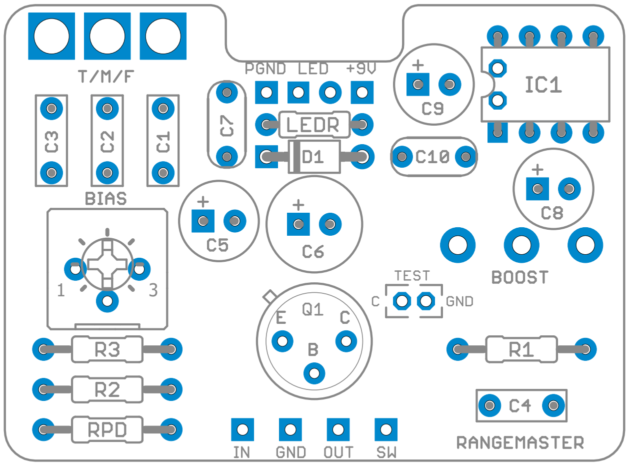
Radian

Based on the Dallas Rangemaster

Overview

Radian Project Link



The Radian is a modified version of the Dallas Rangemaster, a classic 1960s drive unit that is often called a “treble booster” but is really more of an overdrive or fuzz. Originally designed to add treble content to somewhat dark British amplifiers such as the Vox AC30, it took a life of its own in the hands of Brian May, Tony Iommi, Rory Gallagher, and several other highly-regarded guitarists of that era.

The Radian includes a modification to get additional tones out of the circuit. The original Rangemaster emphasized treble frequencies, but the Radian puts the frequency-setting capacitor on a switch, allowing you to change between the stock treble boost, a midrange boost, or a full-range boost.

Controls & Usage

The controls for the Radian are minimal:

- **Boost** controls the signal level of the effect.
- **Tone (T/M/F)** is a switch allowing you to choose between Treble (stock), Mid, or Full-Range boost.

Modifications & Experimentation

The tone switch allows you to put one of two additional capacitors in parallel with the input capacitor, **C1**. By using an on-off-on switch (also known as center-off), we can switch between three different capacitance values. Since the capacitors are arranged in parallel, C2 and C3 will add to C1’s capacitance value when each of them is engaged.

You can adjust **C2** and **C3** to taste, but if you use the suggested values, you’ll end up with the center position as .0047uf (4n7), the C2 position as approximately 27n, and the C3 position as approximately 105n. Each setting produces a very different flavor.

If you’re looking for more subtle tonal changes, perhaps keeping things in the classic treble frequency range but having some options to match better with different guitars and amps, you may choose to use much smaller values for C2 and C3. For instance, **2n2** and **4n7** will produce a very subtle change in frequency (0.0047uF, 0.0069uF, and 0.0094uF), or 4n7 and 10n will change it a little more (0.0047, 0.0094, and 0.0147).

Parts List

Mouser Parts List (Spreadsheet)

Resistors

R1	470k
R2	68k [1]
R3	1k [2]
RPD	2M2
LEDR	4k7

Capacitors

C1	4n7
C2	22n [3]
C3	100n [3]
C4	10n
C5	47uF electro
C6	100uF electro
C7	100n MLCC
C8	47uF electro
C9	47uF electro
C10	100n MLCC

Semiconductors

Q1	OC44 [4]
D1	1N4742A
IC1	LT1054CP [5]
LED	5mm LED

Potentiometers

BOOST	10kA
BIAS	5k trimmer (3386)

Switches

T/M/F	SPDT center off

Build Notes

[1] **Bias resistor value:** Some transistors may require R2 to be reduced to **47k** in order to bias correctly.

[2] **Bias resistor value:** R3 is a 3.9k resistor in the original, but since this implementation adds a 5k bias trimmer, the stock resistor is reduced in value.

[3] **Input capacitors:** C2 and C3 work in conjunction with the Tone switch to provide a different flavor than just the stock treble boost. These are the recommended values, but read the "Modifications and Experimentation" section on the previous page for a discussion of how these could be modified.

