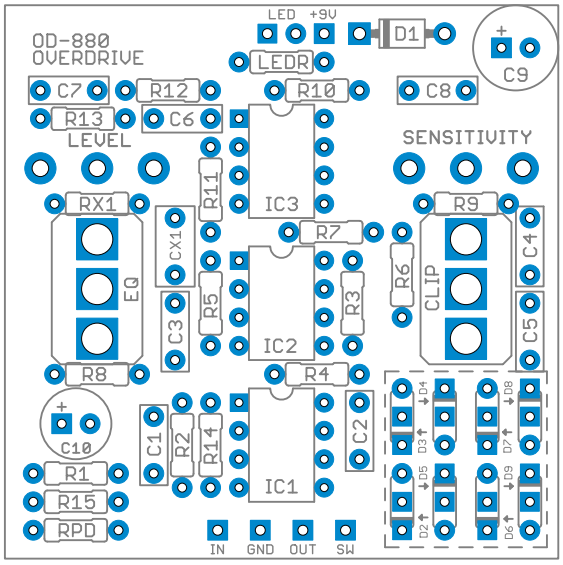
Quark Overdrive

Maxon OD-880 Overdrive / Soft Distortion

Overview



The Quark Overdrive is a clone of the Maxon OD-880 Overdrive / Soft Distortion, originally released in 1976. This pedal was huge in Japan, being the favored overdrive pedal of Char, one of the country's most popular guitarists, who—like the pedal—never really got noticed out of Japan. It was reissued a few times, notably in the mid-90's as well as in the past couple of years.

It's an interesting circuit from a historical perspective, being sort of a "transitional species" between the Distortion+ and the Boss OD-1 OverDrive: it added the input and output buffers and changed the Drive potentiometer to a flat op-amp boost instead of a gain modifier that also affected the bass tone, like in the Dist+. The OD-1 is essentially an OD-880 with feedback clipping diodes instead of diode-to-ground clipping, though this diode arrangement turned out to be a very big deal and made way for the Tube Screamer a couple of years later.

Controls & Usage

The OD-880's controls are as simple as they come:

- **Sensitivity** controls the amount of gain in the op amp stage that is later hard-clipped.
- **Level** controls the overall output.

Modifications & Experimentation

The **Clipping** switch mod allows you to set up a second set of diodes to toggle back and forth from stock. Extra pads have been provided so you can stack two diodes in a row if desired. (The middle two pads are connected in each diode.) If you use a SPDT center-off switch, the middle position becomes a diode lift mode, but you can also use a regular SPDT if you don't care about this.

The **EQ** switch changes the tone stack from stock to modded. The biggest criticism of this circuit is that it cuts too much bass, so this mod gives more bass and fullness and changes the overall gain structure of the effect. Note that this mod doesn't have an "off" mode like a clipping switch or a capacitor mod would, so don't use a center-off switch here. Just use a normal two-position SPDT.

The original uses a few **LM741** op amps. These are fine, and the comparatively low fidelity of these op amps is part of the sound, like in the Distortion+. However, if you want to experiment, you can swap in any single op amp here such as a **TL061/TL071**, **CA3130EZ/CA3140EZ**, or a **NE5534**.

Parts

Resistors

R1	10k
R2	470k
R3	10k
R4	10k
R5	470k
R6	100R
R7	100k
R8	4k7
R9	10k
R10	10k
R11	9k1
R12	100k
R13	68k
R14	10k
R15	10k
RX1	3k3 [1]
RPD	1M to 2M2
LEDR	4k7

Capacitors

C1	47n
C2	47n
C3	47n
C4	100n
C5	100n
C6	4n7
C7	100n
C8	100n
C9	47uF electro
C10	10uF electro
CX1	220n [1]

Semiconductors

IC1–IC3	LM741P
D1	1N4002
D3, D4	1N914
D2, D5	jumper
D5–8	1N914 [2]
LED	5mm LED

Potentiometers

Sensitivity	500kB [3]
Level	10kB

Other

Clip	SPDT center off
EQ	SPDT [1]

Build Notes

[1] **Important:** If you're not using the EQ mod, you need to jumper the middle and bottom pads of the EQ switch for the effect to work properly. If you do use this mod, make sure to use a **regular on-on switch** rather than a center-off switch. The effect would pass only a very weak signal in the center-off position.

