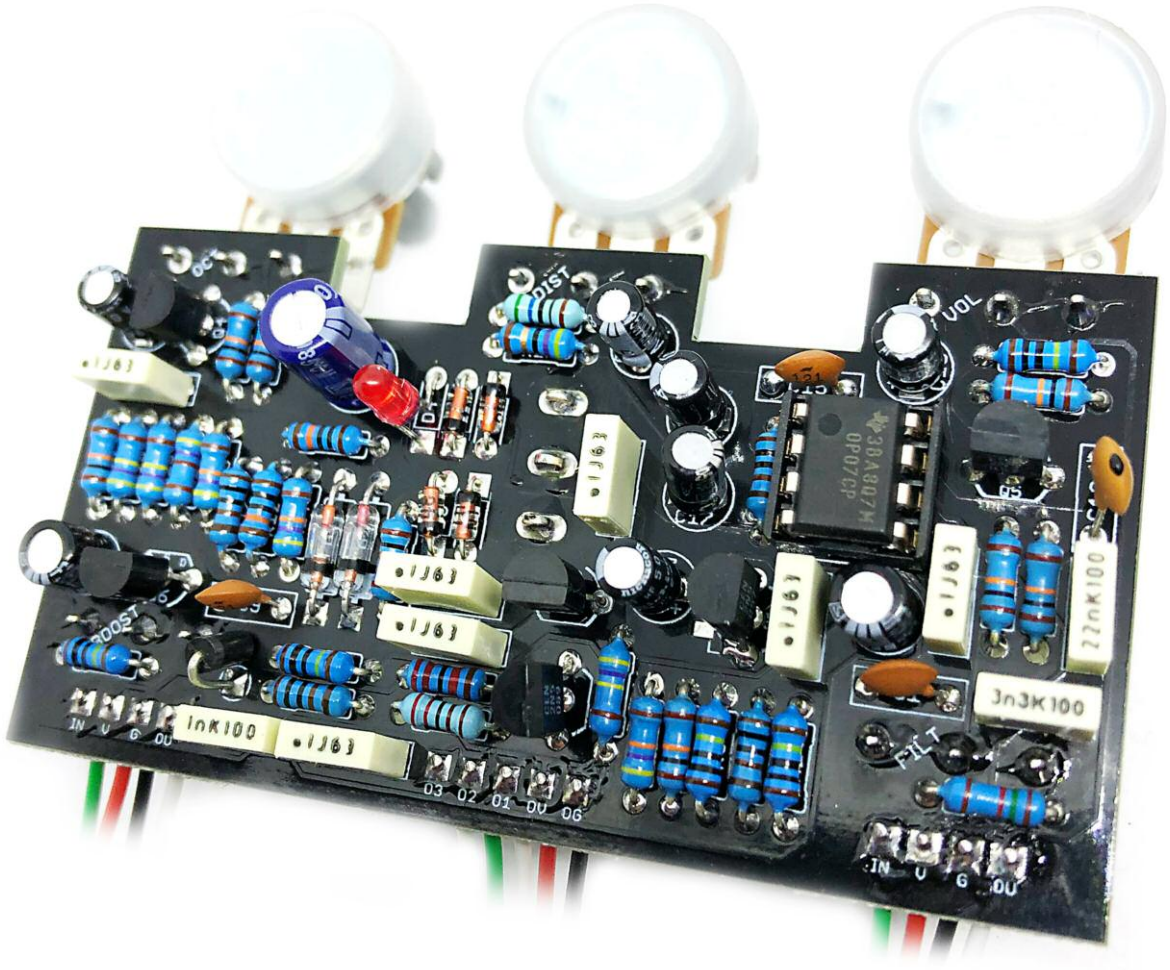


Death Drive

Octave-up into Rat into boost for doomy destruction

Important notes

If you're using any of our footswitch daughterboards, DOWNLOAD THE DAUGHTERBOARD DOCUMENT

- Download and read the appropriate build document for the daughterboard as well as this one BEFORE you start.
- DO NOT solder the supplied Current Limiting Resistor (CLR) to the main circuit board even if there is a place for it. This should be soldered to the footswitch daughterboard.

POWER SUPPLY

Unless otherwise stated in this document this circuit is designed to be powered with 9V DC.

COMPONENT SPECS

Unless otherwise stated in this document:

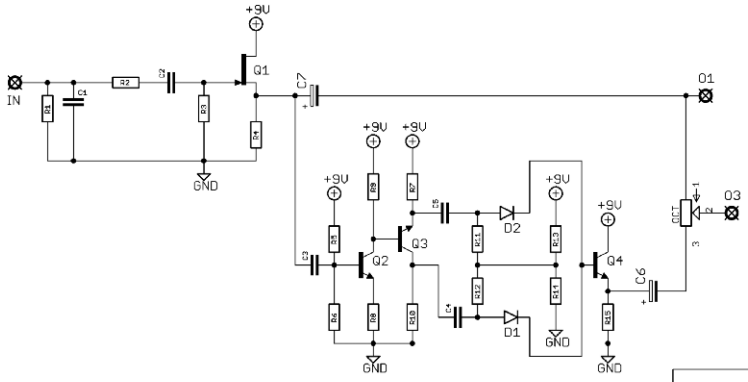
- Resistors should be 0.25W. You can use those with higher ratings but check the physical size of them.
- Electrolytics caps should be at least 25V for 9V circuits, 35V for 18V circuits. Again, check physical size if using higher ratings.

LAYOUT CONVENTIONS

Unless otherwise stated in this document, the following are used:

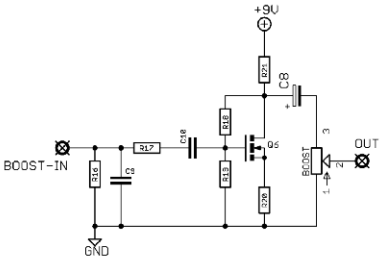
- **Electrolytic capacitors:**
  Long leg (anode) to square pad.
- **Diodes/LEDs:**
  Striped leg (cathode) to square pad. Short leg to square pad for LEDs.
- **ICs:**
  Square pad indicates pin 1.

