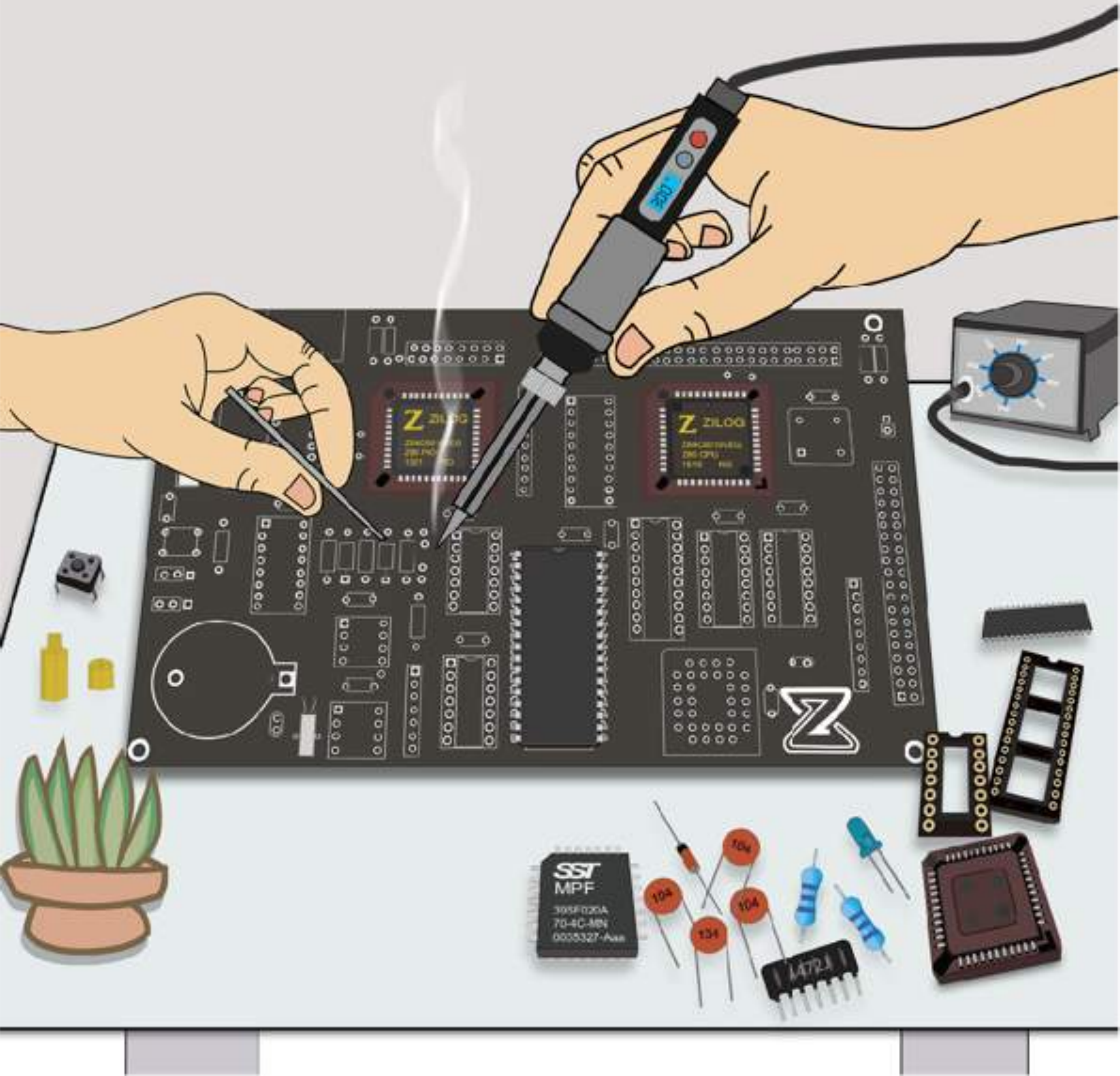
# Soldering Guide for Zeal 8-bit Computer



# Assembling your own Zeal 8-bit computer!

This guide will walk you through the process of soldering the components provided in your "lunchbox" to create a fully functional computer system.

By following this guide from start to end, you'll not only assemble a functional Zeal 8-bit computer but also gain valuable skills and knowledge applicable to your future electronic projects!

**UPDATE 01 Aug 2024**
**The layout of the motherboard v1.3.1 is slightly different from other versions (one more diode, more 2pin jumpers in v1.3.1), but the soldering principles are the same, so you can still refer to this soldering guide.**

# Tools Required

Before starting, make sure you have the following tools:

- ✓ - A multimeter with continuity mode
- ✓ - A soldering iron or soldering station, preferably with adjustable temperature controller
- ✓ - Solder, ideally Sn63/Pb37 or Sn60/Pb40
- ✓ - A small wire cutters, with safety glasses
- ✓ - A heat-resistant work surface, for example, a silicone soldering mat

Nice to have:

- Some solder wick or a desoldering pump, in case any mistake is made and you need to remove a component
- A PCB holder
- Some flux

Here is the list of electronics components you will receive

# Part List 1

| Name | Designator | Qty | Category | Package | Description |
|---|---|---|---|---|---|
| Z84C0010 | U2* | 1 | IC | PLCC44 | Zilog Z80 CPU |
| Z84C2010 | U5* | 1 | IC | PLCC44 | Zilog Z80 PIO |
| SST39SF020 | U4* | 1 | IC | PLCC32 | 256KB NOR Flash |
| AS6C4008 | U3* | 1 | IC | DIP-32 | 512KB static RAM |
| (AT)F16V8/ GAL16V8 | U6* | 1 | IC | DIP-20 | Programmable Logic Device, used as the logic glue |
| 74HC541 | U10* | 1 | IC | DIP-20 | 8-bit buffer |
| 74HC670 | U8, U9* | 2 | IC | DIP-16 | 16-bit static RAM |
| 74HC595 | U11, U12* | 2 | IC | DIP-16 | 8-bit shift register |
| AT24C512/24LC 256 | U14* | 1 | IC | DIP-8 | I2C EEPROM |
| DS1307 | U13* | 1 | IC | DIP-8 | I2C Real-Time Clock |
| 74LS07 | U15* | 1 | IC | DIP-14 | 8-bit open-drain buffer |
| 74HC14 | U7* | 1 | IC | DIP-14 | Hex inverter |
| MCP130-450 DI/ TO | U1 | 1 | IC | TO-92 | Reset supervisor |
| PLCC44 socket | U2, U5 | 2 | Socket | / | |
| PLCC32 socket | U4 | 1 | Socket | / | |
| DIP-32 socket | U3 | 1 | Socket | DIP-32 | |
| DIP-20 socket | U6, U10 | 2 | Socket | DIP-20 | |
| DIP-16 socket | U8, U9, U11, U12 | 4 | Socket | DIP-16 | |
| DIP-14 socket | U7, U15 | 2 | Socket | DIP-14 | |
| DIP-8 socket | U13, U14 | 2 | Socket | DIP-8 | |
| Fuse holder | F1 | 1 | Socket | / | SMD |
| Battery holder | BT1 | 1 | Socket | / | CR2025/2032 battery socket |

