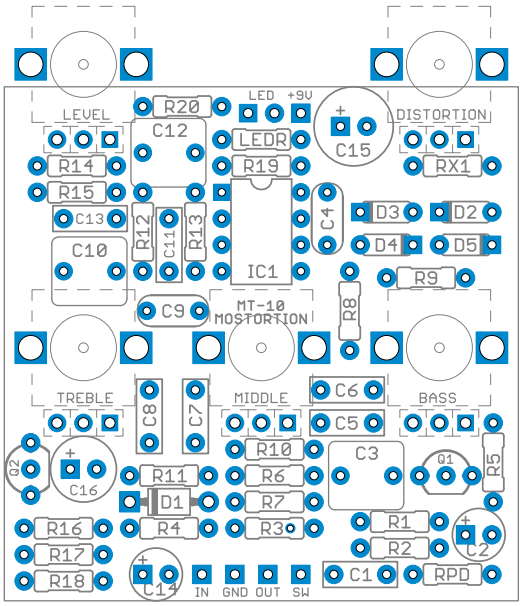
Quantum Distortion

Ibanez MT-10 Mostortion

Overview



The Quantum Distortion is a clone of the Ibanez MT-10 Mostortion MOS-FET Distortion, a fairly obscure pedal in Ibanez's "10 series" released in 1990 and discontinued in 1993.

Also discontinued is the MOSFET op amp used in the design, a CA3260. This op amp is the only MOSFET component in the pedal, and there are no suitable current-production substitutes, so if you're building a clone, you'll have to hop on eBay and find one.

That said, the MOSFET op amp isn't the only thing unique about this circuit and it's still worth building even if you can't find one. It won't be a MOSFET distortion, but it will still have its own character whether you use the CA3260 or a JRC4558 or anything else, and who knows—you may find you like one of those better!

Controls & Usage

This pedal is unique in that it's the only Ibanez pedal with a 3-band tonestack. You don't see that too often in distortions & overdrives in general—the Marshall Guv'nor and Shredmaster are the only ones that come to mind—and this one is a lot different because it uses feedback clipping diodes like a Tube Screamer.

- **Distortion** controls the amount of gain from the op amp that is fed through the feedback clipping diodes.
- **Level** is the output level of the effect.
- **Bass** allows you to adjust the bass response of the circuit.
- **Middle** allows adjustment of the midrange of the circuit.
- **Treble** allows adjustment of the treble of the circuit.

Modifications

The **CA3260** is the only dual op-amp I know of that has MOSFET inputs and CMOS outputs. Most op-amps that are called MOSFETs are actually just MOSFET input and bipolar output, or something else. That said, other MOSFET op-amps include the **CA3240** and **TLC2262**, both of which are in production and much easier to find than the CA3260. Just don't think of them as direct substitutes.

Any other standard-pinout dual op-amps will work in this circuit as well. Try a **JRC4558**, **TL072** or **OPA2104**.

The stock clipping diodes in the MT-10 are two 1N914s in series, which will give a more transparent and less compressed clipping. However, if you use a non-MOSFET op amp, you might try one 1N914 in either direction (Tube Screamer / symmetrical clipping), or two in one direction and one in the other (SD-1 / asymmetrical).

Like a Tube Screamer, you can adjust **R8** and **C5** to change the gain structure and the low-end rolloff frequency. Keep in mind, though, that with a 3-band tonestack you'll have plenty of control over the bass later on in the circuit.

Parts

Resistors

R1	1k
R2	510k
R3	9k1
R4	22k
R5	10k
R6	220R
R7	1M
R8	2k7
R9	47k
R10	10k
R11	1M
R12	10k
R13	47k
R14	1k
R15	510k
R16	10k
R17	470R
R18	100k
R19	10k
R20	10k
RX1	10k [1]
RPD	1M to 2M2
LEDR	4k7

Capacitors

C1	22n
C2	10uF electro
C3	1uF film
C4	51pF (or 47pF) MLCC
C5	220n
C6	68n
C7	33n
C8	15n
C9	330pF MLCC
C10	1uF film
C11	1n
C12	1uF film
C13	100n
C14	10uF electro
C15	100uF electro
C16	47uF electro

Semiconductors

Q1–Q2	2N5088
IC1	CA3260 [2]
D1	1N4002
D2–D5	1N914
LED	5mm LED

Potentiometers

Distortion	500kA 9mm
Volume	100kB 9mm
Bass	250kA 9mm
Middle	50kA 9mm
Treble	250kA 9mm