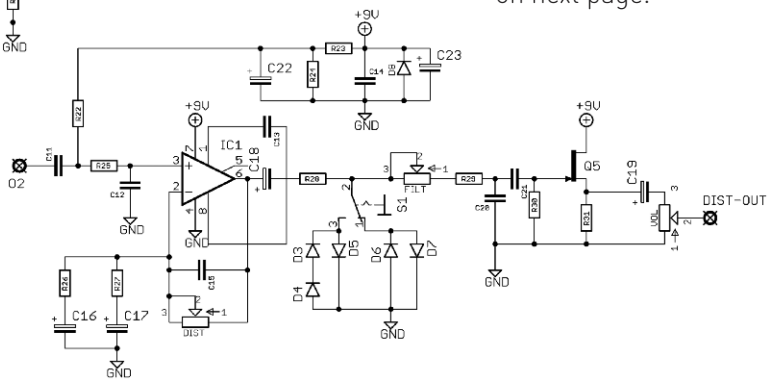
Schematic + BOM



Bigger, legible version on next page.

Based on the trace by Bugg





R1	1M
R2	1K
R3	1M
R4	10K
R5	470K
R6	47K
R7	10K
R8	2K2
R9	22K
R10	10K
R11	100K
R12	100K
R13	47K
R14	47K
R15	10K
R16	1M
R17	1K
R18	470K
R19	470K
R20	470R
R21	10K
R22	1M
R23	10K
R24	10K
R25	1K
R26	560R
R27	47R
R28	1K
R29	1K5
R30	1M
R31	10K

C1	100p
C2	100n
C3	100n
C4	100n
C5	100n
C6	1u elec
C7	1u elec
C8	10u
C9	100p
C10	1u‡
C11	22n
C12	1n
C13	33p
C14	100n
C15	100p
C16	4u7 elec
C17	2u2 elec
C18	4u7 elec
C19	1u elec
C20	3n3
C21	22n
C22	10u elec
C23	100u elec

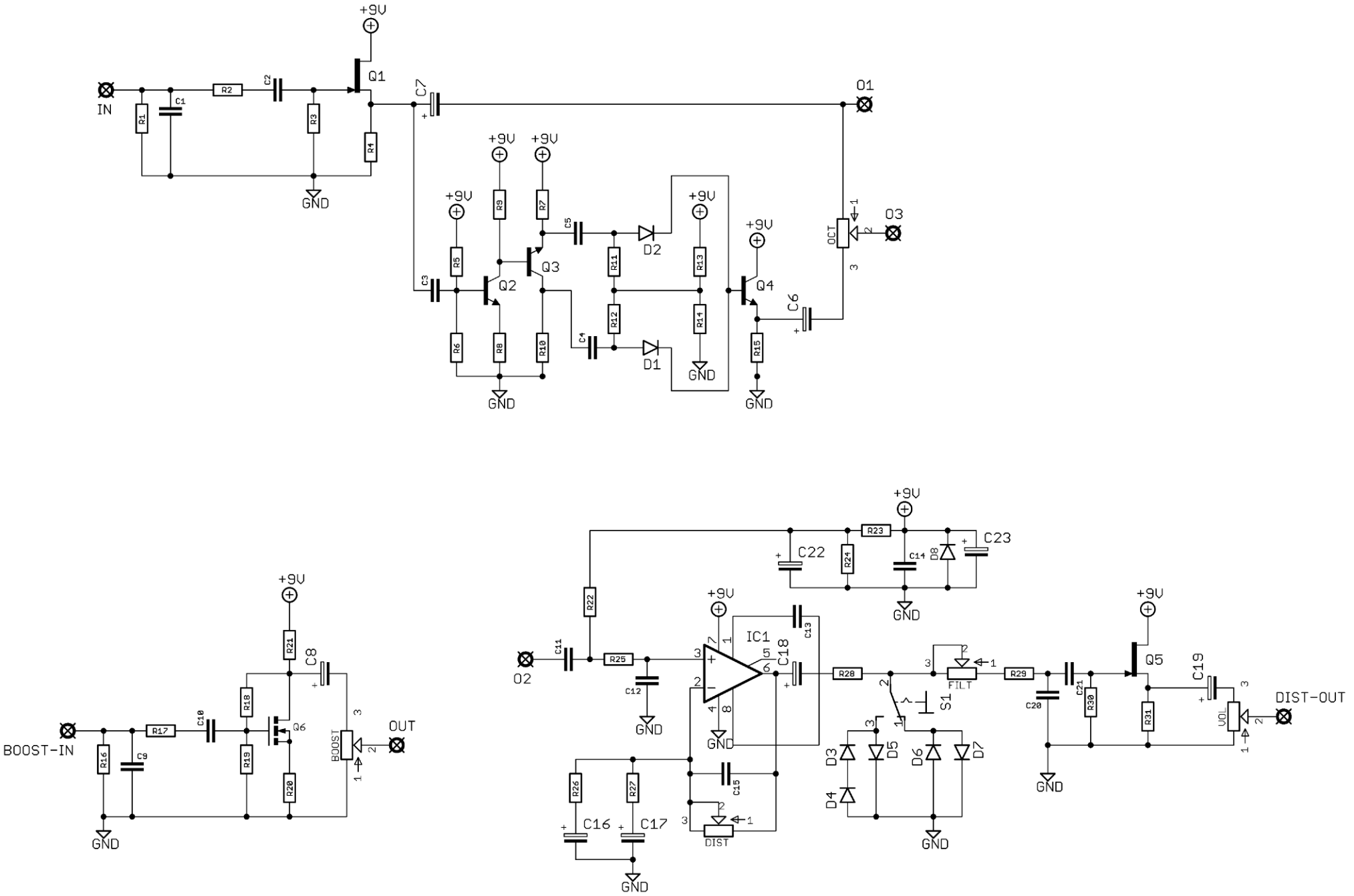
IC1	OP07*
Q1	2N5457**
Q2	2N5089
Q3	2N5087
Q4	2N5089
Q5	2N5457**
Q6	BS170
D1-2	Ge***
D3	1N4148
D4	3mm Red LED
D5-7	1N4148
D8	1N4001
VOL	100KA
DIST	100KA
FILT	100KA
OCT	50KB
BOOS	100KA
S1	SPDT ON-OFF-ON

