NOH

PIANIST - Build Guide

V1.1

Introduction-

This build document aims to help you with the assembly of the module. It is quite difficult to build and is not recommended as a very first build.

It is highly recommended that you read through the entire guide once before starting with the assembly.

Soldering a module together is always a relaxing and gratifying moment when done properly, don't hesitate to take breaks when you feel you need to and double-check steps before progressing in your assembly.

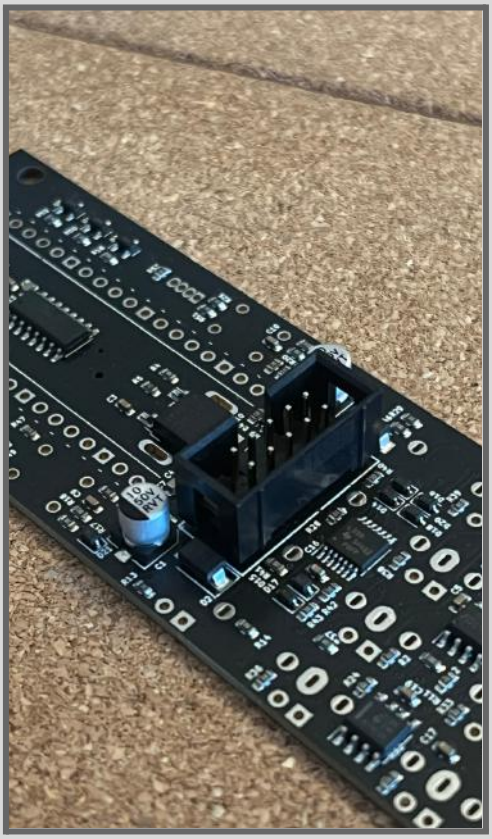
Requirements-

To complete this build you need:

- A Soldering Iron and Solder
- Pliers and Side Cutters
- A Multimeter (optional but **extremely** recommended for debugging)
- Safety Glasses for assembly
- A flat screw-driver
- The Thonny IDE if the firmware needs to be installed / updated- Thonny is available here

Make sure you also take a look at the Bill of Material (BOM) to check whether you have all the required components to complete the build.

Build Guide-



• If not already mounted, start by placing the keyed header, be mindful of its orientation and match it with the indication on the PCB.

• You can solder a pin on each side, check whether it sits flat on the PCB and solder the rest of the pins if it does.

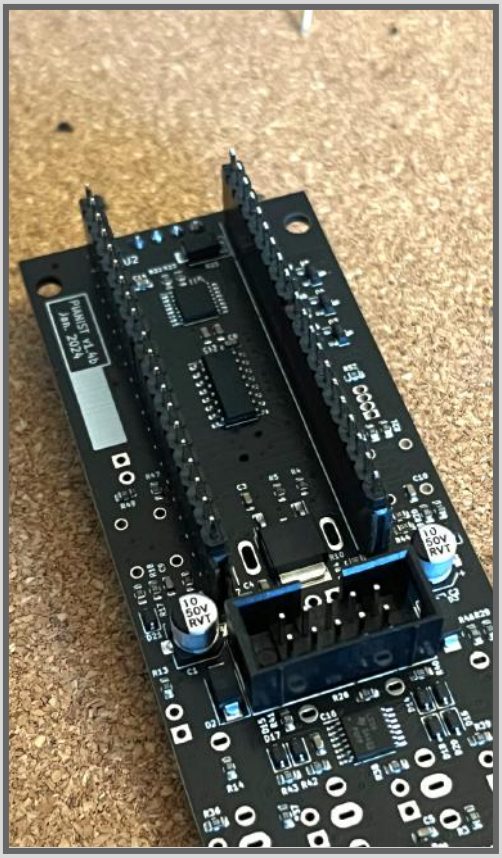
• flip the PCB and mount the SMD joystick, **mind the orientation of the component**. The black “triangle” matches the one shown on the footprint. The placing holes should help keep it in place

• Start by the two bigger pads to secure it in place, and then solder the smaller ones

• **NB-** This is the only SMD component of the build and the big pads should make it approachable

• On the same side, mount the 4-pins low-profile female socket for the screen

• You can start by soldering only one pin, check if the socket is flat and in-line, and solder the rest if it is (or reflow if it isn't).



