C82 MICRO

Build Document last updated march 2025

Version 1.0 2024

The C82 MICRO is a guitar synth inspired by the sounds of 8-bit computers. It has a digital oscillator based on a AVR microcontroller that has a volume envelope, 3 types of modulation and a pseudo random pattern generator with 3 different scales. It can be blended with a typical CMOS square wave fuzz at either unison or one octave down.

The digital oscillator and octave down is monophonic, so play single notes. Use the neck pickup on the guitar to get the most stable tracking.

Have fun building and playing the C82 MICRO!

A 3D render of a complete pedal

Input voltage: 9V DC. Current draw: 52mA

Controls

Switches

- **Select** – toggle between P1 and P1+P2
  A long press will select only P2 (or vice versa, depending on the select priority).

  P1 (player 1) = square wave fuzz (or one octave down).
  P2 (player 2) = the digital oscillator (with envelope and modulation).

  Most buttons only affect the digital oscillator (P2).

- **Start** – Bypass/turn the pedal on/off. A long press will change the select priority.

Potentiometers

- **Trigg** – Controls the trigg threshold. Turn it up until the trigg LED lights up when picking a string – it will turn on the oscillator (P2) and it will capture the input frequency as long as the LED is on. When the LED turns off again it will trigger the volume envelope so that the oscillator starts to fade out.

- **Rate** – Controls the rate of the modulation/sequencer (P2).
  The sequencer has priority over the modulation, so it will control the sequencer rate when the sequencer is on. Otherwise it controls the modulation rate.

- **Decay** – Controls the decay of the volume envelope (P2).
  How long time it takes for the note to fade out after the trigg LED turns off.

- **Level** – The overall output volume.

- **Blend** – Blends between straight square wave fuzz (P1) and the digital oscillator (P2).

  The blend knob is optional. If no blend knob is installed, P1 and P2 will be fixed at 50/50% blend.

Buttons

- **OCT (octave)** – toggles between 3 different octave settings (P2):
  unison, one octave down, one octave up.

  Long press - toggles between unison square wave or one octave down (P1).

- **MOD (modulation)** – Toggles between 4 modulation settings (P2):
  off, vibrato, pwm, alternating octaves.

  Long press - turns the gate on/off, which will gate the oscillator (P2) when there's no guitar signal present at the input.

- **SEQ (sequencer)** – Toggles between 4 sequencer settings (P2):
  off, pentatonic scale, minor scale, ”blues” scale.

  Long press - turns the Drone mode on/off (P2) – Continous oscillation.
  ...**SEQ button** continuation

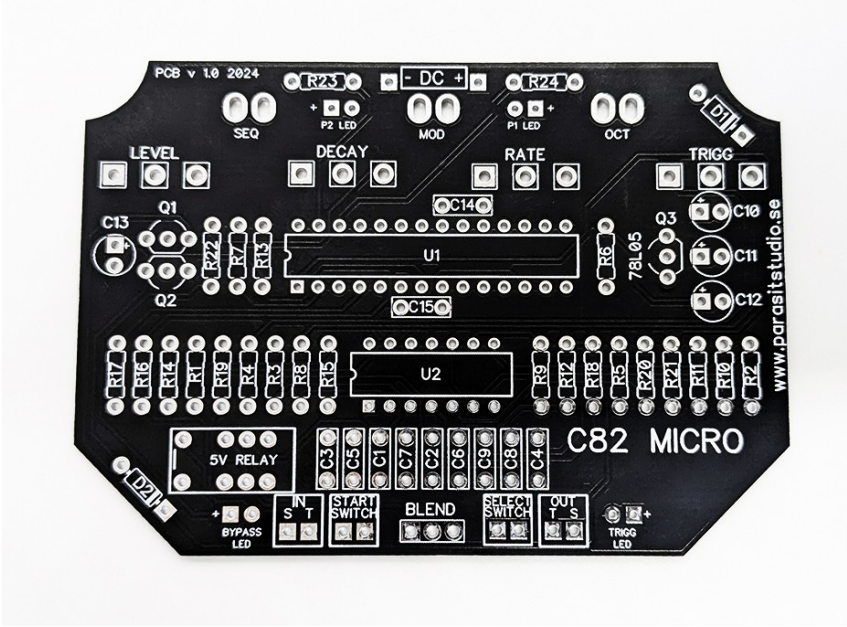
The ”sequencer” is a note generator that is a bit similar to an arpeggiator. It will generate a pseudo random pattern of notes within a selected scale in relation to the current sampled note. The randomness is affected by the general input to the pedal (even if only P1 is selected) but also the rate of the sequencer.

