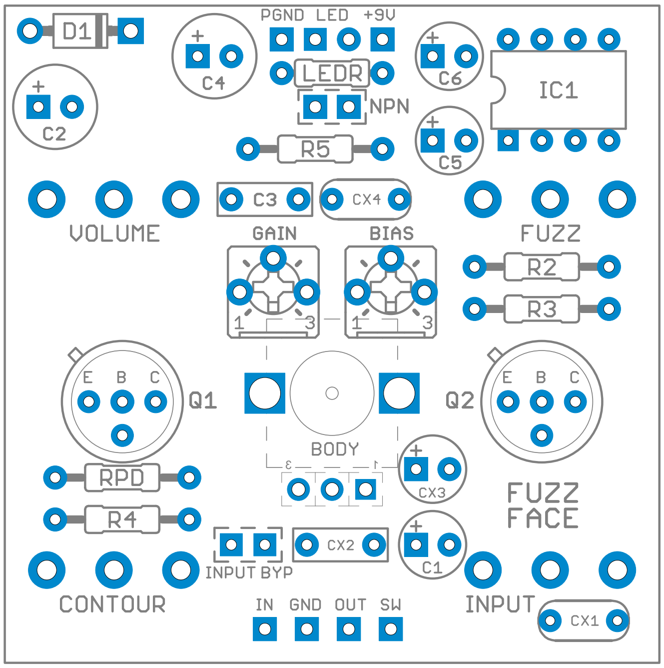
Solaris Fuzz

Dallas Arbiter Fuzz Face (Ge/Si)

Overview

Solaris Project Link



The Solaris Fuzz project is the ultimate DIY clone of the Dallas Arbiter Fuzz Face, one of the most ubiquitous effects in the DIY and boutique effect landscape. While the original Fuzz Face is a very simple circuit, many modifications have been made in the nearly 50 years since it was first introduced.

This PCB will allow you to build with germanium or silicon transistors (or a hybrid like Joe Gagan's Easy Face), positive or negative ground, the original two knobs or as many as five. It also incorporates an optional charge pump, allowing you to use a standard 9v center-negative power supply while still using PNP germanium transistors.

If you do build the 5-knob version, note that all of the original Fuzz Face tones are still available. Each of the extra knobs can be set to a "stock" value which essentially removes their effect on the circuit.

Controls & Usage

The Fuzz Face's controls are as follows:

- **Fuzz** controls the amount of gain from the Q2 transistor which overdrives the signal.
- **Volume** controls the overall output.
- **Contour** affects the midrange (borrowed from the Fulltone '69).
- **Input** allows you to attenuate the input signal, mimicking the effects of turning down your guitar volume. This way you can get similar volume-knob tones even if the fuzz is not the first effect in your chain. Joe Gagan, who came up with this control, recommends turning the Fuzz knob all the way up and using only this knob for the amount of distortion.
- **Body** is an input capacitor blend, which controls the amount of bass.

Modifications & Experimentation

The goal of this project was to make the ultimate tweakable Fuzz Face that incorporated just about every DIY modification imaginable. Accordingly, you can make a ton of different variants using the same PCB. However, since the Fuzz Face is approaching 50 years old and is probably the most analyzed guitar pedal circuit in existence, I couldn't possibly include every bit of research and legend in this build doc. These two pages sum it up better than I ever could:

- The Technology of the Fuzz Face by R.G. Keen
- The FAQ for Fuzz Face Fanatics by Steve Daniels / Small Bear Electronics

Parts

Resistors

R2	33k
R3	100k
R4	220R [5]
R5	1k
RPD	1M to 2M2
LEDR	4k7

Capacitors

C1	2u2 [2]
C2	22uF
C3	10n
C4	47uF
C5	10uF [3]
C6	10uF [3]
CX1	10pF
CX2	10n
CX3	3u3
CX4	10pF