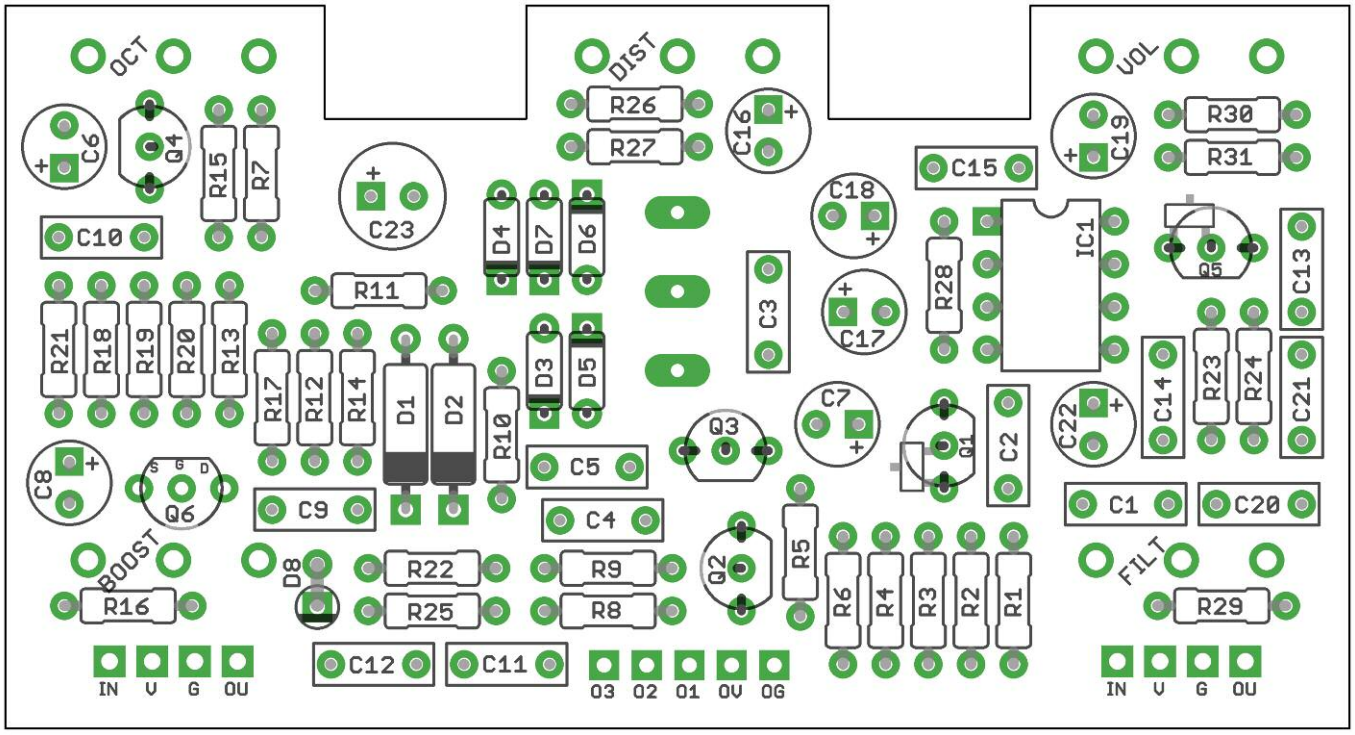
*Original is LM308. Other single op-amps will work.

**Original is PF5102. Other N-channel FETs work in these buffer sections, and will have very little or no impact on tone. Pads are included for through-hole or SMT FETs.

***We supply D9K which have the stipes at the anode rather than cathode. These don't even need to be germanium - 1N4148 work fine. They need to be matched for forward voltage. **These need to be reversed when placed in the PCB, i.e. cathode to square pad.**

‡C10 is 1u on the original. We haven't allowed a large space for that on the PCB, so if you want to use this value you'll need a small format part, such as MLCC. There's really no need to have such a big value in there. Use anything from 100n - 1u.


+9V
IN
R1
C1
R2
C2
Q1
R3
R4
GND
C7
O1
R9
R7
C5
D2
R5
Q3
Q2
C3
R11
R13
Q4
C6
OCT
O3
R12
R14
R6
R8
R10
C4
D1
R15
BOOST-IN
R16
C9
R17
C10
R18
R19
Q6
R21
C8
R20
BOOST
OUT
C22
R23
R24
C14
D8
C23
R22
C11
O2
R25
C12
IC1
C13
C18
R28
FILT
S1
R29
C21
C20
R30
R31
Q5
C19
VOL
DIST-OUT
D3
D5
D6
D7
D4
R26
R27
C15
C16
C17
DIST



PCB layout ©2019 Pedal Parts Ltd.

Two of the power and signal pads on the PCB conform to the FuzzDog Direct Connection format, so can be paired with the appropriate daughterboard for quick and easy offboard wiring. Check the separate daughterboard document for details. The optional Octave-Bypass pads need to be wired differently, but a daughterboard can still be used.

Be very careful when soldering the transistors and diodes. They're very sensitive to heat. Keep exposure to heat to a minimum (under 2 seconds) and leave a few seconds between soldering each leg. Same goes for the IC if you aren't using a socket.