To get the most musical use out of the pedal:

- Turn the pattern generator on by pressing the SEQ button
- Select both P1+P2, then play a single note and let the trigg LED turn off. That note is now sampled and becomes the base note of the selected scale.
- Select only P1
- Turn the Drone mode on by doing a long press of the SEQ button. In Drone mode, the pattern generator will still produce sound even if the P2 oscillator is not selected – every new note in the pattern generator will trigger the envelope. Try adjusting the decay knob for shorter or longer notes.

Now depending on which notes you play on the fuzz side, the pseudo random pattern will change but will still be fixed within a scale determined by the base note that you first sampled. I hope that makes sense... :)

The circuit board



The PCB measures 86mm wide x 61mm tall

There's a 3D render of a populated PCB on the offboard wiring page

General building tips

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

  Recommended order: resistors and diodes, chip sockets, 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.

- Pay special attention to the orientation of the LED's, diodes, the electrolytic capacitors and the orientation of the IC'.
- Always use sockets for IC chips to avoid heating them directly. It also makes it much easier to swap them out if needed.
- IC chips can be very sensitive to static charges and are 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 and buttons I recommend having the enclosure/lid prepared to make sure that they line up with the holes.

  I recommend that you solder only the middle pin of each potentiometer to the PCB (so that the placement matches the silkscreen on the PCB and the pot stays in place when you turn the PCB around but still has some wiggle room). Screw in the toggle buttons in the enclosure and then put the PCB with the pots into the enclosure so that everything fits and finally solder the rest of the pot pins and the buttons from the component side of the PCB.

C82 MICRO BOM (Bill of Materials)

Resistors		Capacitors		IC's	
R1	100R	C1	100nF	U1	AVR32DB28
R2	10K	C2	100nF	U2	LM324N
R3	10K	C3	100nF		
R4	10K	C4	100nF	**Transistors / regulators**	
R5	10K	C5	1nF	Q1	2N3904
R6	10K	C6	2.2nF	Q2	2N3904
R7	1K	C7	2.2nF	Q3	78L05
R8	2.2K	C8	10nF		
R9	22K	C9	47nF	**Potentiometers**	
R10	22K	C10	22uF	TRIGG	A500K
R11	22K	C11	22uF	RATE	B10K
R12	33K	C12	22uF	DECAY	B10K
R13	47K	C13	22uF	LEVEL	A100K
R14	47K	C14 **	100nF	BLEND***	B50K
R15	47K	C15 **	100nF		
R16	100K				
R17	100K	**Diodes**		**Switches**	
R18	220K	D1	1N4001	START	SPST momentary
R19	1M	D2	1N4148	SELECT	SPST momentary
R20	2.2M	4x LED's			
R21 *	1K – 15K	**Relay**		**Buttons**	
R22 *	1K – 15K	1x NA-5W-K		OCT	SPST momentary
R23 *	1K – 15K			MOD	SPST momentary
R24 *	1K – 15K			SEQ	SPST momentary

