Beverly BitCrusher

Build Document last updated july 2024
Version 1.0 2024

The Beverly BitCrusher uses an analog-to-digital converter to turn the signal into a true 8-bit representation. You can change the sample rate from high to low - clean to crushed. It also has a blend control so that you can mix the crushed sound with the clean guitar signal.

Have fun building and playing the Beverly BitCrusher!

Prototype build

Power

Input voltage - 9V DC
Current draw - 14mA

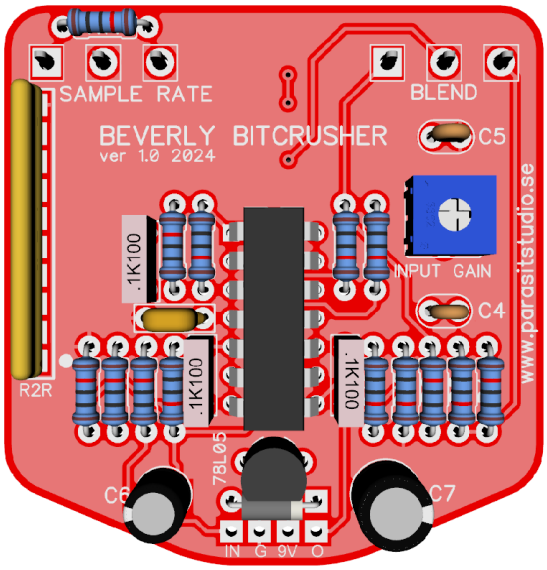
Controls

Potentiometers

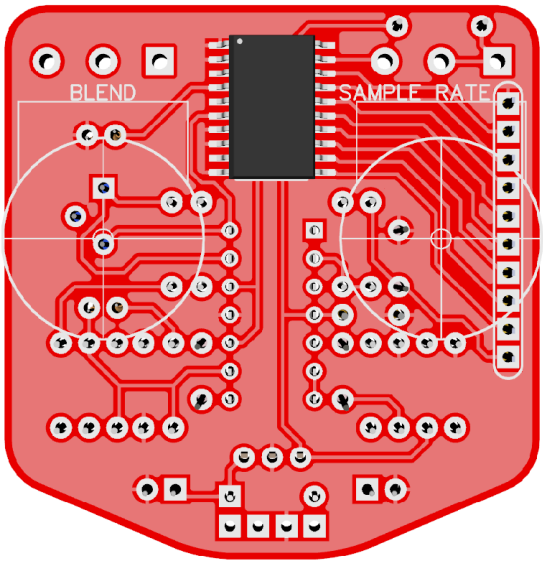
- Blend – blends between the clean signal and the bitcrushed signal
- Sample rate – controls the ADC frequency

The populated PCB

Here's a 3D render approximation of what the fully populated board should look like (without the potentiometers).



Component side (top side)



Solder side (bottom side)

The PCB measures 49mm wide x 50mm tall

The ADC0804 IC

This PCB has a surface mounted chip that is soldered on bottom side of the board. These SOIC-20 packages are pretty easy to solder, so don't be discouraged if you are new to soldering surface mount parts.

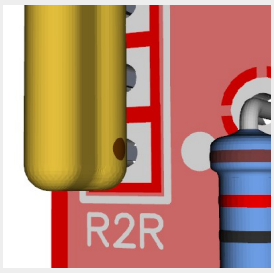
I recommend that you put some solder first on one of the pads on the PCB, then align the chip on the board and reflow that pad to fixate the chip. Then carefully solder each leg, one by one. Just a very small amount of solder is needed. If you happen to bridge two legs, use some copper desoldering braid to remove the excess solder. Take care to not put too much heat on the chip, so don't put your soldering iron on the legs for more than a few seconds.

General building tips

- Just follow the Bill of Materials and solder the low profile components first.

  Recommended order: resistors and diodes, chip sockets, resistor ladders, trim pot, multilayer and ceramic capacitors, film box capacitors, electrolytic capacitors, pots and switches, offboard wiring (jacks and the 3PDT switch). Bend the legs of the components alittle bit to prevent them from falling out, or use tape to hold them in place while soldering.

- The resistor ladder orientation is important! Indicated by the dots on the resistor ladder and the PCB.