Here is the list of electronics components you will receive

# Part List 2

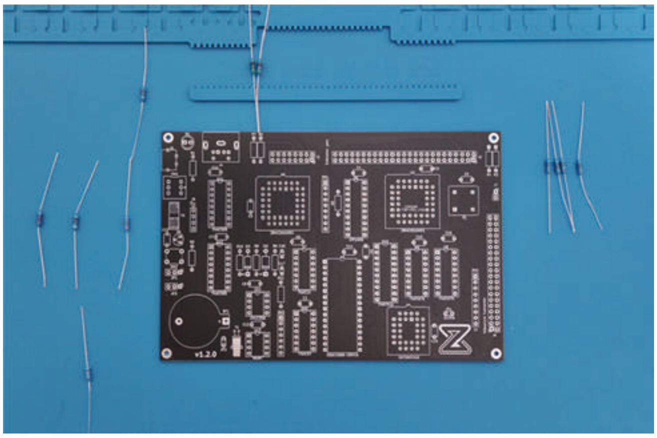
| Name | Designator | Qty | Category | Package | Description |
|---|---|---|---|---|---|
| USB-C 2-pin | J1 | 1 | Connector | / | Power |
| 2x25 pin socket (90 deg) | J2 | 1 | Connector | / | Extension port |
| 2x8 pin socket (90 deg) | J3 | 1 | Connector | / | User port |
| PS/2 port | J4 | 1 | Connector | Mini-DIN | PS/2 keyboard port |
| 5-pin pin header | J5 | 1 | Connector | XH2.54 | Internal I2C connector |
| 6-pin switch | SW1 | 1 | Switch | / | Power switch |
| 4-pin switch | SW2 | 1 | Switch | / | Reset switch |
| 2-pin jumper | JP1, JP2 | 2 | Jumper | / | Optional features |
| 3-pin jumper (w/ cap) | JP3, JP4 | 2 | Jumper | / | Video board interrupts selector |
| 10MHz active oscillator | X1 | 1 | Oscillator | DIP-4 | CPU clock |
| 32.768KHz crystal oscillator | Y1 | 1 | Oscillator | / | RTC oscillator, capacitance of 12.5pF |
| 1 kΩ resistor | R1, R3, R4, R5 | 4 | Resistor | / | |
| 512 Ω resistor | R6, R7 | 2 | Resistor | / | |
| 2 kΩ resistor | R8 | 1 | Resistor | / | |
| 470 Ω resistor | R9 | 1 | Resistor | / | |
| 20 kΩ resistor | R10 | 1 | Resistor | / | |
| 10 kΩ resistor | R11 | 1 | Resistor | / | |
| 1 kΩ resistor network | RN1, RN2 | 2 | Resistor Network | SIP-9 | |
| 4.7 kΩ resistor network | RN3 | 1 | Resistor Network | SIP-7 | |
| 1N4148 Diode | D2, D3 | 2 | Diode | / | |
| 5mm LED | D1 | 1 | Diode | / | |
| 100nF capacitor | C1, C2, C3, C4, C5 , C6, C7, C8, C9, C10, C11, C12, C13, C14, C15, C16 | 16 | Capacitor | / | Bypass capacitor |
| 100nF capacitor (rounded) | C17, C17 | 2 | Capacitor | / | Filters for PS/2 protocol |
| M2 13+3mm screw | / | 4 | screw | / | / |
| M2 4mm cap | / | 4 | cap | / | / |

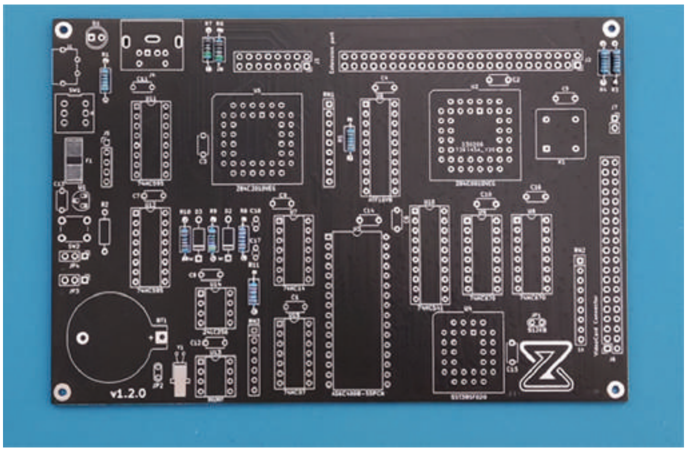
# 01 resistors

The first step is to install the resistors. You can use the 5 bands marked on them to identify their values. If you cannot read the bands, or if you lost the annotation they came with, use a multimeter in resistor mode to get their values.

On motherboards v1.2.0 and below, resistor R2 must be left unpopulated, hence it's absence from the table.

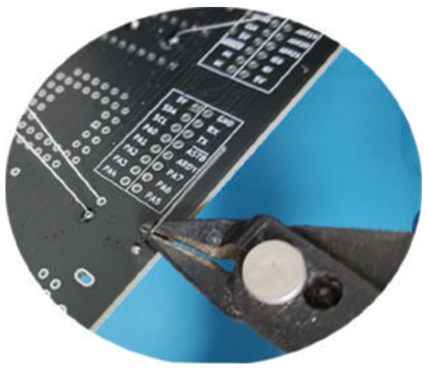
Bend the resistors' legs and put them in their spot. After placing them, return the board and solder them all.



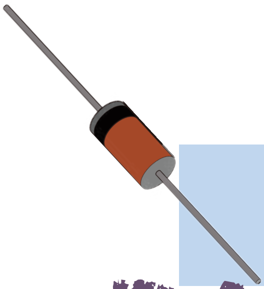


### Resistor Value and Place

| | |
|---|---|
| 1K | R3,R4,R5,R1 |
| 2K | R8 |
| 10K | R11 |
| 20K | R10 |
| 470 | R9 |
| 510 | R6,R7 |



"After soldering the resistors, put on safety glasses, use wire cutters to trim their legs. Cut the legs close to the soldering point without leaving any extra bits that could scratch your hand when touching the board's underside.

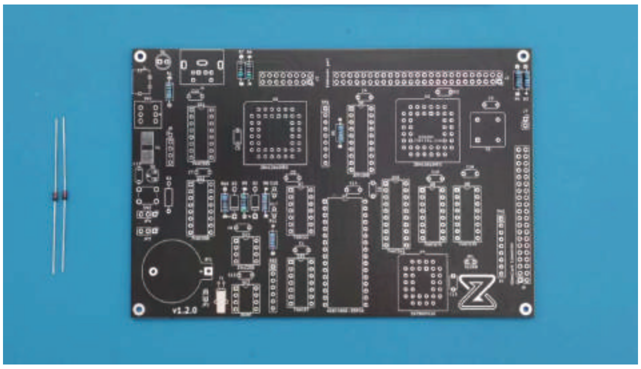


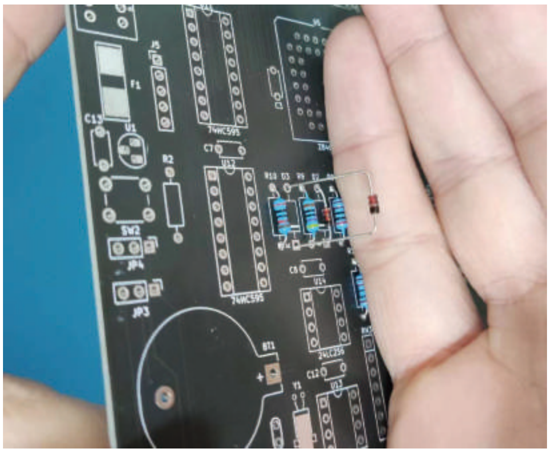
# 02 Diodes

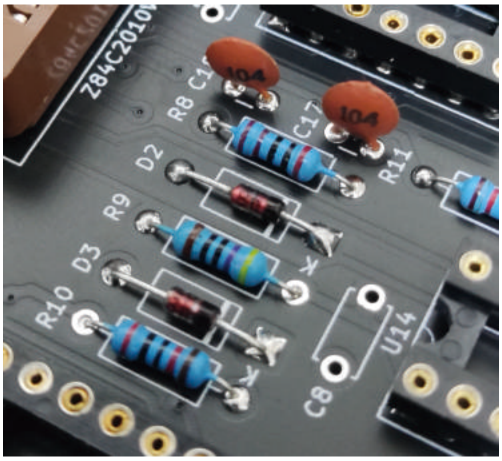
Now come the 1N4148 diodes, the process is similar, start by bending the legs and placing them in their sport. Be careful, the diodes must be placed in the right direction!

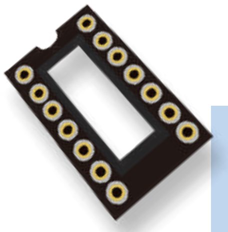
The K on the motherboard corresponds to the black part of the diode:

UPDATE 01 Aug 2024
There is one more diode in PCB v1.3.1









# 03 DIP Sockets

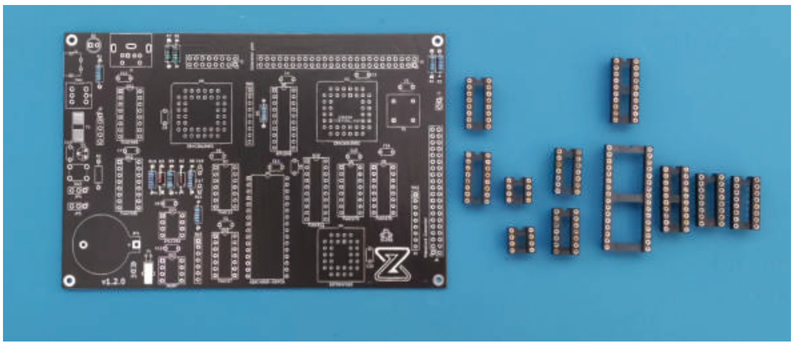
① Place the socket of each DIP integrate circuit in their respective spot. Make sure that the "U" shape at the top of each socket matches with the U shape present on the PCB silkscreen

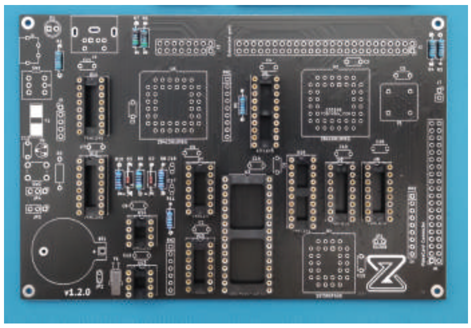
② Flip the board[1]

③ Solder the top left and bottom right pads of each socket

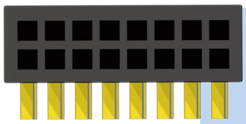
④ Flip the board back to make sure that all the sockets are in the right direction and that none of them is poorly inserted. If any is not inserted properly, press on it with one hand while passing the soldering iron on its two soldered pins with the other hand

⑤ After checking that all the sockets are placed correctly, proceed to soldering the rest of the pins.





