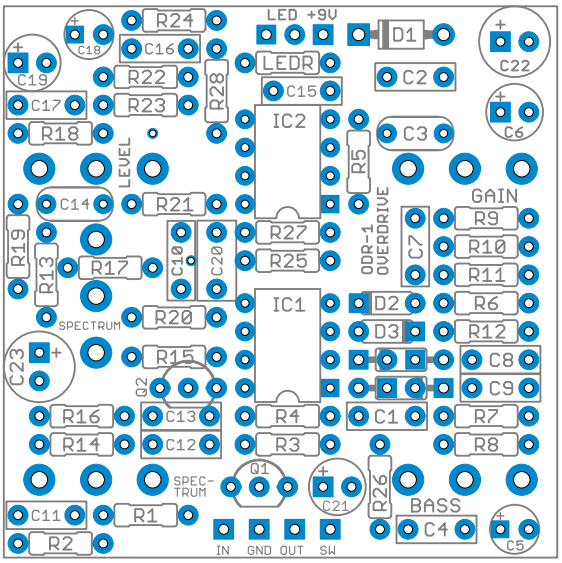
Andromeda

Nobels ODR-1 Natural Overdrive

Overview

Andromeda Project Link



The Andromeda Overdrive is a clone of the Nobels ODR-1, a somewhat obscure pedal from Germany that nonetheless is considered a “secret weapon” among Nashville studio musicians.

The primary complaint of the ODR-1 is that it has way too much bass with no way to dial it out. To that end, this PCB project incorporates a variable bass control inspired by the Timmy and Zen Drive. The stock tones are still there with the knob all the way up, but now you have the option to turn it down.

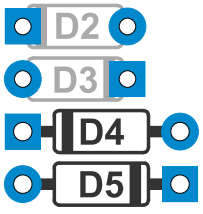
Looking at the schematic, you can see a stark difference with Japanese-designed pedals (i.e. most Ibanez & Boss circuits). This is German engineering at its finest—tons of odd-value resistors and capacitors forming very precise filters and signal shaping. This is not a particularly simple build, but it’s very rewarding and well worth the effort.

Controls & Usage

- **Gain** 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. The stock ODR-1 circuit is the equivalent of having the Bass knob turned all the way up.
- **Spectrum** is a very non-standard tone control of sorts. You can pan between a 700 Hz lowpass filter and a 5 KHz high-pass filter, which is mixed with a fixed 2.1 MHz frequency boost (upper mids). This control is much more complex than your standard hi-cut or Tube Screamer style tone control.

Modifications

This is a tight layout, so you won’t find any of the “standard” Aion modifications such as a clipping diode switch. However, there are extra pads for the hard clipping diodes (D4 and D5) in case you want to stack two diodes in series—for instance, **two 1N914s** or **one 1N914 and one BAT41** on each side. This will reduce the hard clipping of the circuit and result in a less compressed sound and more output volume.



Note that **D4** and **D5** are not labeled on the PCB due to the extra pads. They are located **below D2 and D3** and are oriented as shown to the right.

The **Bass** knob is an addition to this circuit, and while you can omit this control and use the alternate Spectrum orientation for a symmetrical control layout, I don’t recommend it—as mentioned earlier, the main criticism of this circuit is that it has too much bass.

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

Like a Tube Screamer, you can adjust **R7** + **C4** and **R8** + **C5** to change the gain structure and the low-end rolloff frequency.

Parts

Capacitors

C1	68n
C2	22n
C3	120pF MLCC
C4	82n
C5	2u2 electro
C6	2u2 electro
C7	2n7
C8	82n
C9	1n
C10	22n
C11	27n
C12	100n
C13	8n2
C14	560pF MLCC
C15	8n2
C16	4n7
C17	82n
C18	1u electro
C19	2u2 electro
C20	470n
C21	3u3 electro
C22	100uF electro [1]
C23	47uF electro
C24	47uF electro [2]

Resistors

R1	33k
R2	1M
R3	3k3
R4	2k7
R5	10k
R6	1k8
R7	820R
R8	1k5
R9	12k
R10	39k
R11	10k
R12	12k
R13	5k1
R14	2k2
R15	150k
R16	3k3
R17	1k2
R18	43k
R19	10k
R20	20k
R21	4k7
R22	22k
R23	5k1

Resistors, cont.