Build Notes

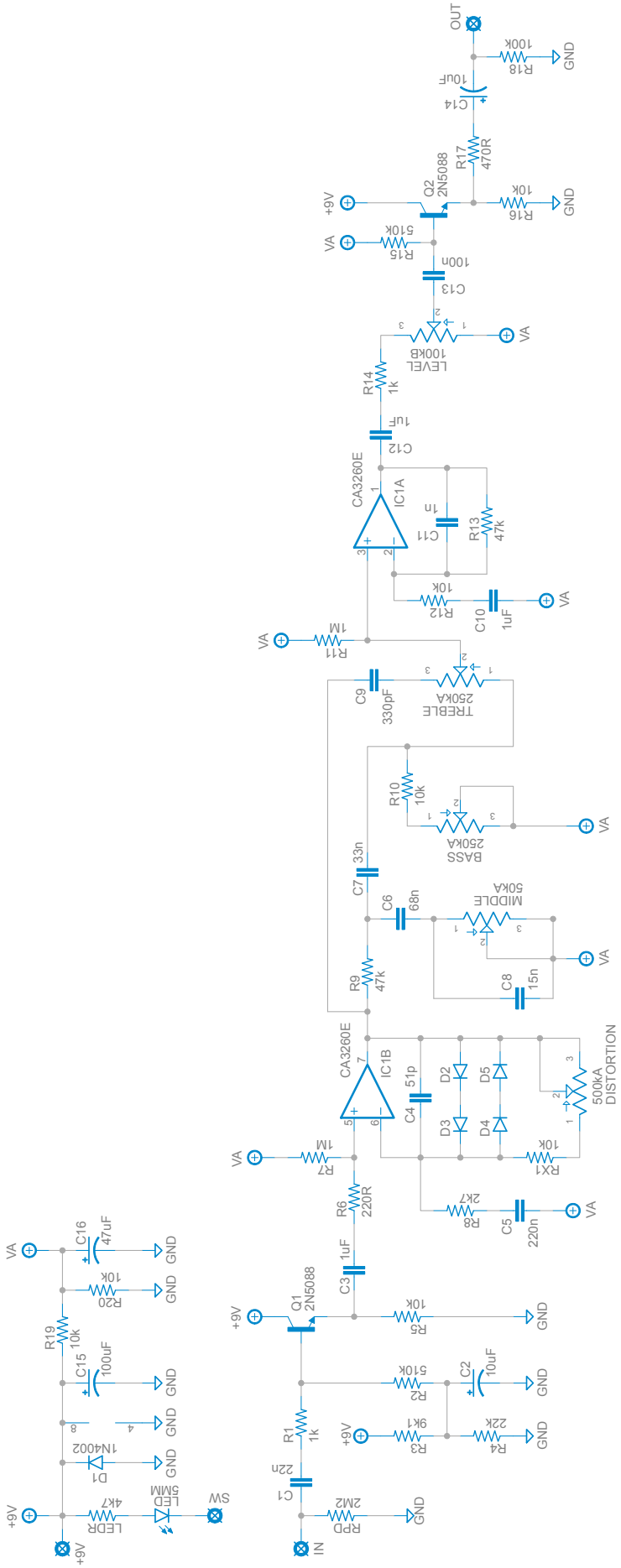
[1] **Optional minimum-drive resistor:** The original circuit doesn't have a resistor to set the minimum drive. This is the only op-amp overdrive or distortion circuit I've seen that leaves off this resistor. I'd recommend setting it to 10k as shown, but if you want to be 100% accurate to the original then jumper it.

[2] **CA3260 availability:** This is an obsolete chip, but only since around 2005 so it's not terribly hard to find. The best place I've found to get them is eBay.

