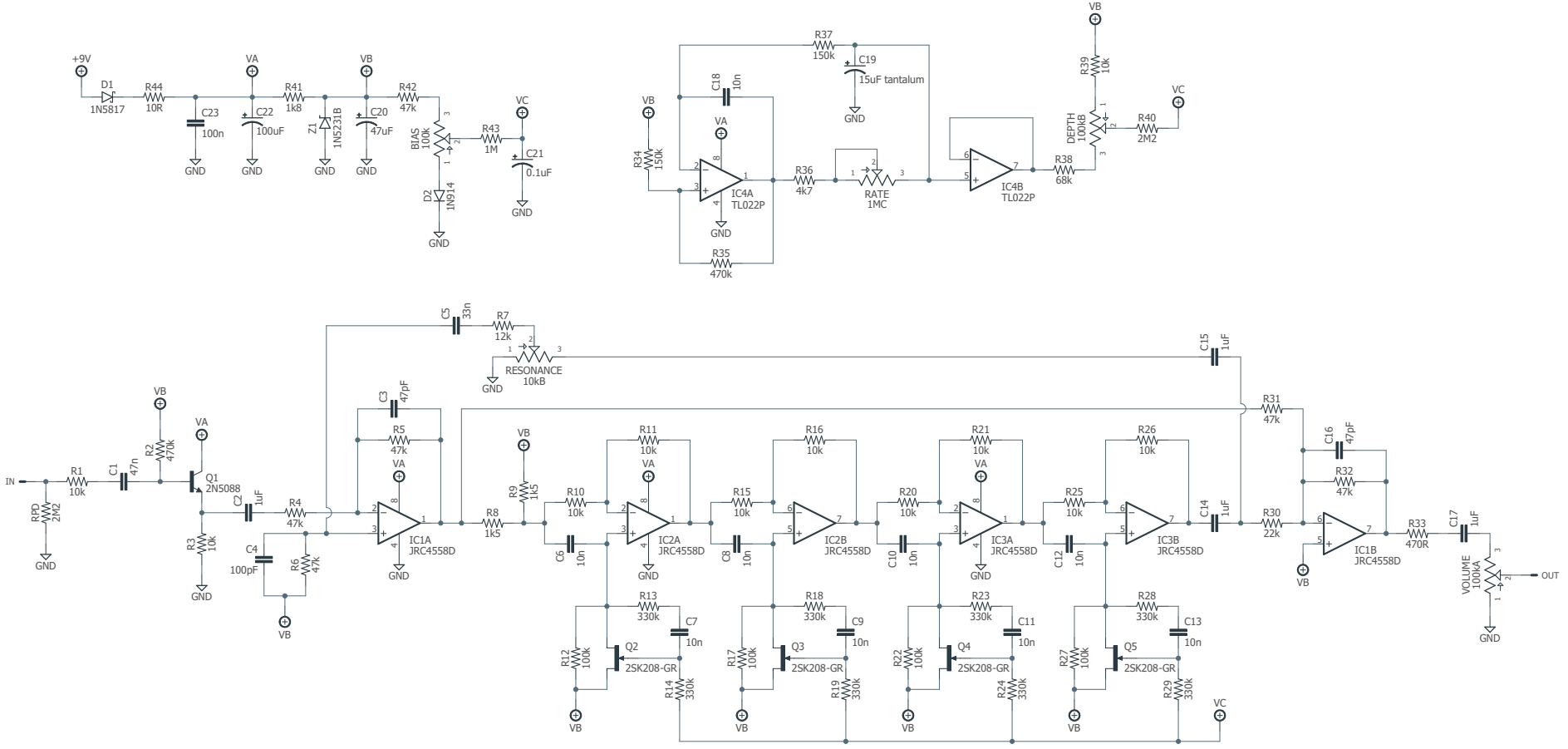
If doesn't work, or doesn't sound quite right after calibration, don't be discouraged. See page 30 for troubleshooting info.

USAGE

The Emerald has four controls:

- **Rate** sets the speed of the phaser effect, from 100 milliseconds to 16 seconds.
- **Depth** sets the intensity of the phaser effect.
- **Resonance** adjusts the amount of frequency-dependent feedback which is fed from the output of the phase stages back to the input.
- **Volume** sets the overall output of the effect signal. This should generally be run at full and only turned down if the settings of the Resonance and Depth controls cause the signal to be higher than unity.

SCHEMATIC


+9V
D1
1N5817
R44
10R
C23
100n
GND
VA
C22
100uF
R41
1k8
Z1
1N5231B
VB
C20
47uF
R42
47k
BIAS
100k
D2
1N914
R43
1M
VC
C21
0.1uF
R37
150k
C19
15uF tantalum
C18
10n
R34
150k
IC4A
TL022P
R35
470k
R36
4k7
RATE
1MC
IC4B
TL022P
R38
68k
R39
10k
DEPTH
100kB
R40
2M2
C5
33n
R7
12k
RESONANCE
10kB
C15
1uF
IN
R1
10k
C1
47n
R2
470k
Q1
2N5088
RPD
2M2
R3
10k
C2
1uF
R4
47k
C4
100pF
R6
47k
C3
47pF
R5
47k
IC1A
JRC4558D
R8
1k5
R9
1k5
R10
10k
C6
10n
R11
10k
IC2A
JRC4558D
R15
10k
C8
10n
R16
10k
IC2B
JRC4558D
R20
10k
C10
10n
R21
10k
IC3A
JRC4558D
R25
10k
C12
10n
R26
10k
IC3B
JRC4558D
C14
1uF
R31
47k
R30
22k
C16
47pF
R32
47k
IC1B
JRC4558D
R33
470R
C17
1uF
VOLUME
100kA
OUT
R13
330k
C7
10n
Q2
2SK208-GR
R12
100k
R14
330k
R18
330k
C9
10n
Q3
2SK208-GR
R17
100k
R19
330k
R23
330k
C11
10n
Q4
2SK208-GR
R22
100k
R24
330k
R28
330k
C13
10n
Q5
2SK208-GR
R27
100k
R29
330k

