mini Ten Key Plus Assembly Guide

Written by minibois, November/December 2021.

2021

github.com/minibois/ten-key-plus

Assembly Guide version: *v1.0a*
Assembly Guide license: *CC BY-SA 4.0 (See section [9])*
Applicable PCB version(s): *rev.1a*
PCB/Schematic source location: https://github.com/minibotis/ten-key-plus

————[0]————

Table of Contents

[0] Introduction **2**
[0.1] Warning 2
[0.2] General tips: 2

[1] Prerequisites **3**
[1.1] Tools 3
[1.2] Supplies 3
[1.3] Components 3
[1.3.1] Optional components 3
[1.4] Inspect/Testing the board 4

[2] Choices to make, general information **4**
[2.1] The layout 4
[2.2] SMD vs THT components 5
[2.3] Reset Switch 5

[3] Case, plate and fasteners **6**

[4] Assembly **6**
[4.1] Soldering techniques 6
[4.2] Diodes 6
[4.2] THT Soldering 7
[4.3] SMD Soldering (Optional) 7
[4.4] Microcontroller's headers / socket 8
[4.4.1] Headers vs. hotswap socket 8
[4.4.2] Soldering in the headers 8
[4.4.3] (Optional) Hotswap socket install 9
[4.5] (Optional) Stabilizers 9
[4.6] (Optional) Rotary Encoder 10
[4.7] Switches and plate 10
[4.8] Microcontroller install 11
[4.9] Case preparation 11
[4.10] Assembling the Ten Key Plus into the case 13

[5] Firmware, testing and tweaking **13**
[5.1] (Simple) flash the supplied firmware 13
[5.1.1] Testing the Ten Key Plus 14
[5.2] (Intermediate) Changing the firmware through the QMK configurator 14
[5.3] (Expert) Editing the keymap manually 15
[5.4] Enjoy your Ten Key Plus! 15

[6] Sources **16**

[7] Differences between revisions **16**

[8] FUT: Frequently Used Terms **17**
Microcontroller 17

Mechanical keyboard components/concepts 17
Electronics 18
Soldering 18
Software/Firmware/QMK 18

[9] License 19
[9.1] Changes 19

[0] Introduction

This assembly guide shows how to assemble the Ten Key Plus, the full source PCB and schematic of which can be found here: https://github.com/minibois/ten-key-plus
This guide will cover the tool/component prerequisites, testing procedures, assembly steps and flashing the firmware.

The instructions in this guide are written for people who are still a bit new to soldering and electronics projects, but do have some experience under their belt.
This assembly guide is written under the CC BY-SA 4.0 license, full details of which can be found in section [9].

[0.1] Warning

Assembling the Ten Key Plus requires you to solder components in place. Soldering carries with itself some inherent risks, especially so when working with leaded solder. Following this guide is at your own risk.
Always follow the necessary and recommended safety procedures, such working in a well-ventilated area and not (directly or indirectly through unwashed hands/equipment) ingesting (leaded) solder. Wear protective goggles too.

Most soldering tasks are through-hole, instead of surface mount (with only one SMD part required), which makes this project accessible to a lot of people. Some familiarity with soldering is required, for which the soldering guide for EEVBlog is a good introduction:
https://www.youtube.com/watch?v=J5Sb21qbpEQ
This video covers the basic tools and techniques. This guide will also cover these, but in lesser detail than this video.

Some practice on a donor board can be helpful too, but it comes down to only attempting this project at your own risk and if you feel confident in your ability and knowledge of soldering.

[0.2] General tips:

When working with soldering, there are some general tips tips to keep in mind:
- A soldering iron is a hot piece of equipment, handle it with care and never leave it unattended.
- Wash your hands after soldering
- Avoid touching other stuff during soldering, to not spread around any dangerous substances
- Work in a well ventilated area, preferably with a fume-extractor or fan running

- Don't keep your iron on the board for a long time, to avoid heat damage
- Add flux when reheating solder joints
- It's easier to clean flux when it hasn't solidified, so clean up the flux as soon as possible
- There is flux included in the soldertin, cleaning is recommended after soldering in each component

[1] Prerequisites

To assemble the Ten Key Plus, certain tools and components are needed, listed below.

[1.1] Tools

- Soldering iron
    - A station - such as the Hakko FX888D or Ksger T12 - is recommended. Cheap tools can make the job much harder
    - If your station of choice does not feature a way to adjust the temperature (at least between 160°C - 350°C / 320°F - 660°F) an additional tool is needed to use the heat inserts discussed in [4.8]
- Multimeter (needs to have continuity mode at the least)
- Sidecutters/flush cutters (if using THT diodes, more on this in section [2.2])
- Screwdriver (PH00 size)
- Tweezers (for the optional testing in step [4.4.3])
- Anti-static bracelet
- Fan: to blow away fumes from yourself
- Safety goggles (solder can splatter, especially when working with solder wick and using side cutters can be dangerous for your eyes)

[1.2] Supplies

- Soldertin 60/40 Sn/Pb 0.7mm recommended. (0.5mm only needed for SMD diodes (see [2.2])
- Isopropyl alcohol (IPA) (70-99% recommended)
- Something to apply the IPA with (cotton swab/toothbrush recommended)
- Flux
- Optional: solderwick/pump (in the unlikely event you need to desolder something)

[1.3] Components

The components needed to assemble the eLiXiVy are described on the BOM.
Octoparts link: https://octopart.com/bom-tool/pvFPXgm8
.txt link: https://github.com/minibois/ten-key-plus/blob/master/Documents/BOM.txt
Excel (.xlsx) link: https://github.com/minibois/ten-key-plus/blob/master/Documents/BOM.xlsx

[1.3.1] Optional components

As mentioned on the BOM, some components are optional. The optional components are the reset switch and rotary encoder, which will be discussed in section [2.3] and [2.1] respectively.
The BOM also mentions getting either 1N4148 *or* 1N4148W diodes, which will be discussed in section [2.2].