The values for **RX1** and **CX1** are arbitrary and you can use whatever you'd like for these. I would recommend something that increases the bass, but **3k3** and **220n** are just the values I settled on.

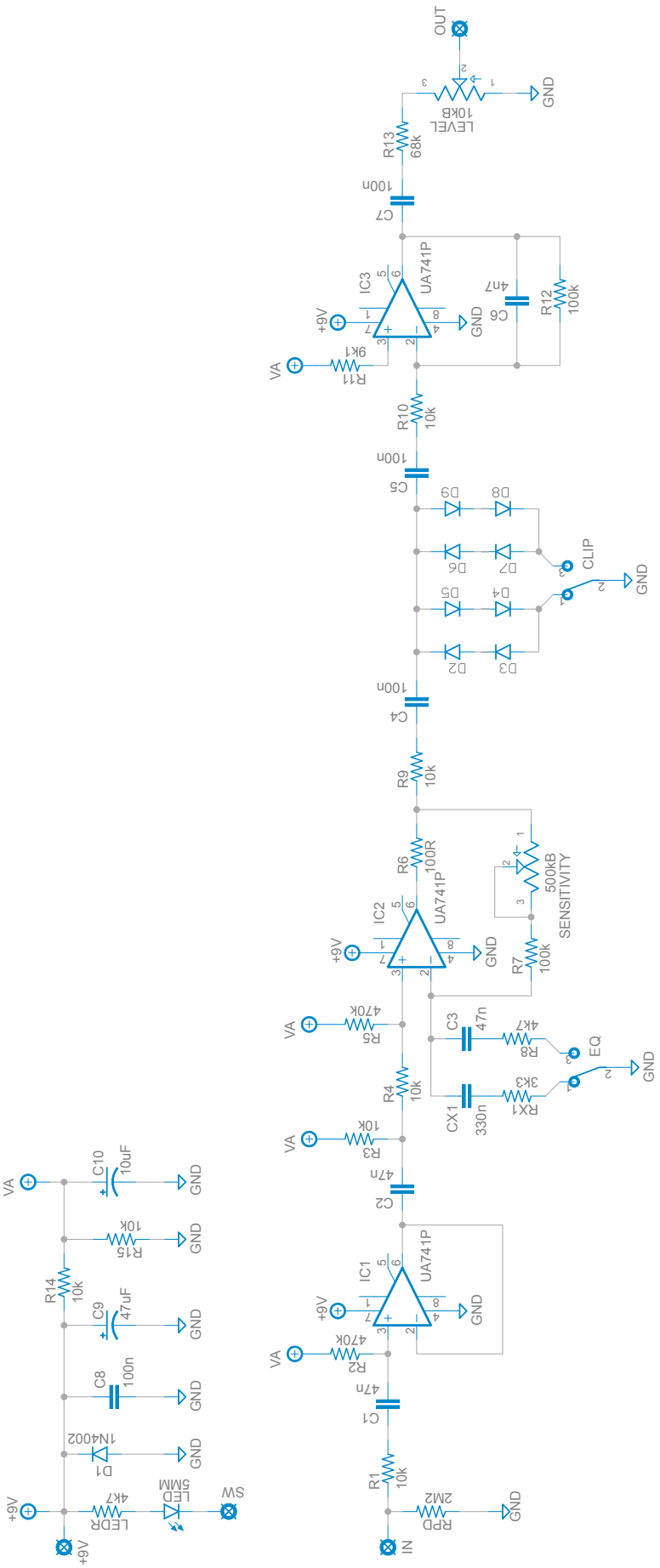
[2] **Clipping diodes:** Try two 1N914s in series for the second side. This will give more output volume and open it up a little bit, so it has more dynamics and less compression.

[3] **Sensitivity pot taper:** The original uses a 500k linear taper (B) pot here, but I found that audio taper (A) gave a smoother range of control.

