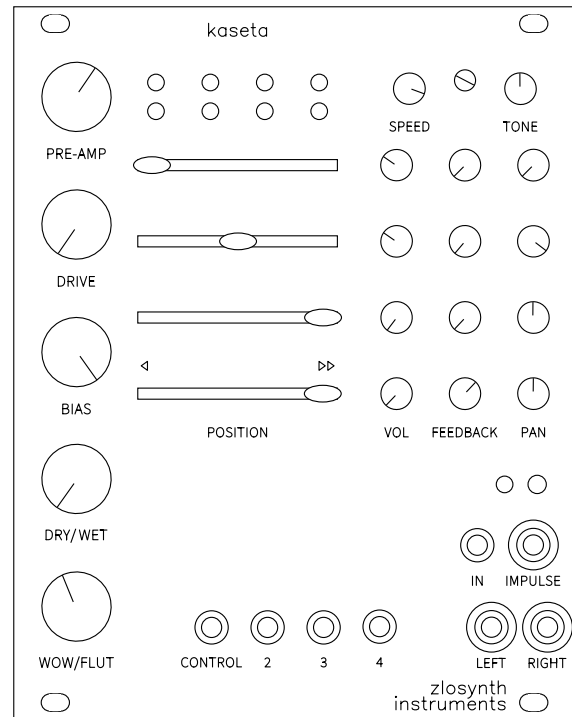
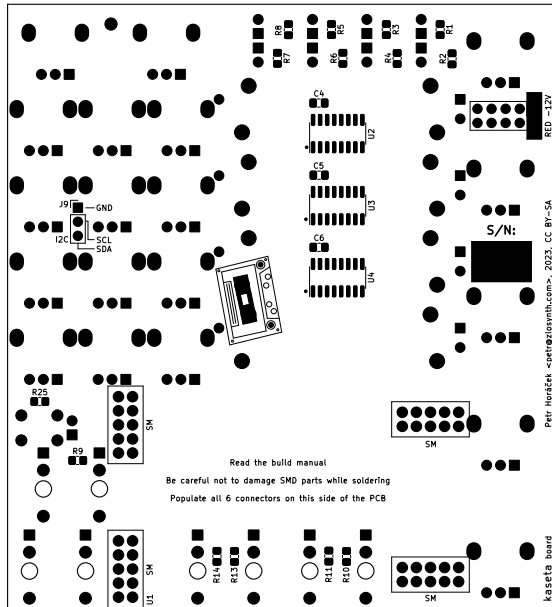


Kaseta – Build Manual



1 Overview

You can always find the latest version of this build manual on <https://zlosynth.com/kaseta-build-manual.pdf>.

This kit contains a printed circuit board (PCB) with all the surface mount device (SMD) parts already pre-soldered. The through-hole components are left to be assembled by you.

Pay attention to the orientation and position of all the parts. Desoldering them would be difficult and may break the module. Also, use care not to touch the pre-soldered SMD parts with the soldering iron. Read through the whole manual first. Make sure you understand all the steps before you start soldering.

2 Tools required

- Soldering iron
- Masking tape
- Slotted head screwdriver
- Phillips head screwdriver
- Side-cutters

3 Bill of materials

Start by unpacking all baggies into a bowl so you don't lose any components.

- 1 × Front panel
- 1 × PCB (the black board)
- 1 × Daisy Patch Submodule (the yellow board)
- 5 × Blue trim potentiometer with center detent*
- 9 × Blue trim potentiometer without a detent
- 1 × Green D-shaped potentiometer with center detent*
- 4 × Green D-shaped potentiometer without a detent
- 5 × M7 nut and washer
- 5 × Knob
- 4 × Slider potentiometer and cap
- 8 × 3.5mm jack socket
- 8 × M6 nut and washer
- 1 × Tactile button
- 9 × Red LED
- 1 × Male connector 2×5
- 4 × Female connector 2×5
- 1 × Male connector 1×3
- 1 × M2 slotted screw, M2 Phillips screw, standoff

* Potentiometers with a center detent snap when they are turned through their center position.

1. Mount the four female 2×5 connectors on the pins of the Daisy Patch Submodule. See Figure 2.
2. Plug the mounted connectors into the black PCB through footprints marked as "SM". Make sure to put them on the correct side of the PCB, where all four connectors are marked.
3. Solder all the pins in.
4. Once done, carefully detach Daisy Patch Submodule to prevent it from getting damaged while progressing with the build. It may be a little difficult. Pull each connector by a millimeter at a time. The result is illustrated in Figure 3.