FULL PARTS LIST

Resistors

PART	VALUE
R1	10k
R2	470k
R3	10k
R4	47k
R5	47k
R6	47k
R7	12k
R8	1k5
R9	1k5
R10	10k
R11	10k
R12	100k

PART	VALUE
R13	330k
R14	330k
R15	10k
R16	10k
R17	100k
R18	330k
R19	330k
R20	10k
R21	10k
R22	100k
R23	330k
R24	330k

PART	VALUE
R25	10k
R26	10k
R27	100k
R28	330k
R29	330k
R30	22k
R31	47k
R32	47k
R33	470R
R34	150k
R35	470k
R36	4k7

PART	VALUE
R37	150k
R38	68k
R39	10k
R40	2M2
R41	1k8
R42	47k
R43	1M
R44	10R
RPD	2M2
LEDR	22k

Capacitors

PART	VALUE
C1	47n film
C2	1uF film
C3	47pF MLCC
C4	100pF MLCC
C5	33n film
C6	10n film

PART	VALUE
C7	10n film
C8	10n film
C9	10n film
C10	10n film
C11	10n film
C12	10n film

PART	VALUE
C13	10n film
C14	1uF film
C15	1uF film
C16	47pF MLCC
C17	1uF film
C18	10n film

PART	VALUE
C19	15uF tantalum
C20	47uF electro
C21	0.1uF tantalum
C22	100uF electro
C23	100n MLCC

Transistors

PART	VALUE
Q1	2N5088
Q2	2SK208-GR
Q3	2SK208-GR
Q4	2SK208-GR
Q5	2SK208-GR

Diodes

PART	VALUE
D1	1N5817
D2	1N914
Z1	1N5231B

ICs

PART	VALUE
IC1	JRC4558D
IC2	JRC4558D
IC3	JRC4558D
IC4	TL022

Sockets

PART
DIP-8 socket (4)
16-pin header & socket

FULL PARTS LIST, CONT.

Potentiometers

PART	VALUE
Rate	1MC (C105)
Depth	100kB
Res.	10kB
Volume	100kA

Trimmers

PART	VALUE
BIAS	100k

Switches

PART
3PDT stomp

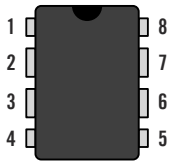
TROUBLESHOOTING INFORMATION

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 **9.6V** supply. Your measured voltages won't be exactly the same due to variance in power supplies and component tolerances. However, if you see anything more than +/-0.5V from the listed voltages, it's a good indicator of an issue, and the exact voltages can help narrow it down.

Note that IC pins are labeled counter-clockwise from the upper-left, as shown in the diagram to the right. Transistors and JFETs have their pins labeled on the PCB.



All voltages taken with Rate at minimum and Depth at maximum. The "~" symbol denotes a voltage range that oscillates with the speed of the Rate control.

IC1

PIN	VOLTAGE
1	4.71
2	4.71
3	4.69
4	0
5	4.71
6	4.71
7	4.71
8	9.23

IC2

PIN	VOLTAGE
1	4.71
2	4.71
3	4.71
4	0
5	4.71
6	4.71
7	4.71
8	9.23

IC3

PIN	VOLTAGE
1	4.71
2	4.71
3	4.71
4	0
5	4.71
6	4.71
7	4.71
8	9.23

IC4

PIN	VOLTAGE
1	0.64 ~ 8.64
2	3.80 ~ 5.60
3	3.68
4	0
5	3.80 ~ 5.60
6	3.80 ~ 5.60
7	3.80 ~ 5.60
8	9.23

Q1

PIN	VOLTAGE
C	9.23
B	4.18
E	3.77

Q2

PIN	VOLTAGE
G	*(see note)*
S	4.72
D	4.70

Q3

PIN	VOLTAGE
G	*(see note)*
S	4.72
D	4.72

Q4

PIN	VOLTAGE
G	*(see note)*
S	4.72
D	4.72

Q5

PIN	VOLTAGE
G	*(see note)*
S	4.72
D	4.71

Note: The voltage on the "G" pins of Q2-5 will oscillate with the position of the Rate control, and the range will vary with the position of the Bias trimmer.

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

BOSS® is a registered trademark of Roland Corporation.

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

1.0.1 (2024-12-06)
Corrected C14 and C15 to 1uF on page 15, mistakenly listed as 10n. The schematic and full parts list were correct.

1.0.0 (2024-06-01)
Initial release.