Semiconductors

IC1	TC1044 [3]
D1	1N4742 zener
Q1/Q2	germanium [1]
LED	5MM

Potentiometers

Fuzz	1kC
Volume	500kA
Input	250kB [4]
Body	100kB 9mm [2]
Contour	1kB [5]
Gain	500R trim (3362P) [6]
Bias	10k trim (3362P) [7]

[1] For this circuit, it's not so much the part number of the germanium transistor as it is the properties (gain and leakage). I highly recommend getting your transistors from Small Bear Electronics—they've been offering tested & matched sets them for 15 years. They will also include a list of resistor values to use. Note that SBE's included resistor numbering is slightly different than mine: their R3 is my **R2**, and their R6 is my **R3**. (R4 and R5 are the same in both.) If using their Q2 bias resistor (R5), keep the Bias trim pot all the way down.

[2] If using the Body control, **omit C1** and use CX2 and CX3. The Body pot can be omitted with no jumpers. Ideally the Body pot would be a 9mm plastic shaft, mounted to the PCB rather than the enclosure.

[3] If you're using NPN germanium transistors, or if you'd rather reverse the DC jack wiring for positive ground, you should omit **IC1**, **C5** and **C6** and jumper the **NPN** pads together. Remember that when using reversed DC jack wiring, you can't daisy-chain with other pedals. The voltage inverter is much more convenient.

[4] If you're leaving off the Input control, jumper the INPUT BYP pads (or else pads 2 & 3 of the Input pot).

[5] If you're leaving off the Contour control, jumper pads 1 & 2 of the Contour pot and raise R4 to **470R**.

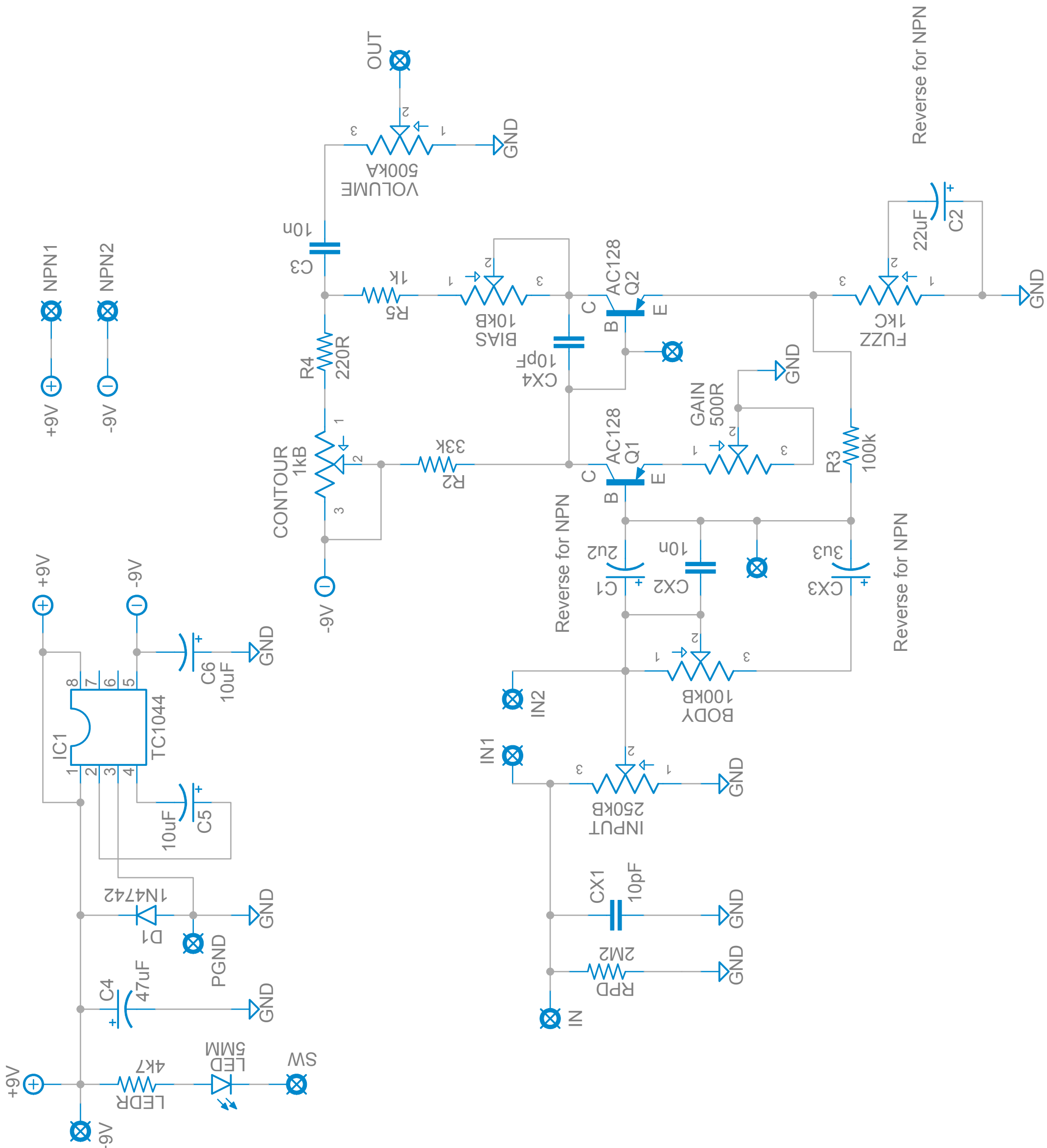
[6] This is just provided for tweakability—the default is all the way down, but you can turn it up to raise the gain of the first transistor stage which can saturate the second stage a bit.