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.
- Switches are Taiway (Small Bear) or Mountain Switch (Mouser) brand with solder lugs. I prefer the short-toggle variety, but that's just a matter of aesthetics.
- 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
LEVEL
10KB
GND
R13
68k
C7
100n
IC3
UA741P
C6
4n7
R12
100k
+9V
GND
VA
R11
9k1
R10
10k
C5
100n
D9
D8
D6
D7
CLIP
D5
D4
D3
D2
GND
C4
100n
R9
10k
R6
100R
IC2
UA741P
500kB
SENSITIVITY
R7
100k
+9V
GND
VA
R5
470k
C3
47n
R8
4k7
EQ
GND
R4
10k
CX1
330n
RX1
3k3
VA
R3
10k
C2
47n
IC1
UA741P
+9V
GND
VA
R2
470k
C1
47n
R1
10k
RPD
2M2
GND
IN
VA
C10
10uF
GND
R15
10k
GND
R14
10k
C9
47uF
GND
C8
100n
GND
D1
1N4002
GND
+9V
LEDR
4k7
LED
5MM
SW
+9V

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

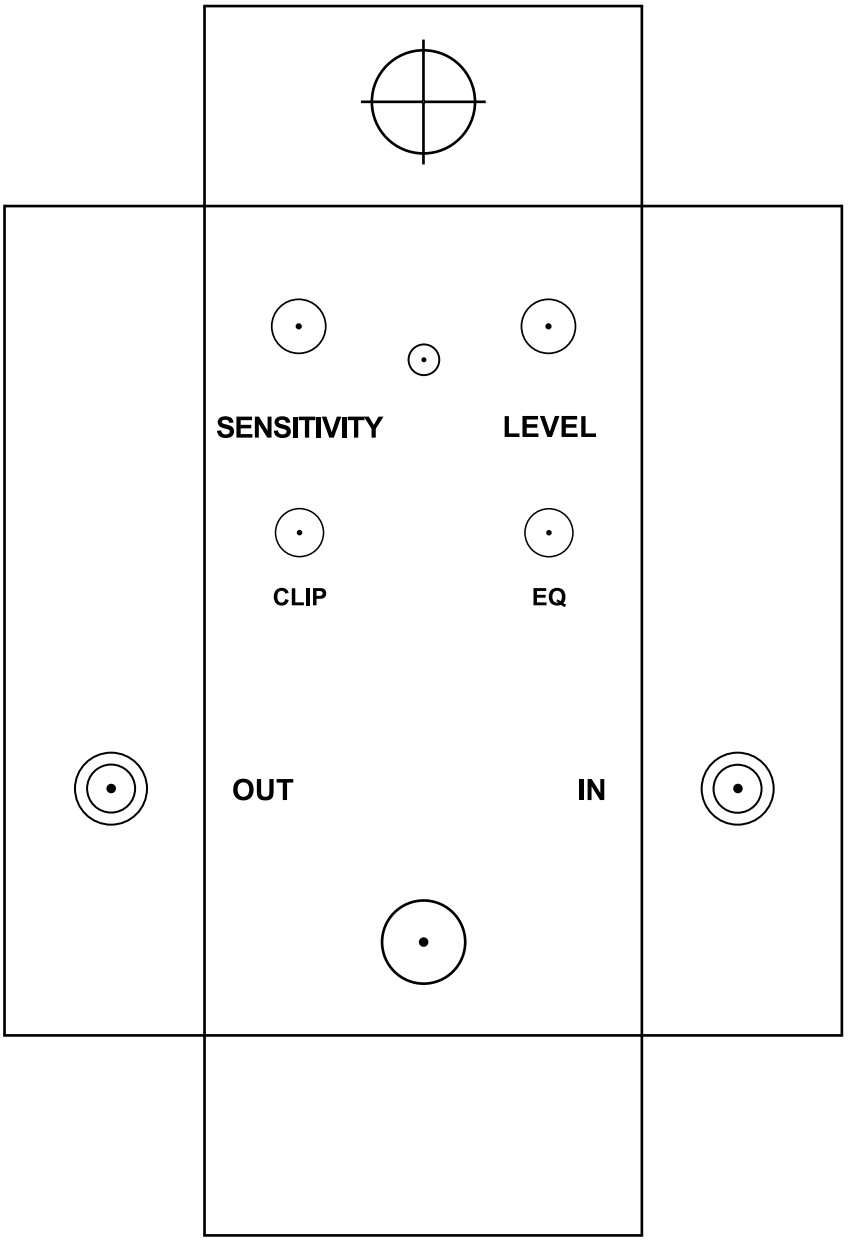
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