BOM Notes

- * = current limiting resistors for the LED's (powered by 5v)
- ** = should be a multilayer ceramic capacitors (yellow capacitors)
- *** = 9mm potentiometer (or wired offboard)
- The momentary switches and buttons are normally open. The PCB is designed for these type of buttons.
- The relay is this type.
- Do no replace the quad opamp with a TL074!
- The AVR32DB28 comes preprogrammed with the PCB / DIY kit

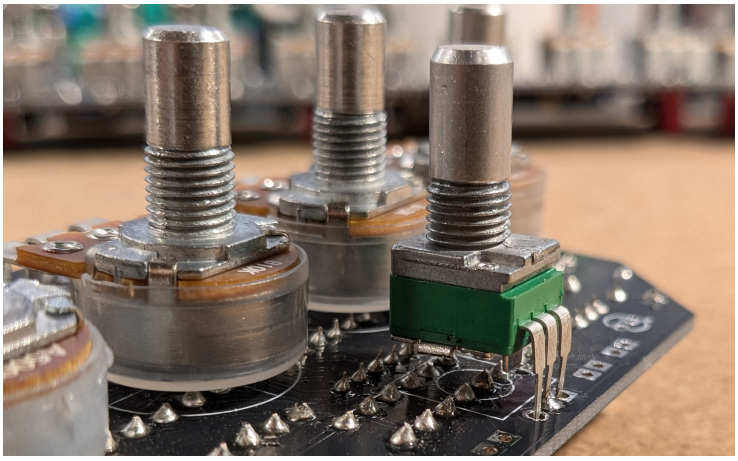
- Things that are not included in the BOM list: enclosure, input and output jacks, Lumberg DC jack, led bezels and knobs.

Special considerations

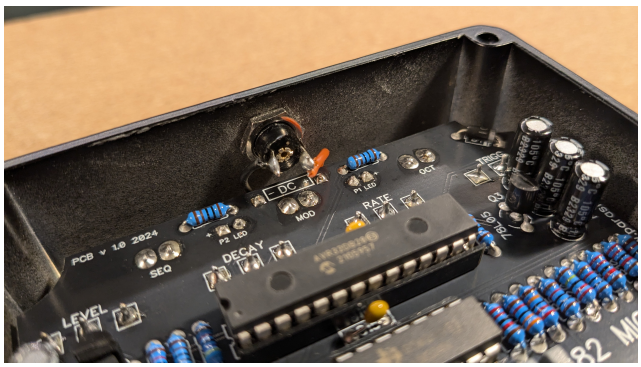
- The optional BLEND potentiometer needs to be a 9mm pot.

  As an alternative you can drill/mount the BLEND pot a bit further down on the enclosure and wire the pot off board. The square hole is pot leg number 1.

- When soldering the 9mm BLEND pot, you need to mount it with only the tips of the legs soldered to the board so that it stands further up from the board, the same height as the other potentiometers. I recommend carefully cutting off the legs on the side that are for mounting (the washer will keep the pot secure). There's also a small tab on the pot that needs to be cut off so that it can sit flush against the enclosure.