• Flip the PCB and mount the two 20-pins female sockets if they are not already mounted. You can follow the same process to make sure they sit flat.

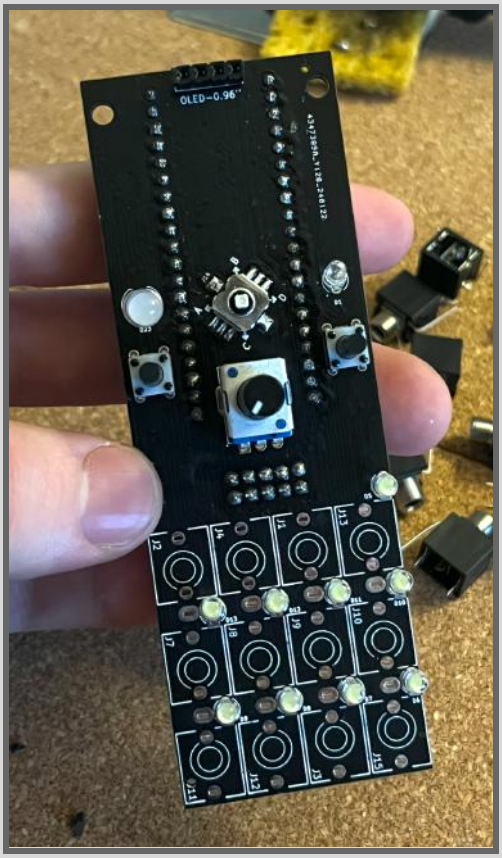
• Once soldered, you can insert the male headers into both of them, with the longer side of the header going in.

• Once the male headers are inserted you can put the microcontroller board on top of both, and solder the pins

• Always be **extremely** careful of where your soldering iron is when you solder, it should alway be oriented as far as possible from components on the board.

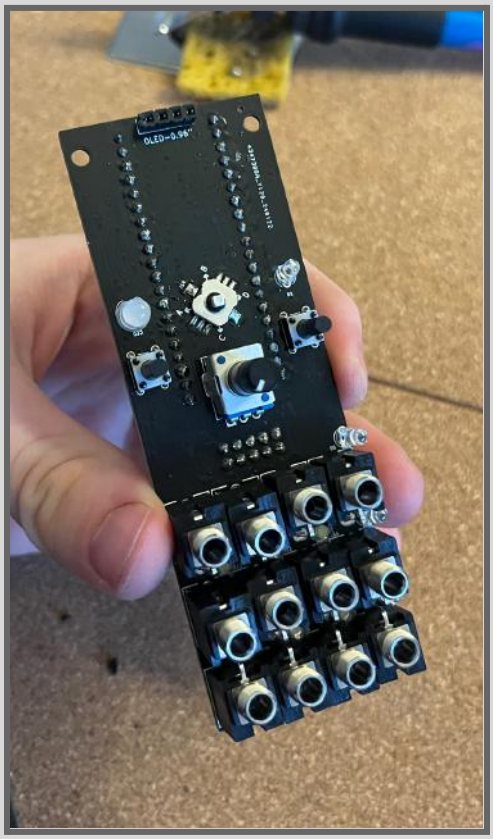
• Flip the PCB, and insert the two buttons, the pot, the RGB and orange LED but-

DO NOT SOLDER ANYTHING.



• The orange LED looks like the white ones, use the image above to help locate the orange one. Watch out for the polarity of the two LEDs. The orange LED has the shorter leg going to the square pad (the top one), and the RGB has the “flat” side on the top, another way is that the longest leg should be second from the top.

• make sure the buttons and pot are fully inserted (again do not solder anything).