[4] **Germanium transistor:** Any PNP germanium transistor with a **gain (HFe) between 65 and 100** will work here. The exact model is not important (and even varied in the original unit).

[5] **Charge pump:** The LT1054CP is recommended, but you can also use a slightly cheaper TC1044SCPA here. If you use the TC1044, make sure to **jumper the two pads** between pins 1 and 8 of the IC or it may be noisy.

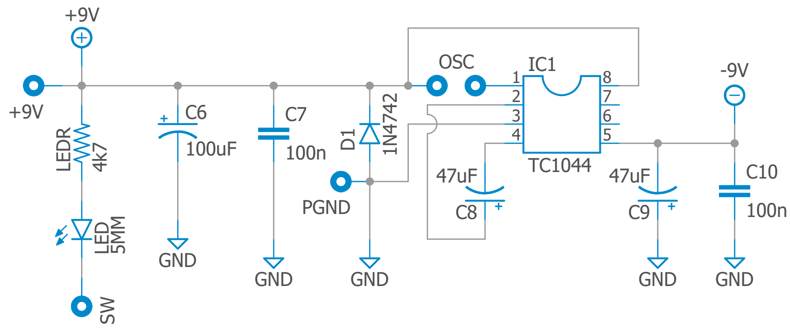
Biasing

Adjust the "BIAS" trimmer until the Q1 collector reads between **-6.8V** and **-7.1V**. The "TEST" pads to the right of the transistor are provided to make this process simpler. Just put the red lead of your DMM on the left pad (marked "C") and the black lead on the right pad (marked "GND") while you adjust the trimmer.

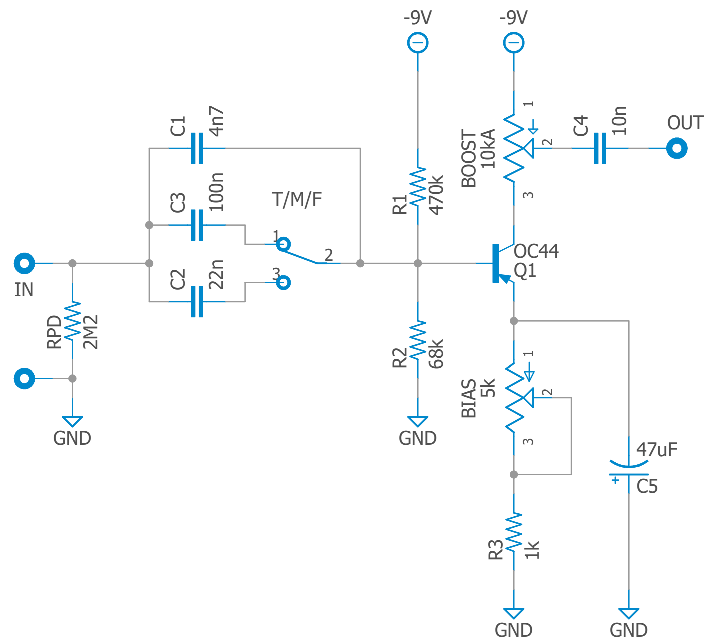
Additional Part Notes

- 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 1uF. I prefer WIMA box film: the FKS2 series for 1n to 10n and the MKS2 series for 10n to 1uF.
- 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



+9V
+9V
LEDR
4k7
LED
5MM
SW
C6
100uF
GND
C7
100n
GND
D1
1N4742
PGND
GND
OSC
IC1
TC1044
47uF
C8
47uF
C9
GND
-9V
C10
100n
GND



-9V
-9V
C1
4n7
C3
100n
T/M/F
C2
22n
IN
RPD
2M2
GND
R1
470k
R2
68k
GND
BOOST
10kA
C4
10n
OUT
OC44
Q1
BIAS
5k
47uF
C5
R3
1k
GND
GND