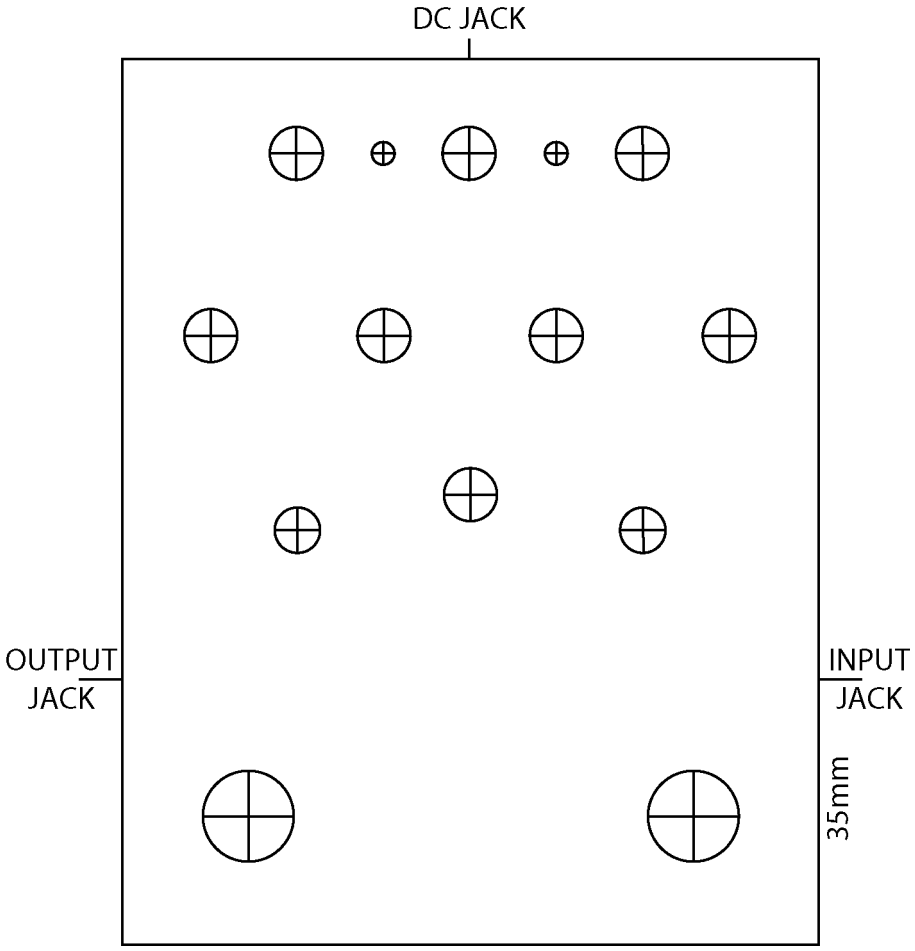


- The DC jack needs to be a small Lumberg 2.1mm type and drilled close to the edge of the enclosure to make it fit.



Screw in the DC jack after the PCB is placed inside the enclosure

Drill Template (1590BB)