[7] This trim pot allows you to bias Q2. The stock value is **8k2** (or around 75% up when using 1k for R5) but should be adjusted based on the transistor to get between **-4.5V** and **-5.5V** on the collector pin. (If your build includes the Contour pot, it should be set at about 20-30% during biasing.)

Additional Part Notes

- Capacitors are shown in nanofarads (n or nF) where appropriate. 1000n = 1uF. Many online suppliers do not use nanofarads, so you'll often have to look for 0.047uF instead of 47n, 0.0056uF instead of 5n6, etc.
- The PCB layout assumes the use of film capacitors with 5mm lead spacing for all values 1nF through 470nF. I prefer EPCOS box film or Panasonic ECQ-B/V-series.
- Potentiometers are Alpha 16mm right-angle PCB mount.
- I recommend using these dust covers / insulators from Small Bear to insulate the back of the pots from the board and prevent shorts. If you don't use these, use some electrical tape or cardboard to act as insulation. The right-angle pots will make direct contact with the solder pads otherwise.

Schematic



OUT
VOLUME
500KA
GND
C3
10n
R5
1K
BIAS
10KB
AC128
Q2
FUZZ
1KC
C2
22uF
Reverse for NPN
R4
220R
CONTOUR
1kB
R2
33K
CX4
10pF
AC128
Q1
GAIN
500R
R3
100k
C1
2u2
CX2
10n
CX3
3u3
BODY
100KB
IN2
IN1
INPUT
250KB
CX1
10pF
RPD
2M2
IN
NPN1
NPN2
+9V
-9V
IC1
TC1044
C6
10uF
C5
10uF
D1
1N4742
PGND
C4
47uF
LEDR
4K7
LED
5MM
SW

General Build Instructions

These are general guidelines and explanations for all Aion Electronics DIY projects, so be aware that not everything described below may apply to this particular project.