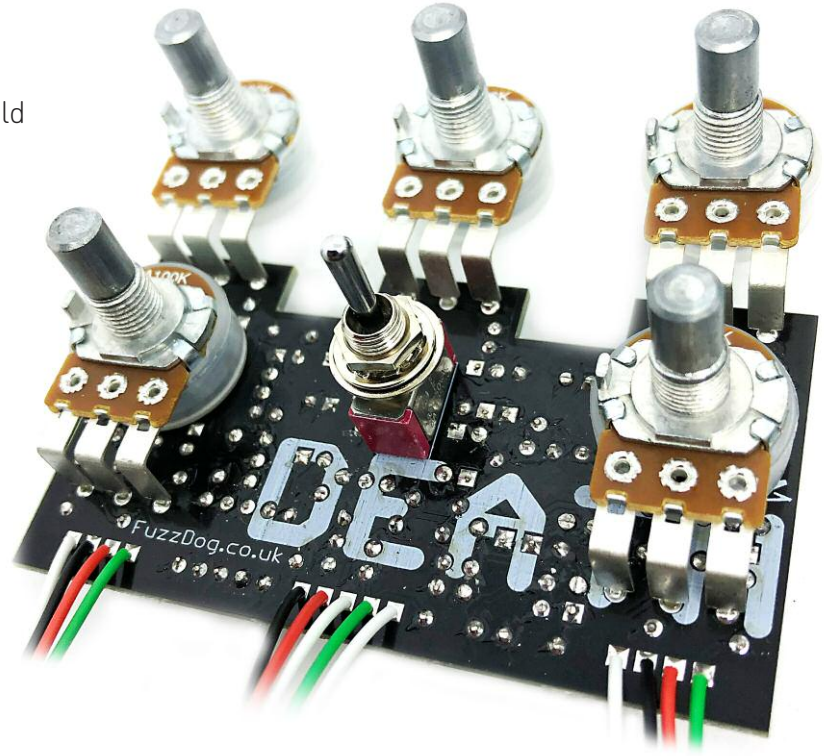
The BS170 is very sensitive to static - handle with care.

The glass case on the Ge transistors is very delicate. Take care when bending the legs. Hold a the leg right up against the body with some small needle-nosed pliers to take the strain, and bend the leg with your finger.

You should solder all other board-mounted components before you solder the pots. Once they're in place you'll have no access to much of the board. Make sure your pots all line up nicely.

The best way to do that is to solder a single pin of each pot in place then melt and adjust if necessary before soldering in the other two pins. Same for the toggle switch. If your pots don't have protective plastic jackets ensure you leave a decent gap between the pot body and the PCB otherwise you risk shorting out the circuit.

Snap the small metal tag off the pots so they can be mounted flush in the box.



Initial wiring

USE WIRES! Our ribbon cables are unlikely to be of any use unless you've compromised your enclosure layout to line up the pads to daughterboards.

If you aren't going to use a footswitch to engage/bypass the OCTAVE section of the circuit, add a jumper as shown below.

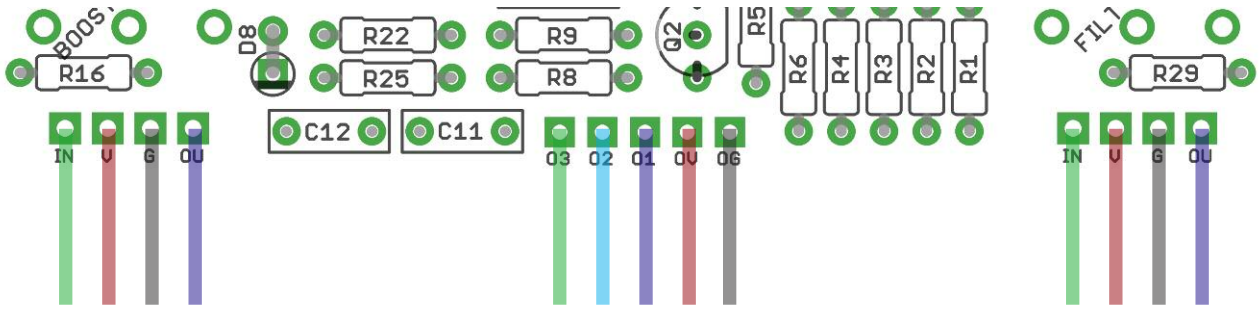
...otherwise add wires to all five of those pads. OV is for +supply, OG is ground.

O1 is Octave Bypass

O2 is the input to the Distortion section

O3 is Octave Out

Add long wires to the other two sets of pads for offboard wiring.



BOOST **OCTAVE** **DISTORTION**

Testing

You can test the Distortion and Boost sections individually, as if they're separate boards. Do the test wiring as shown below for each section.