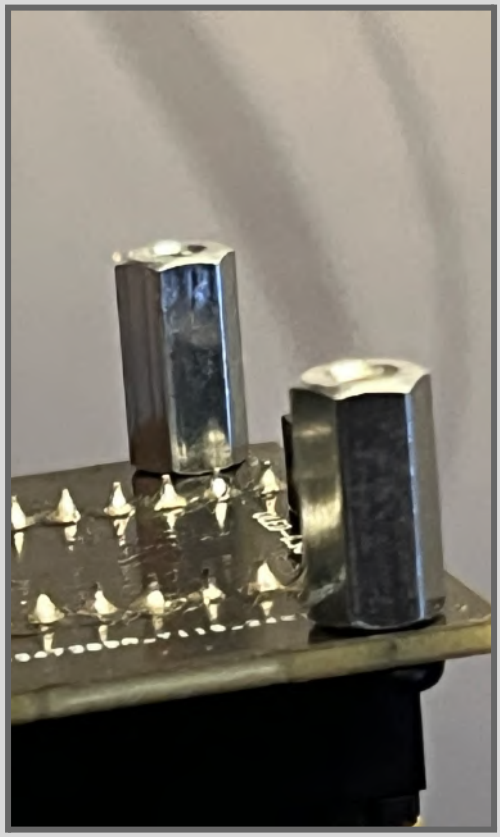


• Now insert the jack sockets and the white LEDs, but again-

DO NOT SOLDER ANYTHING.

• Watch out for the jack sockets, the middle row of jack sockets share a common pin with the row **below**, not above it (as shown in the picture).

• The LEDs will have their shorter legs go in the square pads (the bottom one this time). Again do not solder anything yet.



• Once all components have been inserted (without soldering them) you can screw in the two hex standoffs.

• **TIP-** considering the oled screen boards could have a bigger size, try to make the side of the hex standoff facing the centre of the board parallel by rotating it around (like shown in the picture).

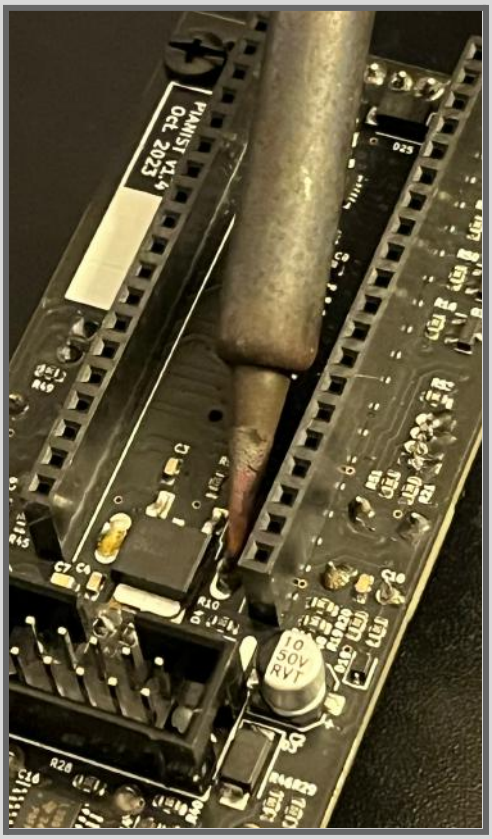
• Once that is done, insert the front panel. Be mindful that nothing has been soldered in so components might fall out of place.

• You can screw in the hex standoffs (as shown in the picture) to secure the panel.

AGAIN, DO NOT SOLDER ANYTHING.

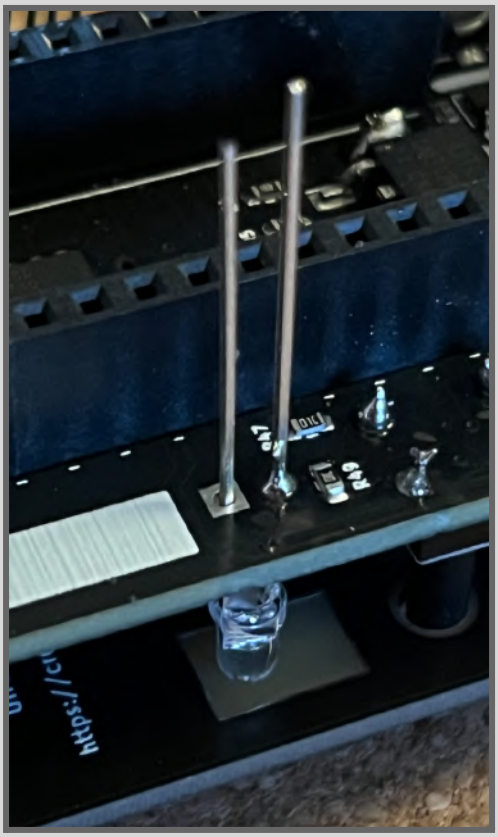
• To further secure the front panel, hand-tighten four knurled nuts to the four jack sockets in each corner.