How many diodes/switches/keycaps are needed depends on the layout chosen, which will be discussed in section [2.1].

[1.4] Inspect/Testing the board

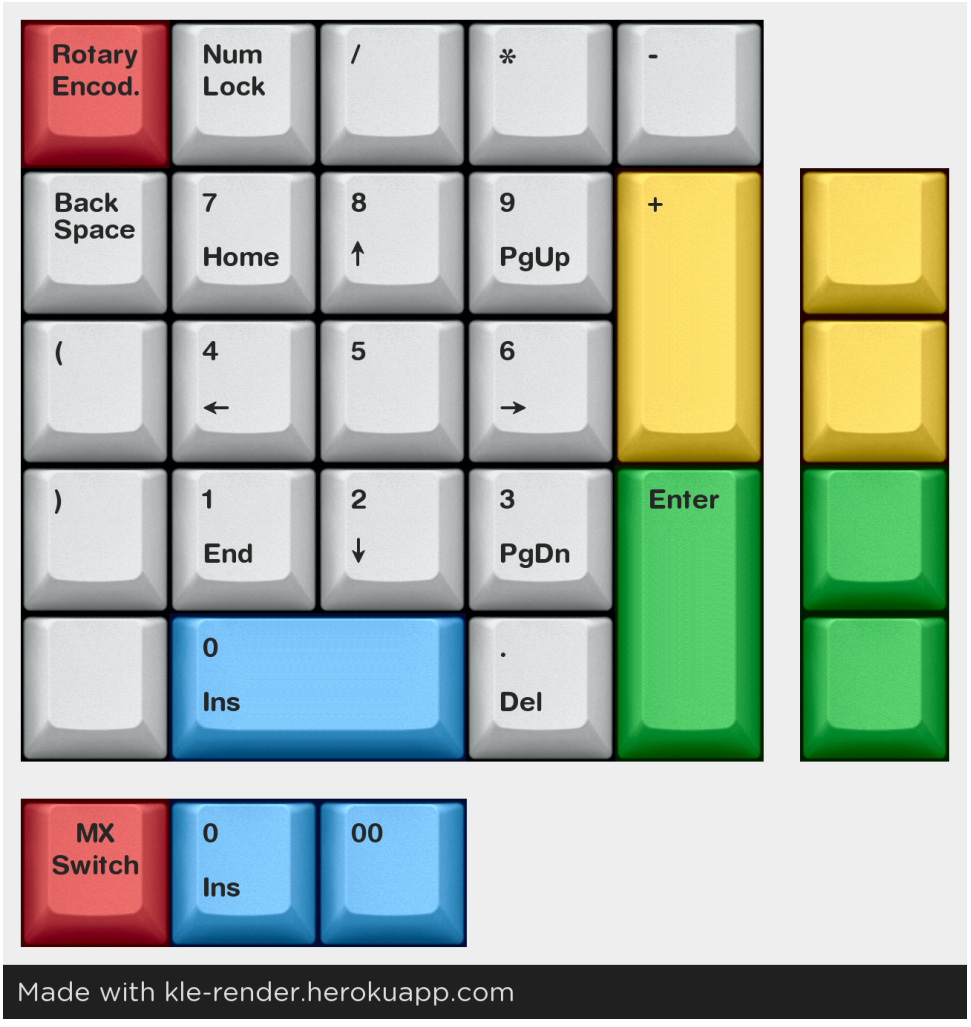
It is important to first know the PCB is in a good condition - before assembly of the board - as it may be difficult to find out later on if the fault is in the components or board itself.
There are a plethora of things you could test. Usually a visual inspection should suffice, as there is not much that could go wrong with a reputable PCB manufacturer.

[2] Choices to make, general information

There are some choices that can be made with the Ten Key Plus, specifically choices about what layout you want to use and whether you want to use THT or SMD diodes.

[2.1] The layout

The layout of a (mechanical) keyboard is defined by the sizes of key(cap)s you use and where.
The Ten Key Plus' layout can be described as a numpad with an extra column, an option to exchange 2u keys for 1u and an option for a rotary encoder instead of a switch.
As is usually the case, a picture speaks a thousand words:



Made using https://kle-render.herokuapp.com/

This layout can be described as the numpad with an extra column, with some more choices to make regarding rotary encoder and 1u/2u options.

Depending on the layout chosen (and whether or not a rotary encoder is used), the Ten Key Plus requires between 21 and 25 keyswitches and diodes.

[2.2] SMD vs THT components

This guide has mentioned surface mount device (SMD) and through-hole technology (THT) a few times.

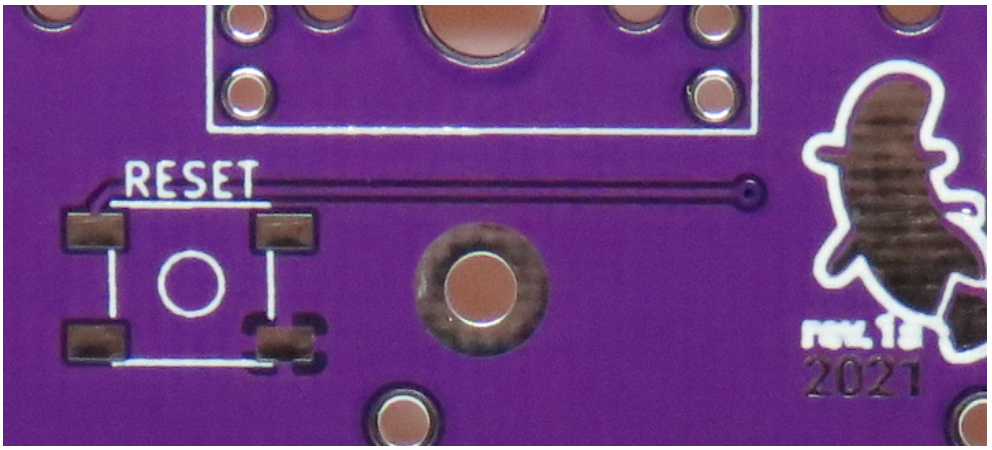
The difference between these two types of devices is the way they are secured to the PCB.
SMD components adhere to the board with metal pads. This includes parts like the microcontroller and passive components (such as the capacitors)
THT components have legs that stick through the board and adhere like that. This includes the switches, header pins and optionally the diodes.