Build Order

When putting together the PCB, it's recommended that you do not yet solder any of the enclosure-mounted control components (pots and switches) to the board. Instead, follow this build order:

1. Attach the **audio jacks**, **DC jack** and **footswitch** to the enclosure.
2. Firmly attach the **pots** and **switches** to the enclosure, taking care that they are aligned and straight.
3. Push the **LED**[1] into the hole in the enclosure with the leads sticking straight up, ensuring that the flat side is oriented according to the silkscreen on the PCB.
4. Fit the **PCB** onto all the control components, including the leads of the LED. If it doesn't fit, or if you need to bend things more than you think you should, double-check the alignment of the pots and switches.
5. Once you feel good about everything, **solder them from the top**[2] as the last step before wiring. This way there is no stress on the solder joints from slight misalignments that do not fit the drilled holes. You can still take it out easily if the build needs to be debugged, but now the PCB is "custom-fit" to that particular enclosure.
6. Wire everything according to the wiring diagram on the last page.

[1] **For the LED**: You can use a bezel if you'd like, but generally it's easier just to drill the proper size of hole and push the LED through so it fits snugly. If you solder it directly to the PCB, it'll stay put even if the hole is slightly too big. Make absolutely sure the LED is oriented correctly (the flat side matches the silk screen) before soldering, as it'll be a pain to fix later! After it's soldered, clip off the excess length of the leads.

[2] **Note on soldering the toggle switch(es):** It will require a good amount of solder to fill the pads. Try to be as quick as possible to avoid melting the lugs, and be prepared to feed a lot of solder as soon as the solder starts to melt. I recommend waiting 20-30 seconds between soldering each lug to give it time to cool down.

"RPD" and "LEDR" resistors

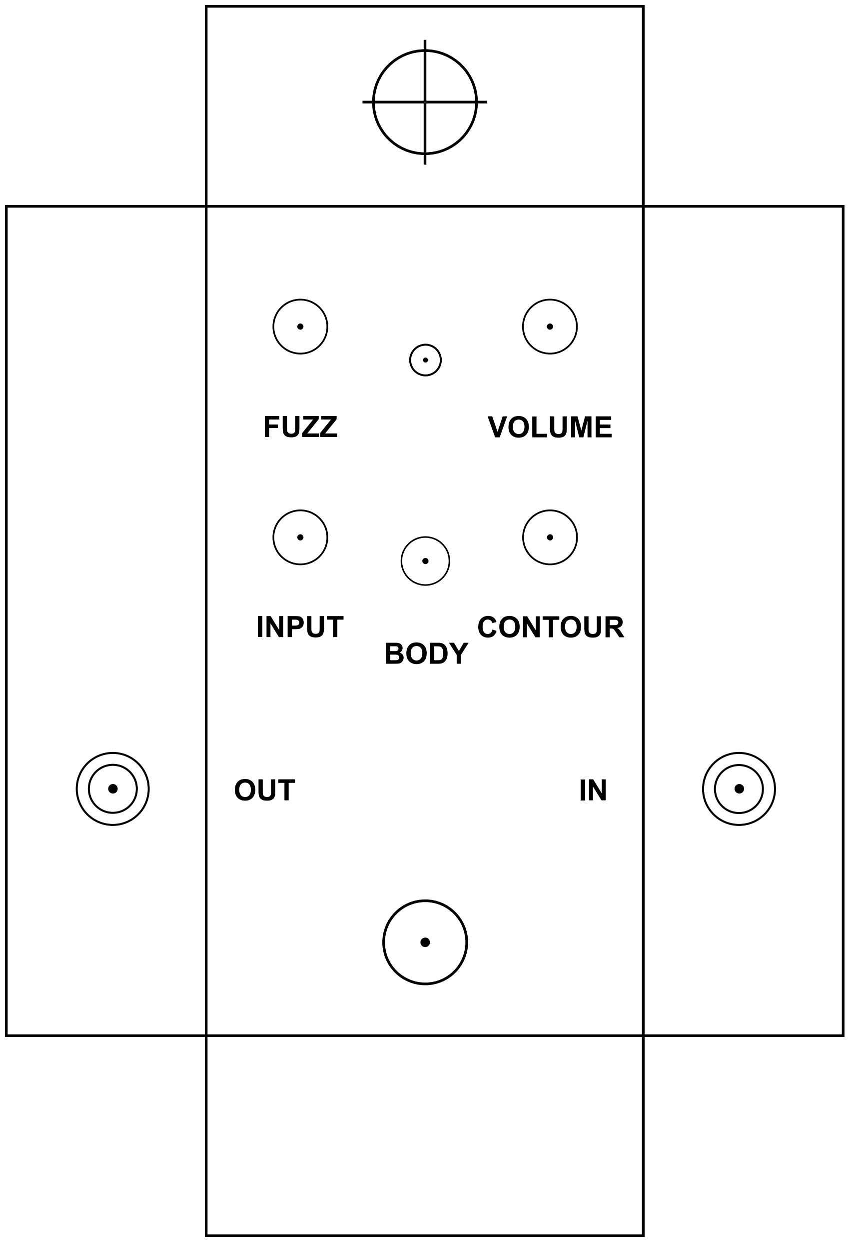
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