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.
- 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
100K
R18
GND
10uF
C14
470R
R17
Q2
2N5088
+9V
10K
R16
GND
VA
510K
R15
100n
C13
VA
100KB
LEVEL
R14
1k
1uF
C12
CA3260E
IC1A
1n
C11
R13
47k
10K
R12
C10
1uF
VA
VA
1M
R11
C9
330pF
250KA
TREBLE
R10
10k
250KA
BASS
VA
33n
C7
C6
68n
50KA
MIDDLE
VA
R9
47k
C8
15n
CA3260E
IC1B
51p
C4
D2
D3
D5
D4
500kA
DISTORTION
VA
1M
R7
10K
RX1
R6
220R
2K7
R8
C5
220n
VA
C16
47uF
GND
10K
R20
GND
R19
10k
C15
100uF
GND
GND
8
4
1N4002
D1
GND
+9V
4K7
LEDR
LED
5MM
SW
+9V
C3
1uF
Q1
2N5088
+9V
10K
R5
GND
510K
R2
C2
10uF
GND
R1
1k
+9V
9K1
R3
22K
R4
GND
22n
C1
2M2
RPD
GND
IN

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

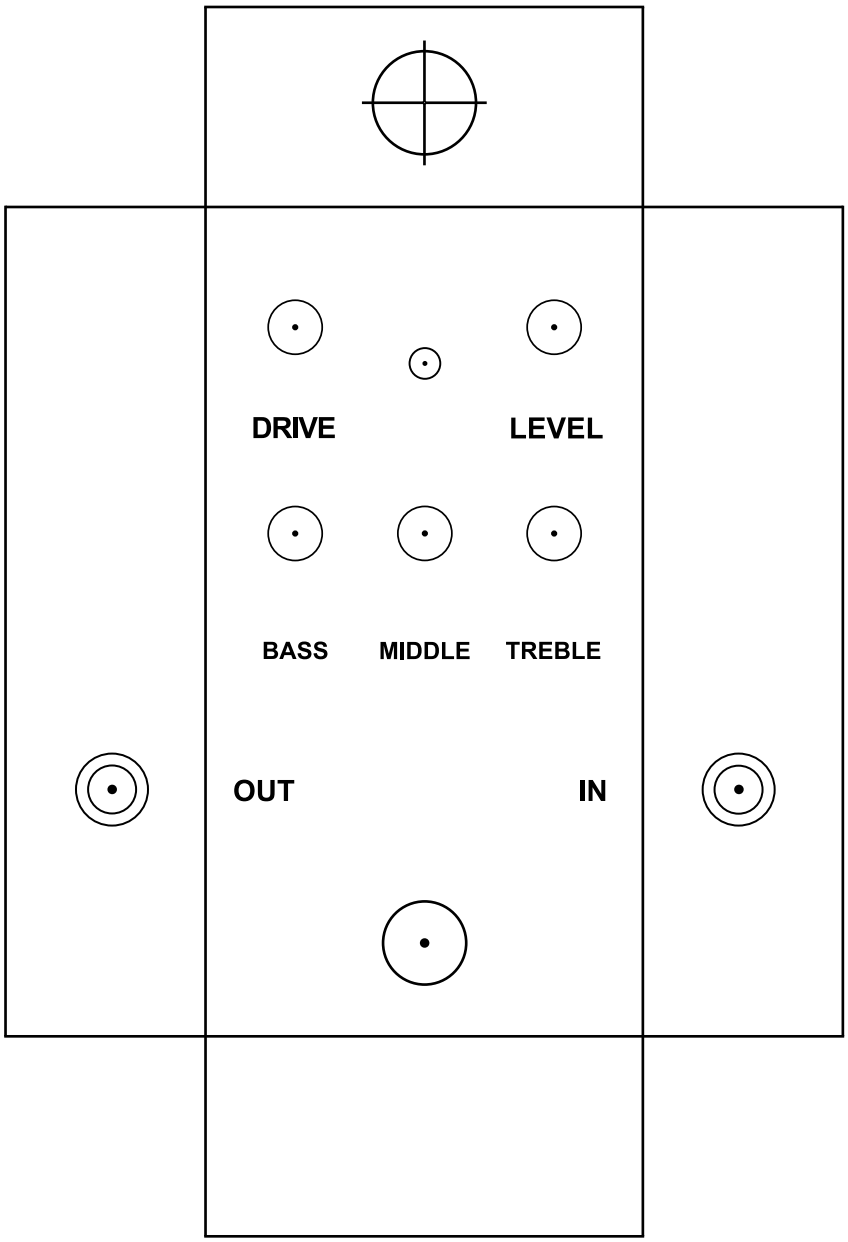
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