Most components on the Ten Key Plus are THT (switches, microcontroller/socket)
THT: Keyswitches, rotary encoder and ICSP headers.
SMD *or* THT: diodes (discussed in more detail in [4.2]).

[2.3] Reset Switch

The reset switch is installed on the bottom of the PCB and as the name implies is used to reset the Ten Key Plus. When installing the firmware on the microcontroller, the reset has to be pressed.
It is an optional - but recommended - component, as it's possible to reset the microcontroller without the button itself.

The button simply connects the 'Reset' and 'Ground' pins of the microcontroller, which can also be done by touching tweezers/a paperclip/something metal to the reset and ground pins here:



Short the top left and bottom right pads together, by touching both at the same time with a paperclip/tweezers/etc.

It is still recommended to use the button, as it's much easier and more reliable to push a button rather than the alternative of shorting the pins together.

[3] Case, plate and fasteners

The Ten Key Plus uses a 3D printed plate and case, to hold it all together in a presentable package.
The switches are snapped into the plate and the whole PCB/switches/plate assembly is attached to the case using heat inserts, standoffs and screws.
Information on how to install the heat inserts, use the plate and attach it all together will be given in the relevant sections.

The provided case and plate files have been confirmed to work with PLA material on a Flashprint Adventurer 3. The plate was printed with 2 perimeter, the case with 4 perimeter (to better support the heat inserts).

The heat inserts are 3mm diameter, 3mm height, the standoffs are 3mm height and the screws are 4mm height.
All three - heat inserts, standoffs, screws - are M2 standard.

[4] Assembly

After gathering the components, tools, and supplies, testing the PCB, and making the layout-/diode choices it's now time for assembly.
Assembly is essentially six steps:

1. Soldering the diodes (SMD/THT)
2. Soldering the microcontroller socket
    a. Optional: test the PCB if you're using a hotswap socket
3. Install the stabilizers
4. Placing the switches in the case/plate and soldering them in
5. Testing it
6. Putting the case together
7. Use it!

[4.1] Soldering techniques

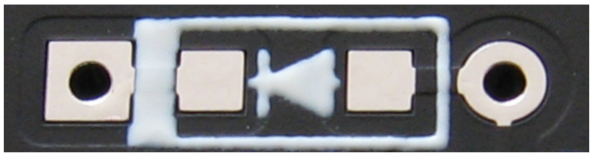
The Ten Key Plus assembly features THT soldering and optionally SMD soldering.
Be sure to consult section [0.1] and [0.2] for some general information/warnings on soldering, including a link to a video by EEVBlog on how to solder.
Refer to the different sections of [1] for a list of tools, supplies, and components needed.

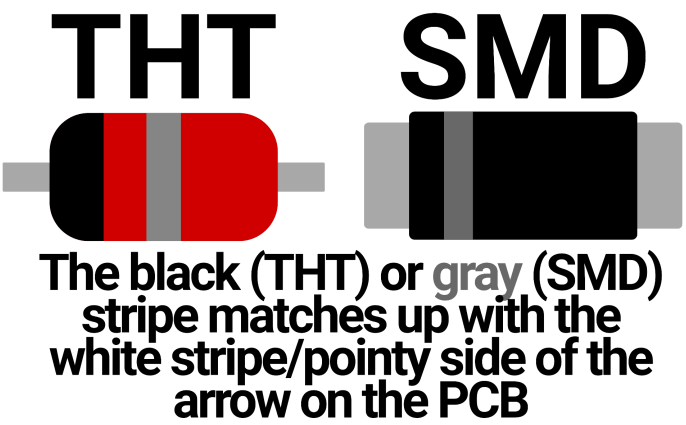
[4.2] Diodes

The diodes can be installed on either the front or the back of the PCB (up to personal preference) and either THT or SMD variants can be chosen.
THT variants are easier to solder in place, but take up space on both sides of the board.
SMD variants are smaller, but more difficult to solder in place.

Regardless of the type of diode chosen, it is important to keep in mind the orientation of the diode. They only work in the intended way when installed in a certain direction, indicated by a stripe on the diode and a graphic on the silkscreen of the PCB:

Diode direction





[4.2] THT Soldering

A throughhole component has legs that stick through the PCB, which allow it to be soldered to the PCB.

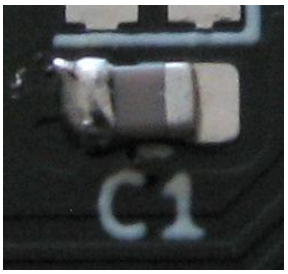
The way these are installed is by sticking the component's leg through the board, holding your soldering iron to the pad and leg and after about a second adding solder tin from the other side
A nice shiny and concave connection should remain. Repeat on any of the other leg(s) of the component.
With components with long legs (such as THT diodes), cut the excess of the leg off. Be sure to wear protective eye equipment so the lead doesn't land in your eyes.

[4.3] SMD Soldering (Optional)

SMD soldering is only needed should SMD diodes be chosen (instead of THT diodes).
The steps are shown off with a capacitor in the images, but the same applies with SMD diodes

1. Hold your iron to one of the pads and apply some solder to that pad, with your 0.5mm tin

2. Grab the component with your tweezers and bring it to the previously made solderblob on the pad (The capacitors, resistors and fuse do not have a specific orientation)
3. Heat up the solderblob and bring the component into it
    a. The component may not be laying flat on the board, fix this by heating up the solderblob again and pushing the component down with your tweezers



4. Now hold your soldering iron to the other pad of the PCB and component apply some soldertin

These are the basic steps to solder the passive components in place. Be sure to use your multimeter after every component, to know for sure there are no wrong shorts made.
Soldertin also has some flux inside (which is the golden/yellow substance you see on the board above). Be sure to clean it off after installing each component as well. It is much easier to clean at this step than it is at a later time.