Tip: Using kapton tape, blu-tak or a hard surface can help you keeping the sockets in place before flipping the board. If you don't manage to get all the sockets in place when flipping the board, it is okay, solder them one by one!



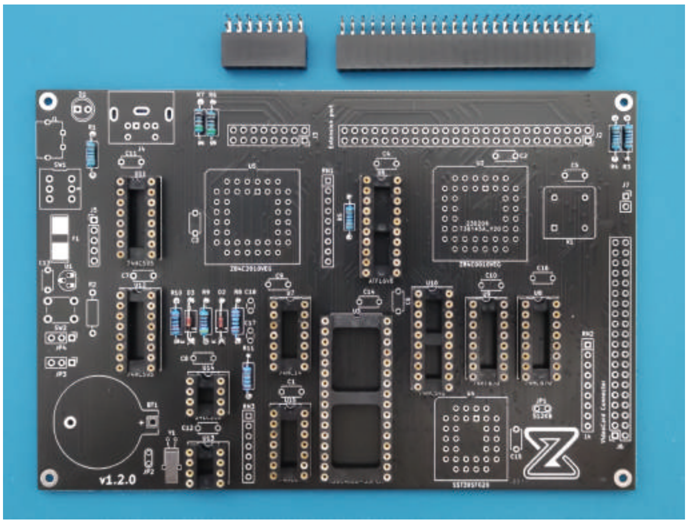
# 04

# 2.54mm Connectors

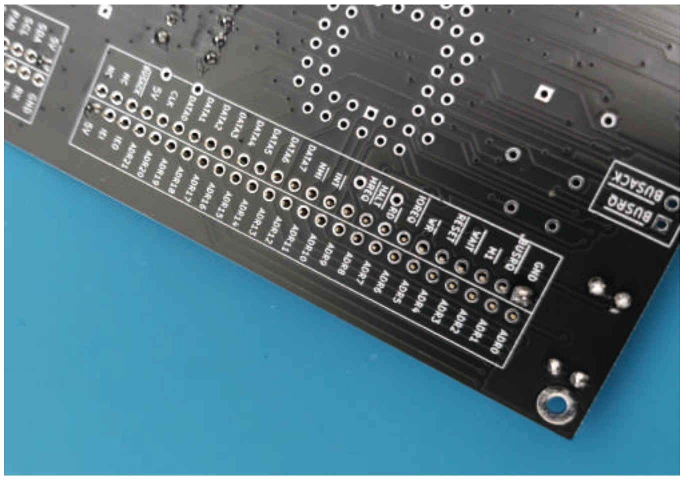
① Place the extension port, which is a standard 2.54mm 2x25 pins connector, and the user port, which is a standard 2.54mm 2x08 pins connector, on the PCB

②Flip the board. Since there are only two connectors on the same side, it should be easier than previously.

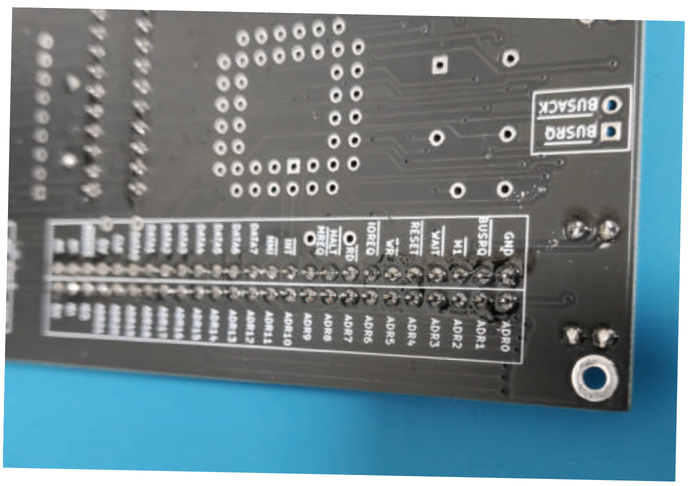
③Similarly to the previous step, solder the top left and bottom right pins just to make sure everything is correct. Afterwards, proceed to soldering the rest.



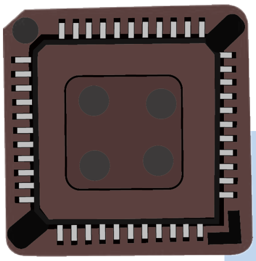

the components




solder the diagonal pins to stabilizer the boards



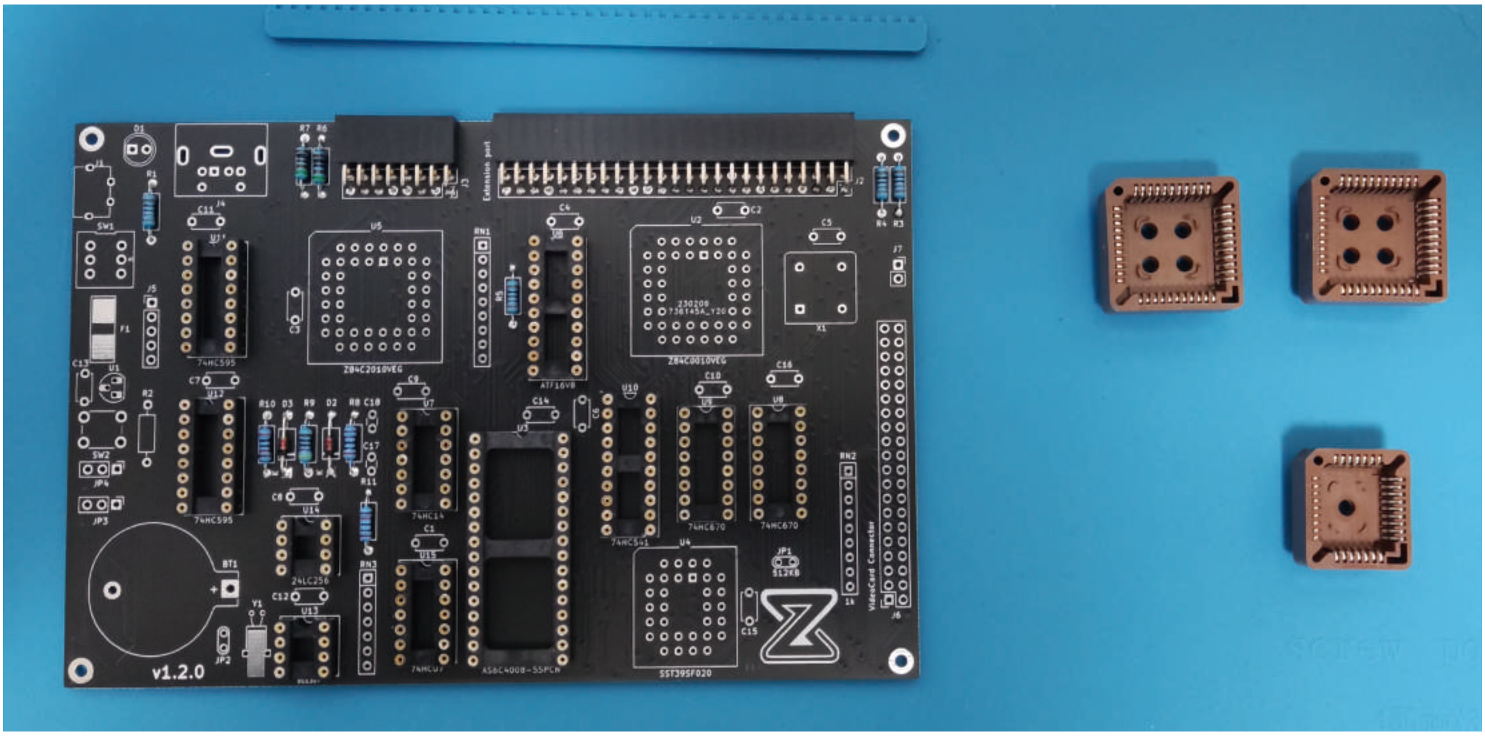

All soldered !