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

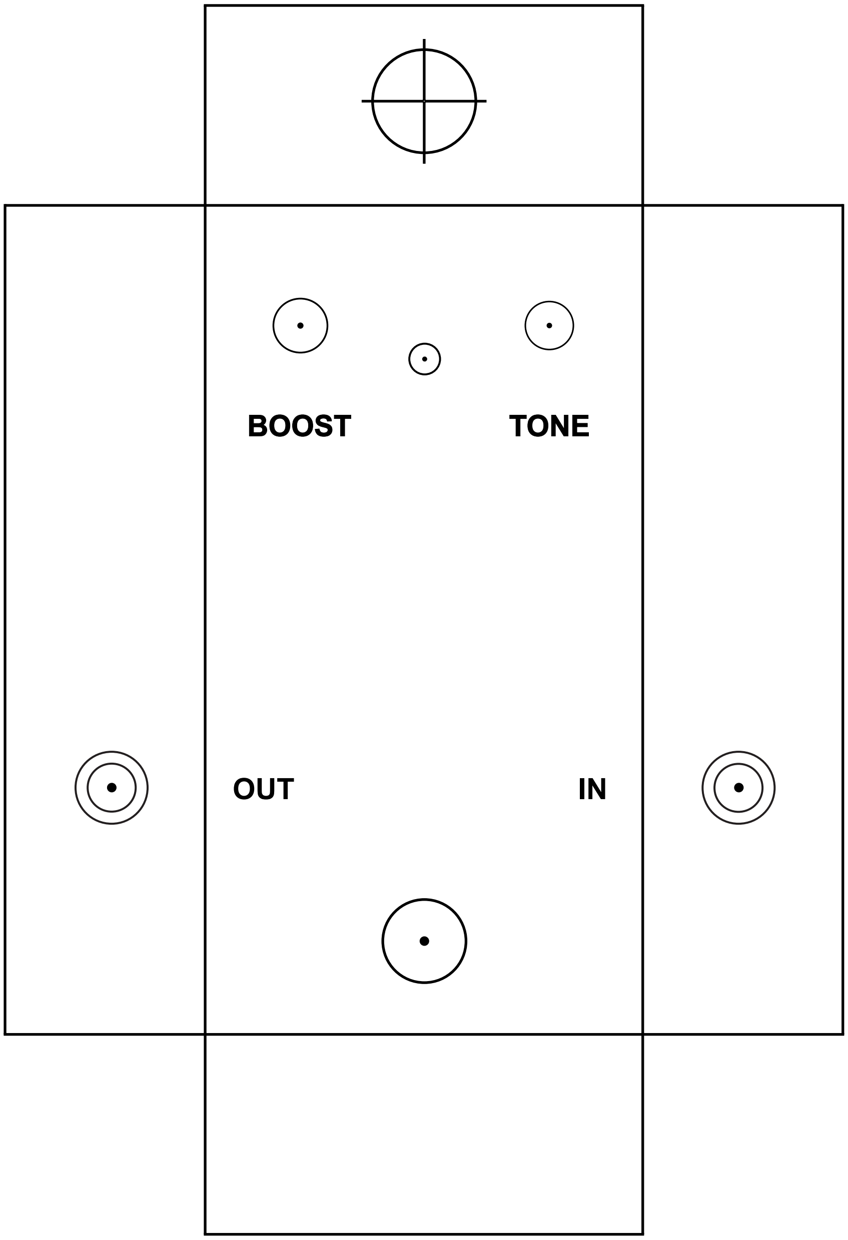
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

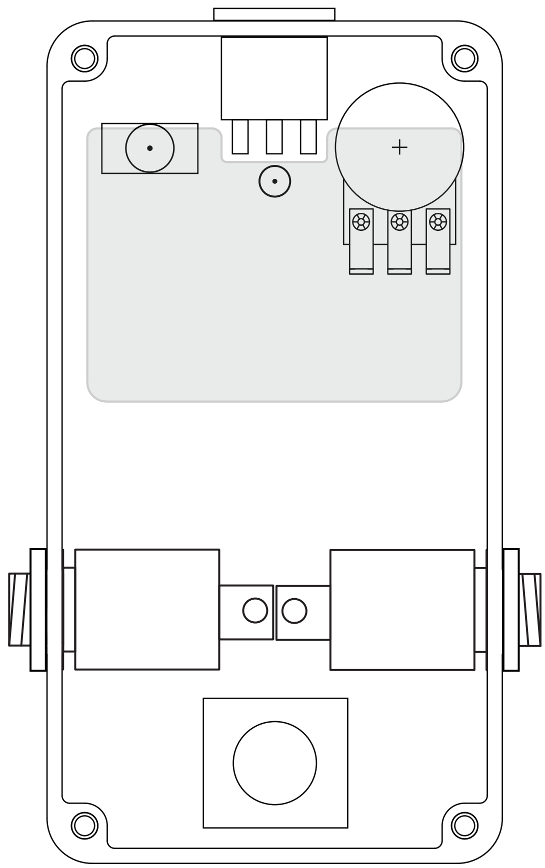
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