[4.4] Microcontroller's headers / socket

*Note: do **not** install the microcontroller in this step, unless you're using a hotswap socket which allows the removal of the microcontroller. Two switches have to be installed under the microcontroller, which cannot be done once the microcontroller is in place.*

[4.4.1] Headers vs. hotswap socket

Most (if not all) Pro Micro or Elite-C microcontrollers will come with a set of headers, which allow the microcontroller to be soldered to the Ten Key Plus PCB. It's also possible to purchase a hot-swap socket for the microcontroller, so it can easily be removed.
There are various reasons to use a hot-swap socket, ultimately it makes it easy to replace the microcontroller, when upgrading (from a micro USB to a USB Type C microcontroller) or when replacing a defective microcontroller.

These types of hotswap sockets are optional, soldering in a microcontroller with the included headers is still the most affordable option.

[4.4.2] Soldering in the headers

To solder in the headers is much like soldering the THT diodes described in [4.2], except now with more pins.

Insert a header in the board and solder one of the pins. After that one pin is soldered in place, check if the header is placed flush with the board and sitting straight (not at any sort of angle).
If it's not flush/straight, hold your iron on the soldered pin and adjust the header as necessary.
Now solder in all the legs, one at a time, making sure to avoid adding too much solder.

Repeat these steps for the other header.

[4.4.3] (Optional) Hotswap socket install

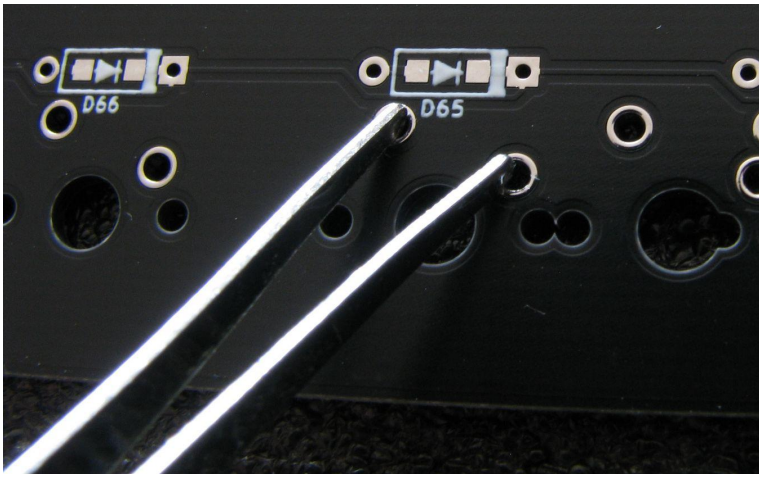
A hotswap socket allows the microcontrolller to be removed from the Ten Key Plus and later added back, without any soldering (after soldering it all in place first).
When the microcontroller is installed, the legs for the switches underneath are not accessible anymore.

Because a hotswap socket allows the microcontroller to be removed, this step can be done before the switch install step, while if the microcontroller is soldered in place the switches could not be reached.
There is a small chance the hotswap socket will hold the microcontroller in place if something goes wrong in assembly, because of that it is recommended to only follow these steps *after* section [4.6] which goes over the install of the plate and switches.

The hotswap socket install consists of two steps, the install of the sockets on the PCB and the install of the pins in the microcontroller.

The install of the hotswap sockets is the same as the headers described above.
For the install of the microcontroller, follow the steps provided by splitkb:
https://docs.splitkb.com/hc/en-us/articles/360011263059

To test the Ten Key Plus at this point, follow the steps in [5.1] to install the firmware, then follow the steps in [5.1.1], but instead of pressing the switches, short the switch legs with tweezers (or something else metal) like this:



[4.5] (Optional) Stabilizers

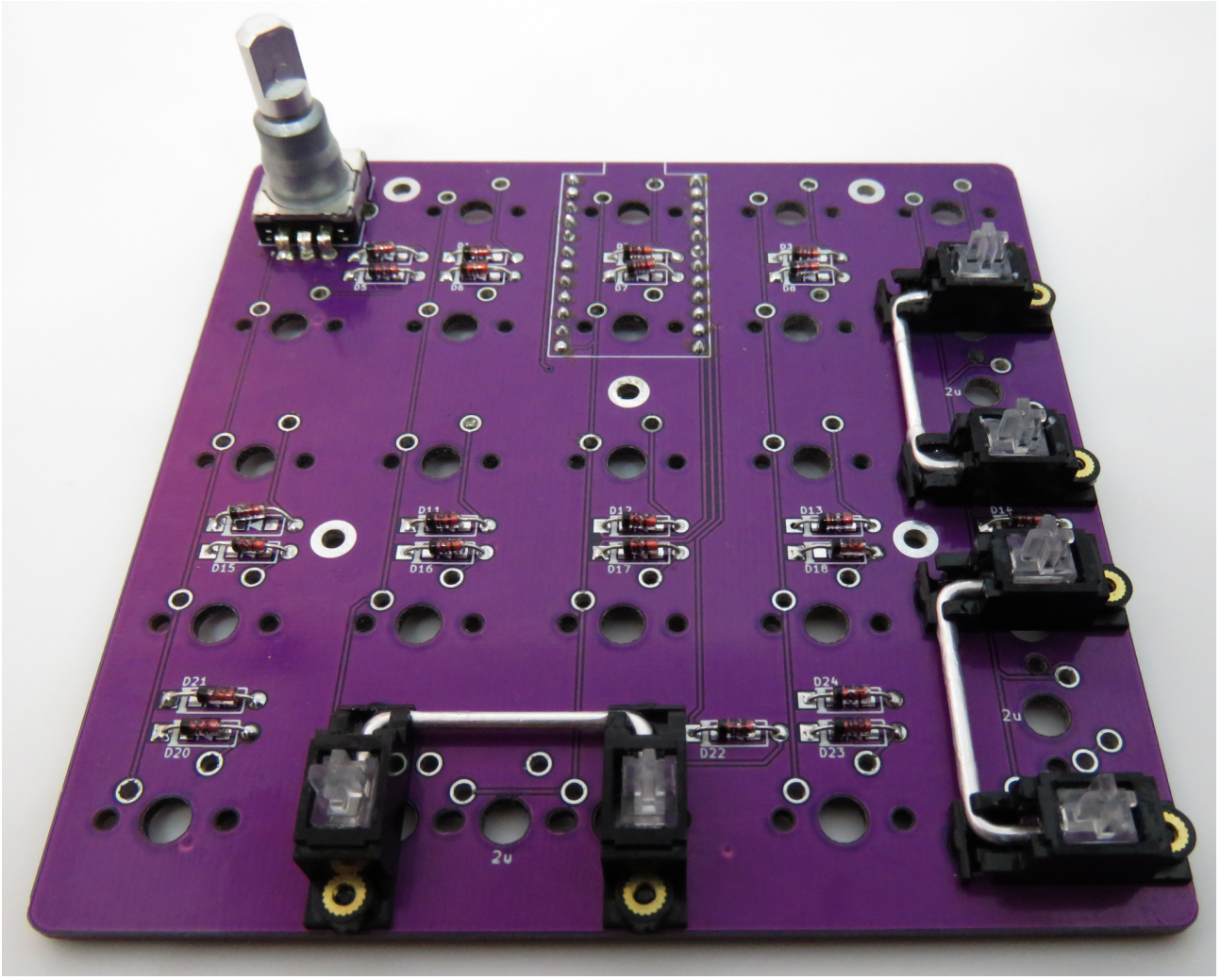
Depending on the layout chosen in section [2.1], these stabilizers may be optional. Stabilizers are only needed with keys of a 2u size, which are the larger '*0*', '*Enter*' and '+' keys.
The Ten Key Plus supports PCB screw-in / clip in stabilizers.