# 05 PLCC Sockets

①Place the PLCC sockets in their respective spots. There should be two PLCC44 sockets, the right one on the PCB being the Z80 CPU and the left one being the Z80 PIO, and one PLCC32 socket, which is for the NOR Flash.

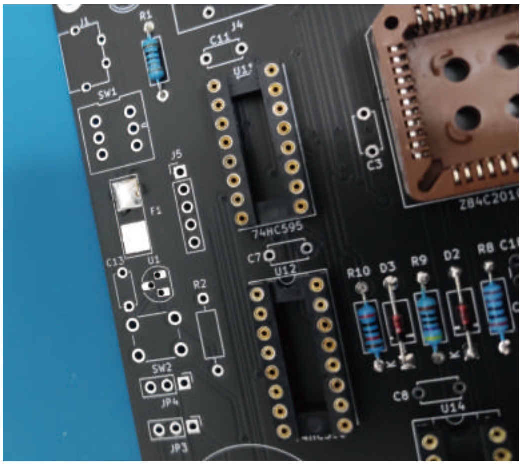
②Proceed to the same steps as before: flip the board, solder few pins, perform a visual inspection and proceed to the rest.

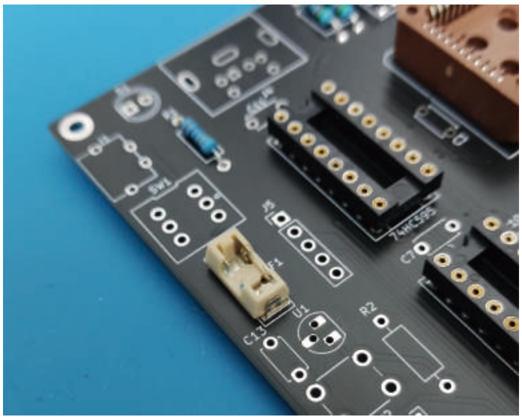


# 06
# Fuse holder

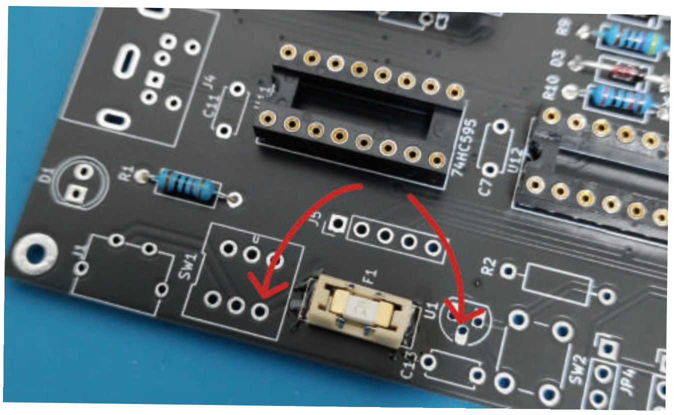
This part may be the trickiest since the fuse holder is an SMD component.

① Put some solder on the top pad
② Heat up the solder on that pad so that it is liquid, while placing in the fuse
③ Once placed properly, remove the soldering iron, the holder should be soldered on that pad
④ Heat up the other part of the holder (be careful to no melt the plastic) while putting some solder underneath





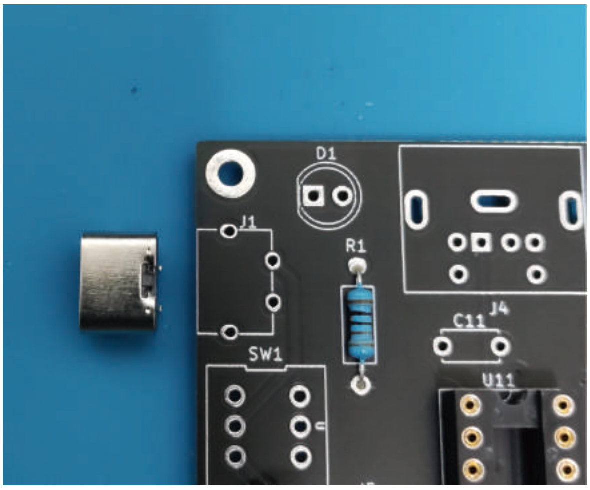
① Once soldered, place the 1A fuse inside
② Put your multimeter in continuity mode and make sure that the two pads shown in the following picture are connected:

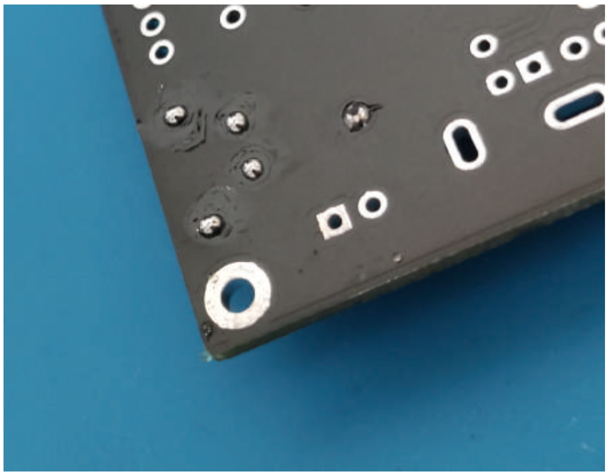


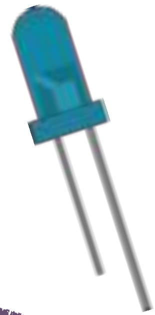
# 07

# USB-C connector

Now comes the USB-C connector, it shall be placed in J1, at the top left of the PCB. ITs legs are not very long, so when soldering, you should let the solder enter inside the holes. The resulting joints should look like the following:



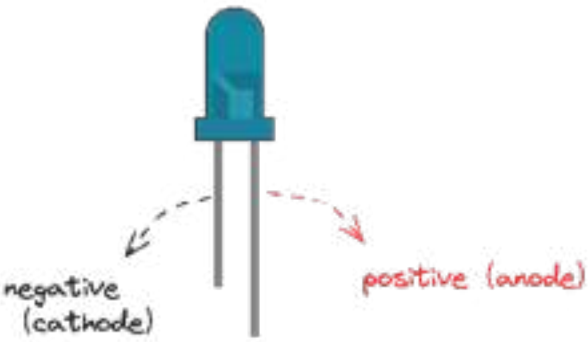


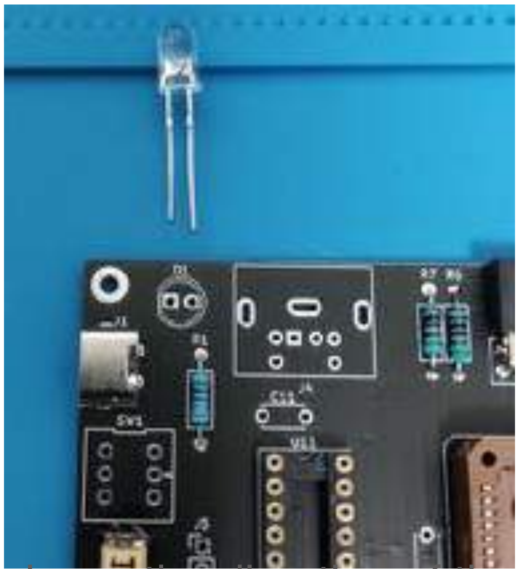
# 08&09

# LED and PS/2 Connector

The LED is located just next to the USB-C connector. Just like the other diodes, the direction is important: the long leg is the positive side (anode) and the short leg is the negative side (cathode).

On the PCB, the silkscreen is rounded on the positive side and straight on the negative side.



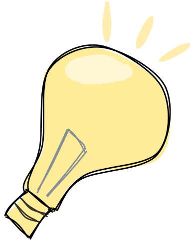
The picture shows the direction of the LED.

*Note: this picture only shows where the connector sits, not its direction*

The PS/2 connector, referenced J4, must be soldered just by the LED.

It can be tricky to make it enter its footprint if the pins are not straight, if that's your case, bend the pins a bit and try again.