• This is important as it will level all components and make sure that the buttons will stick out enough to feel good to the touch.



• Once you have checked that no components got out of place, you can start soldering.

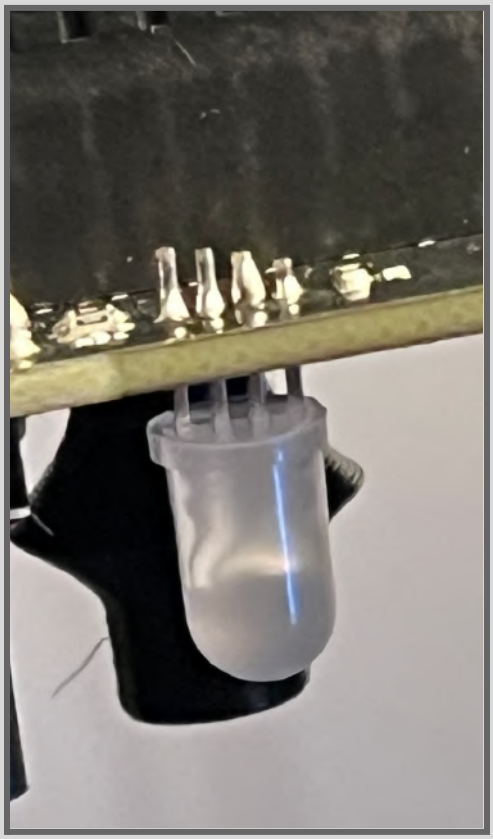
• You can start with the potentiometer, which is in a pretty complex spot! Simply remember to have your soldering iron in an orientation that puts it as far away from other components as possible.



• To solder the orange LED in place, push it towards the front panel and solder **only one** pin of that LED.

• Then, reflow the pin (i.e. make the solder melt again) and while the solder is liquid use the leads to orient it correctly (**make sure you do not burn yourself as the reflowed lead will be hot**).

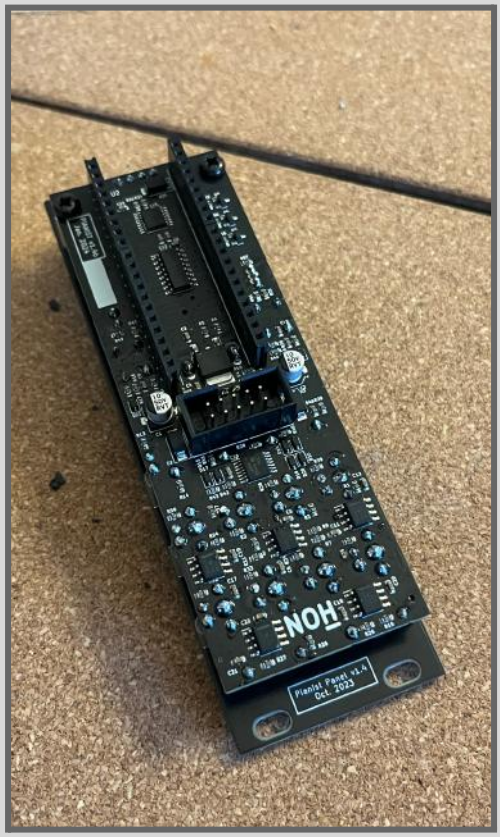
• Once you are happy with how the LED sits, remove the soldering iron and the LED will stay in place. You can then solder the second pin and clip the leads.



• The RGB LED will require more attention as the pads are closer, but the same process applies.