Mod the stabilizers by personal preference (i.e. clipping, lubing, etc.) and then install them on the board. First insert the "wire side" of the stabilizer in the large holes and then the clip in/screw in the smaller holes:

[4.6] (Optional) Rotary Encoder

The rotary encoder is an optional component, which can be installed in the top left corner of the board. It only inserts one way and the way it's soldered into place is using the THT method outlined in [4.2].

With the optional rotary encoder and stabilizers, the board now looks like this:



[4.7] Switches and plate

Grab the 3D printed plate and the amount of switches needed for the layout chosen, which would be somewhere between 22 and 25 (depending on the optional choice of stabilizers and a rotary encoder). Insert the switches into the plate, by gently pressing the switches in place. Do not put too much pressure on the plate, if the keyswitch doesn't snap in place, press gently with tweezers/a spudger on this part of the switch so it can more easily snap into place:

Keep in mind the switches in the right column are installed rotated 90° clockwise, regardless of if a 2u layout is chosen or not.
After all switches are installed in the plate, the whole assembly can be lowered on the PCB, ensuring all the switches are properly aligned into the PCB. Check if all the metal switch pins are sticking through the PCB, before proceeding.

Soldering these switches is the same as the THT diodes discussed in [4.2], just with more solder, because of the larger size of these legs.

To make sure the switches are installed totally flat on the board, some extra steps need to be taken. Solder one leg of one switch in a corner. After that one leg is soldered, squish the switch and PCB together and heat up that solder joint again, to make sure it's entirely flat on the PCB.

Repeat this on all the corner switches, or generally all switches that seem to be bowing out. After one leg is soldered on all switches, double-check if all switches are installed flat in place. If that is the case, solder the other legs of all these switches. If the originally soldered leg of the switches doesn't look that good, add some flux, heat up the joint (and optionally add some solder).

[4.8] Microcontroller install

The microcontroller is installed on the underside of the Ten Key Plus, on the headers installed in [4.4.2]. This section will focus on how to solder a microcontroller in place.
If a hotswap socket is used, follow the steps in [4.4.3].

Place the microcontroller on the board, with the components facing *up* (*away* from the Ten Key Plus). The steps to solder the microcontroller are the same as explained in section [4.2].
Start with a singular joint, in a corner. Ensure the microcontroller is installed flat on the plastic spacers, if it's not flat heat up that joint again and press the microcontroller down so it's flat. Once it's flat, solder the joint in the opposite corner. Once it's totally flush, solder all the other legs in place.

It is recommended to skip every other leg, so the microcontroller doesn't have too much heat at one place at one time. So go from leg 1 to 3 to 5, etc. then afterwards solder leg 2, 4, 6, etc.
Repeat this on the other side of the microcontroller until all legs are soldered in place.

After soldering the microcontroller, be sure to test that no shorts are present in the board. This can be tested using a multimeter in its continuity mode, preferably the beeper mode of continuity.
In continuity mode, when the two probes of the multimeter are touched together, they will cause the multimeter to make a noise.

Now you can touch the two probes on all the adjacent solderpoints on the microcontroller, to ensure none of them are *attached together* (i.e. making noise).
The only thing to keep in mind is that two pins are connected together and adjacent. Look at the bottom of the PCB (the side with the Ten Key Plus logo), on the left column of microcontroller pins. Counting from the top, pins 3 and 4 are connected together. All adjacent pins shouldn't be connected together.

At this point it's possible to move to section [5], which goes into detail about flashing the microcontroller, adjusting the firmware and testing the Ten Key Plus. The next section is about the preparation of the case, to install the Ten Key Plus in it.

[4.9] Case preparation

Now grab the 3D printed case, as it needs to be outfitted with some fasteners to install the board in it. There is a hole for the USB connection, one for the reset switch and two holes for the (optional) feet. Before using any of those, firstly it's time to install the heat insert fasteners.