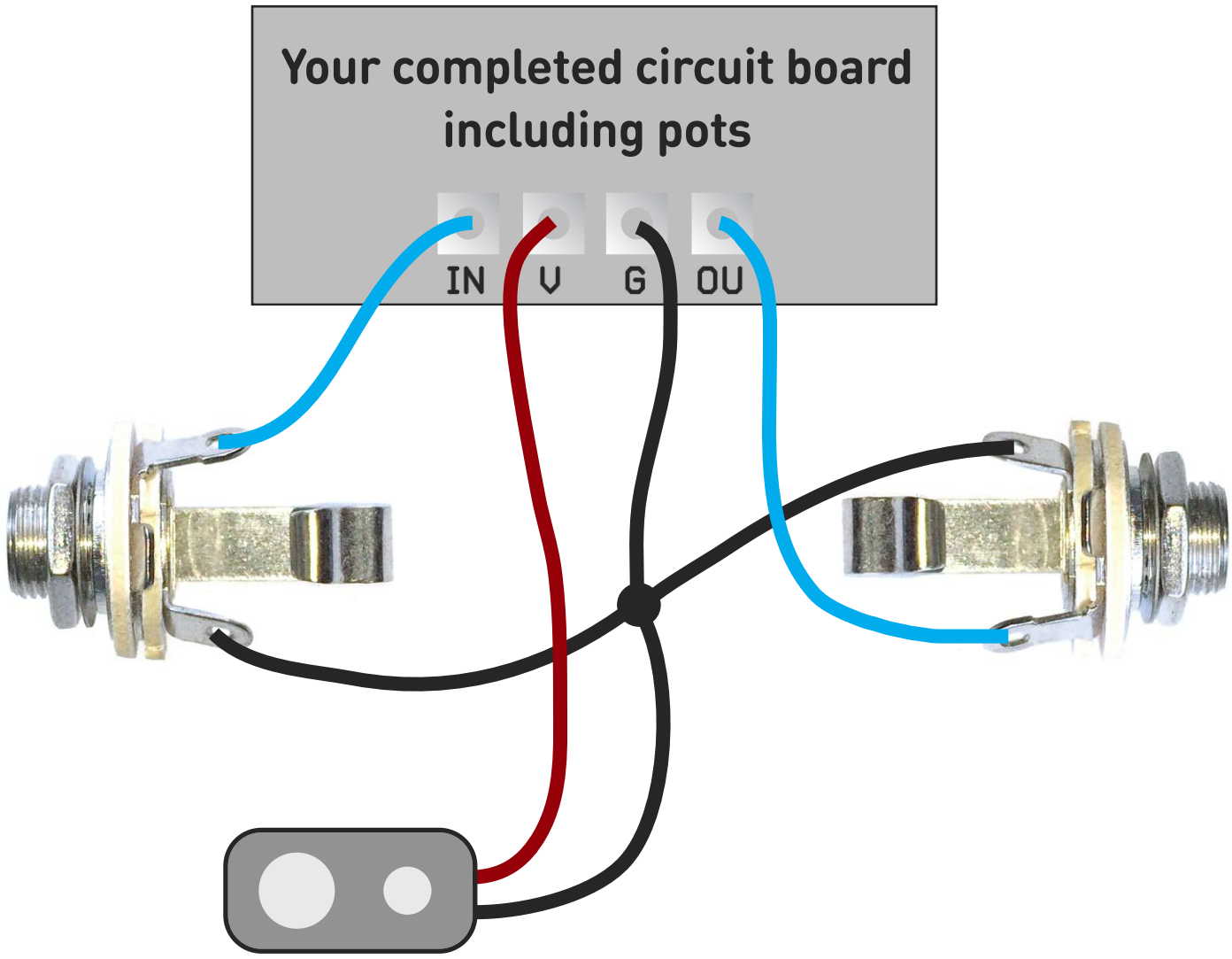
UNDER NO CIRCUMSTANCES will troubleshooting help be offered if you have skipped this stage. No exceptions.

Solder some nice, long lengths of wire to the board connections for 9V, GND, IN and OUT. Connect IN and OUT to the jacks as shown. Connect all the GNDs together (twist them up and add a small amount of solder to tack it). Connect the battery + lead to the 9V wire, same method. Plug in. Go!

You'll have to connect the wire from pad O2 to either O1 or O3, or you'll get no signal from the Distortion section.

There's no need to connect power to the OV and OG pads - these are purely to supply power to an LED if you're adding the Octave footswitch.

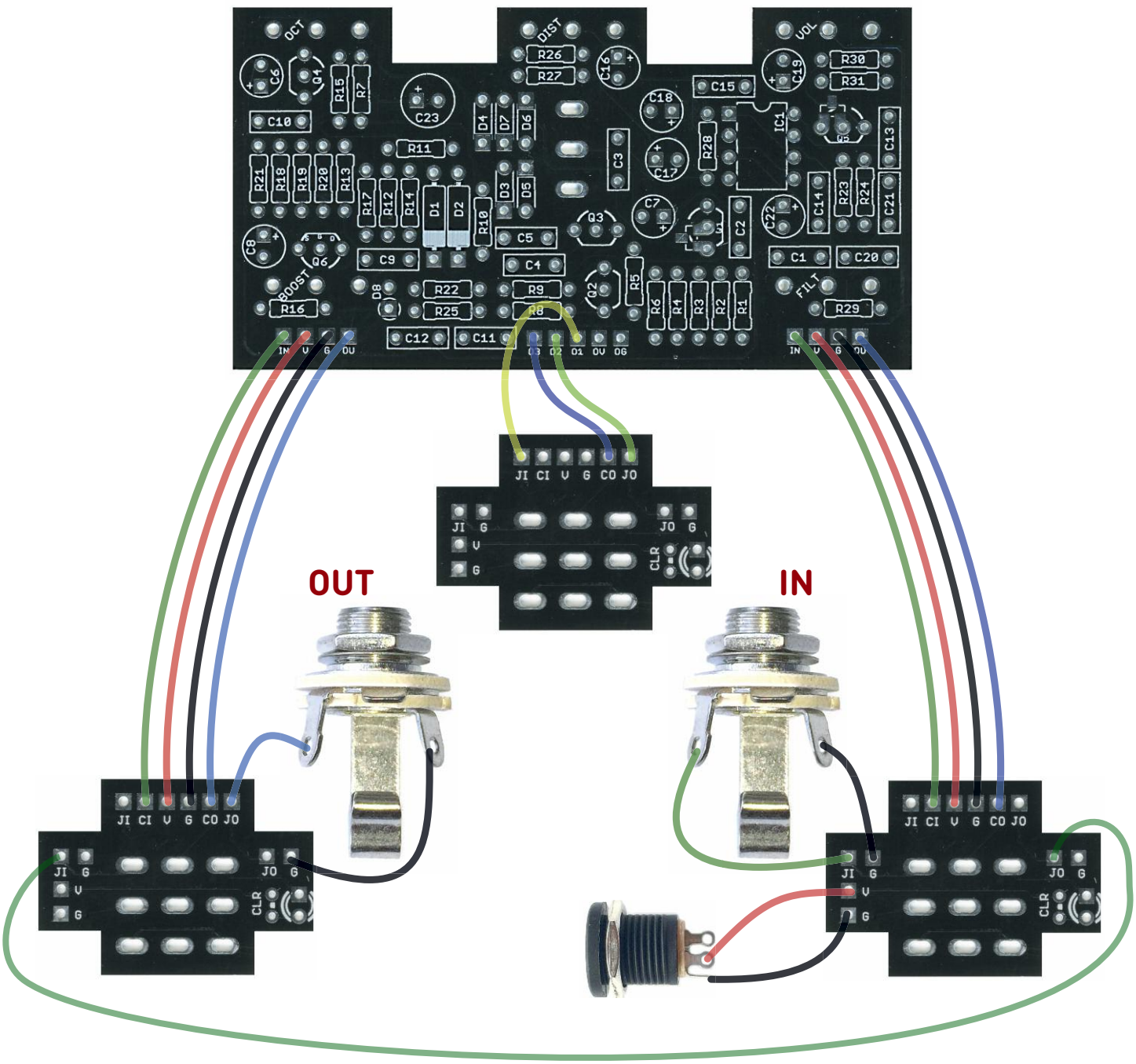
If you have a circuit tester (we sell a few!) just use that as normal.