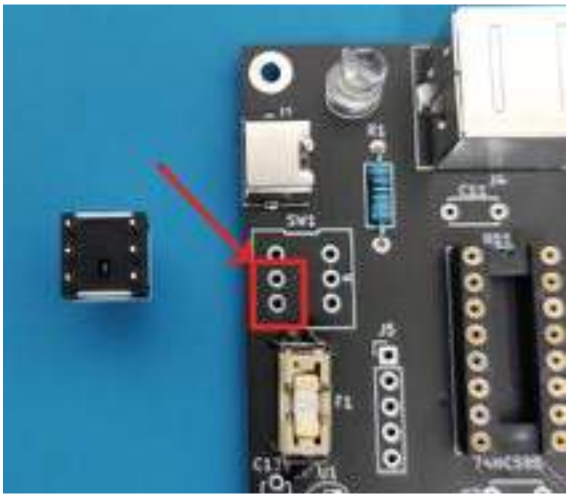
# 10



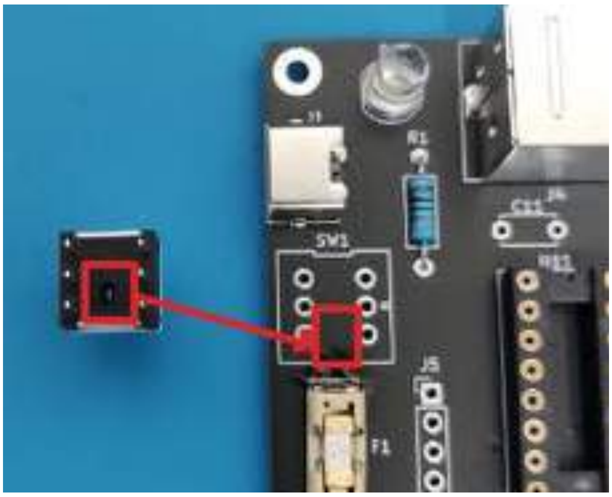
# Power switch and first test!

Despite being the tallest component, the power switch is the next one, the idea is to have a way to test if the board is already working.

The power switch will make contact between the following pads on the PCB:

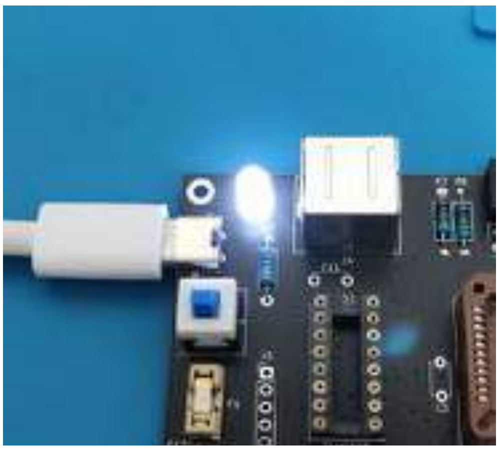


The right position for is as shown in the picture below, the small square at the bottom should be close to the fuse (do not follow the silkscreen on the PCB)



## Test Now!

Once the power switch is in place and soldered, it's time for the first test!



① Make sure there is no short by putting the multimeter in continuity mode and testing **+5V** and **GND** pins. The first one corresponds to bottom pad of the fuse, while the latter is located on the USB-C connector shield. Of course, they which must **NOT** be shorted.

② If the previous step was a success, plug a USB-C cable and a power supply (a USB phone power supply preferably)

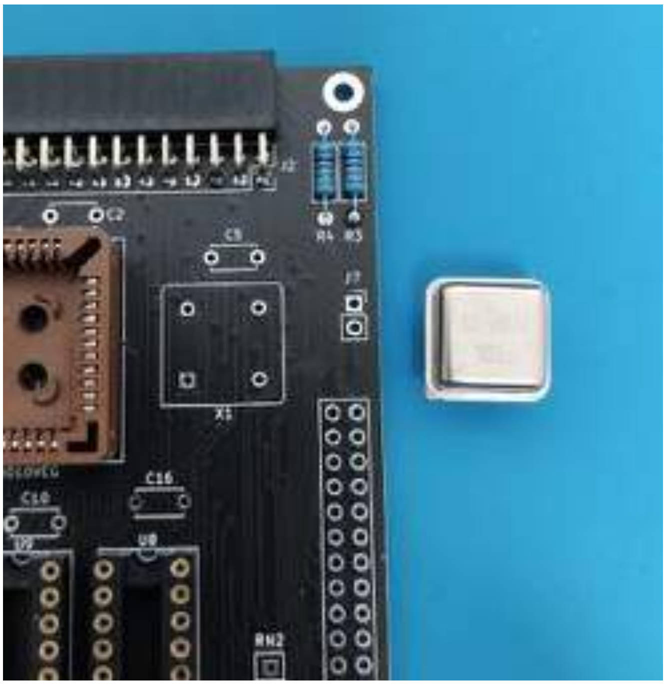
③ Press the power switch, the LED should light up!

④ Power off the board and unplug the USB cable

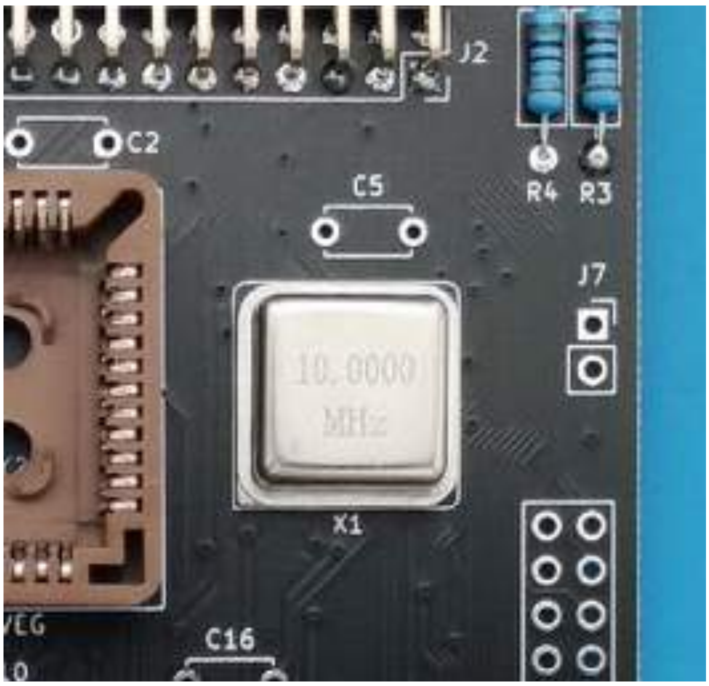

# 11

# Active oscillator

The 10MHz active oscillator sits in the footprint marked X1, on the right side of the board.



The squared side of it should be located at the bottom left, just like shown on the PCB. It should look like this:

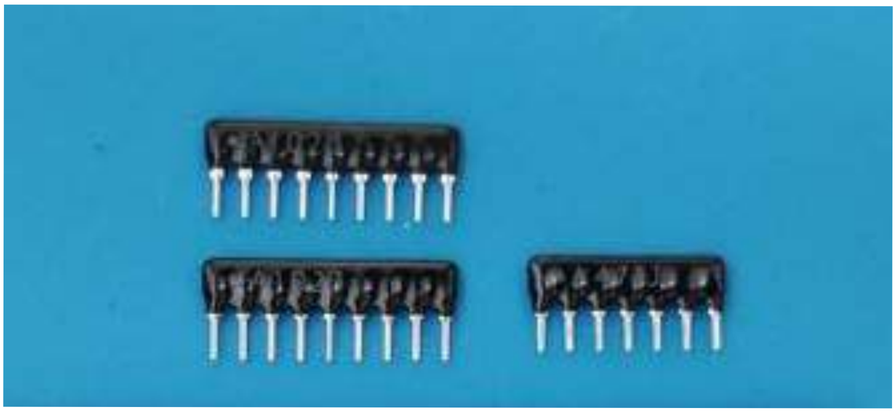


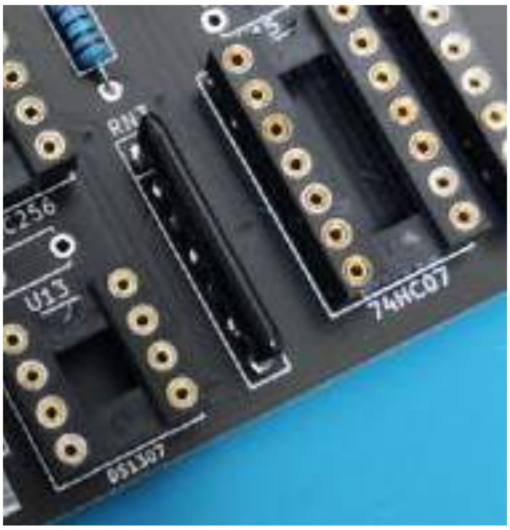


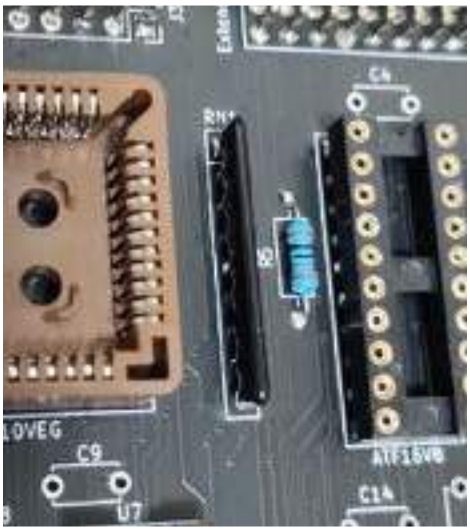
# 12 Resistors Network

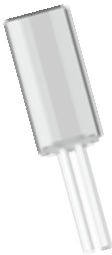
There are three resistor networks on the board: two 10kΩ one referenced RN1 and RN2, and a third one of 4.7kΩ referenced RN3.