These heat inserts are installed in the five smaller circular holes in the board. Specifications of the heat inserts needed:

- M2 screw type
- 3mm diameter
- 3mm height

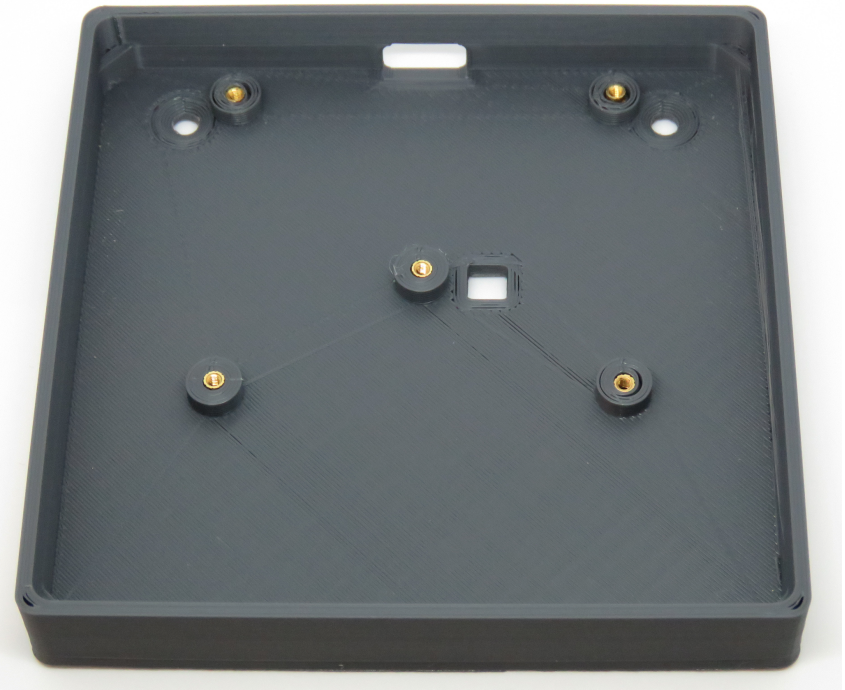
They are installed using a soldering iron, at a much lower temperature than what is used to solder using tin. If the temperature on your soldering iron cannot be adjusted, it won't work for this step. A simple heat insert iron could be an alternative.
Other than the iron and heat insert, another tool is needed to make sure the inserts are installed flat. A device with a flat metal end - such as a hammer - is needed too.

First set your iron's temperature to the regular soldering temperature (usually around 350°C/660°F), so the iron can be cleaned. Apply some fresh solder to the tip and then clean it using your steel wool. Ensure a thin layer of solder is on the tip, as this will help in the transfer of heat.

Now change the temperature to 170°C/340°F and let the iron cool down to that temperature. Working at this temperature gives maximum control when installing the heat insert, so the process doesn't go too fast.

Once the iron is at temperature, place a heat insert on top of the hole, with the narrow side of the insert down. Most inserts will have a narrower lip on the underside, that will make it easy to stand the insert in the hole.
Now place the iron into the insert and keep it perpendicular to the case (*note: the iron is very hot, so is the heat insert now, don't touch either with your hand*). The heat of the iron and the gentle pressure applied by pressing the iron down will allow the insert to move into the hole in the case.

Once the insert is 90% in the hole, remove the iron and use the hammer to gently press on the insert, to push it down the last 10%.
Now inspect the insert is properly installed and repeat the steps above for the other 4 inserts.



[4.10] Assembling the Ten Key Plus into the case

If you want to use the optional feet, now is the time to install them. Insert the screws into the holes and screw the feet onto them. These feet have a rubberized end, to keep the Ten Key Plus in place. Place some additional rubberized/soft feet on the other side of the bottom and if you aren't using those optional, place some rubberized/soft feet there too.

To install the PCB/plate/switch assembly in the case, first install the standoffs into the inserts. Make sure these standoffs are well in place, by either using the right size driver for them, or using some pliers to tighten them. Don't apply too much pressure, as otherwise the inserts may suffer damage.

Now the PCB/plate/switch assembly can be placed on top of those standoffs and secured using some screws.
Afterwards install your keycaps of choice (and optionally a rotary encoder knob) and the assembly of the Ten Key Plus is complete!