R24	1k2
R25	150k
R26	150k
R27	15k
R28	15k
LEDR	4k7

Semiconductors

Q1	2N5457
Q2	2N5088
IC1–IC2	JRC4558D
D1	1N4002
D2–D5	1N914
LED	5mm LED

Potentiometers

Gain	250kA
Level	50kA
Spectrum	25kB
Bass	50kC [3]

Build Notes

[1] The stock circuit has a generous **220uF** for the main filter capacitor (C22). I dropped this to a more reasonable 100uF. There shouldn't be any increase in noise, but I wanted to make mention of it.

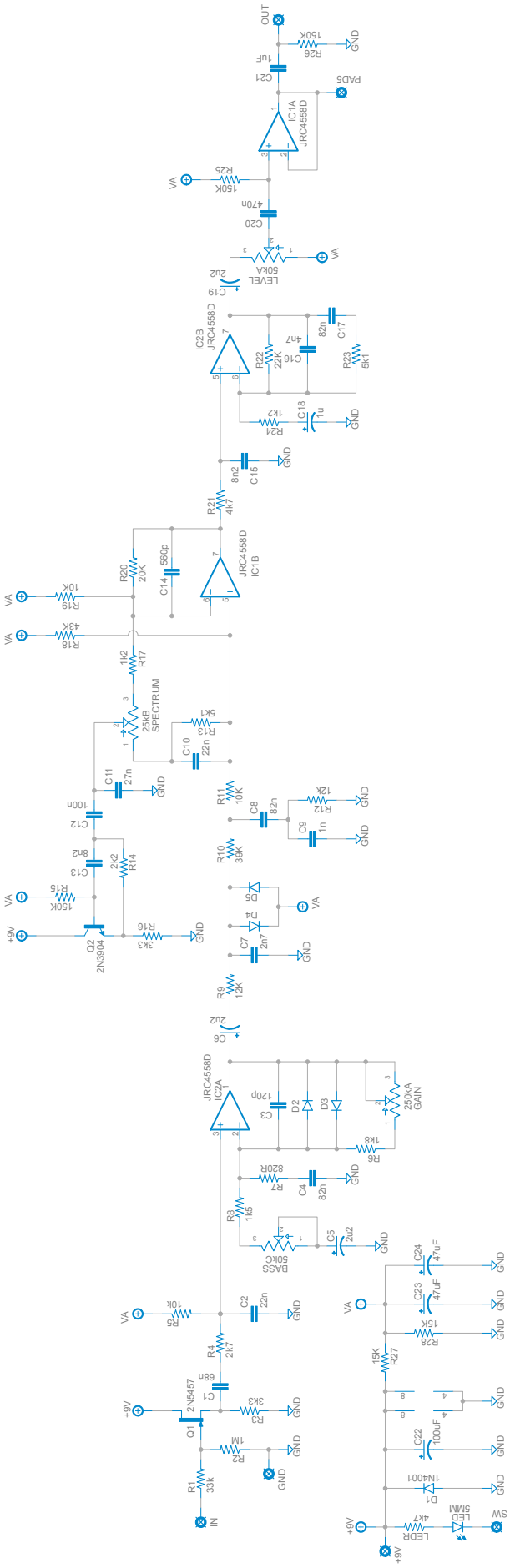
[2] I added **C24** to counteract some low-frequency oscillations at high drive settings. 22uF is probably OK here.

[3] The **Bass knob** is not present in the original, but I'd recommend including it. If you do leave it off, you'll need to **jumper pads 2 and 3** of the Bass pot, and then use the center orientation of the Spectrum 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.
- 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
150K
R26
GND
1uF
C21
PAD5
IC1A
JRC4558D
VA
R25
150K
470n
C20
50KA
LEVEL
VA
4u2
C19
IC2B
JRC4558D
82n
C17
4n7
C16
R22
22K
R23
5k1
1k2
R24
C18
1u
GND
8n2
C15
GND
R21
4k7
JRC4558D
IC1B
560p
C14
R20
20K
10K
R19
VA
43K
R18
VA
1k2
R17
25kB
SPECTRUM
5k1
R13
C10
22n
C11
27n
GND
100n
C12
R11
10K
C8
82n
12K
R12
GND
C9
1n
GND
8n2
C13
2k2
R14
R10
39K
150K
R15
VA
D5
D4
VA
+9V
Q2
2N3904
R16
3k3
GND
C7
2n7
GND
R9
12K
4u2
C6
JRC4558D
IC2A
120p
C3
D2
D3
250kA
GAIN
1k8
R6
820R
R7
C4
82n
GND
R8
1k5
50KC
BASS
C5
2u2
GND
VA
10K
R5
C2
22n
GND
R4
2k7
68n
C1
2N5457
Q1
+9V
3k3
R3
GND
1M
R2
GND
R1
33k
IN
GND
C24
47uF
GND
VA
C23
47uF
GND
15K
R28
GND
15K
R27
GND
C22
100uF
GND
1N4001
D1
GND
LED
5MM
SW
+9V
LEDR
4k7
+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