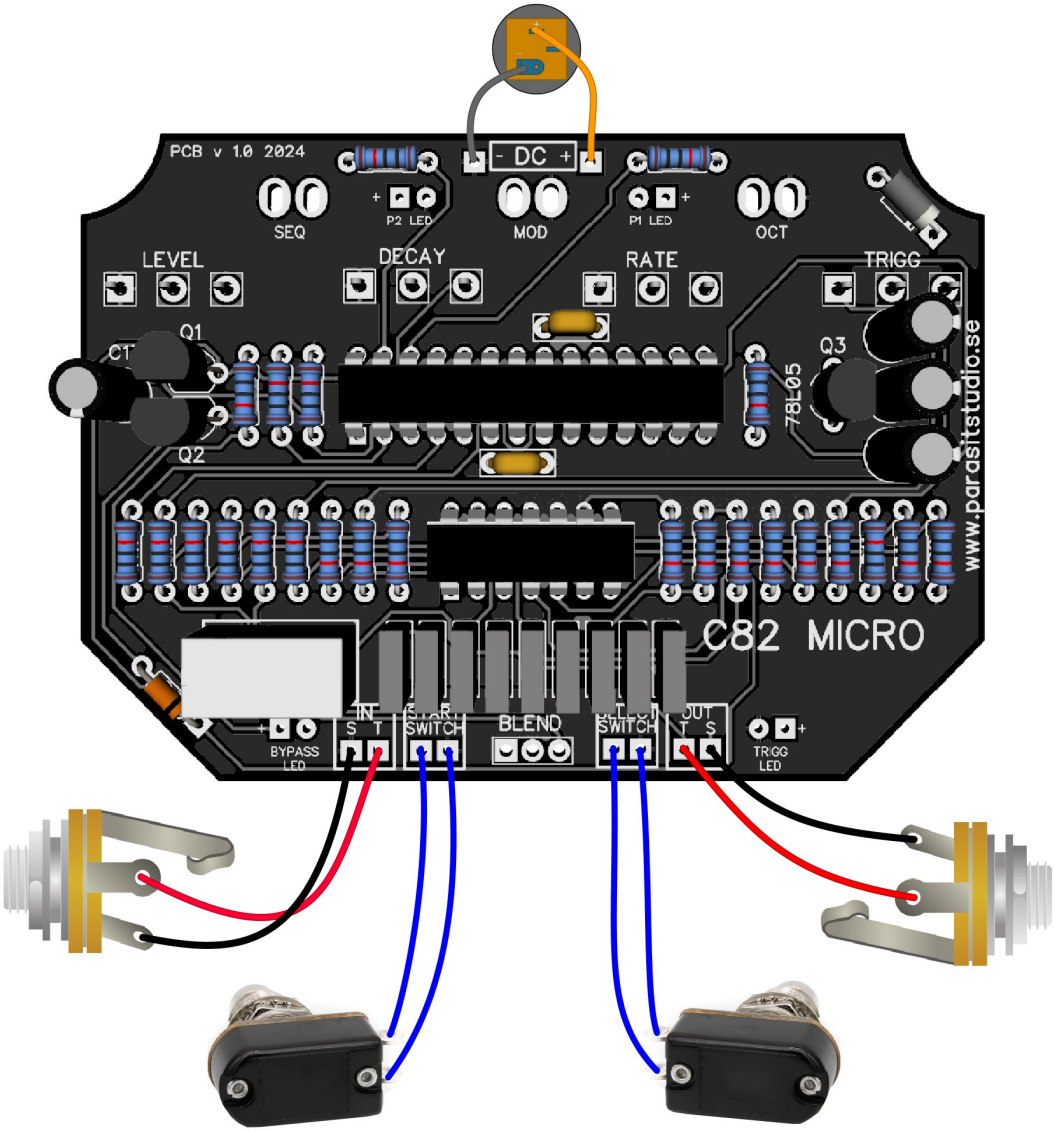


- Make sure your printer isn't doing any scaling (100% print size).
- Jacks are measured from the edge at the bottom of the enclosure.
- On my build I only use bezels for the BYPASS and TRIGG LED's (3mm LED's with 6mm bezels), but I mount the P1/P2 LED's directly through 3mm holes without any bezels, but you can do it however you like if you want bezels on all LED's.
- The center of the DC jack is drilled approximately 8mm from the bottom edge (see the picture on the previous page)

- Typical drill sizes are:
  - LED bezel (for a 3mm LED) - 6mm
  - potentiometers / push buttons - 7mm
  - DC jack (for lumberg style) - 8mm
  - input/output jacks (Neutrik style) – 9,5mm (10mm is ok) 9mm for Lumberg style jacks
  - BYPASS / SELECT Footswitches - 12mm

Measure and confirm before drilling!

Off Board Wiring



- DC negative to the short DC jack lug
- DC positive to the long DC jack lug

- IN T (tip) to INPUT JACK tip
- IN S (sleeve) to INPUT JACK sleeve (the lug with a connection to the inner ring)

- OUT T (tip) to OUTPUT JACK tip
- OUT S (sleeve) to OUTPUT jack sleeve (lug with a connection to the inner ring)

- Wire the START and SELECT footswitches as shown. Which wire goes to which lug doesn't matter.

I recommend that you solder wires to the PCB before installing it into the enclosure. After you have mounted the PCB inside the enclosure, trim the wires to lenght and solder the other ends to the external parts. Don't use unnecessary long wires, but keep some slack. Be careful when trimming wires so you don't make it too short.

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

Smallbear Electronics / Synthcube DIY kit

If you have bought the Smallbear Electronics DIY kit and got a faulty, incorrect or missing component, please contact Smallbear Electronics.

smallbearelec@synthcube.com

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