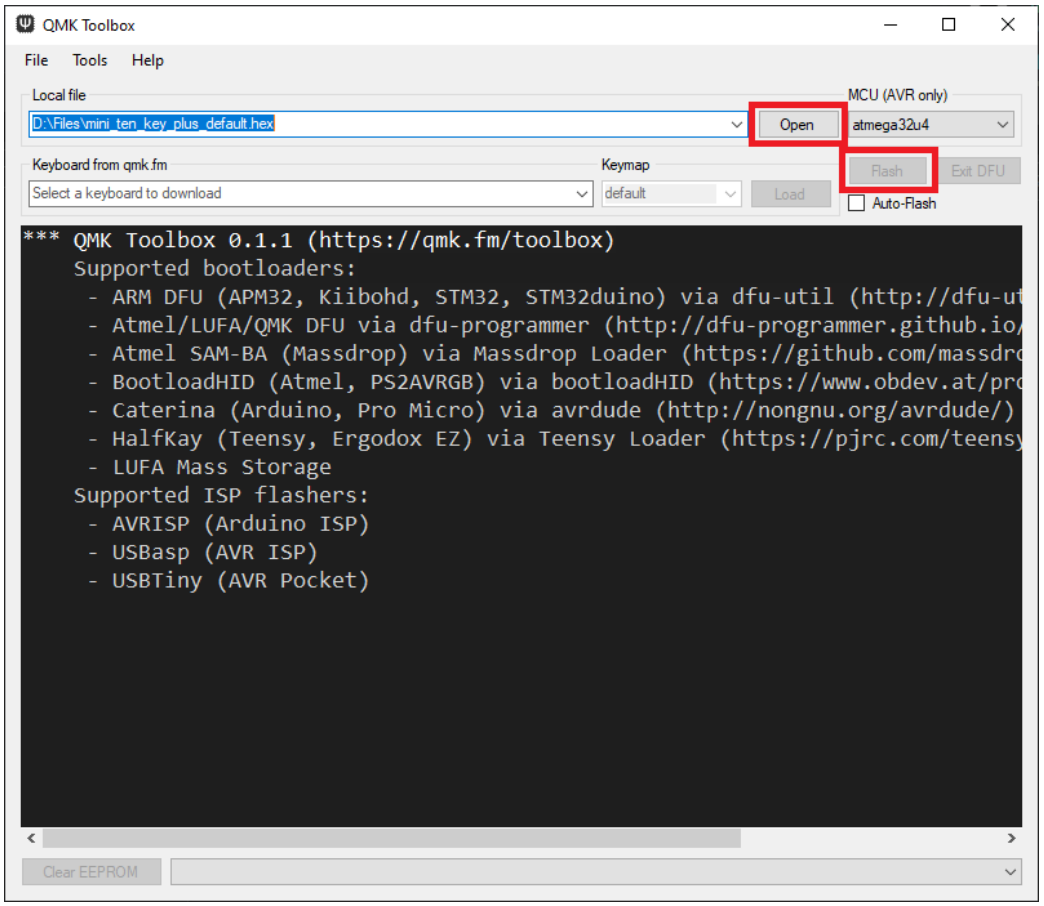
[5] Firmware, testing and tweaking

Once the Ten Key Plus is physically put together, it's time to install the firmware on it, test it out and optionally tweak the firmware to fit your need(s).

[5.1] (Simple) flash the supplied firmware

To install the firmware on the microcontroller, a program called QMK Toolbox is needed, which can be download from this location: https://github.com/qmk/qmk_toolbox, a .hex-file is also needed to install on the board.
A .hex-file is included in the Github repository, in the 'Firmware'-folder. Info on how to tweak the firmware is explained in [5.2].

After downloading and installing QMK Toolbox, start the program, press the 'Open'-button and locate the .hex-file you downloaded previously.
During the first-time setup, you can now press 'Flash' right away, but when the QMK firmware is already installed, you need to first press the 'Reset'-button on the Ten Key Plus to make the 'Flash'-button available for use.



It will play the connect/disconnect sound once and will install as the ‘mini Ten Key Plus’, which is when it’s ready for use!

[5.1.1] Testing the Ten Key Plus

The easiest way to check if everything is working, is to go to the QMK keyboard tester (https://config.qmk.fm/#/test) and pressing all keys one after another. All keys should give an output, if one - or multiple - don’t, something may have gone wrong.

It’s important to have double-checked all diodes were installed in the right orientation, as otherwise any key with the wrong orientated diode wouldn’t work.

[5.2] (Intermediate) Changing the firmware through the QMK configurator

Should the included key bindings not be to your liking, it’s easily possible to adjust them using the QMK configurator.
Head over to the QMK configurator page for the Ten Key Plus:
https://config.qmk.fm/#/mini_ten_key_plus/LAYOUT_all

Here it is possible to choose one of two layouts; with or without the large keys. Change the keys according to your preference and once you’re happy with the changes, press the ‘Compile’ button in the top right and wait for it to compile the firmware. Now press the ‘Firmware’ button in the bottom right to grab the .hex file and flash it on the microcontroller using the steps described in [5.1].

You can optionally also press the button for “Full source”, which allows you to work on your configuration at a later date, should you want to tweak your layout any further.

[5.3] (Expert) Editing the keymap manually

Should the online QMK configurator not have the right settings for your purpose (such as being able to edit the rotary encoder) or if you want to share your keymaps to the greater QMK community, you can edit and compile the keymaps locally.
The first steps include downloading the repository locally (using your Git client of choice, like Github) and setting up the different dependencies QMK has.

These steps are described as such here: https://beta.docs.qmk.fm/tutorial/newbs
Where the default keyboard can be set to “mini_ten_key_plus” of course and the name to your Github name.

There are a few things to keep in mind though, if you plan on publishing your keymaps to the main project though. If you are familiar with the Git workflow, these will seem obvious, but it doesn’t hurt to mention it again.
After cloning the repository locally you can work in your own branch. After you are happy with your keymap and want to include it in QMK, you can publish your branch (which will make a fork of the project) and then make a pull request, to be reviewed by QMK members to include your keymap in the full project.
Please read this page thoroughly before making a new pull request, to be sure your changes can be accepted in the QMK repository: https://beta.docs.qmk.fm/developing-qmk/pr_checklist

If you’re unsure about what to do, do not hesitate to contact someone about your questions. It’s possible to contact the QMK community via Discord, Reddit or Github:
https://beta.docs.qmk.fm/tutorial/support
Should you run into an issue with the keymap you think should be solved by me, please open an issue on the Github page: https://github.com/minibois/ten-key-plus/issues

[5.4] Enjoy your Ten Key Plus!

Now that you’ve assembled, tested and perfected your own Ten Key Plus, it’s time to use and enjoy your new keypad, be sure to show off your work where-ever you prefer!

I showed off the project on the LinusTechTips forum:
https://linustechtips.com/topic/1400294-mini-ten-key-plus-numpad-build-log-chip-shortage-effect-on-hobbyists-and-3d-printing-for-prototypes/

Should there be any questions, concerns of corrections, do not hesitate to open an issue on the Github page for this project: https://github.com/minibois/ten-key-plus/issues

[6] Sources

Ten Key Plus source: https://github.com/minibois/ten-key-plus
rev.1a source: https://github.com/minibois/ten-key-plus/releases/tag/rev.1a

Assembly guide license: CC BY-SA 4.0, See [9]
PCB and schematic license: https://ohwr.org/cern_ohl_s_v2.pdf

ai03's PCB design guide, which this keypad and previous creations were based on:
https://wiki.ai03.com/books/pcb-design/page/pcb-guide-part-1---preparations

Libraries adapted in the creation of the PCB/schematic:
That-Canadian's KiCad_labs (for the D-SOD123_axial-dual):
https://github.com/That-Canadian/KiCad_Libs
MX_Alps_Hybrid: https://github.com/ai03-2725/MX_Alps_Hybrid
random-keyboard-parts (for a reset switch): https://github.com/ai03-2725/random-keyboard-parts.pretty