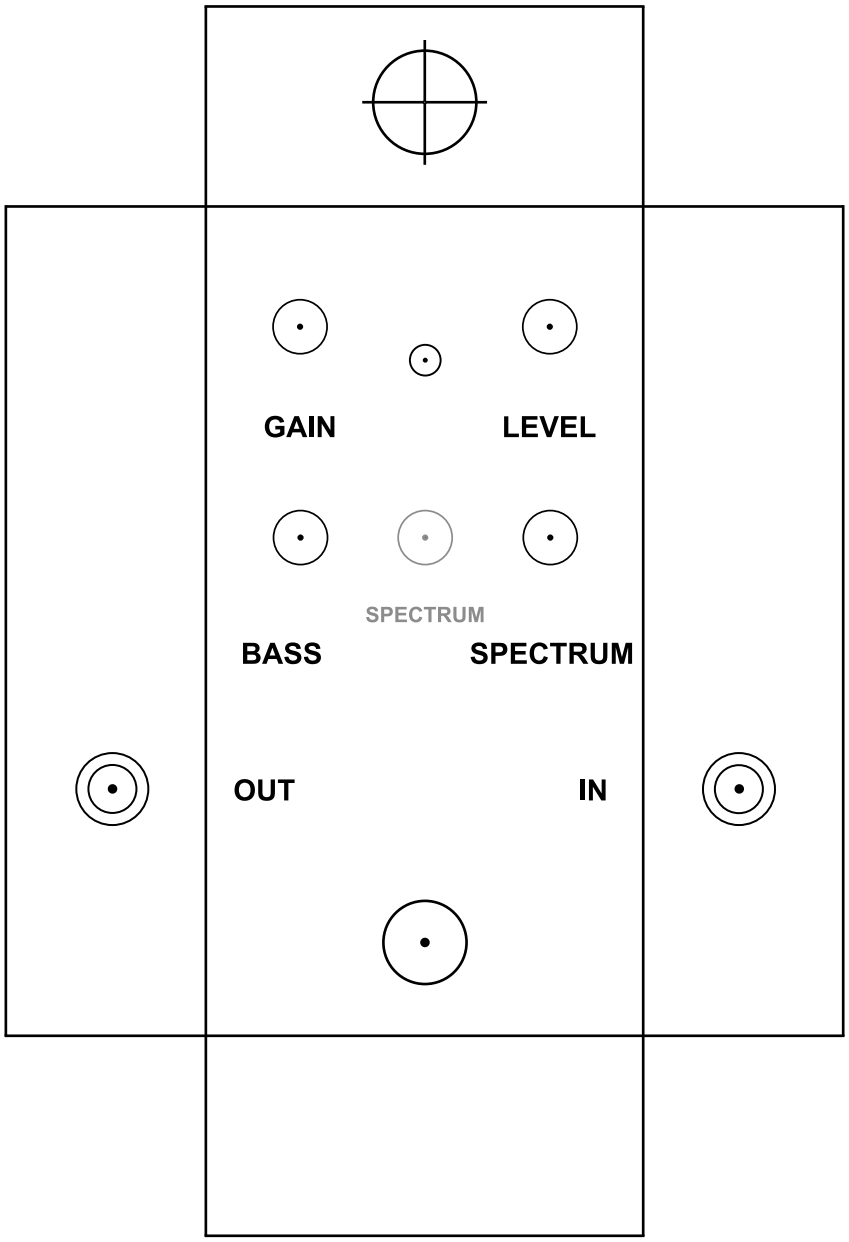
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