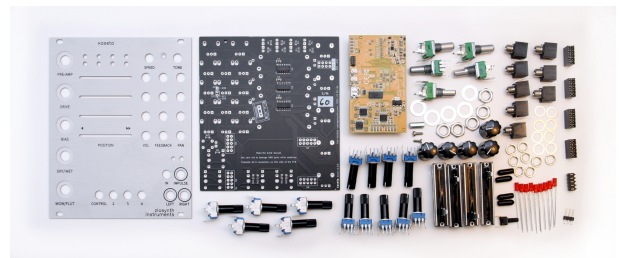


Figure 1: All the components laid out

4 Power and I2C

Start by soldering the power and I2C connectors.

1. Take the male 2×5 connector and put the side with shorter legs through the footprint marked with a white stripe and a label "RED -12V". Make sure to put it on the marked side of the PCB.
2. Solder a single pin in and check that the connector is upright. If it is not, heat up the pin and align the connector by pushing it against the board.
3. Solder all the pins in.

Repeat the process for the male 1×3 connector, placed in the "I2C" footprint.

5 Daisy Patch Submodule

The Daisy Patch Submodule is connected to the main PCB through a set of connectors.

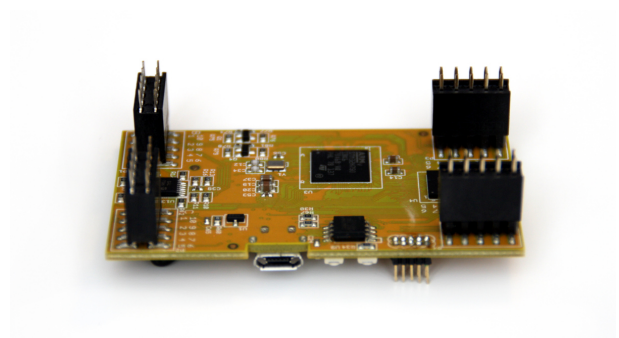


Figure 2: Connectors mounted on the submodule

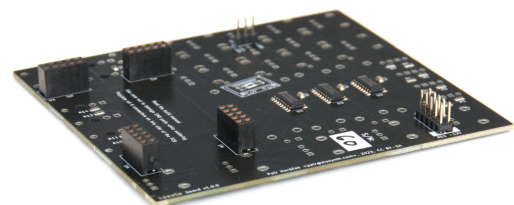


Figure 3: The power, I2C and submodule connectors

6 Front panel

Now when all the internal parts are soldered, the next step is to assemble parts sitting in the front panel.

1. Put the hex standoff on top of the hole labeled "Standoff" and secure it there with the Phillips screw.
2. Snap in the button.
3. Place the 5 green D-shaped potentiometers in "Green pot" footprints, do not solder them yet. The big legs on the sides are used to snap the pot in. Use the appropriate potentiometer for the bottom-most place marked as "Green pot w/ center detent".
4. Similarly, place the 14 blue trimmer potentiometers in the "Blue pot" footprints, some of which are marked for center detent.
5. Place sliding potentiometers in their marked places. Do not put their caps on yet.
6. Put all jack sockets into the PCB.
7. Put LEDs in place. Pay attention to the markings on the PCB explaining their correct orientation.
8. Now when all parts are in, carefully put the front panel on them. This is probably the most difficult part of this build. Be patient aligning all parts so they fit through holes. If you see that some are not getting through, use tweezers to align them. Leave the LEDs hanging freely in their holes for now.
9. Put washers on the pots and jacks.
10. Secure the front side of the standoff with the slotted screw. Tighten all the potentiometers and 3.5mm jack sockets in place with their nuts. Use care not to scratch the panel. Plastic tools are preferred, and steel drivers should also serve well. If you only have pliers, put them in a thick plastic bag. Protect the panel!
11. Go over all the trimmer pots and check that they turn smoothly. Double-check that WOW/FLUT, TONE, and all four PAN potentiometers have a center detent.