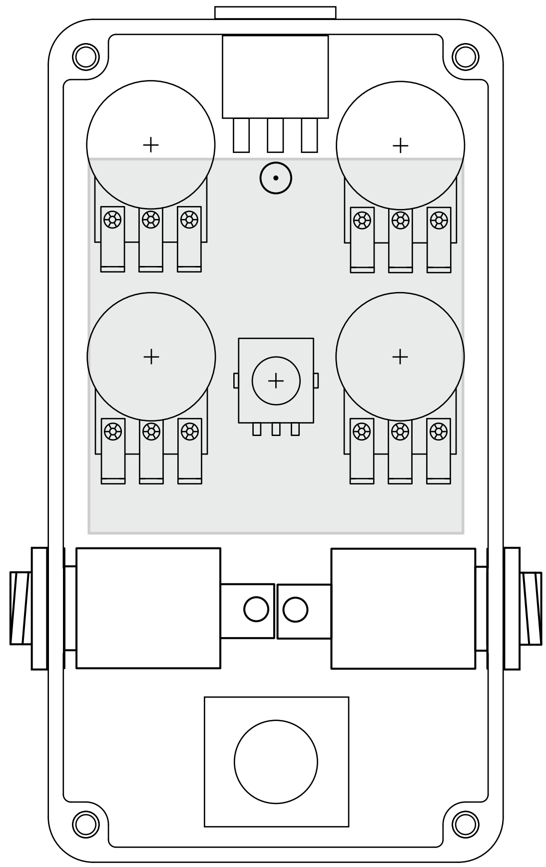
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

Drilling & Placement

Print this page and have an adult cut out the drilling template below for you. Tape it to the enclosure to secure it while drilling. Note that the holes are shown slightly smaller than they need to be, so drill out the holes as shown and then step up until they are the correct size for the components.



Hammond 1590B
(bottom/inside view)