- Also pay special attention to the orientation of the LED's, diodes and the electrolytic capacitors.
- Always use sockets for IC chips to avoid heating them directly. It also makes it much easier to swap them out if needed.
- CMOS chips are very sensitive to static charges and can be easily damaged. It's a good idea to wear a anti-static wristband. Or at least don't wear a woolen jumper and pat your dog while building, and keep the circuit away from rugs... Put the chips in last, after everything else is soldered in place.
- Break off the small tap on the potentiometers, so they can sit flush against the top cover.
- Make sure that the backside of your pots are covered so they don't short anything on the PCB. If you not have pot covers I recommend pvc electrical tape.
- When it's time to solder the potentiometers, switches and buttons I recommend having the enclosure/lid prepared to make sure that they line up with the holes. I usually solder just the middle leg of each potentiometer to the board before putting the board inside the enclosure, and then solder the rest of the legs.

Beverly BitCrusher (Bill of Materials)

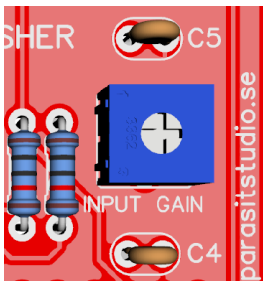
Resistors		Capacitors		IC's	
R1	1M	C1	100nF	U1	LM324
R2	1M	C2	100nF	U2	ADC0804LCWM
R3	10K	C3	10nF		
R4	10K	C4	100pF	**Potentiometers**	
R5	10K	C5	220pF	BLEND	B25K
R6	10K	C6	47uF	SAMPLE RATE	B100K
R7	10K	C7	100uF		
R8	10K	C8	100nF **	GAIN (trimpot)	500K
R9	10K				
R10	10K	**Diodes**		**Resistor Ladder**	
R11	150K	D1	1N4001	1x 4610X-R2R-103LF	
R12	33K	1x LED	(bypass)		
R13	100K				
R14	2.2K			**Voltage regulator**	
R15	4.7K*			1x 78L05	

BOM Notes

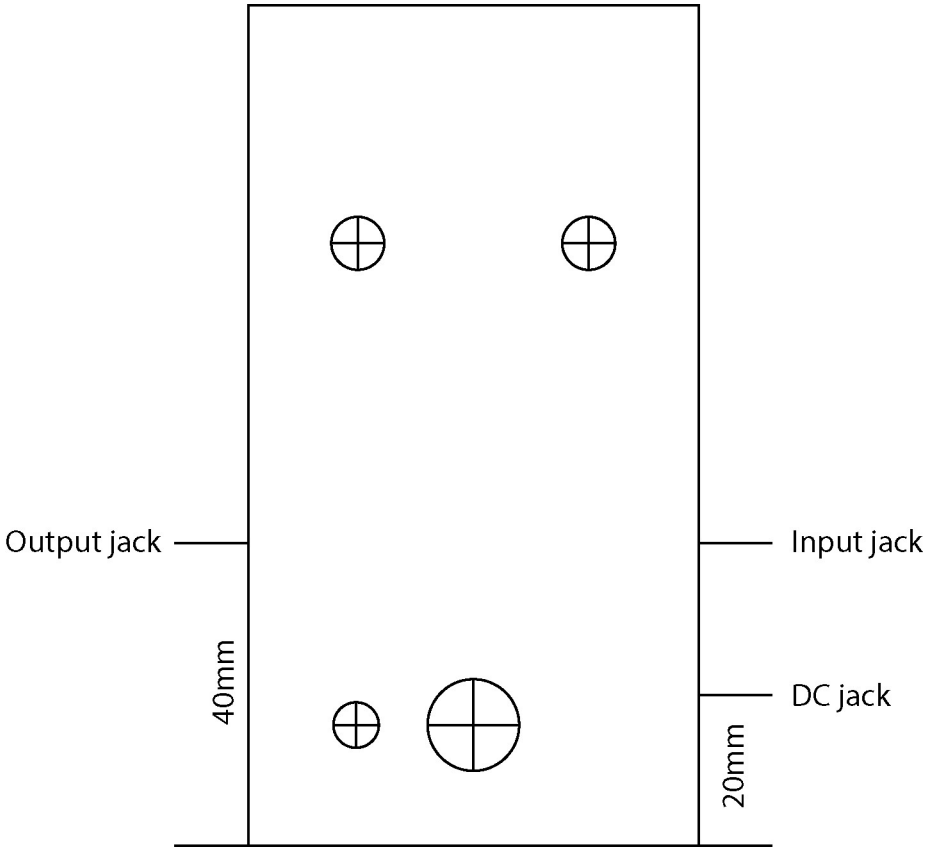
- * = current limiting resistors for bypass LED. This is mounted off board or on the optional 3PDT daughterboard PCB.
- ** = should be a multilayer ceramic capacitor (yellow capacitor)
- Things that are not included in the BOM list: enclosure, input and output jacks, DC jack, 3PDT stomp switch, led bezel and knobs.