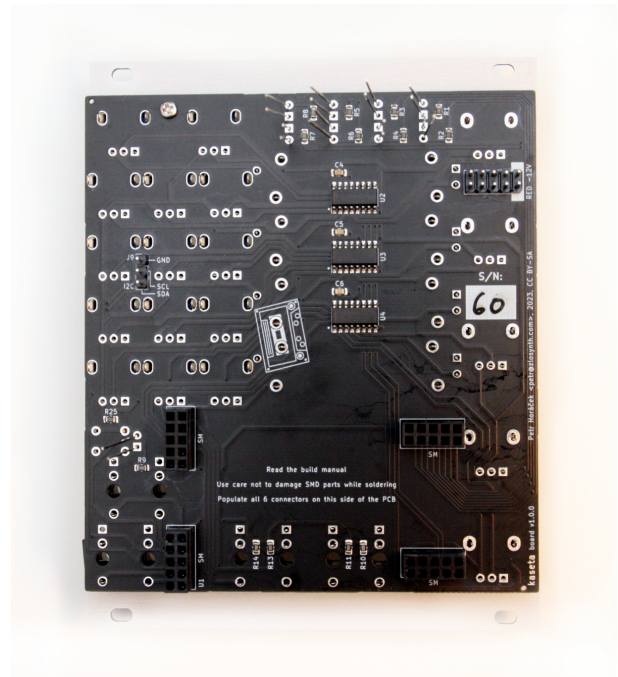


Figure 4: Rear of the panel before soldering



Figure 5: Taped LED holes

Once all the components are placed correctly, move on to soldering.

1. Solder in all the rotary potentiometers and jacks. Only solder the three smaller terminals of each potentiometer.
2. For each sliding potentiometer, start by soldering one of its smaller terminals. After that, slide its shaft to the side of the just-soldered terminal. Then heat up the terminal while pressing on the shaft to make it sit flat on the PCB. Repeat the process on the other end of the potentiometer. Finally, solder the third small terminal.
3. Use a masking tape on the portion of the panel with holes for LEDs, see Figure 5. Push the LEDs against the tape, so they are even with the panel surface.
4. Solder the LEDs. Then snap their legs off and remove the tape.
5. Use tweezers to align the button. About 2 mm of it should be sticking out of the panel. Test that it can be easily clicked and returns to its resting position. If it does not, try to angle its body.
6. Once the button is in a satisfying position, solder one of its legs, double-check it can be clicked and that it returns, and solder the remaining legs.

7 Knobs and caps

You can now put knobs and caps on the potentiometers.

1. Put black knobs on the five pots on the left. Align them with the D-shaped shaft and press them in. You may need to pull them a little bit if you see they are scratching the nut.
2. Mount the black caps onto the shafts of slider potentiometers.

8 Final assembly

Connect the Daisy Patch Submodule to the main black PCB to complete the build.

9 Congratulations

The module is now complete. Have fun!

You can find the user manual on <https://zlosynth.com/kaseta-user-manual.pdf>.

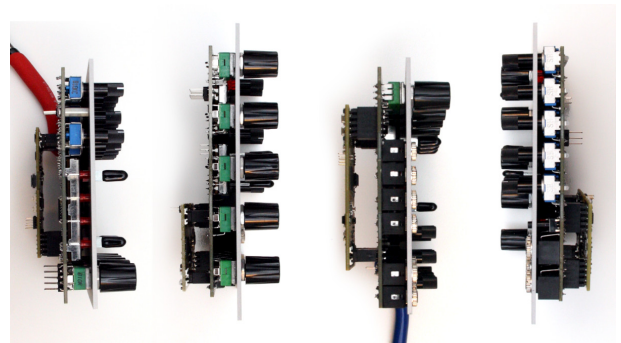


Figure 6: Top, left, bottom and right side view of the assembled module

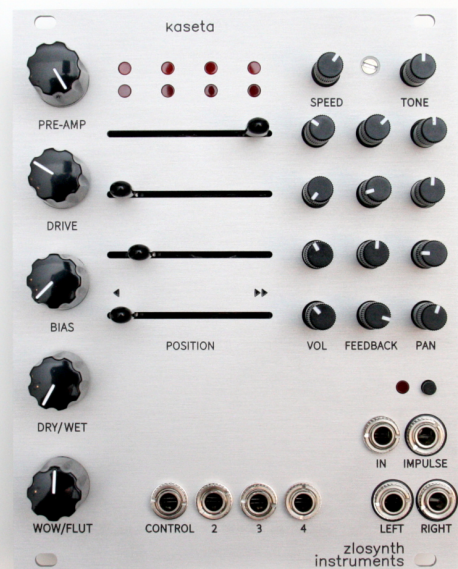


Figure 7: Front view of the assembled module