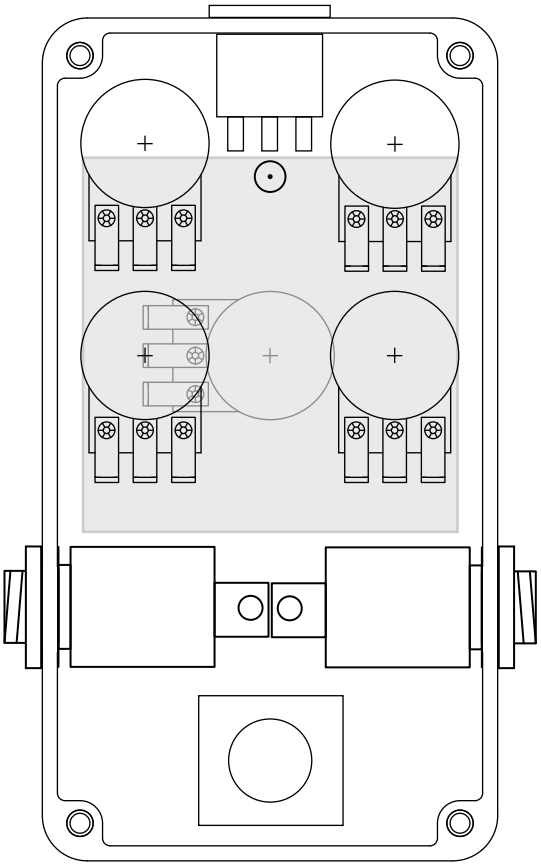
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

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

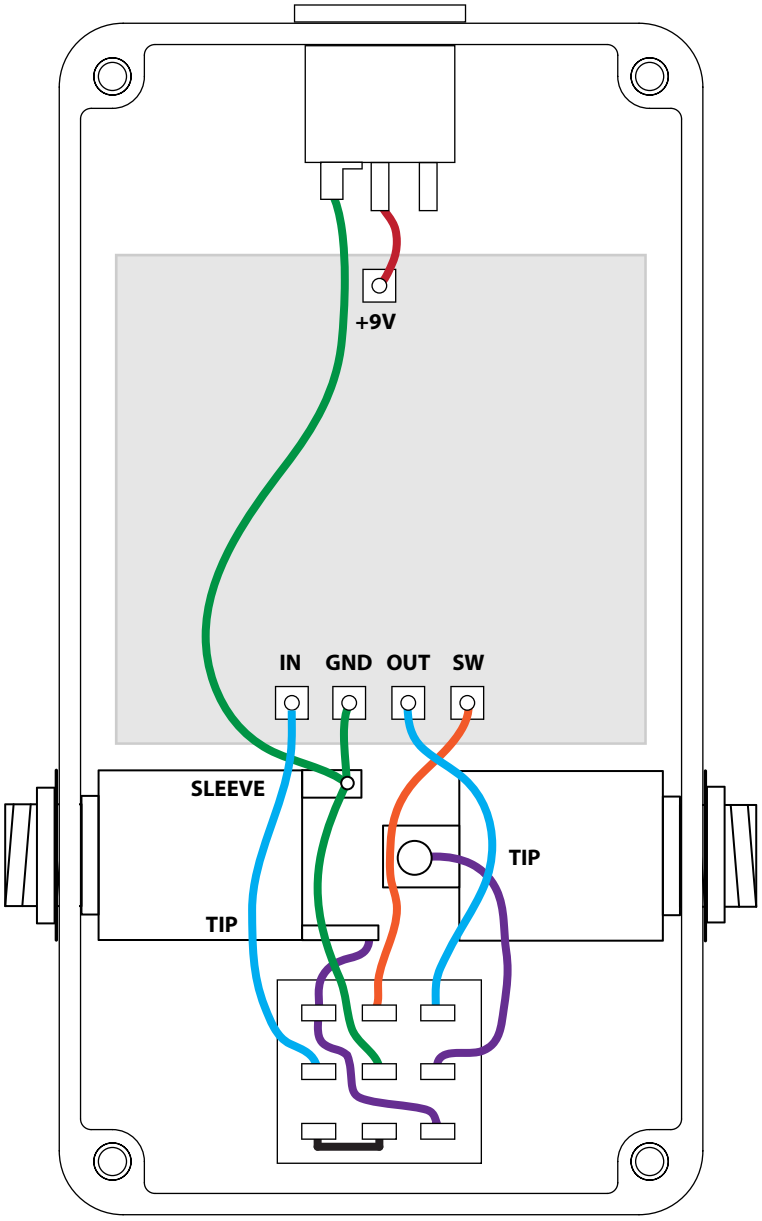
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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!)