The small dot present on their marking shows their direction: this dot shall be in the square pin on the PCB silkscreen.



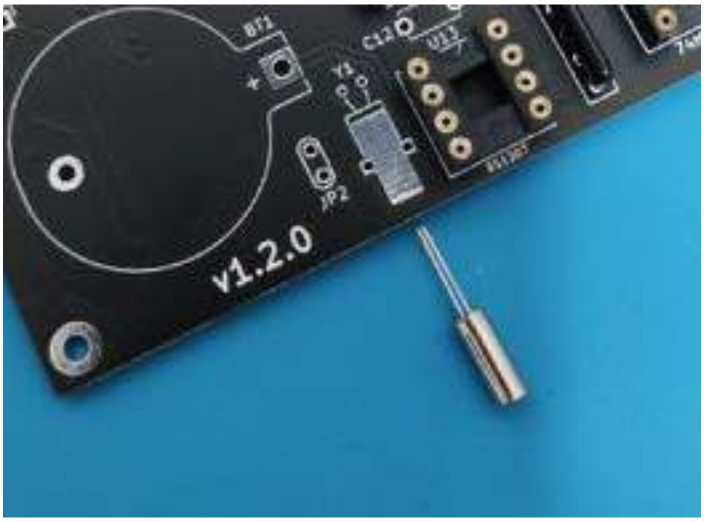


# 13 Crystal oscillator

The crystal oscillator has a lower frequency (~32KHz) than the previous active oscillator. Its role is to provide a clock signal to the Real-Time Clock, to let it count the elapsed time.



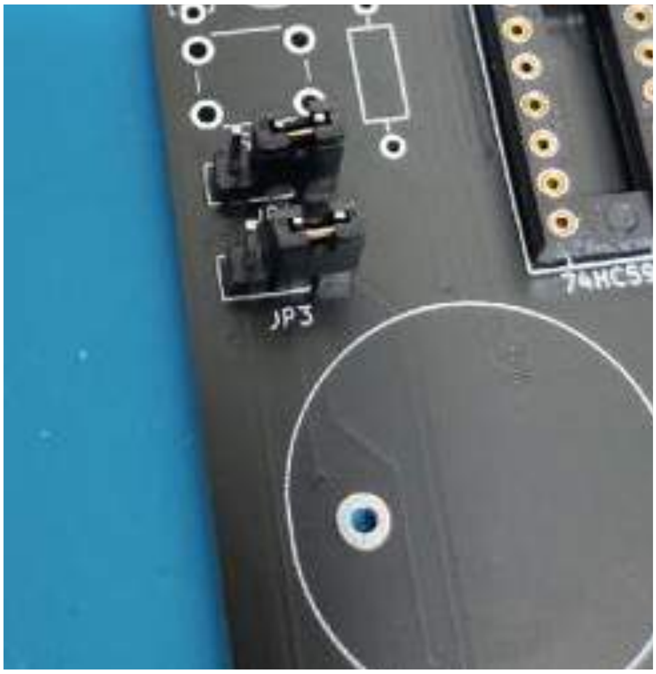

The footprint, referenced Y1, shows how it should be soldered. There is no direction this time for this component.


After soldering the two legs, apply some solder between the oscillator case and the footprint on the top of the PCB. It will guarantee that the case is connected to *GND*.

# 14 Jumpers

The board presents 4 jumpers. The jumpers JP3 and JP4 are three-pin wide and are **mandatory**, they are responsible for enabling or disabling the interrupt signals from the video board connector to the Zilog Z80 PIO. They must be soldered and connected as follows:

Putting the caps on the right side wil disabled the video interrupts.

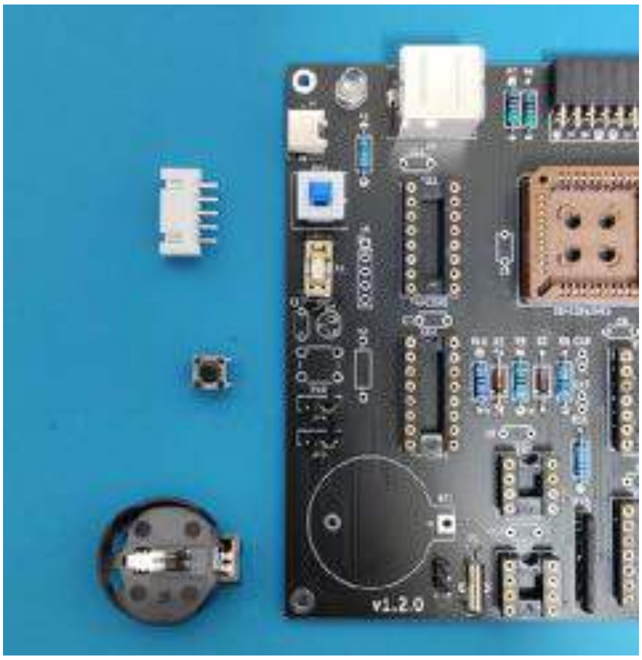




The two other jumpers are two-pin wide, referenced JP2 and JP1. The first one is used to catch the square signal from the RTC chip (check the documentation for more info), and the second one is used to enable the 512KB range for the NOR Flash (when installed) instead of the default 256KB range. They can be omitted if you are not going to use any of these features.

# 15 Miscellaneous

## Internal I2C connector, reset switch and battery connector



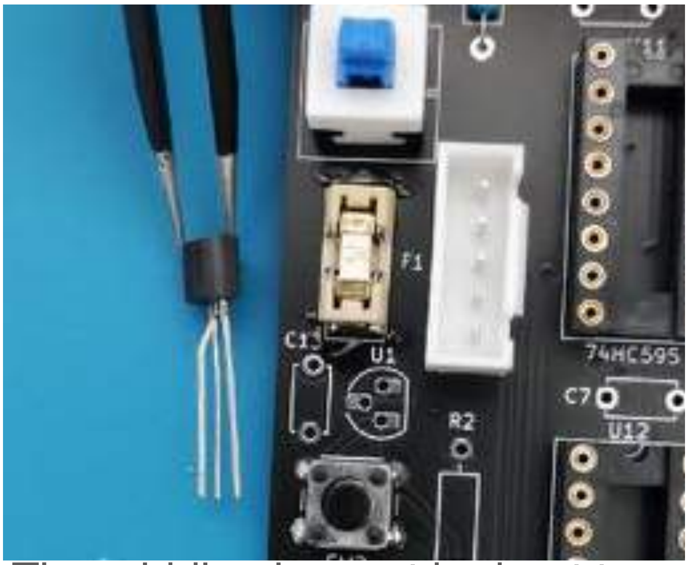

The internal I2C connector, a standard XH 2.54mm connector, should be soldered with the lock on the right.

The reset switch can only be plugged in a single direction. For the battery connector, follow the footprint on the board.

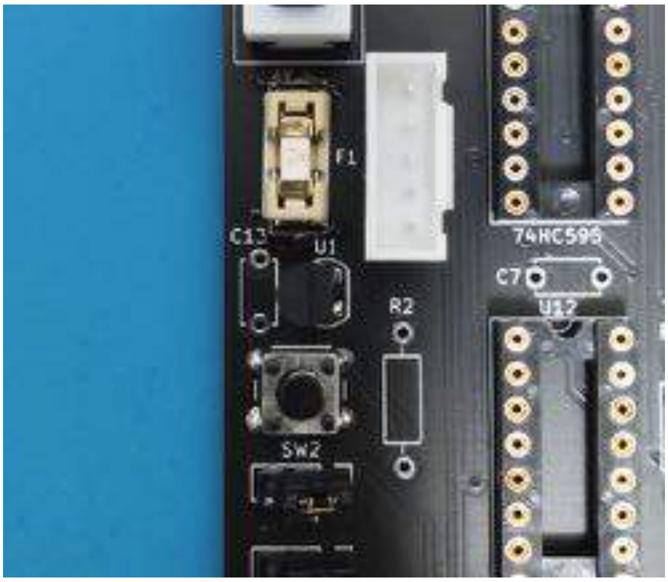
# 16

# Reset supervisor and second test!

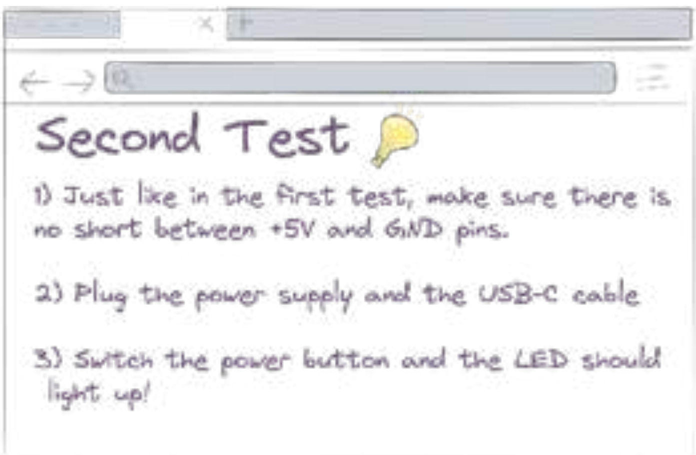
The reset supervisor component, referenced U1 is the only integrated circuit that doesn't sit on a socket. Its package resembles the very common NPN transistors', the TO-92 package. It is responsible for generating a reset signal when the power is connected to the board or when the voltage drops suddenly.