Wire it all up

The image below assumes you're using an Octave Bypass footswitch. If not just leave that out, having added the jumper shown earlier in this document.

You should also connect OV and OG to V and G on the Octave daughterboard, we just left them out of the image as it would get messy. The wiring shown will light up the LED when Octave is engaged.



Please refer to the separate daughterboard document for further info regarding the current limiting resistors and LEDs. The DC socket can connect to any of the three daughterboards.

See next page for more info if you're building a 1590BB, two switch version.

Keeping up appearances...

On the bigger box versions of this kit, the LEDs all sit to the same side of the footswitches. However, that would look decidedly weird on the two switch, 1590BB version.

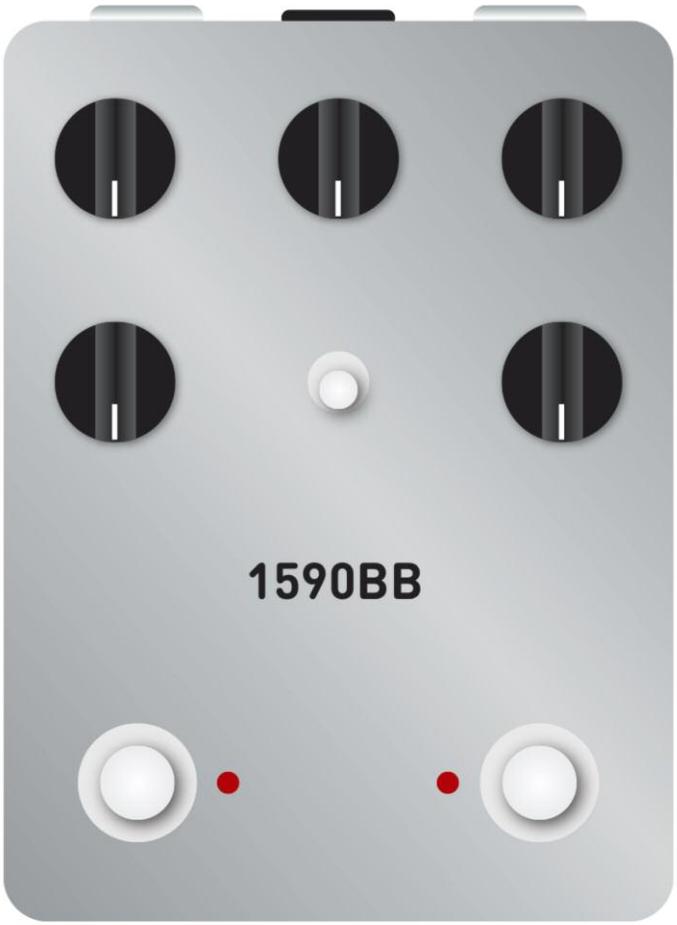
To keep things sweet you'll have to flip the Distortion side daughterboard over before soldering in the footswitch and LED. That way the LED can still sit in the daughterboard and line up with the hole.

Everything will function exactly the same, you'll just have wires twisted around between the boards. Keep them slack and you'll have no issues.