The Input Gain trimpot

The input gain trimpot is there for making the output of the pedal at unison, as it doesn't have a volume control. Turn the blend fully clockwise, then adjust the trimpot so that you get equal volume compared to the clean or bypassed signal.



Drill Template (1590B)

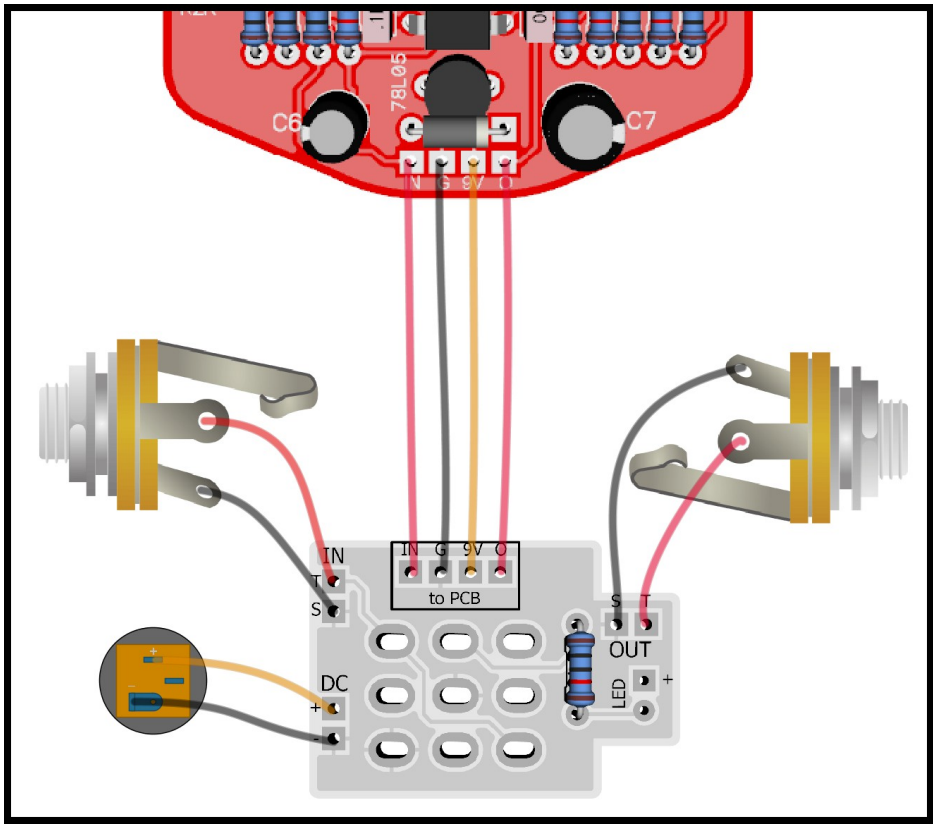


- Use at your own risk! This template is approximate.
- Make sure your printer isn't doing any scaling (100% print size).
- Jacks are measured from the edge at the front/top of the enclosure (as they were drilled on the prototype), but you can drill the positions for the footswitch, DC jack and input/output jacks to your own preference.

- Typical drill sizes are:
  - switches / LED bezel (for a 3mm LED) - 6mm
  - potentiometers - 7mm
  - DC jack / 3PDT footswitch – 12mm
    (8mm for lumberg style DC jacks)
  - input/output jacks (Neutrik style) – 9,5mm
    (9mm for Lumberg style jacks)

Measure and confirm before drilling!

Off Board Wiring



The top row of connections on the 3DPT daughterboard connects directly across to the main PCB as shown.

Input jack sleeve → ”S” IN pad *the lug that connects with the inner ring of the jack*
Input jack tip → ”T” IN pad *the lug that connects to the tip bracket on the jack*

Output jack sleeve → ”S” OUT pad *the lug that connects to the inner ring of the jack*
Output jack tip →”T” OUT pad *the lug that connects to the tip bracket on the jack*

DC jack negative → ”–” DC pad *the widest lug, or the short lug (Lumberg style jack)*
DC jack positive → ”+” DC pad *the outer lug if it's a 3 pin DC connector, or the long lug if using a Lumberg style jack*

The short leg of bypass LED is the negative side (the side with the flat edge of the LED)

If you are not using the 3PDT daughterboard PCB, have a look at the offboard wiring diagram here (fig1/3): [wiringrev3.pdf (parasitstudio.se)]

Never use wires that are much longer than they need to be. Cut down the wire to just a bit longer than needed, then strip the ends and solder. Just be careful when cutting the wires so that you still have some slack.