• You can start by pushing it towards the panel, soldering only one pin, reflow to nudge it in place, and solder/clip the rest.

• **Make sure you do not short circuit pins** between them by putting too much solder. Use the image as a reference.



• You can then solder the two buttons, the jack sockets, and the bottom LEDs.

• To solder the white LEDs in place, you should use the same process shown previously for the orange LED. For those in the middle that you cannot see, simply move them so that the pins look upright.

• **NB-** Again, **make sure your soldering iron doesn't touch other components** as it will damage them.

• You are done with the main board, congrats! You can un-tighten the four knurled nuts, and unscrew the top screws of the hex standoffs to remove the front panel from the PCB

• Take the oled screen and the M2 nylon screws, nuts, and washers.

• Once the front panel has been removed from the board, you can take one nylon screw, pass a washer through it and then insert it in the hole shown on the right ->

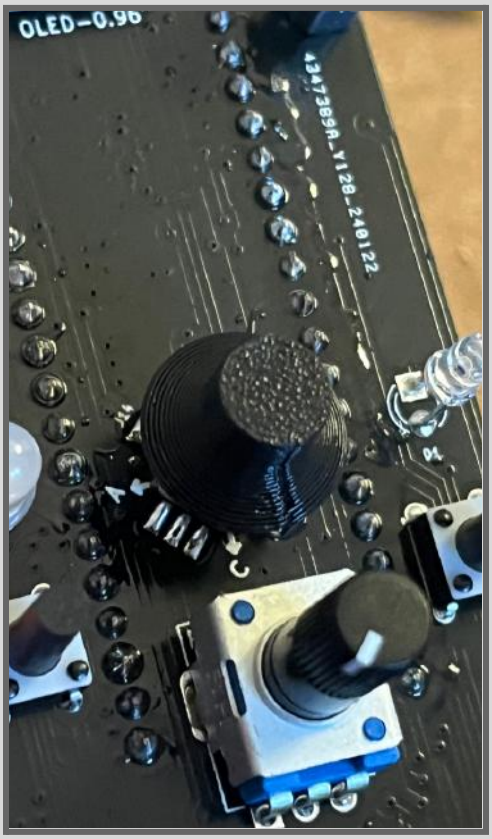
• **NB-** This process will have to be done twice on the two similar-looking holes

• Flip the panel, add another washer, then screw in a nylon nut to tighten the screw in place

• The layers should look like what is shown on the picture, and the screw should be well-tightened.

• Once the process has been done on both holes, you can pass the oled screen board through the screws.

• To secure it in place, put another nut on top of the screw and tighten it, as shown in the picture.



• Press the joystick cap on the top of the joystick. The square hole on the bottom of the cap should key in the square joystick.

• You might have to put a little bit more force to press it until you hear a click, just make sure that force isn't unreasonable.

• To finalise the hardware part of the assembly, insert the front panel on top of the PCB.

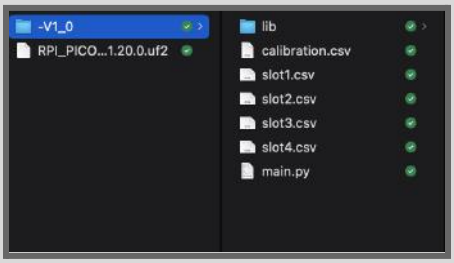
• It is normal for the screen headers to not fully go in the female socket underneath. Also, It might take some nudging to level it with other components, but once that is done you can screw back the hex standoffs and tighten the knurled nuts on the jacks.

• The next step only applies if your Pi Pico is **BRAND NEW**. Pianist DIY kits should have the firmware pre-installed.

Installing/Updating the Pianist Firmware

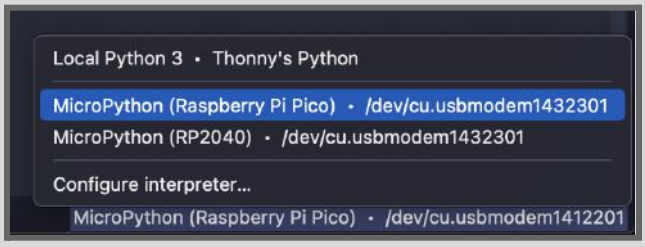
• Get on the Pianist product page and download the firmware file. It should be a compressed zip file with the correct version of MicroPython (1.20.0) and another folder inside of it.