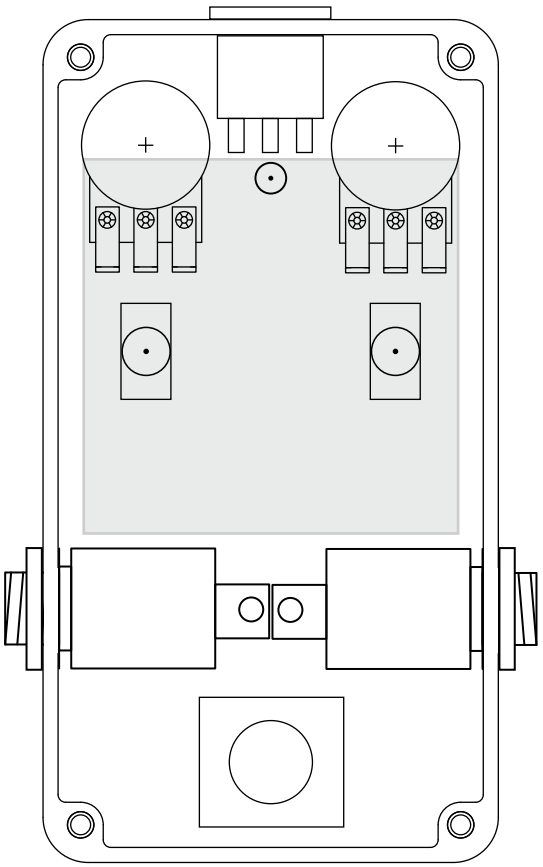
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

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

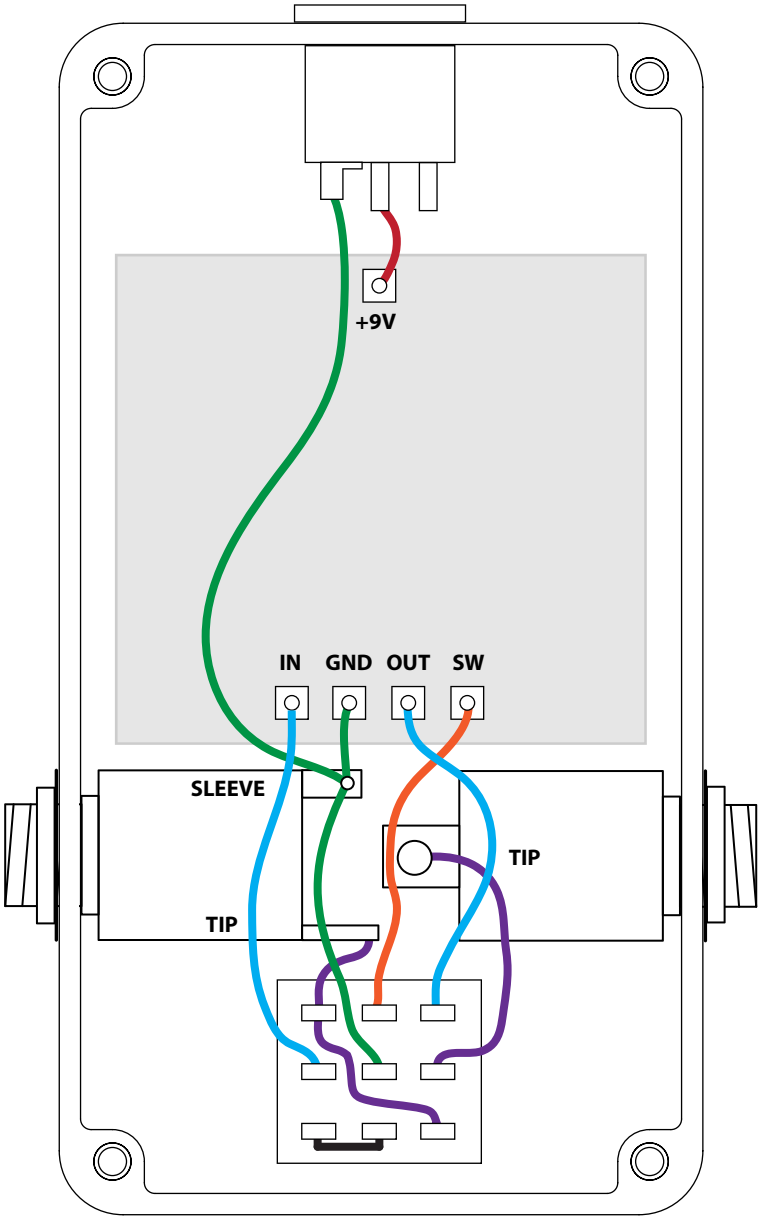
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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.

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 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!)