On/Offboard wiring

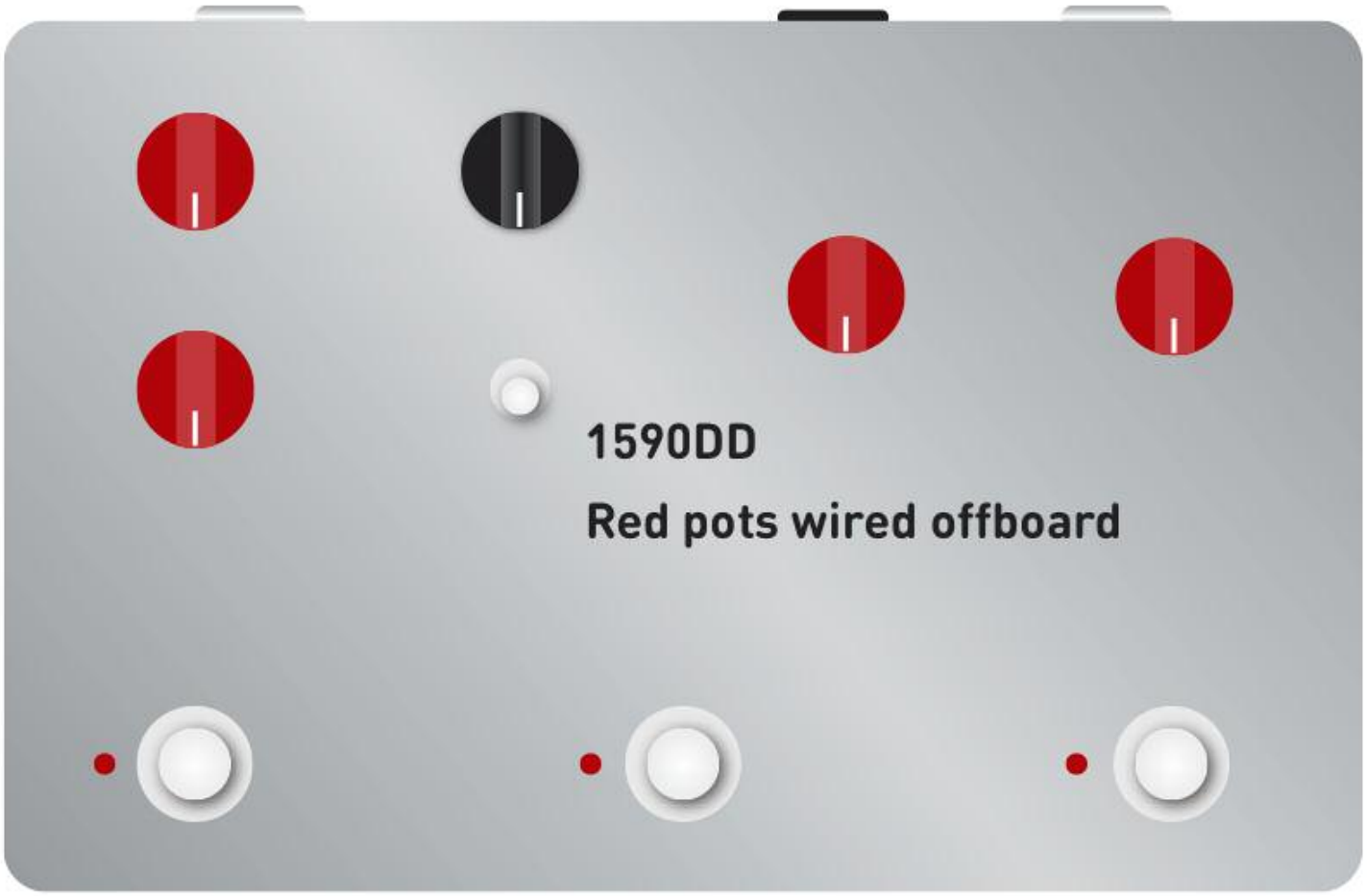
We designed the board to fit nicely in a 1590BB with all controls mounted on the PCB. Using a bigger enclosure means some pots need to be wired offboard.

The images on the next two pages show which pots need to be wired. You may need to bend the legs down to ensure they don't touch the PCB in some places.




1590XX
Red pot wired offboard


1590DD
Red pots wired offboard


1590DD
Red pots wired offboard


1590DD
Red pots wired offboard

Drilling templates

The templates on the following pages are a guideline only. Ensure correct marking of your enclosure, and compare component positions on your built circuit to your markings before drilling.

The only critical positions in relation to the main PCB are any board mounted components.

We've only provided one template for the 1590DD size enclosure, as there are no rules for that one. Position your controls however you see fit, as long as you have one or two board mounted components to hold your PCB securely.

Our templates are based on using Lumberg KLBM-3 jacks which fit nicely above the pots in these cases. If you're using a different type you'll have to work out for yourself how far down the pots need to be to allow clearance for any lugs.

On the 1590DD enclosures there's an extra rib in the centre, so take that into consideration when drilling your middle footswitch and your DC socket hole.