Parts Used

- Switchcraft 111X enclosed jacks
- Kobiconn-style DC jack with internal nut

Standard Wiring Diagram

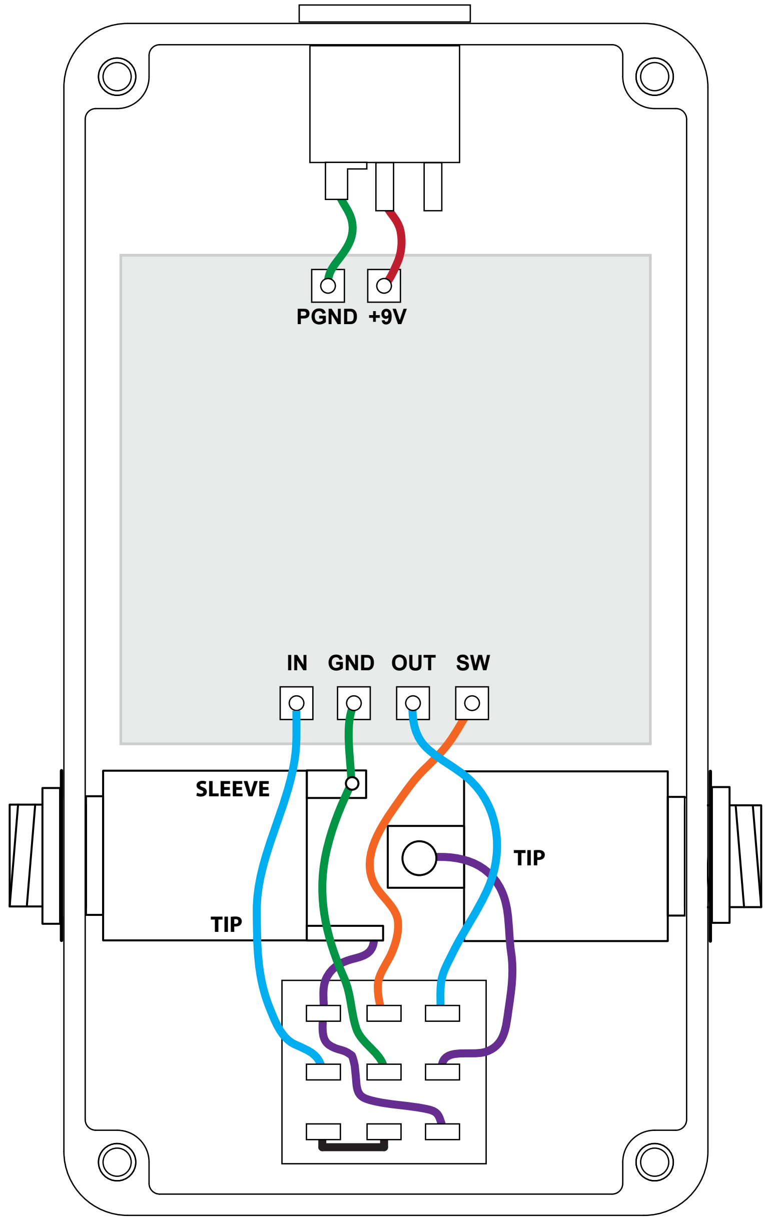
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

The **SW** pad is the cathode connection for the LED. This will connect to ground to turn it on when the switch is on. Usage of the on-board LED connection is not required if you have specific placement needs for your enclosure, but's incredibly convenient.

The wiring diagram also makes use of **star grounding** principles where all of the grounds connect to a single ground point (in this case the sleeve of the input jack). This is best practice to avoid added noise caused by improper grounding. The sleeve of the output jack is unconnected.

This PCB has been specially designed for separate power and signal grounds, which converge at the GND pad. Run the ground wire from the DC jack to the "PGND" pad at the top.

If using a painted or powdercoated enclosure, **make sure both jacks have solid contact with bare aluminum** for grounding purposes. You may need to sand off some of the paint or powdercoat on the inside in order to make this happen.



License / Usage

No direct support is offered for these PCBs beyond the provided documentation. It is assumed that you have at least some experience building pedals before starting one of these. Replacements and refunds will not be offered unless it can be shown that the circuit or documentation are in error. I have in good faith tested all of these circuits. However, I have not necessarily tested every listed modification or variation. These are offered only as suggestions based on the experience and opinions of others.

Projects may be used for commercial endeavors in any quantity unless specifically noted. No bulk pricing or discounting is offered. No attribution is necessary, though a link back is always greatly appreciated. The only usage restrictions are that **(1) you cannot resell the PCB as part of a kit**, and **(2) you cannot "goop" the circuit, scratch off the screenprint, or otherwise obfuscate the circuit to disguise its source**. (In other words: you don't have to go out of your way to advertise the fact that you use these PCBs, but please don't go out of your way to hide it. The guitar effects pedal industry needs more transparency, not less!)