[7] Differences between revisions

After the first version of the Ten Key Plus - rev.1a - small changes were made after assembly to reflect better compatibility between the PCB and its plate/case.
The changes made between rev.1a and rev.1 were:

- Four of the 1u keyswitches on the right side were rotated 90° clockwise, so they can be installed using the same plate as for the 2u keys
- Increased size of the Github-link text

- Moved microcontroller forward a couple mm
- Moved three of the screwholes, so they're at a 4-way between keys

[8] FUT: Frequently Used Terms

Microcontroller

MCU: Microcontroller Unit. See it as a microprocessor, with a few more components (such as memory) to make it a full computer. In this project it will refer to the Arduino Pro Micro (or compatible, such as the Elite C)
Arduino: The company behind the ATmega MCU powered microcontroller boards, such as the Arduino UNO.
Firmware: A program on the MCU that allows it to function as a device (QMK in this project).
Flashing the firmware]: The process of installing the firmware (QMK) to the MCU.

Mechanical keyboard components/concepts

Be sure to check out Eschew's introduction to mechanical keyboards, for a lot more in-depth keyboard information: https://linustechtips.com/topic/1214368-an-introduction-to-custom-mechanical-keyboards/
Check out the mini_cardboard build log, for some more detailed information on some of these concepts (particularly a matrix, its diodes and the MCU):
https://linustechtips.com/topic/1328547-mini_cardboard-a-4-keyboard-build-log-and-how-keyboards-work/
mini eLiXiVy build log/info about open source licenses:
https://linustechtips.com/topic/1366493-elixivy-a-65-mechanical-keyboard-build-log-pcb-anatomy-and-how-i-open-sourced-this-project/

Keyswitch: a component that can be pressed down to connect two pieces of metal together, which in turn can be used to detect a keypress
Cherry MX: The largest/most well-known manufacturer within mechanical keyswitches.
Cherry MX-like: Keyswitches with a design based on Cherry MX switches, because of Cherry's expired patent on keyswitches. Includes companies like Kailh, Gateron and much more. PCBs and keycaps typically advertise compatibility with "MX-Like" switches, so indicate it works with all of those.
QMK: One of the most well-known open-source keyboard firmwares.
Plate: Holds the keyswitches in place together with the PCB. Either the PCB or case is mounted to the case. While off-the-shelf options are available in aluminium, brass or polycarbonate, this project uses a 3D printed plate. This is an optional part, but a recommended part.
Stabilizer (stabs): Makes sure the larger keys (2u and larger) are properly supported when pressed down. PCB mount stabs are supported.
Keycap: The (usually) plastic covers that sit on the keyswitches. Popular keycap materials include - but are not limited to - ABS and PBT. Refer to Eschew's introduction to mechanical keyboards for much more info. A keyboard needs different sizes of keycaps, depending on the layout chosen.
Layout: The different sizes of the keycaps decide the layout of the keyboard. The layouts supported by the Ten Key Plus can be found in [2.1]
Rotary Encoder: The Ten Key Plus supports a rotary encoder, which is a knob that can be turned

around, much like a volume wheel. Popular choices on how to use the rotary encoder include - but are not limited to - volume up/down, page up/down, (Shift+)Alt+Tab and more.

Electronics

PCB: Printed Circuit Board. A board made from fiberglass, with its purpose in a keyboard being to hold the components in place and connecting them together with traces (wires on a PCB)
Ground: The 'return' part of electricity, such as Anti-static discharges.
Diodes: An electrical component that allows power to flow one way, but not another. Used in the keyboard matrix.
THT: Through Hole Technology components have long legs that stick through a PCB and get soldered to a plate through hole to the board. These components include the switches, header pins and diodes.
SMT: Surface Mount Technology uses pads on the board to solder SMD (Surface Mount Device) components in place. SMD components can be smaller than THT components and because they don't stick through the board, it's possible to design a PCB with SMD components on both sides.

Soldering

Soldering: The process of joining two components together (like the PCB and component) with soldering tin
Soldering iron: A tool to heat up the two components, to make the soldering tin flow between them
Soldering station: A device that includes a soldering iron, but also has a base station which allows you to change the temperature of the iron.
Solder(tin): a metal (combination) used to melt between the components you want to solder together. Typically consists of Sn (tin) and Pb (lead) in a 60/40 ratio. Lead free options typically include - other than tin - metals like Ag (silver), Cu (copper), etc.
Lead-free/leaded solder: Leaded solder is easier to work with, because of its lower melting temperature, but additional safety precautions need to be taken and it may be more difficult to get.
Flux: A substance used to make soldertin flow better to the pads and flow away from the rest of the PCB.
Desoldering braid/wick: A braid from copper strands with flux in it, used to wick away soldertin. Used in drag-soldering too.
Drag soldering: A popular method to install a chip with a lot of pins, like the MCU. Place a lot of tin over the legs/pads and wick it away. More info in "The Build IV: The First* Board"
IPA: Isopropyl alcohol, a cleaning agent to clean off the flux from the board.
Tacking down: (Relevant to SMD components) soldering one pad of a component to the board, so it's held in place when soldering in the other pad(s).

Software/Firmware/QMK

QMK: One of the most well-known open-source keyboard firmwares.
Git: A version control system, to make it easy to work together on a project, or just keep track of different versions in general
Github issue: Github is the chosen Git version control platform of this project.
Keymap: An inherent part of the firmware that decides what key outputs what and what the rotary encoder does. This is generated by QMK and can be adjusted to fit everyone's personal needs.

[9] License

[9.1] Changes

Please note dates in the ISO 8601 (YYYY-MM-DD) format. You may include your website/a way to contact you if you wish.

Revision: mini_ten_key_plus_AssemblyGuide_v1a
Date: 2021-12-17
Change(s): Assembly guide created.
Author: minibois
Contact: https://github.com/minibois/