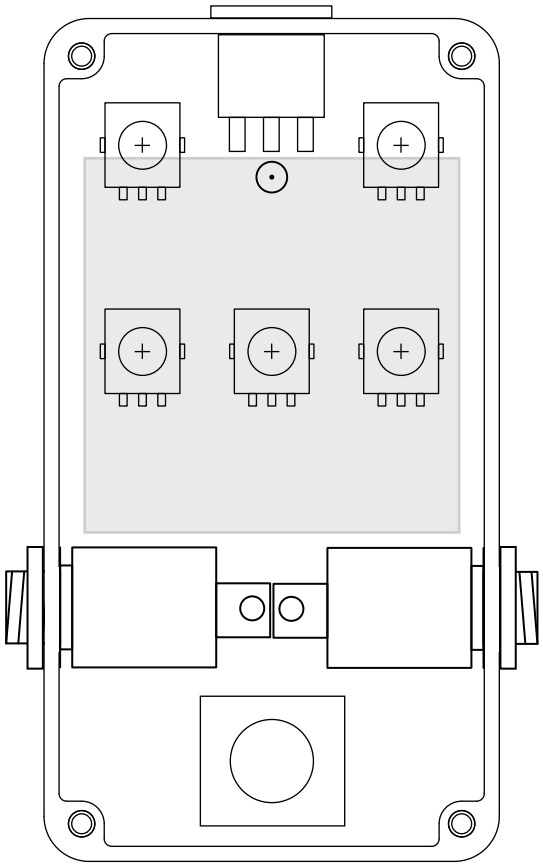
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

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

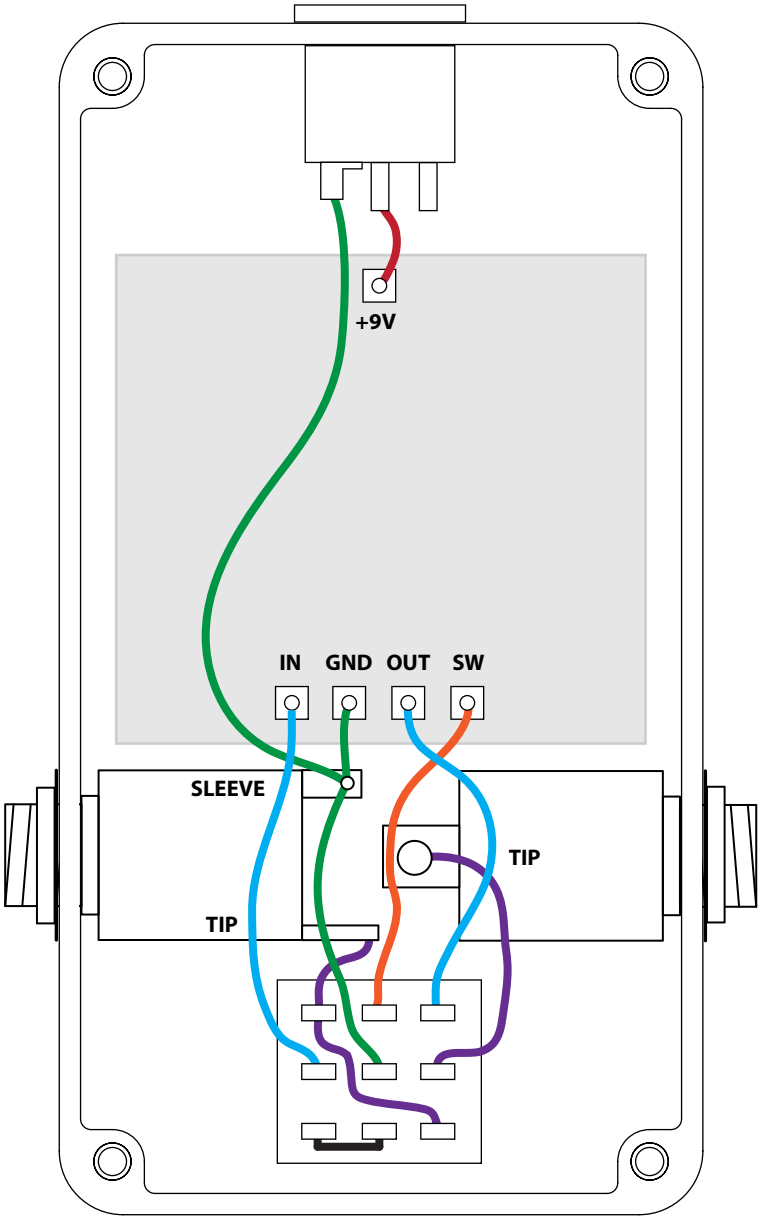
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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!)