Drilling & Placement

Print this page and cut out the drilling template below. 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

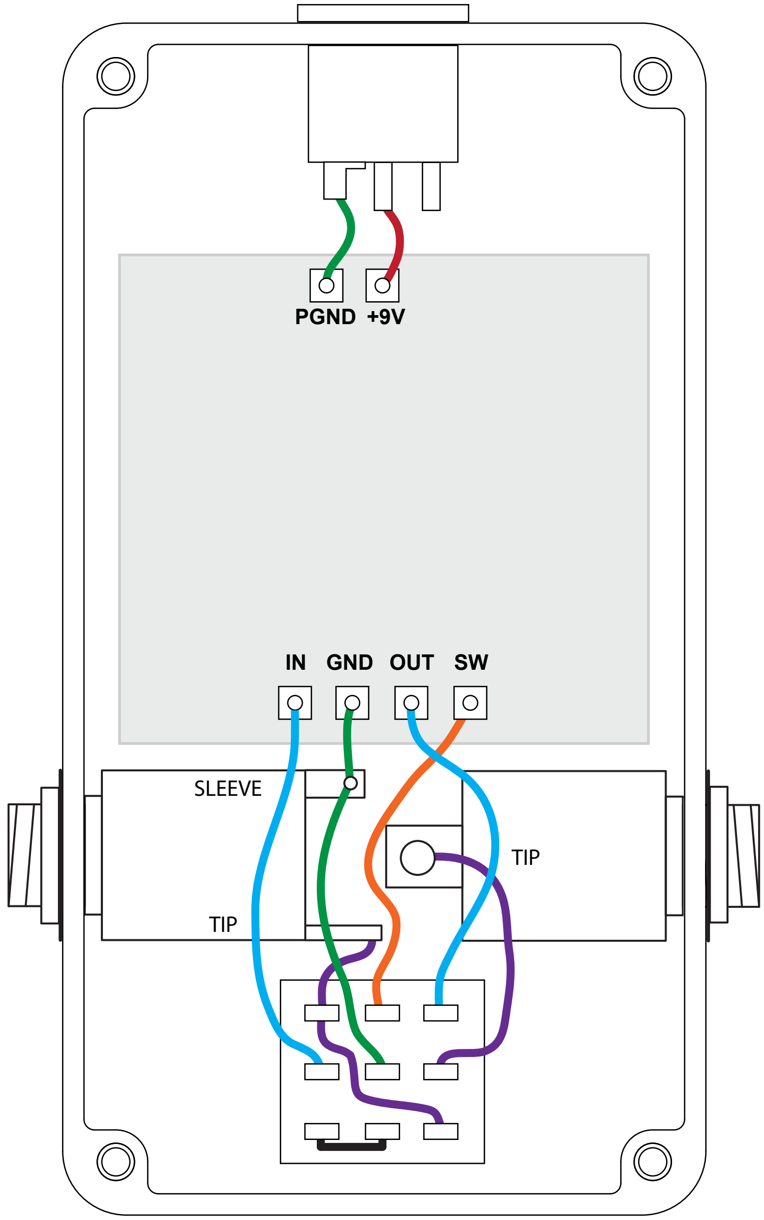
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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.

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.

Make sure to double-check the markings of the pads on the PCB for your particular project – they are not always in the order shown here!



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 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!)