Troubleshooting

There's always a chance of running into trouble. To minimize error, follow the BOM and general building tips carefully. Take your time and don't rush. Take a break now and then. Use good solder, and it helps to have a decent soldering station insted of a cheap iron.

Musikding DIY kit

If you have bought the Musikding DIY kit and have recieved a faulty faulty, incorrect or missing component, please contact musikding.

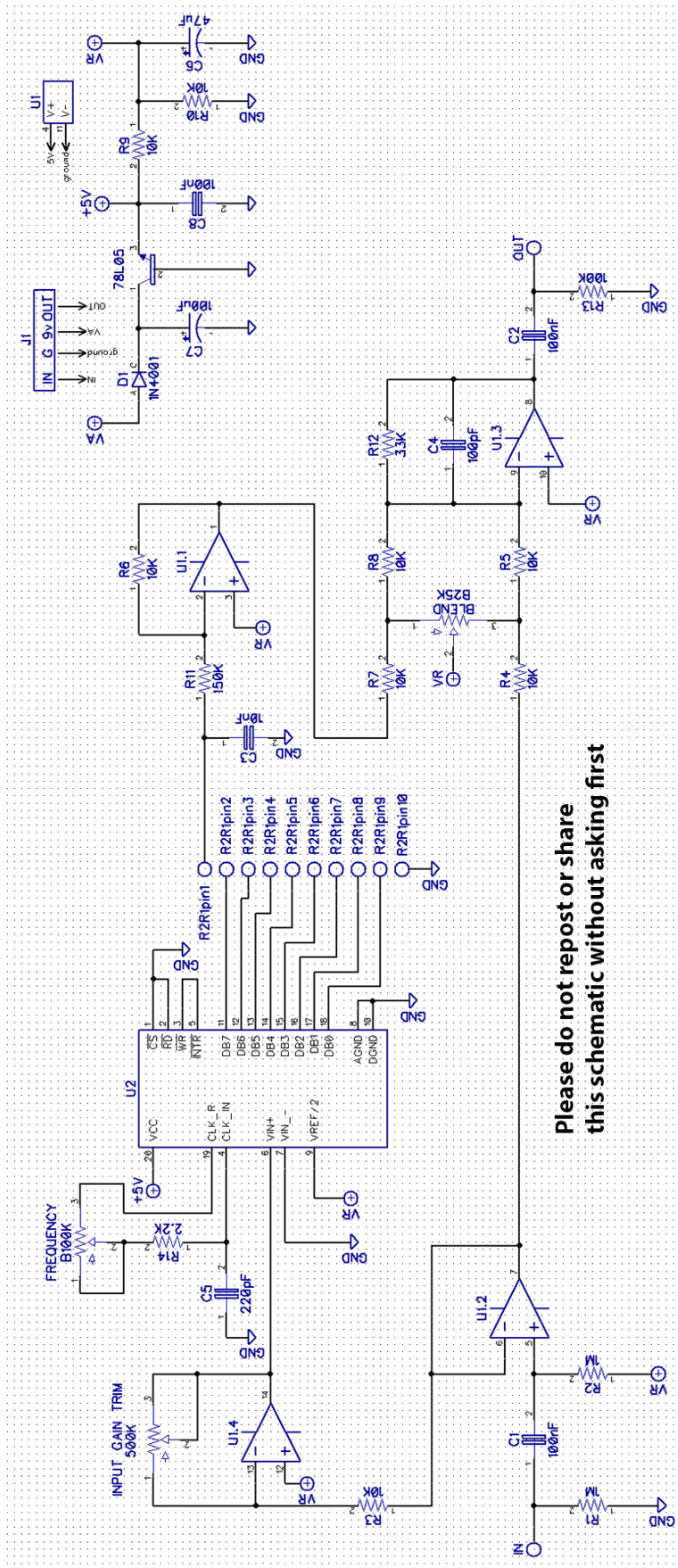
Contact us

Terms of use – please read

PCB's and circuit designs from Parasit Studio are intended for personal use only. No commersial use. It's not ok to build and sell these pedals without permission. However, it's totally ok to build a few pedals and give to your friends and bandmates. After all, that's what this hobby is about. :)

www.parasitstudio.se
parasitstudio@gmail.com

Schematic


Please do not repost or share
this schematic without asking first