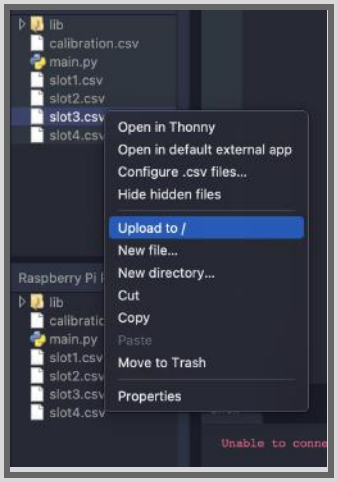
• Plug the Raspberry Pi Pico to the computer, make sure you use a USB cable that has data lines.



• If the Pi Pico is new you should see it as a peripheral device once plugged in. Go to the firmware folder and drag and drop the ".uf2" file onto the Pi Pico. This will install the correct version of MicroPython on the Pico (v1.20.0).

• Once MicroPython is installed on the Pi Pico, install/open the Thonny IDE and connect to the Pi Pico as a device from the bottom right menu.





• After connecting to the Pi Pico, erase all of the documents in it if you are updating the firmware by selecting and deleting everything.

• Once that is done or if you are installing it for the first time, select all **seven** files from the firmware folder you downloaded and select "upload to /" to add all needed firmware files to the pico.

Unplug the pico, do a module check, and enjoy the new firmware :)

Inspection -

While the assembly might be finished, **it is not a good idea to plug power through before inspecting the module.**

• Start with a visual inspection, looking for any pins you forgot to solder, pins you soldered too much and shorted to other pins, or bad solder joints.

• Check the polarities of components where polarity matters (i.e. the power header, the MCU, the LEDs, and the joystick).

• Look for any damage on the board, specifically whether tracks or components got damaged.

• Finally, using a Multimeter, check for shorts between the power pins (12V to GND, 12V to -12V, and GND to -12V).

Module Check (debugging) -

Once you are happy with all the inspection steps above, you can follow the checks detailed below to be sure that the Pianist will operate normally:

• Start by giving power to the module. The expected behaviour is for the gate LED to shine a Blue/White colour and for the clock to run. Turn the internal clock speed all the way up to make sure it works.

• After a small amount of time, the user interface will load and display the main menu on the screen. That’s good news!

• If the Module never boots up and is instead frozen with a black screen and the left button still shining Blue/White, it means that the firmware could not load.

• If that's the case, keep the Pianist powered and plug the Pi Pico back to the Thonny IDE, open its “main.py”, run it and see what error message it gives. Some errors are:

- Wrong version of MicroPython (usually a storage error)
- Broken / unconnected screen
- Missing files for the Firmware
- Badly soldered DAC IC

• To check if all LEDs are alive, change the root of the chord to an Ab (as shown above). If any LED does not shine it means that it either got soldered the wrong way or that it died.

• Go to the “GLOBAL” menu and load slot number 2. This will return automatically to the main menu and the 1st chord should be a Bmin(add9).

• Take a patch cable and connect it to the v/oct input of an oscillator. Test out all eight outputs individually while listening to the oscillator.

• All should give a higher pitch than the one you hear when nothing is patched. You can also move the root of the chord around and see that the outputs follow.

• Finally, press the “Play/Stop” button to launch the sequence and turn the internal clock all the way up.

• Test all three inputs using either a square wave LFO or any gate you can send to the module.

- The gate input will turn the gate LED purple and randomise chord colours.
- The Play/Stop input will toggle between main menu and sequence
- The clock input will replace the internal clock

That’s all!

Conclusion -

This guide went through the assembly process of the module and what to be careful of when building it. If you haven't already, you can read the user manual to know what the power requirements are and how to operate it.

You can find this build guide, and other important documents regarding this module and others on the NOH-Modular website.

I hope you'll find good and interesting use out of this module.