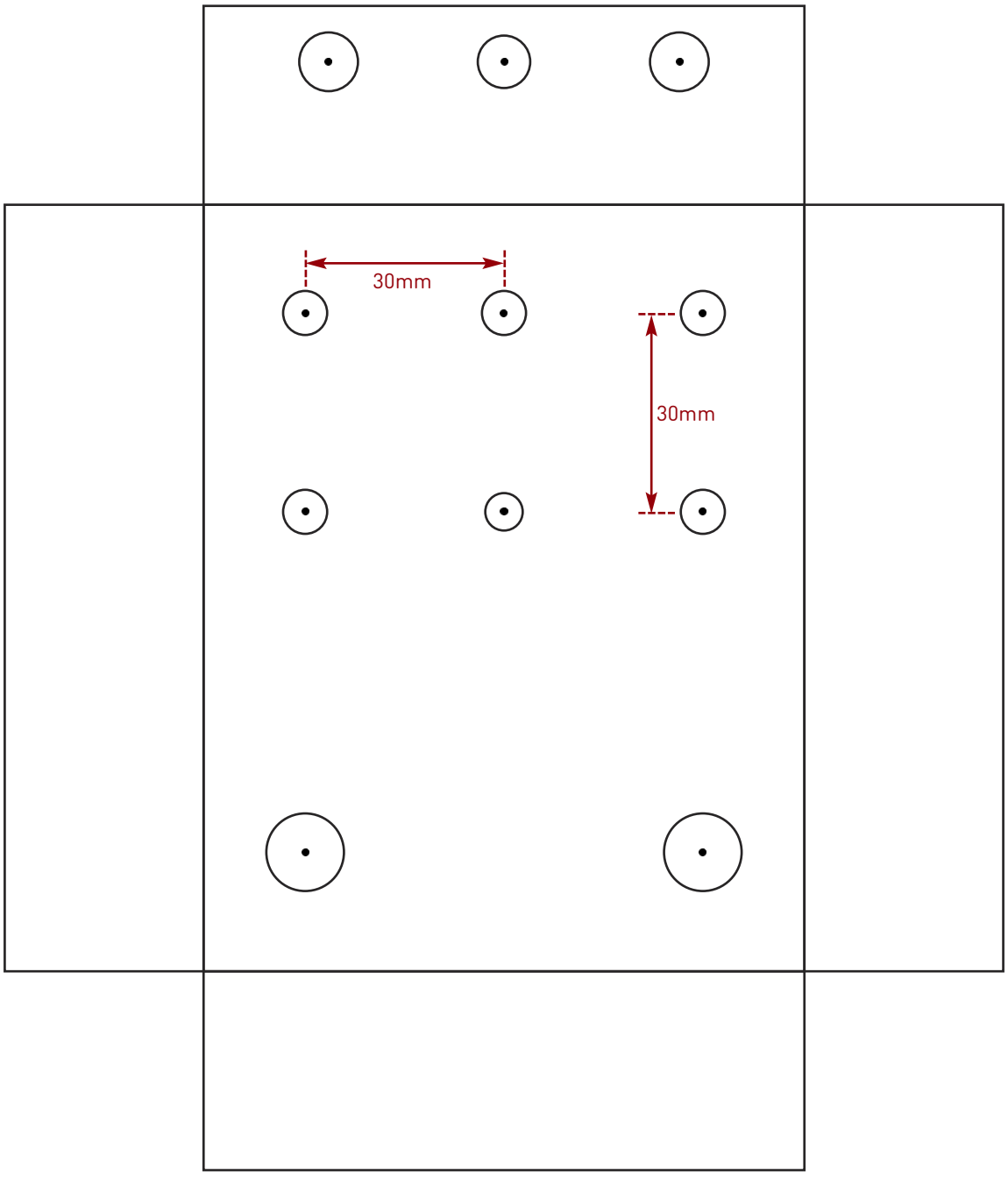
Drilling template

Hammond 1590BB

It's a good idea to drill the pot and toggle switch holes 1mm bigger if you're board-mounting them.

Recommended drill sizes:

Pots	7mm
Jacks	10mm
Footswitch	12mm
DC Socket	8/12mm
Toggle switches	6mm



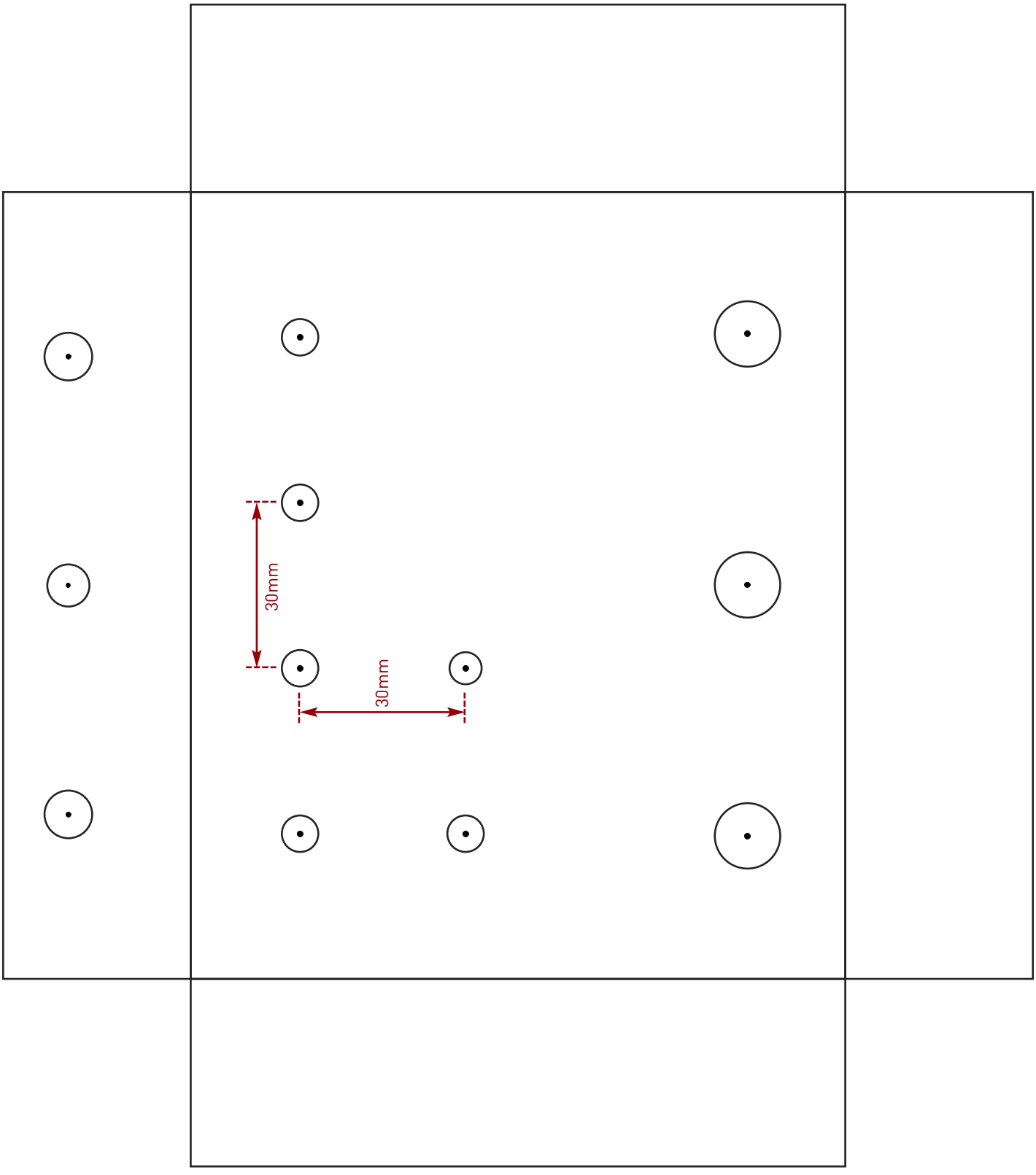
Drilling template

Hammond 1590XX

It's a good idea to drill the pot and toggle switch holes 1mm bigger if you're board-mounting them.

Recommended drill sizes:

Pots	7mm
Jacks	10mm
Footswitch	12mm
DC Socket	8/12mm
Toggle switches	6mm



Drilling template

Hammond 1590DD