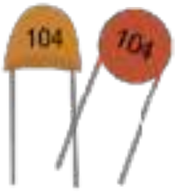



The middle pin must be bent to the back (the front being the flat surface), just like the picture above




After soldering it, you should get something like the following:




# 17&18

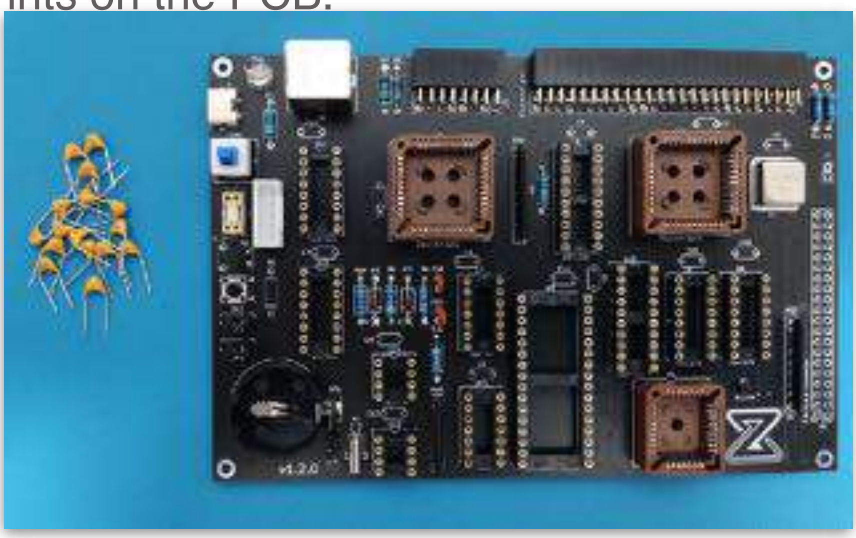
## PS/2-related capacitors
## Decoupling Capacitor

The PS/2 protocol decoding requires two 100nF ceramic capacitors, which are the two rounded orange ones provided in the kit.

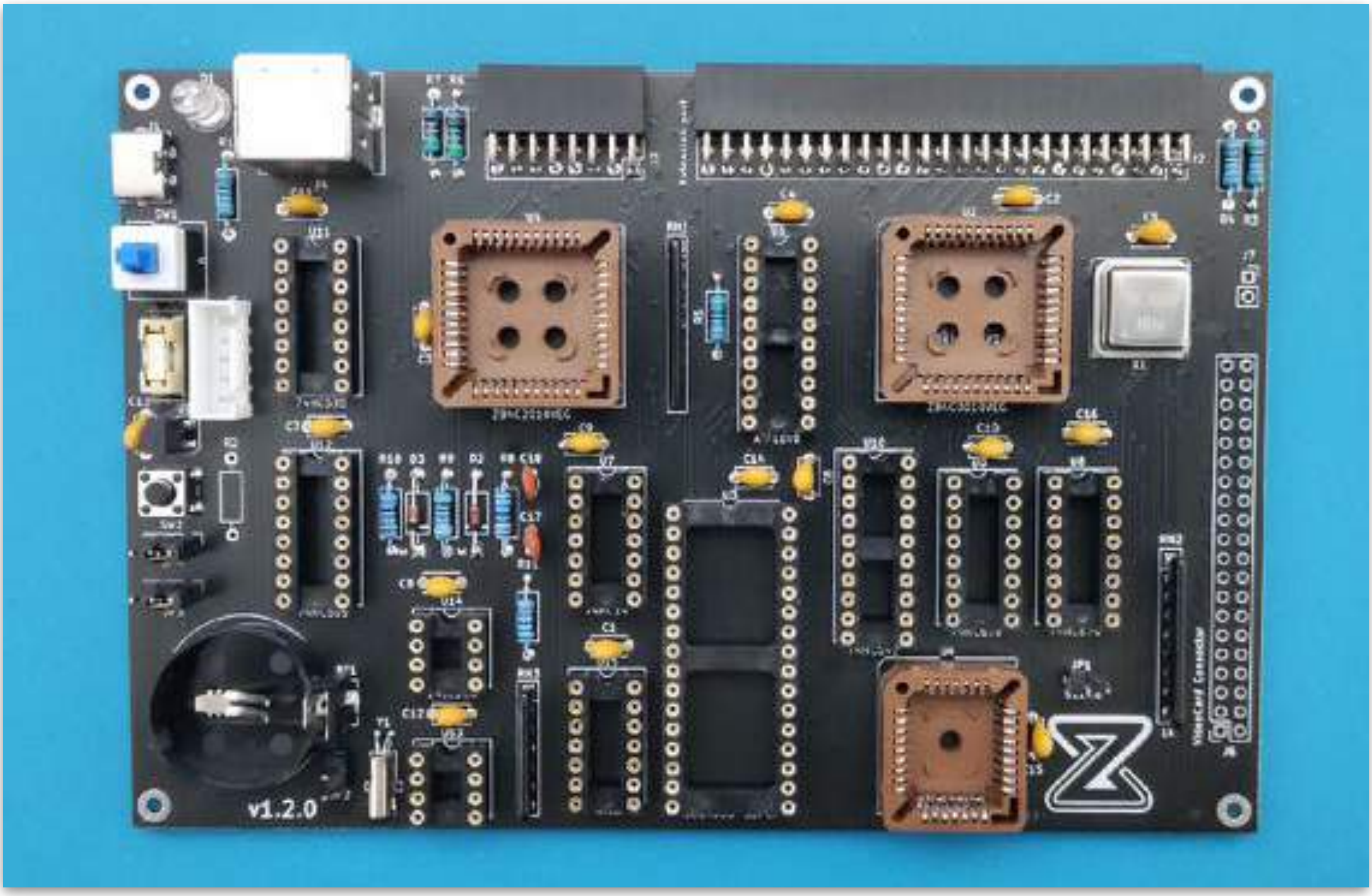
They must be soldered in footprints marked C17 and C18. They can be put in not particular direction.

The last soldering step is to solder all the decoupling capacitors, which are also 100nF ones. They are marked with Cxx footprints on the PCB.

There are quite a few of them so take your time and make sure to cut their legs after finishing soldering them.

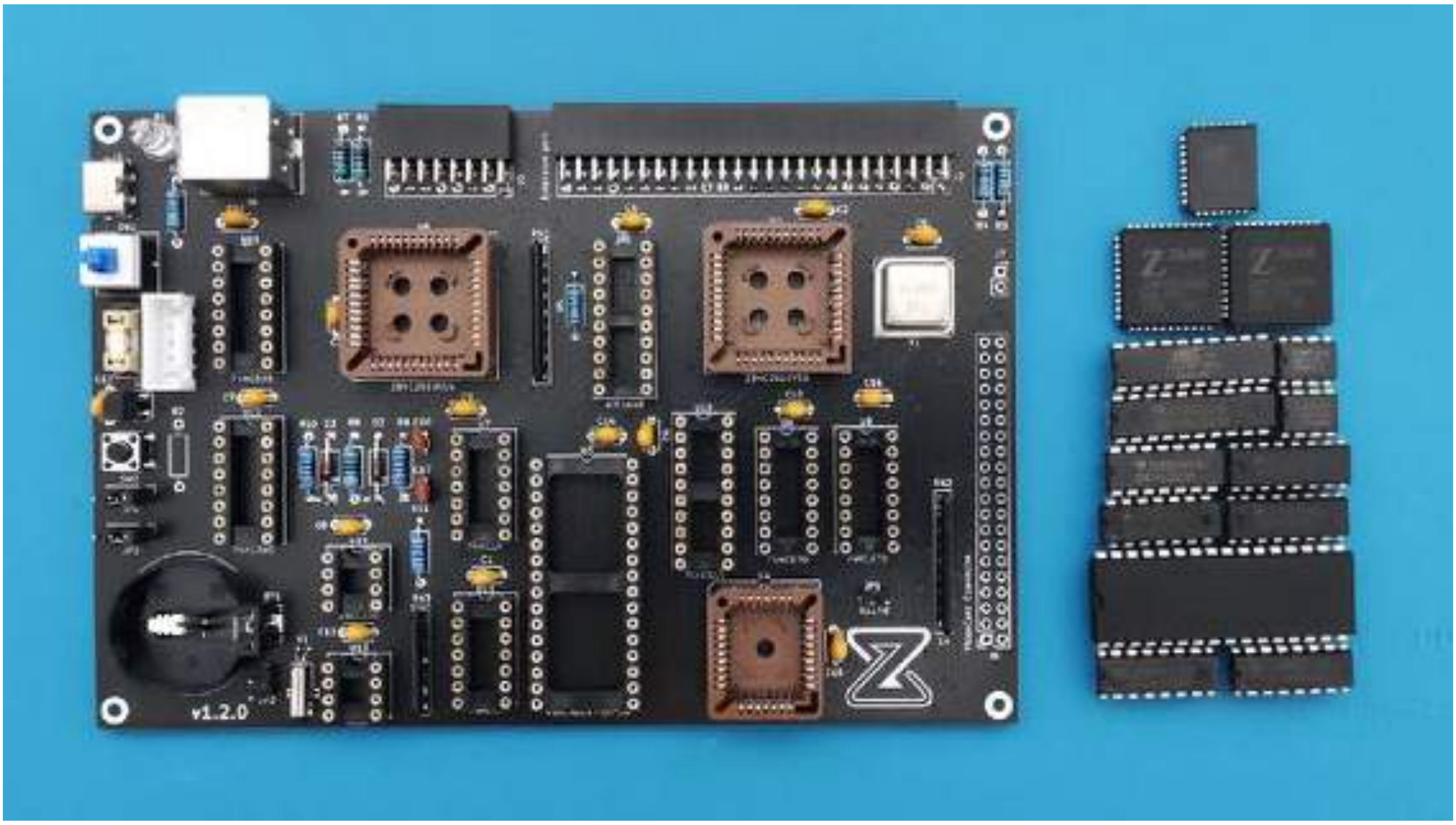


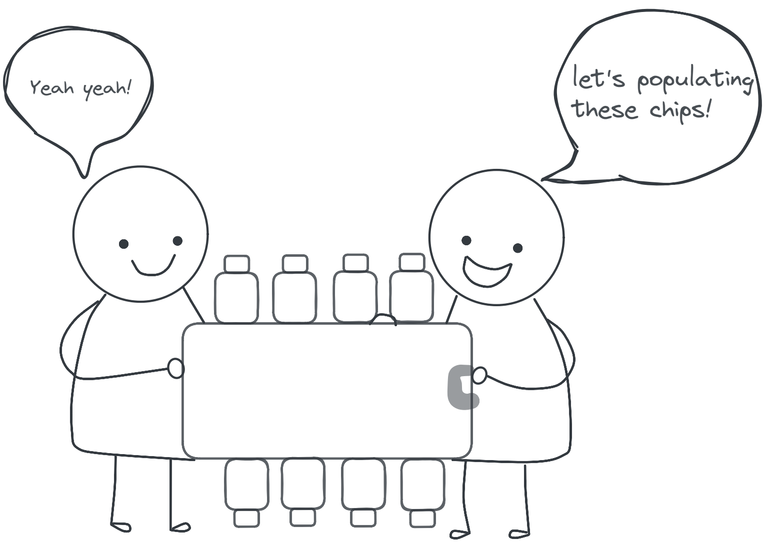
# End result after soldering



# Populating!

Now you can test the board like we did twice previously,
if there is no short and the LED is lighting up,
you can start populating the board with the integrated circuits!



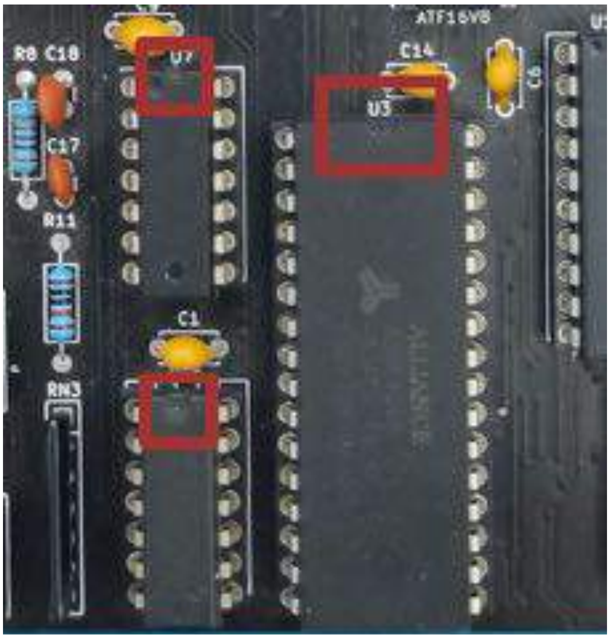



# Populating!

It's now time to put the components in their respective socket!

This part is much easier than the soldering one, there are two important things to remember when you are going to plug in the components:


1. Make sure the components are put in the right direction. For the PLCC components, the squared ones, there is a dot on the top, that shall be placed as close as possible to the socket's dot, located on the top left corner. The following picture takes the Z80 CPU as an example:



For the DIP components, there is a small **u** at the top that shall match the **u** marking present on both the socket and the PCB silkscreen, as shown on the picture on the left:

When inserting the components, mainly the DIP ones, **do not force** if only few pins entered the socket, else you can end up when bent pins, or worse, broken pins!

# Populating!

To straighten the pins, you can make the use of a pin straightener, such device can also be 3D printed. If you don't have such tool, you can put your component on its side on a flat surface and push a bit to make sure all the pins are bent at the same time. Again, do not force too much or it would result in bent or broken pins.


To know which components must be placed in which socket, you can refer to the names at the bottom of the sockets on the PCB, or, the component list present at the beginning on this guide.

After placing all the components, you can continue to the: http://zeal8bit.com/getting-started/

# Troubleshooting Tips & Additional Resources

### Cold Joints
Joints that appear dull or grainy may indicate insufficient heat during soldering. Reheat and apply additional solder if needed.

### Solder Bridges
If solder forms a bridge between two connections, use desoldering wick or a desoldering pump to remove the excess solder.

### Additional Resources
For more detailed soldering techniques or troubleshooting, refer to the: zeal8bit.com

# FAQ

## Why do we need to test the board that many times while soldering?

The idea is to be sure that the board works and that there is no short-circuit. If a short appears later, it will be easy to discriminate which components could potentially be the problem.

## After soldering the capacitors, I have a short-circuit between GND and Vcc! What should I do?

First of all, make sure that there is no solder bridge on the board. If no short was present before soldering the capacitors, then one of the capacitor may be bad, try to find it and desolder it. This can be done with the multimeter in resistor mode, the closer you are to the short, the less the resistance should be.
Do **NOT** try to power on the board if a short is present.

## I powered on the board, LED turn on for a split second and turned off directly, why?

The first suspect is the fuse, use a multimeter in continuity mode to check if it blew up, if this is the case, it means that there is a short-circuit on the board. You will have to fix it first (check the answer above) and then replace the fuse. Do **NOT** bypass the fuse since its role is to protect the integrated circuits. By omitting it, all the components on the board would get damaged in case of a short-circuit.

## I soldered the board, and placed all the components, what should I do now?

Congratulations for this! The PLD and the ROM are pre-flashed already, so you can proceed to the regular procedure that you can find here: https://zeal8bit.com/getting-started.

# FAQ

**When I power on the board, nothing appears on the UART, why?**

This means that the something is wrong with the integrated circuits, make sure they are all inserted in the right direction (check the ICs top tab), they are in the right socket, which should be soldered properly.

**When I type a key on the keyboard, multiple characters appear instead of one, why?**

The PS/2 protocol decoding is done with discrete logic and filters. The filters are made with the two 100nF rounded ceramic capacitors: **C17** and **C18**. Capacitors value is not something very precise, in some cases, it is off, so the solution is to replace one or both capacitors. The kit contains a few extra capacitors

**Buy it now:**
https://www.tindie.com/stores/zeal8bit/

**Contact:**
contact@zeal8bit.com

**Resources:**
zeal8bit.com
https://github.com/Zeal8bit

**Join the community:**
Youtube: https://www.youtube.com/@zeal8bit
Twitter: https://twitter.com/zeal8bit
Discord: https://discord.gg/CtyGsSkzDY